GOLDENSOFTWARE

Surfer®

Powerful contouring, gridding & surface mapping system

User's Guide

Surfer® Registration Information

Your **Surfer** product key is located in the download instructions email and in your account at MyAccount.GoldenSoftware.com.

Register your **Surfer** product key online at www.GoldenSoftware.com. This information will not be redistributed.

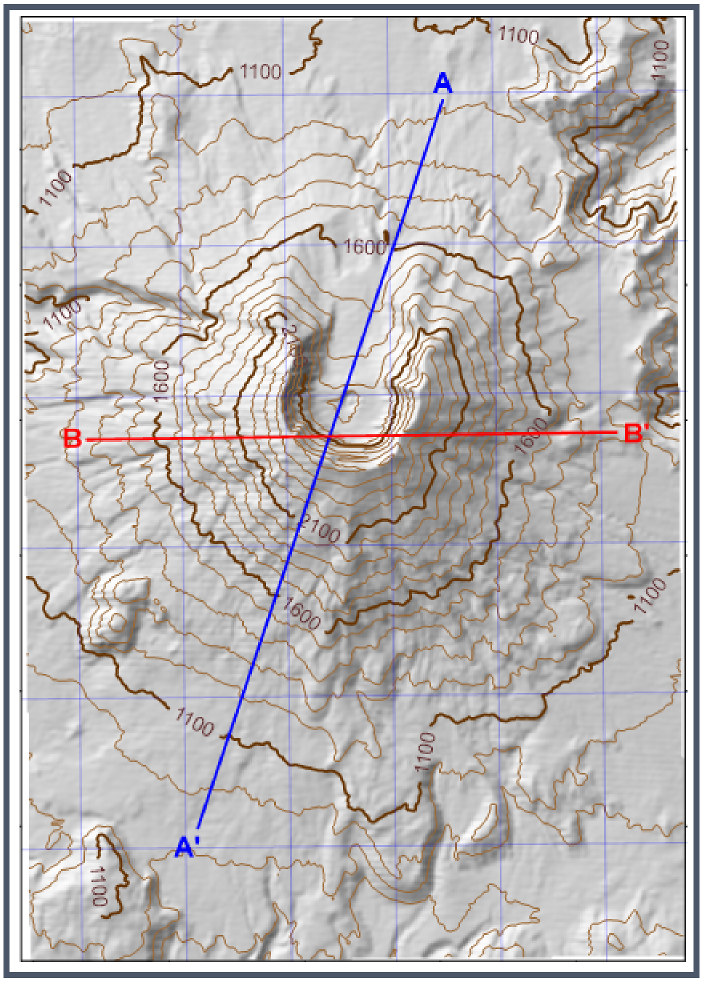
Registration entitles you to free technical support, download access in your account, and updates from Golden Software.

For future reference, write your product key on the line below:

____

Surfer® User's Guide

Contouring and 3D Surface Mapping
for Scientists and Engineers



Golden Software, LLC
1301 Arapahoe St Suite 105, Golden, Colorado 80401, U.S.A.
Phone: 303-279-1021
www.GoldenSoftware.com

COPYRIGHT NOTICE

September 2022

Contents

Chapter 1 - Introduction

Introduction to Surfer

Welcome to **Surfer**, a powerful contouring, gridding, and surface mapping package for scientists, engineers, educators, or anyone who needs to generate maps quickly and easily. Producing publication quality maps has never been quicker or easier. Adding multiple map layers and objects, customizing the map display, and annotating with text creates attractive and informative maps. Virtually all aspects of your maps can be customized to produce the exact presentation you want.

Surfer is a grid-based mapping program that interpolates irregularly spaced XYZ data into a regularly spaced grid. Grids may also be imported from other sources, such as the United States Geological Survey (USGS). The grid is used to produce different types of maps including contour, color relief, and 3D surface maps among others. Many gridding and mapping options are available allowing you to produce the map that best represents your data.

An extensive suite of gridding methods is available in **Surfer**. The variety of available methods provides different interpretations of your data, and allows you to choose the most appropriate method for your needs. In addition, data metrics allow you to map statistical information about your gridded data. Surface area, projected planar area, and volumetric calculations can be performed quickly in **Surfer**. Cross-sectional profiles can also be computed and exported.

The grid files can be edited, combined, filtered, sliced, queried, and mathematically transformed. For example, grids can be sliced to create cross-sectional profiles, or the **Grids | Calculate | Isopach** command can be used to create an isopach map from two grid files. Grids can be edited with an intuitive user interface in the grid editor.

Scripter

The **Scripter ™** program, included with **Surfer**, is useful for creating, editing, and running script files. A script is a text file containing a series of instructions for execution when the script is run that automates **Surfer** procedures. By writing and running script files, simple mundane tasks or complex system integration tasks can be performed precisely and repetitively without direct interaction. **Surfer** also supports ActiveX Automation using any compatible client, such as Visual BASIC. These two automation capabilities allow **Surfer** to be used as a data visualization and map generation post-processor for any scientific modeling system.

New Features

The new features in **Surfer** are summarized:

- Online at www.GoldenSoftware.com/products/surfer
- Online at the *What's New in Surfer* Knowledge Base article

Who Uses Surfer?

People from many different disciplines use **Surfer**. Since 1984, over 100,000 scientists and engineers worldwide have discovered **Surfer's** power and simplicity. **Surfer's** outstanding gridding and contouring capabilities have made **Surfer** the software of choice for working with XYZ data. Over the years, **Surfer** users have included hydrologists, engineers, geologists, archeologists, oceanographers, biologists, foresters, geophysicists, medical researchers, climatologists, educators, students, and more! Anyone wanting to visualize their XYZ data with striking clarity and accuracy will benefit from **Surfer's** powerful features!

System Requirements

The system requirements for **Surfer** are:

- Windows 7 SP1, 8 (excluding RT), 10 or higher
- 32-bit or 64-bit operation system support
- 1024x768 or higher monitor resolution with a minimum 16-bit color depth
- At least 500MB free hard disk space
- At least 512MB RAM for simple data sets, 1GB RAM recommended
- 3D View requirements:
    - Graphics card with OpenGL v3.2 or later, including the following support:
        - Vertex shaders
        - Fragment shaders
        - Geometry shaders
        - GLSL (GL Shading Language) 150
    - Graphics emulators, such as VMs and Parallels for Mac, may not support all the required features for viewing data in the 3D view

Fine Tuning Surfer Performance

- Issues with 3D View features may be corrected by upgrading to the latest graphics drivers.
- Many heavily computational operations, including gridding and contouring, are multi-threaded and processor reliant. A faster processor will improve **Surfer's** performance.
- For processing very large data files, such as LiDAR or some vector data files, fast and large RAM storage capacity is recommended.
- Click the **File | Options** command and on the **General** page, set the *Max number of processors* to use all processor cores.
- To improve performance, click the **File | Options** command, and, on the **General** page, uncheck the check box for *Save auto recovery information*.

Installation Directions

Golden Software recommends installing only the latest version of **Surfer**.

Installing **Surfer** requires Administrator rights. Either an administrator account can be used to install **Surfer** or the administrator's credentials can be entered before installation while logged in to a standard user account. If you wish to use a **Surfer** single-user license, the product key must be activated while logged in to the account under which **Surfer** will be used. For this reason, we recommend logging into Windows under the account for the Surfer user, and entering the necessary administrator credentials when prompted. Golden Software does not recommend installing the current version of **Surfer** in the same location as any previous versions of **Surfer**.

To install **Surfer** from a download:

- Log into Windows under the account for the individual who is licensed to use **Surfer**.
- Download **Surfer** according to the emailed directions you received.
- Double-click on the downloaded file to begin the installation process.
- Once the installation is complete, run **Surfer**.
- License **Surfer** by activating a single-user license product key or connecting to a license server.

Updating Surfer

To update your version of **Surfer**, open the **Surfer** program and choose the **File | Online | Check for Update** command. This will launch the Internet Update program which will check Golden Software's servers for any updates. If there is an update for your version of **Surfer**, you will be prompted to download and install the update.

You can also email your registered **Surfer** product key to *surfersupport@goldensoftware.com* and request to download the full product update. See the *Check for Update* topic in the help for additional information.

Uninstalling Surfer

To uninstall **Surfer**, follow the directions below for your specific operating system.

Windows 7

To uninstall **Surfer** go to the Windows Control Panel and click the *Uninstall a program* link. Select **Surfer** from the list of installed applications. Click the *Uninstall* button to uninstall **Surfer**.

Windows 8

From the *Start* screen, right-click the **Surfer** tile and click the *Uninstall* button at the bottom of the screen. Alternatively, right-click anywhere on the *Start* screen and click *All apps* at the bottom of the screen. Right-click the **Surfer** tile and click *Uninstall* at the bottom of the screen.

Windows 10

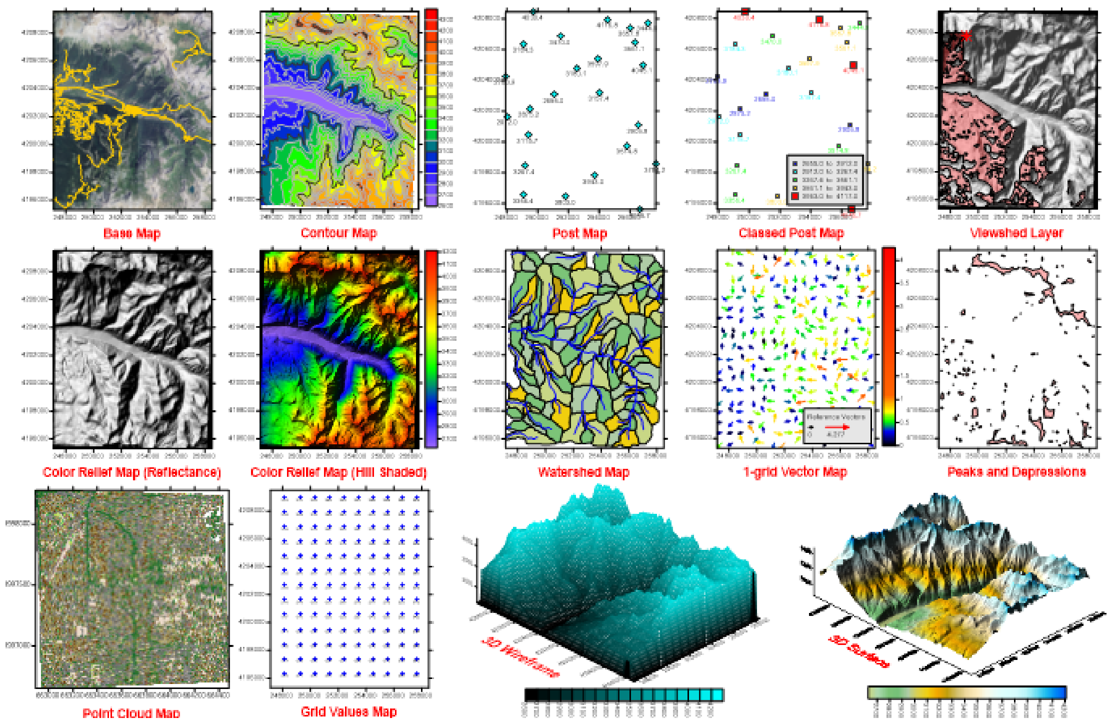
Select *Settings* in the **Start** menu. In *Settings*, select **System | Apps & features**. Select **Surfer** and then click *Uninstall*. To uninstall **Surfer** from the Windows Control Panel, click **Programs | Programs and Features**. Select **Surfer** and click *Uninstall*.

Surfer Trial Functionality

The **Surfer** trial is a fully functioning time-limited trial. This means that commands work exactly as in the full program for the duration of the trial. The trial has no further restrictions on use. The trial can be installed on any computer that meets the system requirements. The trial version can be licensed by activating a product key or connecting to a license server.

Three-Minute Tour

We have included several sample files with **Surfer** so that you can quickly see the variety of **Surfer's** capabilities. Only a few files are discussed here, and these examples do not include all of **Surfer**'s many map types and features. The **Contents** window is a good source of information as to what is included in each **Surfer** file. The different types of maps that can be created is found in the program help in the *Map Types* topic.

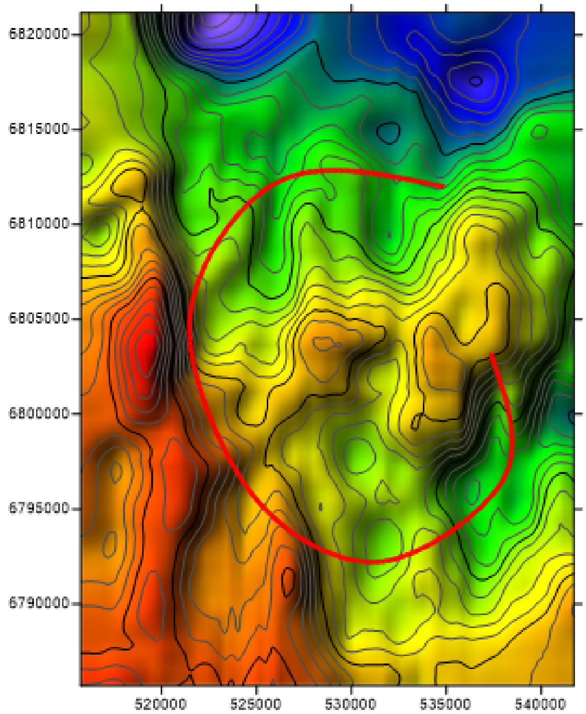


***Surfer**, a powerful contouring, gridding, and surface mapping package, produces publication quality maps. Virtually all aspects of your maps can be customized to produce the exact presentation you want.*

To access the example files from your computer:

1. Open **Surfer**.
2. Click the **File | Open** command.
3. In the **Open** dialog, navigate to the **Surfer** Samples folder located in C:\Program Files\Golden Software\Surfer\Samples by default.
4. Select the sample .SRF file of interest and click *Open*. The sample file is now displayed. Repeat as necessary to see the files of interest.

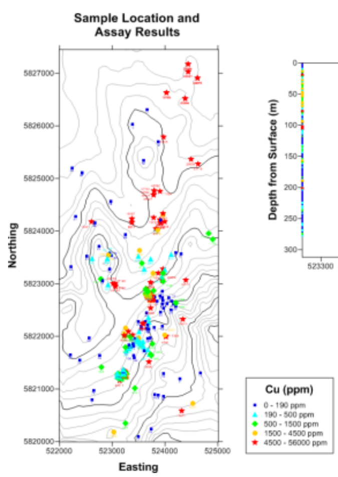
Examples of Surfer Capabilities

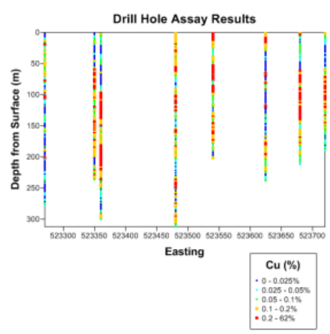


View Data in 2D and 3D

The *3DView.SRF* sample file includes contour and color relief layers, as well as a base (vector) layer that is used for a 3D view fly-through.

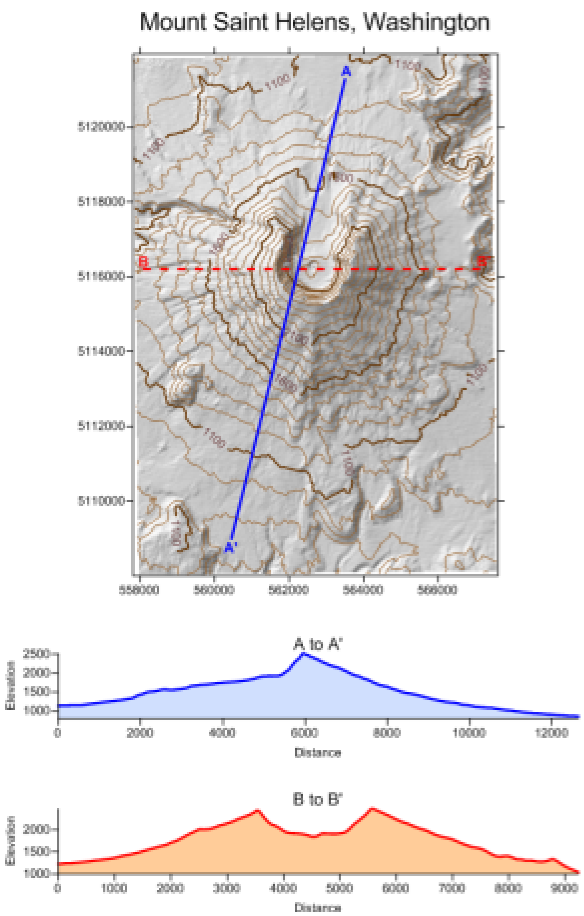
To view the 3D and fly-through, open the *3DView.SRF* file and select the map in the **Contents** window. Next, click **Map Tools | View | 3D View** to open a 3D view. Click **3D View | Fly-Through | Play** to view the example fly-through.





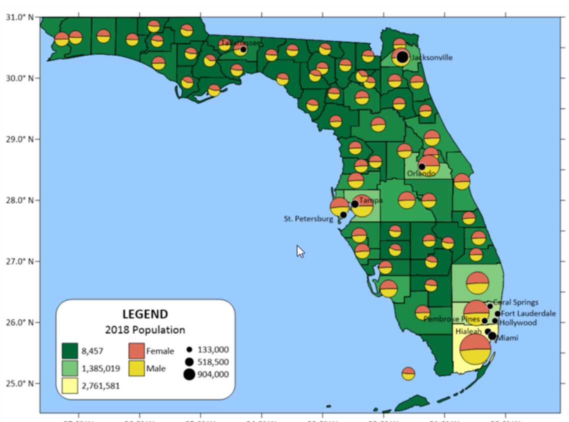
Present Scientific Research

The *Classed Post.SRF* sample file displays two maps. The left map is a contour map with a classed post map layer displaying the location of the sample of copper in parts per million and assay results over a study area. The right map is a classed post map that displays the drill hole assay results by comparing the depth from surface to the Easting. A classed post map legend has been added to each map.



Display Complex Spatial Data

The *Profile.SRF* file contains a map with two base map layers, a contour layer, and a shaded relief layer. The base maps were created with the **Map Tools | Add to Map | Profile** command. At the bottom of the page, the A and B profile lines are displayed, showing two elevation profiles across the Mount St. Helens map.



Layer Multiple Types of Data

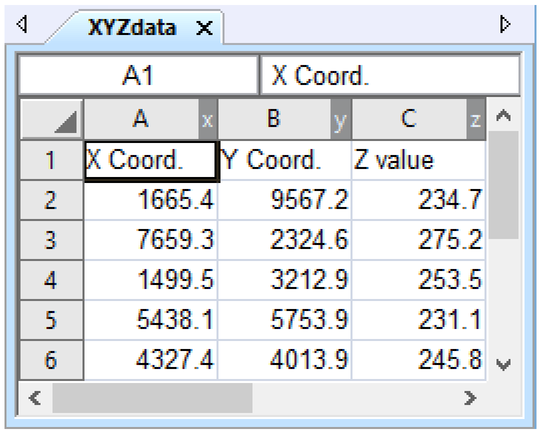
The *BaseSymbology (PieChart).SRF* sample file was created from one post layer and two base layers. The post layer displays circular symbols relatively sized according to population count in various cities throughout Florida. The base layer depicts population in all the counties in Florida using Unclassed Colors symbology. The Pies base layer uses Pie Chart symbology to depict the proportion of women to men in each county.

Create a Grid from XYZ Data

The most common application of **Surfer** is to create a grid-based map from an XYZ data file. An XYZ data file has X data, Y data, and Z data delimited into separate columns (for example, longitude, latitude, and elevation). The **Grid Data** command uses an XYZ data file to produce a grid file. The grid file is then used by most of the **Home | New Map** commands to produce maps. Post maps, base maps, point cloud, and drillhole maps do not use grid files.

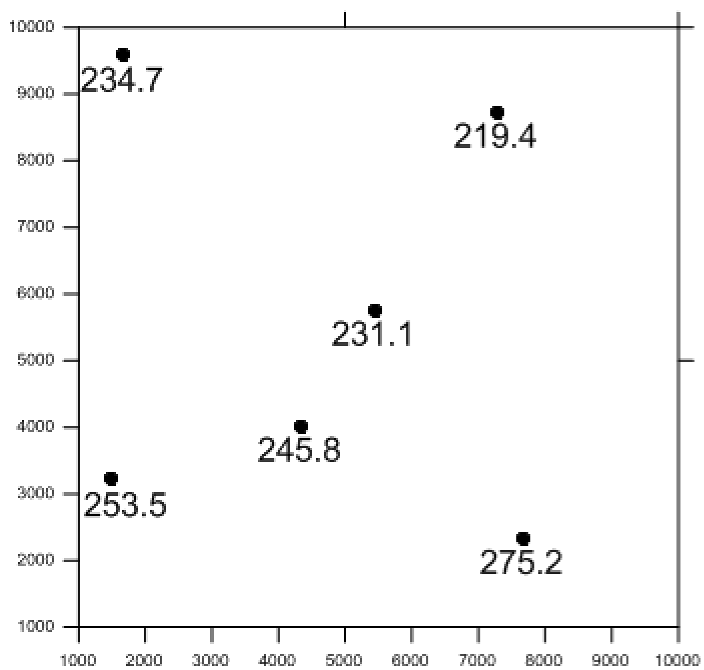
The general steps to progress from an XYZ data set to a finished grid-based map are as follows:

1. Create an XYZ data file. This file can be created in a **Surfer** worksheet window or outside of **Surfer** (using an ASCII text editor or Microsoft Excel, for example).



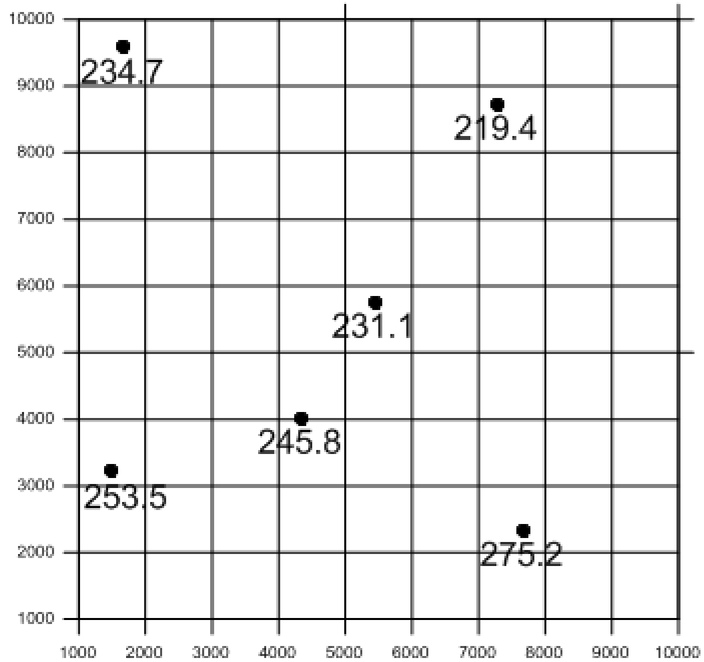
Start with irregular XYZ data in three columns.

2. To display the data points, click the **Home | New Map | Post** command.



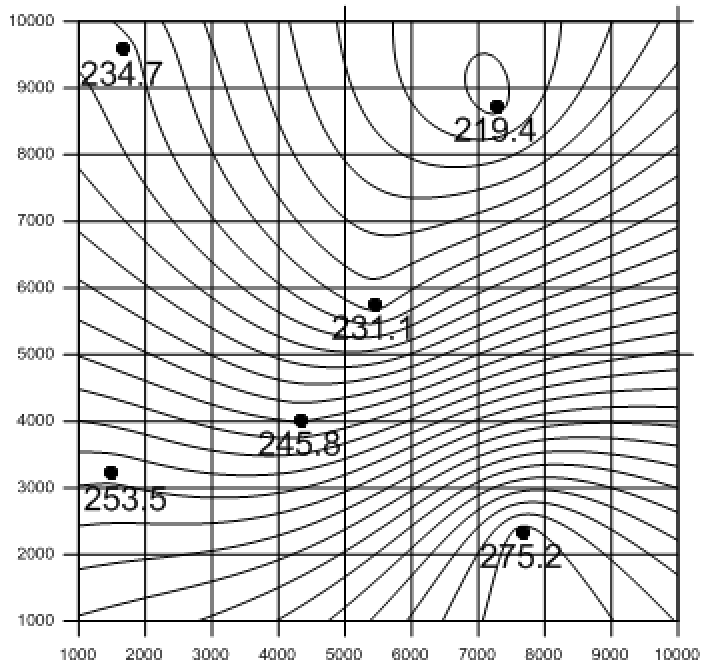
A post map displays the original XYZ data locations.

3. Create a grid file .GRD from the XYZ data file using the **Home | Grid Data | Grid Data** command.



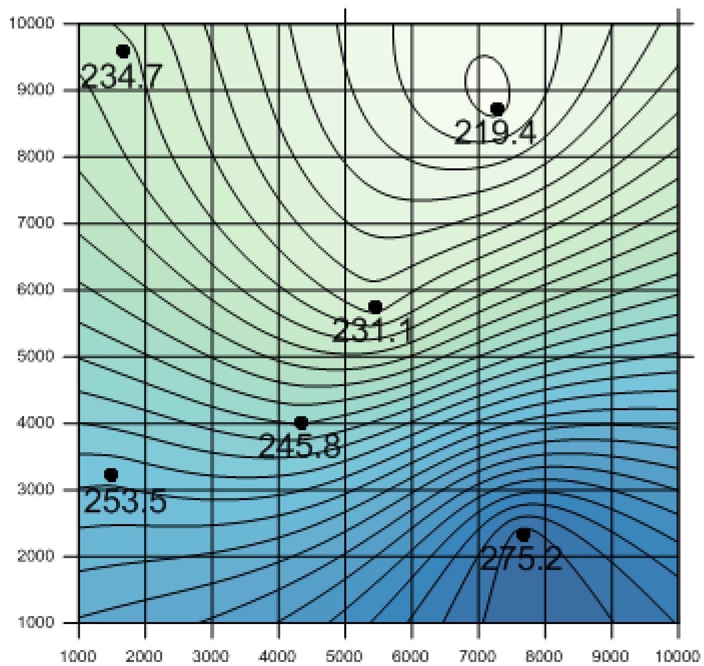
Gridding interpolates a Z value at the intersection of each row and column in the grid file. This fills the holes in the data. Here the rows and columns are represented by grid lines.

4. To create a map, select the map type from the **Home | New Map** commands. Select the grid file from step three. New grid-based maps that can be created include contour, 3D surface, 3D wireframe, color relief, peaks and depressions, 1-grid or 2-grid vector, watershed, and grid values maps.



The post map layer shows the original data points. The contour map layer shows the grid based contour map.

5. Click on the map to display the map properties in the **Properties** window where you can customize the map to fit your needs.

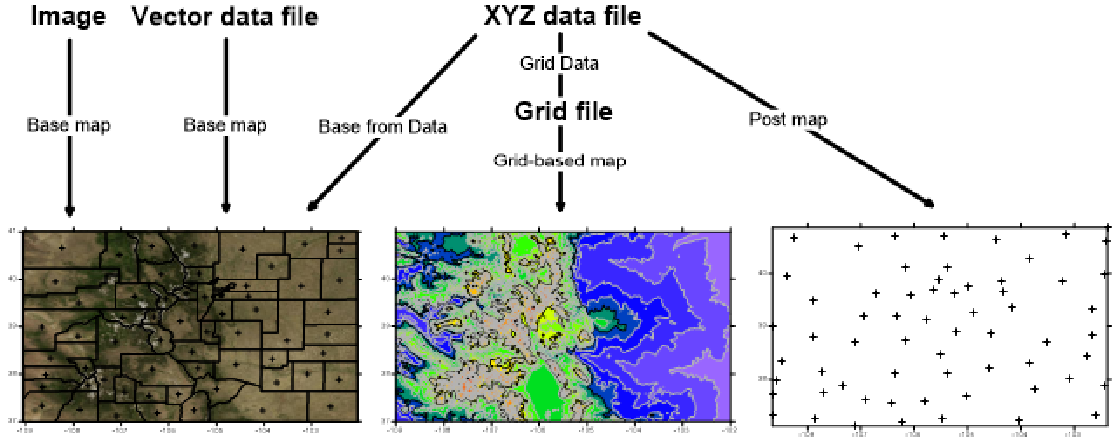


The contour map layer is filled with a gradational color fill.

6. Click the **File | Save** command to save the project as a **Surfer** .SRF file which contains all the information needed to recreate the map.

Surfer Flow Chart of Data and Maps

This flow chart illustrates the relationship between XYZ data files, grid files, vector files, image files, and various maps. This example displays only one of the grid based maps, a contour map.



This flow chart illustrates the relationship between different data files and different map types.

Sample Files

Sample files are included in **Surfer**'s Samples directory. The samples include interesting maps, data files, and scripts that can be used to automate **Surfer**. The sample files are located in: C:\Program Files\Golden Software\Surfer\Samples.

Using Scripter For Automation

Tasks can be automated in **Surfer** using Golden Software's **Scripter** program or any ActiveX Automation-compatible client, such as Visual BASIC. A script is a text file containing a series of instructions for execution when the script is run. **Scripter** can be used to perform almost any task in Surfer. Scripts are useful for automating repetitive tasks and consolidating a sequence of steps. **Scripter** is installed in the same location as Surfer. Refer to the *Surfer Automation* topic in the help for more information about **Scripter**. We have included several example scripts so that you can quickly see some of **Scripter's** capabilities.

To run a sample script file:

1. Open **Scripter** by navigating to the installation folder, C:\Program Files\Golden Software\Surfer\Scripter. If you are running a 32-bit version of **Surfer** on a 64-bit version of Windows, navigate to C:\Program Files (x86)\Golden Software\Surfer\Scripter. Right-click on the *Scripter.exe* application file and select **Run as administrator**.

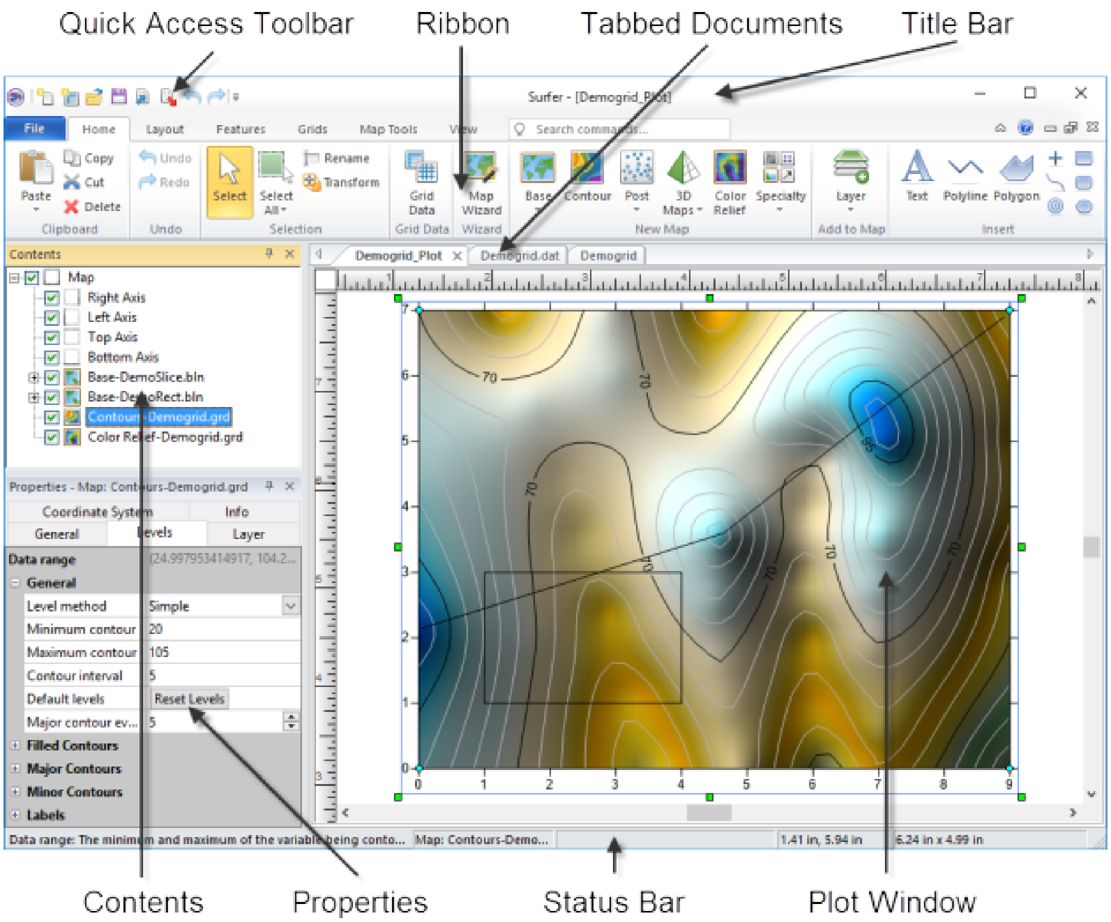
2. Choose the **File | Open** command.

3. Select a sample script .BAS file. These are located in the C:\Program Files\Golden Software\Surfer\Samples\Scripts folder or, if you are running a 32-bit version of Surfer on a 64-bit version of Windows, the C:\Program Files (x86)\Golden Software\Surfer\Samples\Scripts folder.

4. Click the **Script | Run** command and the script is executed. Most sample scripts open **Surfer** and display a map in the plot window.

Surfer User Interface

Surfer contains four document window types: the plot document, worksheet document, 3D view, and grid editor. Maps are created and displayed in the plot document and 3D view. The worksheet document displays, edits, transforms, and saves data in a tabular format. The grid editor displays and edits Z values for the grid with various editing tools.



*This is the **Surfer** plot window with the **Contents** and **Properties** windows on the left and the worksheet and grid editor tabs on the top of the horizontal ruler.*

Surfer Layout

The following table summarizes the function of each component of the **Surfer** layout.

Component Name	Component Function
Title Bar	The title bar lists the program name plus the saved Surfer .SRF file name (if any). An asterisk after the file name indicates the file has been modified.
Quick Access Toolbar	All window types in **Surfer** include the quick access toolbar to the left of the title bar. The quick access toolbar contains buttons for many common commands. The quick access toolbar can be customized to add or remove buttons with the **Customize Ribbon** command.

Ribbon	The ribbon includes all of the commands in **Surfer**. Commands are grouped under the File menu and various tabs. Some commands and tabs are only available in specific views. For example, the **Features | Insert | Polyline** command is only available in the plot window. The ribbon commands can be modified and rearranged with the **Customize Ribbon** command. On the upper right side of the ribbon is a flag icon that will display badges when there are user-specific notifications to be read.
Tabbed Documents	The plot, 3D view, worksheet, and grid editor windows are displayed as tabbed documents. The tabs may be reordered by clicking and dragging. When more than one window is open, tabs appear at the top of the document, allowing you to click on a tab to switch to a different window. When a document contains unsaved changes, an asterisk (*) appears next to its tabbed name.
Contents	The **Contents** window contains a hierarchical list of all the objects in a Surfer plot document, grid editor, or 3D view window displayed in a tree view. The objects can be selected, added, arranged, or edited. Changes made in the **Contents** window are reflected in the plot document, grid editor, or 3D view and vice versa. The **Contents** window is initially docked at the left side of the window.
Properties	The **Properties** window contains all of the properties for the selected object or objects. Changes made in the **Properties** window are reflected in the plot document, grid editor, or 3D view. The properties in the **Properties** window are grouped by page. The **Properties** window is initially docked below the **Contents** window.
Status Bar	The status bar displays information about the current command or activity in **Surfer**. The status bar is divided into five sections. The sections display basic plot commands and descriptions, the name of the selected object, the cursor map coordinates and units, the cursor page coordinates, and the dimensions of the selected object.

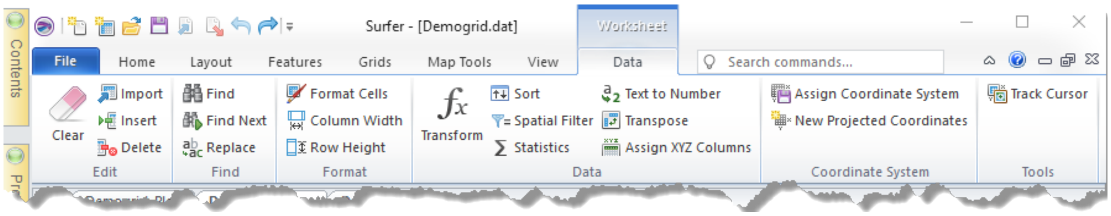
Opening Windows

Selecting the **File | Open** command opens grid files and data files as maps. The **File | New | Plot** command creates a new plot window. The **File | New | Worksheet** command creates a new worksheet window. The **Map Tools | View | 3D View** command opens a 3D view of the selected map. The **Grids | Editor | Grid Editor** command opens a grid in the grid editor.

Ribbon

The ribbon is the strip of buttons and icons located above the manager and view windows. The ribbon replaces the menus and toolbars found in earlier versions of **Surfer**. The ribbon is designed to help you quickly find the commands that you need to complete a task.

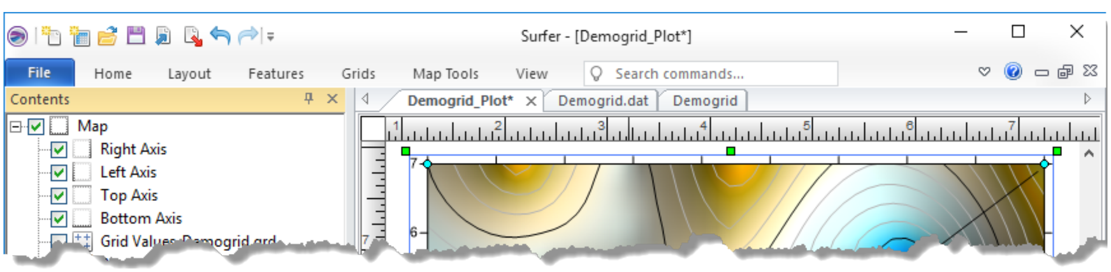
Above the ribbon are a number of tabs, such as **Home**, **Features**, and **Map Tools**. Clicking or scrolling to a tab displays the commands located in this section of the ribbon. The tabs have commands that are organized into a group. For instance, all the commands for adding drawn objects are on the **Features** tab in the **Insert** group.



The Ribbon is displayed with the ***Data*** *tab selected.*

Minimizing the Ribbon

The ribbon can be minimized to take up less space on the screen. To minimize the ribbon, right-click on the ribbon and select **Minimize the Ribbon** or click the button in the top right portion of the **Surfer** window. When displayed in a minimized mode, only the tabs at the top of the screen are visible. To see the commands on each tab, click the tab name. After selecting a command, the ribbon automatically minimizes again. Double-click any tab name to quickly minimize or maximize the ribbon.



The Ribbon displayed with the Minimize the Ribbon *option selected. Clicking any tab name displays the ribbon.*

Customizing the Ribbon

The ribbon is customizable in **Surfer**. To customize the commands in the ribbon, right-click on the ribbon and select **Customize the Ribbon**.

In the **Customize Ribbon** dialog, you can add new tabs, add groups, hide existing tabs or custom groups, and add commands to any custom group. You can also rearrange the tabs into an order that fits your needs better.

To customize the commands in the **Customize Ribbon** dialog, right-click on the ribbon and select **Customize the Ribbon**. In the **Customize Ribbon** dialog, use the following options.

Tab options

- To add a custom tab, set the *Customize the Ribbon* section to *All Tabs*. Click in the list on the right side of the dialog where the custom tab should be located and click the *New Tab* button.
- To delete custom tab, right-click on the tab name in the list on the right side of the dialog and select **Delete**.
- To rename a default or custom tab, click on the tab name in the list on the right side of the dialog. Click the *Rename* button. Type the new name and press OK to make the change.
- To hide a default or custom tab, uncheck the box next to the tab name on the right side of the dialog. Only checked tabs will be displayed.
- To change the order of default or custom tabs, click on the tab name that should be moved in the list on the right side of the dialog. Click the up and down arrow buttons on the far right side of the dialog to move the selected tab up or down. Default tabs must remain in their major group.

Group options

- To add a custom group to a default or custom tab, click on the ⊞ next to the tab name. Click in the list of group names where the new group should be located and click the *New Group* button.
- To delete a default or custom group on any tab, right-click on the group name in the list on the right side of the dialog and select **Delete**.
- To rename a default or custom group on any tab, click on the group name in the list on the right side of the dialog. Click the *Rename* button. Type the new name and click OK to make the change.
- To change the order of default or custom groups on any tab, click on the group name that should be moved in the list on the right side of the dialog. Click the up and down arrow buttons on the far right side of the dialog to move the selected group up or down in the list.

- To replace a default group with a custom group, right-click on the default group name and select **Delete**. Click the *New Group* button. Add the desired commands to the new group that you want displayed. Rename the new group, if desired.

Command options

Commands can only be added to or deleted from custom groups. Commands can only be rearranged or renamed in custom groups. If you wish to edit the commands in default group, the default group should be hidden and a new custom group should be created with the same commands.

- To add a command to a custom group, set the *Choose commands from:* list to *All Tabs* so that all commands are listed on the left side of the dialog. Select the desired command that should be added. On the right side of the dialog, click the ⊞ next to the custom group name. Click on the desired position in the list of commands. If no commands exist in the group yet, click on the group name. Click the *Add>>* button and the command is added to the custom group.

- To delete a command from a custom group, right-click on the command name in the list on the right side of the dialog and select **Delete**. Only commands from custom groups can be deleted.

- To rename a command in a custom group, click on the command name in the list on the right side of the dialog. Click the *Rename* button. Type the new name and click OK to make the change. Only commands in custom groups can be renamed.

- To change the order of commands in a custom group, click on the command name that should be moved in the list on the right side of the dialog. Click the up and down arrow buttons on the far right side of the dialog to move the selected command up or down in the list.

Reset the Ribbon

To reset all customizations on the ribbon, click the *Reset* button at the bottom of the **Customize Ribbon** dialog.

Command and Help Search

The ribbon also includes a command search to the right of the last tab (**View**, **Data**, or **Grid Editor** depending on document type). Begin typing a command name to search for commands. Click on a command in the search results to use the command. Press ENTER to quickly use the top search result command. For example type *post* into the command search bar and the **Home | New Map | Post** command group, **Map Tools | Add to Map | Layer** command group, and **Map Tools | Edit Layer | Post Labels** commands are displayed in the search results. You can also click the *Search help file* at the bottom of the results list to search the help file for the search term.

The command search will return commands from all ribbon tabs. No more than five commands are displayed in the results list. A command may be disabled in the results list if the command is not applicable to the current document or selection.

Quick Access Toolbar

The quick access toolbar is at the top of the **Surfer** window. This toolbar has frequently used commands and can be customized by the user. The commands in the quick access toolbar are the same regardless of the type of window displayed in **Surfer** .

*The **Quick Access Toolbar** is displayed at the top of the **Surfer** window.*

Customizing the Quick Access Toolbar

The quick access toolbar is a customizable toolbar. One method that can be used to add commands to the quick access toolbar is to right-click on the command in the ribbon and click **Add to Quick Access Toolbar**. The command is automatically added to the end of the toolbar.

To customize the commands on the quick access toolbar, right-click on the quick access toolbar or ribbon and select **Customize Quick Access Toolbar**.

In the **Quick Access Toolbar** dialog,

1. To add a command, select the command from the list on the left that you want to add. Click the *Add>>* button and the command is added to the list on the right.

2. To add a separator between commands, set the *Choose commands from* to Main on the left side of the dialog. Select *<Separator>* and click *Add>>* .
Move the separator to the desired position.

3. To delete a command, select the command from the list on the right. Click the *<<Remove* button and the command is removed from the list on the right.

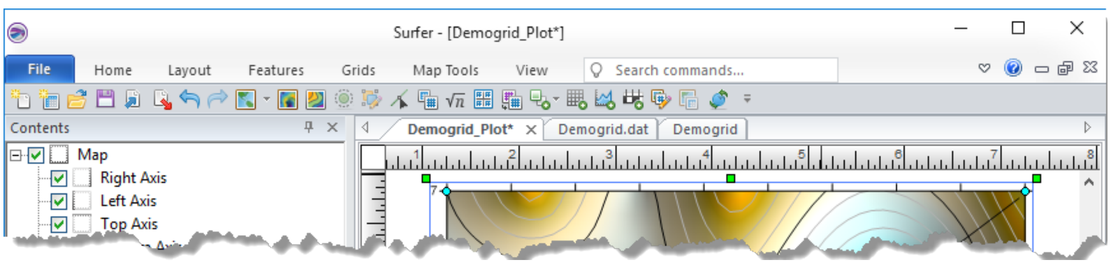
4. To rearrange commands or move separators, click on the command or separator name from the list on the right that you want to move. Click the up and down arrow buttons on the far right to move the command up or down the list. Commands are shown in the exact order that they are displayed in the Quick Access Toolbar.

5. To reset the Quick Access Toolbar to the default display, click the *Reset* button below the list on the right side of the dialog.

6. Click OK and all changes are made.

Displaying the Quick Access Toolbar Below the Ribbon

To display the quick access toolbar below the ribbon, right-click on the quick access toolbar or ribbon and click **Show Quick Access Toolbar Below the Ribbon**. This setting is useful if you have added many commands to the quick access toolbar. More commands display, by default, when the quick access toolbar is below the ribbon. When combined with the minimized ribbon appearance, this can give single click access to all your most used commands and maximize the viewing area for the plot.



Customize the Quick Access Toolbar to display all the commands you frequently use. Then, display the Quick Access Toolbar below the ribbon bar. When the ribbon bar is minimized, it appears that all of your commands are in a single toolbar, ready to create exactly what you want with a single click.

Tabbed Documents

The plot, 3D view, worksheet, and grid node editor windows are displayed as tabbed documents. When more than one window is open, tabs appear at the top of the screen, allowing you to click on a tab to switch to that window.

Selecting and Closing Windows

To select a tab to view, click the tab name. To close a tab, right-click and select *Close* or click the X next to the tab name. If unsaved changes are present in the document, you will be prompted to save the changes before the file is closed.

Change Order of Tabs

When viewing in tabbed document mode, the tabs may be dragged to reorder them. Left-click on a tab, hold the left mouse button, drag to a new location, and release the mouse button to move the tab to a new location.

To move to the next tab, you can use the Next command. Alternatively, press CTRL + F6 to move to the next tab.

The and buttons on the sides of the tabs are used to scroll the tabs should there be more tabs than can fit along the top of the window.

Unsaved Changes

When a document contains unsaved changes, an asterisk (*) appears next to its tabbed name. The asterisk disappears once the unsaved changes have been saved.

The Plot1 *tab has unsaved changes, indicated by the (*) asterisk. The* Sheet1 *and* Sheet2 *tabs do not have unsaved changes.*

Tab Style

The style of the tab can be changed in File | Options | User Interface. Select a new tab style from the *MDI tab style* list.

No Tabs

Tabs can be turned off in **Options** dialog User Interface page. Select *None* from the *MDI tab style* list.

Changing the Layout

The plot, worksheet, grid editor, 3D view window, **Properties** window, and **Contents** window are in a docked view by default. However, they can be displayed as floating windows. The visibility, size, and position of each item may be changed.

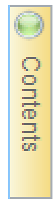
Visibility

Use the **View | Show/Hide** commands to toggle the display of the rulers, drawing grid, status bar, **Contents** window, and **Properties** window. Alternatively, click the or buttons in the **Contents** and **Properties** windows to auto-hide or close the windows.

Right-click the ribbon or quick access toolbar to minimize the ribbon, move the quick access toolbar above or below the ribbon, and customize the ribbon or quick access toolbar.

Auto-Hiding the Contents or Properties Windows

Click the button to auto-hide a docked **Contents** or **Properties** window. The window slides to the side of the **Surfer** main window and a tab appears with the window name.



*The **Contents** appears as a tab on the side of the window.*

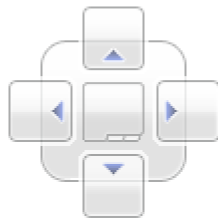
Position the mouse pointer over the tab to view the window. Move your mouse away from the window and the window "hides" again. Click inside the window to anchor it at its current position. Click in another window to release the anchor and hide the window. Click the button to return the window to a docked position.

Size

Drag the sides of the application window, **Contents** window, **Properties** window, or document window to change its size. If a window is docked, its left and right bounds are indicated by a cursor, and its upper and lower bounds are indicated by a cursor. Click and drag the cursor to change the size.

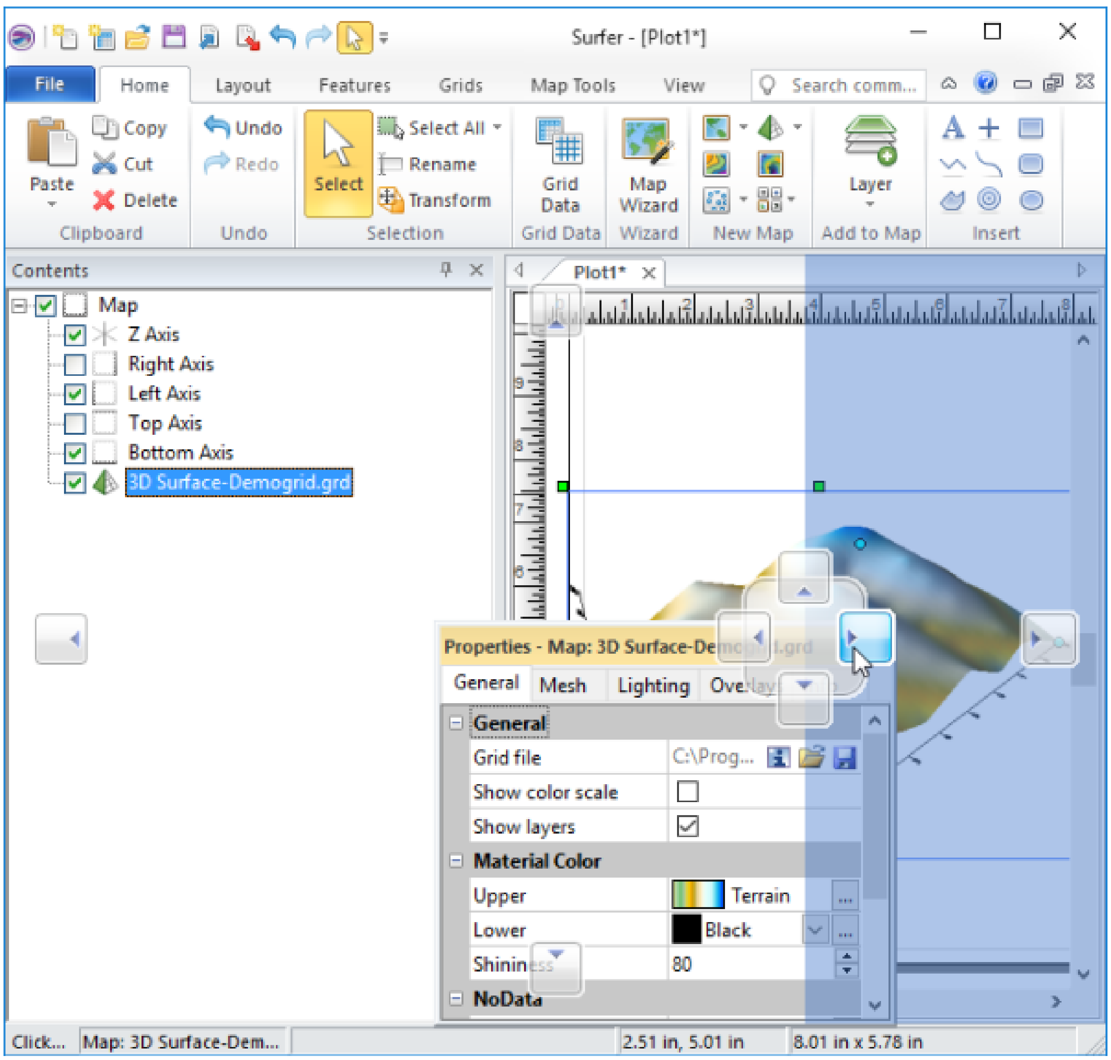
Position

To change the position of a docked window, click the title bar and drag it to a new location. To dock the **Contents** or **Properties** windows, use the docking mechanism. Double-click the window's title bar to toggle between floating and docked modes. Left-click the title bar of a window and drag it to a new location while holding the left mouse button. The docking mechanism displays with arrow indicators as you move the window.



*The docking mechanism makes it easy to position the **Contents** and **Properties** windows.*

When the cursor touches one of the docking indicators in the docking mechanism, a blue rectangle shows the window docking position. Release the left mouse button to allow the window to be docked in the specified location.



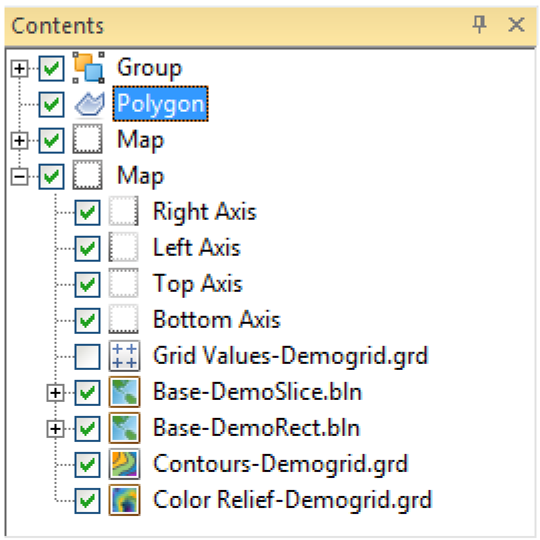
*This image displays the **Properties** window being docked to the right side of the **Surfer** plot window.*

Restoring the Windows to Their Original Locations

If the **Contents** or **Properties** windows have moved or become invisible, or if they are in undesired locations, you can use the **View | Windows | Reset Windows** command to move them back to their original locations. You must restart **Surfer** for the changes to take effect.

Contents

The **Contents** window contains a hierarchical list of all objects in the plot, grid editor, or 3D view window. The objects can be selected, arranged, moved, renamed, or deleted in the **Contents** window. Changes made in the **Contents** window are reflected in the plot, grid editor, or 3D view window and vice versa.



*The **Contents** window displays the structure of all the objects in the plot window.*

Displaying or Hiding the Contents Window

The **Contents** window is opened and closed with the **View | Show/Hide | Contents** command. Clear the **Contents** check box to hide the **Contents** window. Check the **Contents** check box to display the **Contents** window. Alternatively, you can click on the  button in the title bar of the **Contents** window to close the window. You can also right-click on the **Contents** window title bar and click **Hide**. To activate the **Contents** window, click inside the **Contents** window or press ALT+F11 on the keyboard.

Auto Hide the Contents Window

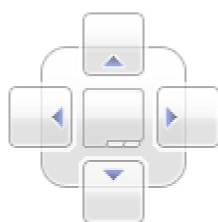
You can increase the plot document space by minimizing the **Contents** window with the *Auto Hide* feature. To hide the manager, click on the button in the upper right corner of the **Contents** window. The window hides on the left, top, or right side of the plot window as a small tab labeled **Contents**.

To view the contents of the **Contents** window while in tab view, place the cursor directly over the tab. Click in the window to keep it open for use. Click outside the window to return it to the hidden position. Click on the button to return it to the normal display mode. Alternatively, right-click the **Contents** window title bar and select **Auto Hide**. You can also drag the sides the **Contents** window to change the size of the window.

Changing the Contents Window Location - Floating vs. Docking

The **Contents** window can be docked on the edge of the **Surfer** window or floated as a dialog. The **Contents** window is displayed in a docked view by default. The manager can also be detached to display as a floating window. Double-click on the **Contents** window title bar to toggle between floating and docked modes. Alternatively, right-click the **Contents** window title bar and select **Floating**, **Docking**, **Auto Hide**, or **Hide**.

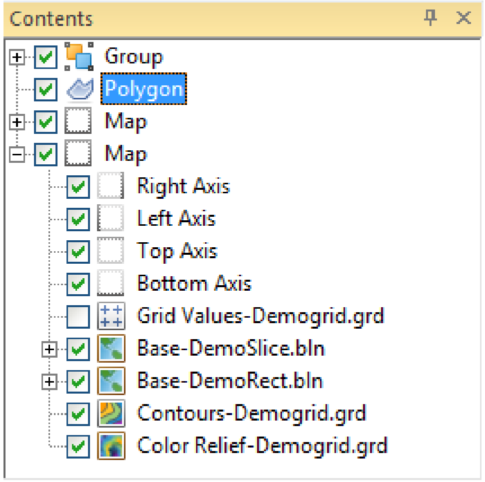
To change the position of the docked **Contents** window, left-click the title bar of the manager and drag it to a new location while holding the left mouse button. The docking mechanism displays with arrow indicators as you move the window. When the cursor touches one of the docking indicators in the docking mechanism, a blue rectangle shows the window docking position. Release the left mouse button to allow the manager to be docked in the specified location.



The docking mechanism displays with docking indicators.

Object Tree

If an object contains sub-objects, a ⊞ or ⊟ is located to the left of the object name. Click on the ⊞ or ⊟ button to expand or collapse the list. For example, a map object normally contains at least one map layer (e.g. Contours) and four axes. The *Map* object may contain many other objects. To expand the *Map* tree, click on the control. You can also select the item, and press the PLUS key on the numeric keypad or press the RIGHT ARROW key on your keyboard. To collapse a branch of the tree, click on the control. You can also select the item, and press the MINUS key on the numeric keypad or press the LEFT ARROW key. The expansion state of sub-objects in the **Contents** window is retained in the **Surfer** file .SRF. Use the *Expand new Contents window items* option in the **Options** dialog to control the expansion state of new objects in **Contents** window.



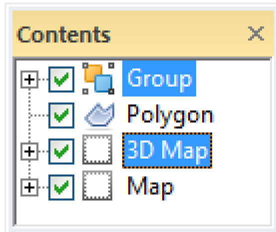
The + sign to left of the Group indicates it is collapsed. The - sign to the left of the Map indicates it is expanded.

Selecting Objects

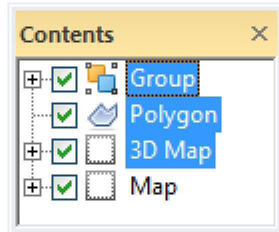
To select an item in the **Contents** window, click on the item or press the arrow keys, and the object text is highlighted. The selection handles in the plot change to indicate the selected item. If you select an object in the plot window, its name is selected in the **Contents** window as well. More than one nested object can be selected at a time.

To select multiple objects at the same level in the tree, hold down the CTRL key and click on each object. To select multiple contiguous objects at the same level in the tree, select the first object, and then hold down the SHIFT key and click on the last object.

Click on a layer or a group in the **Contents** window and an orange left-hand arrow with a small pushpin appears. Clicking on the pin either pins or unpins the layer or group for editing. When a layer or group is pinned, only objects within the pinned layer or group can be selected. This feature is useful for selecting objects in the plot window.



The Group and 3D Map objects were clicked while holding CTRL.



The Group and 3D Map objects were clicked while holding SHIFT.

Arranging Objects

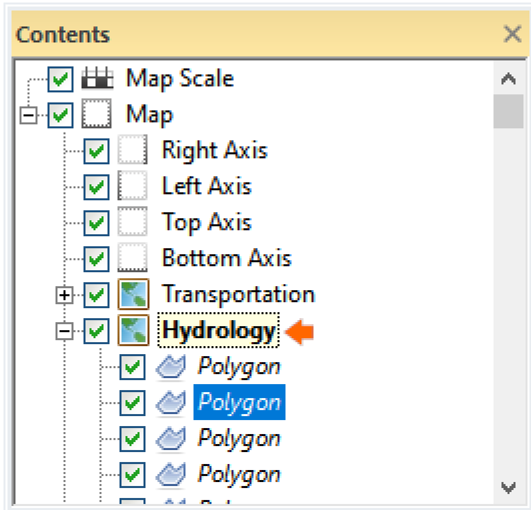
To change the display order of the objects with the mouse, select an object and drag it to a new position in the list above or below an object at the same level in the tree. The pointer changes to a black arrow if the object can be moved to the cursor location, or a red circle with a diagonal line if the object cannot be moved to the indicated location. Alternatively, select an object and use the **Bring to Front**, **Send to Back**, **Bring Forward**, and **Send Backward** commands. These commands can be accessed in the **Layout | Arrange** command group or by right-clicking on an object in the **Contents** window.

Moving Features

Features such as points, polylines, and polygons can be moved between base (vector) layers and the plot document. The **Move/Copy to Layer** command can be used to move or copy features. Features can also be moved in the **Contents** window. To move a feature to another base (vector) layer, select the feature and drag it to a new position within another base (vector) layer. To move a feature to the plot document, select the feature and drag it to a new position above, between, or below the top-level objects in the **Contents** window.

Editing Features in Groups

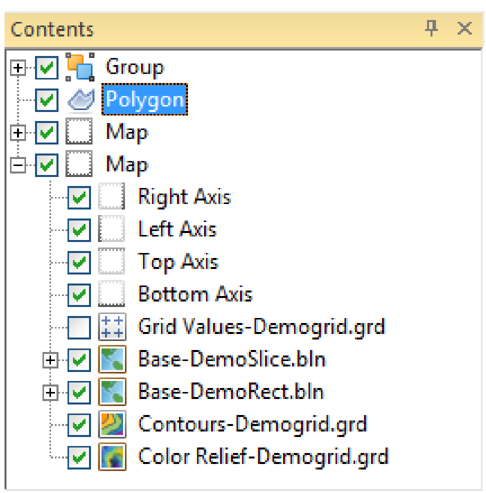
Features such as points, polylines, and polygons can be added, edited, and removed from composite objects such as groups and base (vector) layers. A special edit mode is enabled to do so. Edit mode is started and stopped automatically by the application. Ensure that edit mode is not enabled before using the **Export** command either by clearing the selection or selecting a non-composite object.



An orange arrow and italic text indicates edit mode is enabled.

Object Visibility

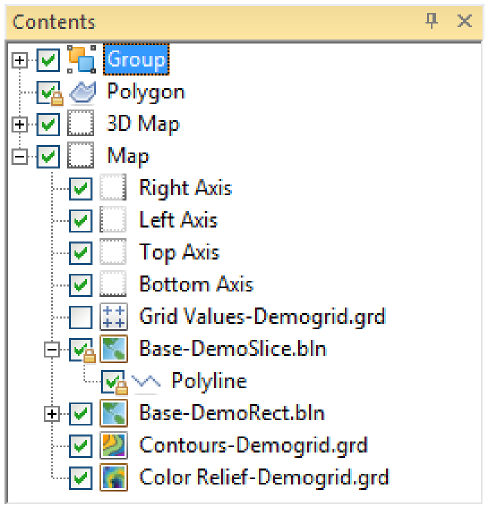
Each object in the **Contents** window includes an icon indicating the type of object and a text label for the object. All objects also have a check box that indicates if the object is visible. A indicates the object is visible. A ☐ indicates the object is not visible. Click on the check box to change the visibility state of the object. Invisible objects do not appear in the plot window and do not appear on printed output. The visibility check box also controls the visibility for all of its sub-objects. For example, if a *Map* object is made invisible then the axes and layers within the *Map* will also be hidden. Note that if a surface is made invisible, any overlays also become invisible. Select multiple objects to toggle the visibility for multiple objects at one time.



A check mark indicates the object is visible. In this example, the grid values layer is not visible.

Locked Objects

Objects and layers can be locked to prevent changes to their size and position with the **Lock Position** command. When an object or layer is locked, a small lock icon appears in the lower-right corner of the visibility check box. When a map, group, or base layer object is locked, all of its sub-objects are automatically locked.



The lock icon indicates the object is locked. In this example a polygon and base map layer are locked.

Opening Object Properties

To display the properties for an object, click once on the object in the **Contents** window or in the plot window. The properties are displayed in the Properties window. To display a context menu of available actions for an object, right-click on that object. When the **Properties** window is hidden or closed, double-clicking on an object in the **Contents** window opens the **Properties** window with the properties for the selected object displayed. The map properties control the map's *View, Scale, Limits*, *Frame,* and *Coordinate System*. Each map layer has specific properties that controls the options for the specific map type. Each map axis also has properties.

Renaming Objects

To edit an object's text ID, select the object in the **Contents** window and then click again on the selected item (two slow clicks) to edit the text ID associated with an object. Allow enough time between the two clicks so it is not interpreted as a double-click. Enter the new name into the box. Alternatively, right-click on an object name and select *Rename Object*, select the object and click the

Rename command, or select the object and press F2 on the keyboard. Enter an ID in the **Rename Object** dialog and click *OK*.

Deleting Objects

To delete an object, select the object and press the DELETE key. To move a map layer from one map to a new map, click on the map layer and click the **Map Tools | Layer Tools | Break Apart** command. Or right-click on the map layer and select **Break Apart Layer**. Select multiple objects and press DELETE to delete multiple objects at one time.

Scroll the Contents Window

If the list of objects in the **Contents** window is long, you can use the scroll bar on the side of the **Contents** window to scroll down to an object. Alternatively, you can use the mouse scroll wheel to scroll down. To scroll down using the mouse, click once in the **Contents** window to select the window. Roll the mouse wheel backward to scroll lower in the **Contents** window. Roll the mouse wheel forward to scroll higher in the **Contents** window.

Properties

The **Properties** window allows you to edit the properties of a selected object, such as a contour map or axis. The **Properties** window contains a list of all properties for the selected object. The **Properties** window can be left open so that the properties of the selected object are always visible.

To display the properties for an object, click once on the object in the **Contents** window or in the plot window. The properties are displayed in the **Properties** window. When the **Properties** window is hidden or closed, double-clicking on an object in the **Contents** window opens the **Properties** window with the properties for the selected object displayed. To activate the **Properties** window, click inside the **Properties** window or press ALT+ENTER on the keyboard.

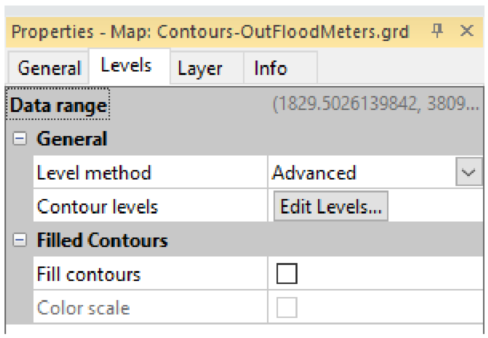
For information on a specific feature or property that is shown in the **Properties** window, refer to the help page for that **Properties** window page. For instance, if you are interested in determining how to set the *Fill colors* for a contour map or how to save data for a post map, refer to the contour map *Levels* topic or post map *General* topic in the program help, respectively.

Opening and Closing the Properties Window

The **Properties** window is opened and closed with the **View | Show/Hide | Properties** command. Clear the **Properties** check box to close the **Properties** window. Check the **Properties** check box to open the **Contents** window. Alternatively, you can click on the  button in the title bar of the **Properties** window to close the window. You can also right-click on the **Properties** window title bar and click **Hide** . To activate the **Properties** window, click inside the **Properties** window or press ALT+ENTER on the keyboard.

Auto Hide the Properties Window

You can increase the plot document space by minimizing the **Properties** window with the *Auto Hide* feature. To hide the **Properties** window, click on the button in the upper right corner of the **Properties** window.



Click on the autohide button to display the ***Properties*** *window as a tab.*

The window hides on the left, top, or right side of the plot window as a small tab labeled **Properties**.

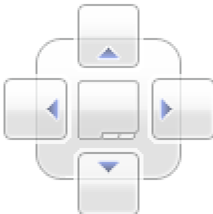
The ***Properties*** *tab view.*

To view the contents of the **Properties** window while in tab view, place the cursor directly over the tab. Click in the window to keep it open for use. Click outside the window to return it to the hidden position. Click on the button to return it to the docked display mode. Alternatively, right-click the **Properties** title bar and click **Auto Hide** . You can also drag the sides of the **Properties** window to change the size of the window.

Changing the Properties Window Location - Floating vs. Docking

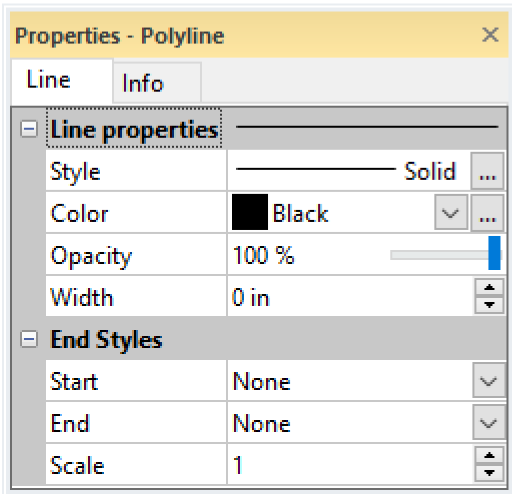
The **Properties** window can be docked on the edge of the **Surfer** window or floated as a dialog. The **Properties** window is displayed in a docked view by default. The window can also be detached to display as a floating window. Double-click on the **Properties** window title bar to toggle between floating and docked modes. Alternatively, right-click the **Properties** window title bar and select **Floating**, **Docking**, **Auto Hide**, or **Hide**.

To change the position of the docked **Properties** window, left-click the title bar of the window and drag it to a new location while holding the left mouse button. The docking mechanism displays with arrow indicators as you move the window. When the cursor touches one of the docking indicators in the docking mechanism, a blue rectangle shows the window docking position. Release the left mouse button to allow the manager to be docked in the specified location.



The docking mechanism displays with docking indicators.

Changing Properties



*The **Properties** window displays the properties associated with the selected object.*

The **Properties** window displays the properties for selected objects. To change a property, click on the property's value and select a new property from the pop up box, scroll to a new number using the buttons, select a new value using the slider , select a new value from the list or palette, or type a property value. Objects in the plot, grid editor, or 3D view window automatically update after you select an item from a palette, use one of the controls, or press ENTER after typing a new value.

For example, a polyline has *Style, Color, Opacity, Width,* and *End Styles* properties. Changing the *Color* requires clicking on the current color and selecting a new color from the *color palette*. Changing the *Width* requires highlighting the current width and typing a new number or scrolling to a new number. Changing the *Opacity* requires highlighting the current value and typing a new number or clicking on the slider bar and dragging it to a new value.

You can modify more than one object at a time. Only shared properties can be changed are when multiple objects are selected. For example, you can click on a polyline in the **Contents** window. Hold the CTRL key and click on a polygon. You can then change the *line* properties of both objects at the same time. *Fill* properties, which are available if only a polygon is selected, are not available as the polyline does not have fill properties.

Some properties are dependent on your other selections. For example, there is a *Pattern Offset* section on the **Fill** page. This section is only available when an image fill type is selected as the *Pattern.*

Expand and Collapse Features

Features with multiple options appear with a ⊞ or ⊟ to the left of the name. To expand a group, click on the ⊞ icon. To collapse the group, click on the ⊟ icon. For example, the expanded *Filled Contours* section in the **Levels** page contains three options, *Fill contours, Fill colors,* and *Color scale*.

Keyboard Commands

To activate the **Properties** window, press ALT+ENTER on the keyboard. When working with the **Properties** window, the up and down arrow keys move up and down in the **Properties** window list. The TAB key activates the highlighted property. The right arrow key expands collapsed sections, e.g., *Filled Contours*, and the left arrow collapses the section.

Property Defaults

Use the **File | Options** command to change the default settings. Default settings for rulers, drawing grid, line, fill, text, symbol, label format, and advanced settings that control each map type can be set from the **Options** dialog.

Property Information Area

If the *Show property info area* is checked on the **Options** dialog **User Interface** page, a short help statement for each selected command is presented in the **Properties** window.

Status Bar

The status bar is located at the bottom of the **Surfer** window. The status bar displays information about the current command or activity in **Surfer**. Click the **View | Show/Hide | Status Bar** check box to show or hide the status bar. A check mark next to **Status Bar** indicates that the status bar is displayed. Clear the **Status Bar** check box to hide the status bar.

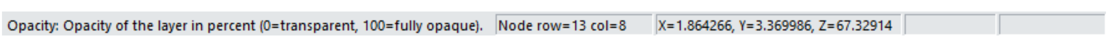
Status Bar Sections

The status bar is divided into five sections. The left section displays information about the selected command or item in the **Properties** window. The second section shows the selected object name or the number of objects/points in the selection. The middle section shows the cursor coordinates in map units, if the cursor is placed above a map. The fourth section shows the cursor coordinates in page units of inches or centimeters. The right section displays the dimensions of the selected object. In the worksheet, the status bar displays tool tips.

The status bar has five sections of information.

Grid Editor Status Bar Sections

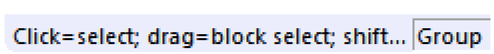
When viewing a grid in the **grid editor**, the first three sections of the status bar display a description for the selected property in the **Properties** window, the active grid node grid coordinates, and the map coordinates of the cursor location.



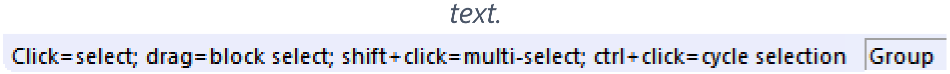
The status bar displays different information when viewing a grid in the grid editor.

Adjust Section Width

The status bar section widths can be adjusted to display additional text. If "..." is displayed at the end of the text, additional text can be displayed. To change the width, place the cursor over a section division. When the cursor changes to a ⟺, left-click and drag the divider left or right to a new location.



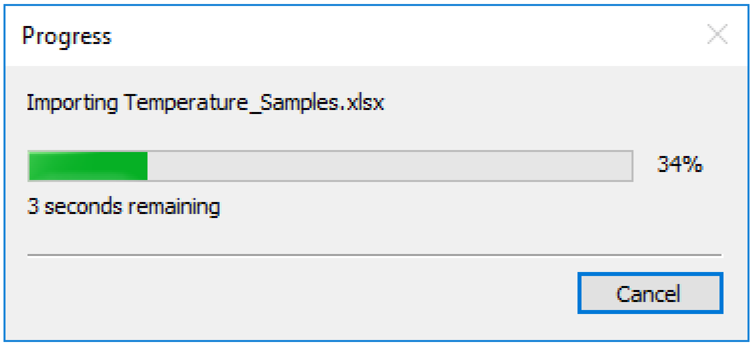
A portion of the status bar. The "..." in the left section indicates there is additional text.



A portion of the status bar after making the left section larger.

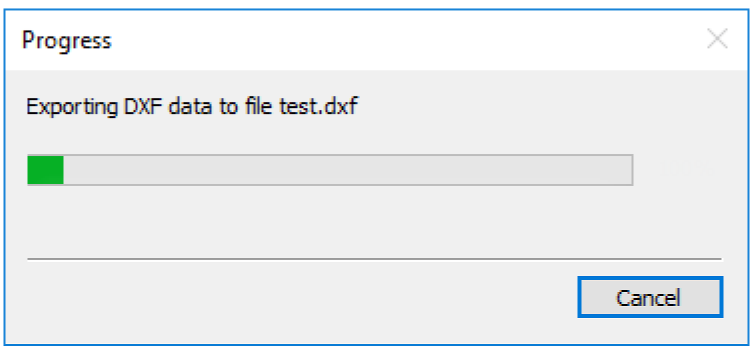
Progress

The **Progress** dialog indicates the progress of a procedure, such as gridding. The percent of completion and time remaining will be displayed. Click *Cancel* to stop the current process.



*The progress of a procedure is shown in the **Progress** dialog.*

When the program does not know how much time is required to complete a task, the *Indeterminate* mode is displayed in the **Progress** dialog. This indicates that the program is actively completing the task, with an unknown time of completion. The program is not frozen.



*The **Progress** dialog does not display a percentage or time estimate in Indeterminate mode.*

Menu and Tab Commands

The ribbon contains the commands that allow you to add, edit, and control the objects on the plot, worksheet, grid editor, or 3D view window page.

Plot Document Commands

When viewing a plot document, the main ribbon tab commands are available:

File	Open and save files, import or export data, print, and set options and defaults
Home	Contains common editing, selection, feature, grid, and map commands
Layout	Set the page display and arrange or position maps and objects in the plot document
Features	Draw features and perform geoprocessing
Grids	Perform grid operations
Map Tools	Add map layers, and edit or analyze maps and map layers
View	Controls the display of toolbars, status bar, rulers, grids, and managers, resets window positions, tracks cursor between map and worksheet, and controls the zoom level of the plot

Point Cloud Commands

The commands for editing the points within a point cloud layer are located on the **Point Cloud** ribbon. The **Point Cloud** tab is only displayed when a point cloud layer is selected.

Point Cloud	Contains commands for selecting, classifying, modifying, and exporting points

3D View Commands

The commands for changing the view, creating fly-throughs, and copying images in the 3D view window are located on the **3D View** ribbon. The **3D View** tab is only displayed while viewing a map in the 3D view.

3D View	Contains commands for modifying the 3D view, creating a fly-through, and copying an image of the 3D view to the clipboard.

Worksheet Commands

The primary commands when viewing a worksheet window are located on the **Data** ribbon. However, many of the **File** menu and **Grid** tab commands are also available when viewing a worksheet window, and a few of the **Home** and **View** tab commands are available as well.

Data	Contains commands to sort data, filter data, compute statistics, assign projection, re-project data, and calculate mathematical transformations

Grid Editor Commands

The primary commands when viewing a grid in the grid editor are located on the **Grid Editor** tab. The **Grid Editor** tab includes commands and tools for editing the grid values.

Grid Editor	Contains commands to open, edit, and save grids

The **Application/Document Control Menu** commands control the size and position of the application window or the document window.

Worksheets

Worksheet windows are a view of the data file and are designed to display, edit, enter, and save data. The worksheet windows have several useful and powerful editing, transformation, and statistical operations available. In addition, a coordinate system can be assigned to the data file. Several import and export options are available for opening data files from other spreadsheet programs. The components of the worksheet window are displayed below.

Worksheet Commands

The worksheet commands include commands on the following tabs:

File	Open and save files, import or export data, print, and set options and defaults
Home	Contains clipboard and undo commands
Grids	Perform grid operations
View	Controls the display of status bar and windows and resets window positions
Data	Edit, find, format data in the worksheet. Manipulate, transform, and perform calculations with worksheet data. Assign or project coordinates. Track the cursor between the plot, worksheet, and grid windows.

Not all of the **File**, **Home**, **Grids**, and **View** commands are available in the worksheet view.

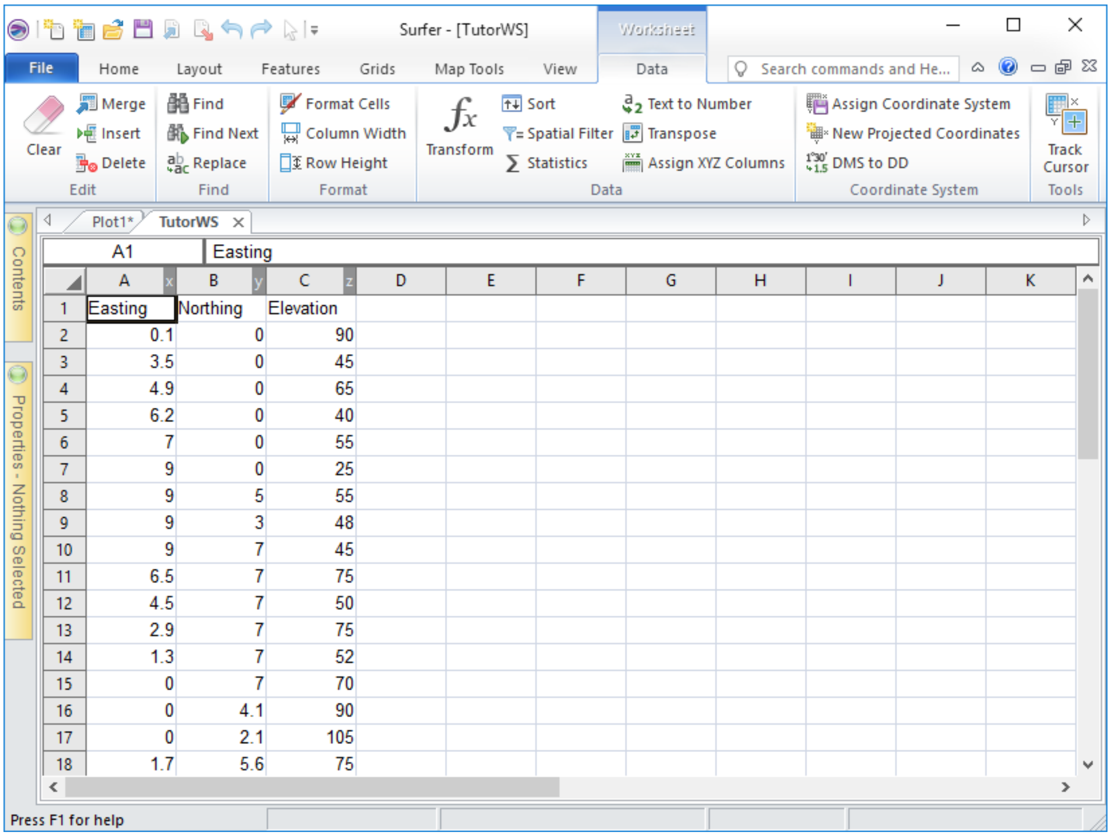
The Application/Document Control menu commands control the size and position of the application window or the document window.

Tab View

The plot, worksheet, and grid node editor windows are displayed as tabbed documents. When more than one window is open, tabs appear at the top of the document, allowing you to click on a tab to switch to a different window. The tabs may be dragged to reorder them. When a document contains unsaved changes, an asterisk (*) appears next to its tabbed name. The asterisk is removed once the changes have been saved.

Worksheet Window

The image below displays the parts of the worksheet document.



*This is the **Surfer** worksheet document with the **Contents** and **Properties** windows in auto hide mode on the left, and the plot document and worksheet tabs at the top of the worksheet.*

Grid Editor

The **File | Open**, **Grids | Editor | Grid Editor** command, the  button, and the **Map Tools | Edit Layer | Grid** commands open the grid editor as a new document.

- The **Grids | Editor | Grid Editor** command and the  button open a grid file with the **Open Grid** dialog.
- The **Map Tools | Edit Layer | Grid** command opens the grid file from the selected map layer in the plot document. You can also edit the grid for a map

layer by right-clicking on the map layer and clicking **Edit Grid**. This command enables the **Update Layer** command in the grid editor. The **Map Tools | Edit Layer | Grid** command is not available for 1-grid vector and 2-grid vector layers. When accessed this way, the Grid Editor will display any Post, Classed Post, and vector or raster Base layers in the Map as context layers.

The grid editor contains various methods for editing the grid Z values. Editing the grid Z values will change the appearance of any grid-based maps. For example, the grid editor can be used to edit contours on a contour map or change the surface in a 3D surface map.

Each grid node is indicated with a black "+" in the grid editor window by default. Each NoData grid node is indicated with a blue "x" by default. The active node is highlighted with a red diamond. To move between grid nodes, press the arrow keys, or click a node with the **Select** tool active to make it the active node. The grid editor also includes contours, node labels, and a color fill. The grid appearance is controlled by the items in the **Contents** window and the properties displayed in the **Properties** window. Note the **Undo** command does not undo changes in the **Properties** window in the grid editor.

Context Layers

When the grid editor is accessed either using the **Map Tools | Edit Layer | Grid** command or by right-clicking on the map layer and clicking **Edit Grid**, any Post, Classed Post, and vector or raster Base layers will be displayed as context layers to assist with editing the grid. Any layers being used as context layers will be displayed in the **Contents** window. Visibility for these context layers can be controlled by toggling the layers on and off in the **Contents** window, or, by toggling visibility for those layers in the 2D plot window.

The context layer will have labels in the grid editor if the layer has labels applied in the 2D plot window. These labels can be toggled on and off by selecting the desired context layer in the **Contents** window and toggling the *Show labels* option in the **Properties** window. If a layer does not have labels applied in the 2D plot window, then there are no properties for the context layers in the grid editor – all the properties are changed in the 2D plot window.

Note, no context layers will be displayed if the grid file being edited has been geotransformed. To remove a geotransform from a grid layer, set the map target coordinate system to the same coordinate system as the grid layer source coordinate system.

Images in the Grid Editor

The grid editor also allows you to open an image file and save as a grid file.

A grid requires a single floating point value at each grid node. Images contain colors which are three separate values (Red, Green, Blue) at each pixel.

Color Image

Color image formats are converted to a single floating point value by calculating the intensity of each color value using the intensity equation:

$$I = A(.30R + .59G + .11B)$$

where I = intensity, R,G,B,A are the red, green, blue, and normalized alpha values.

For example, a pixel from a color image with Red=255, Green=0, and Blue=0 would be mapped to a grid node with the value of:

$$I = .30*255 + .59*0 + .11*0 = 77$$

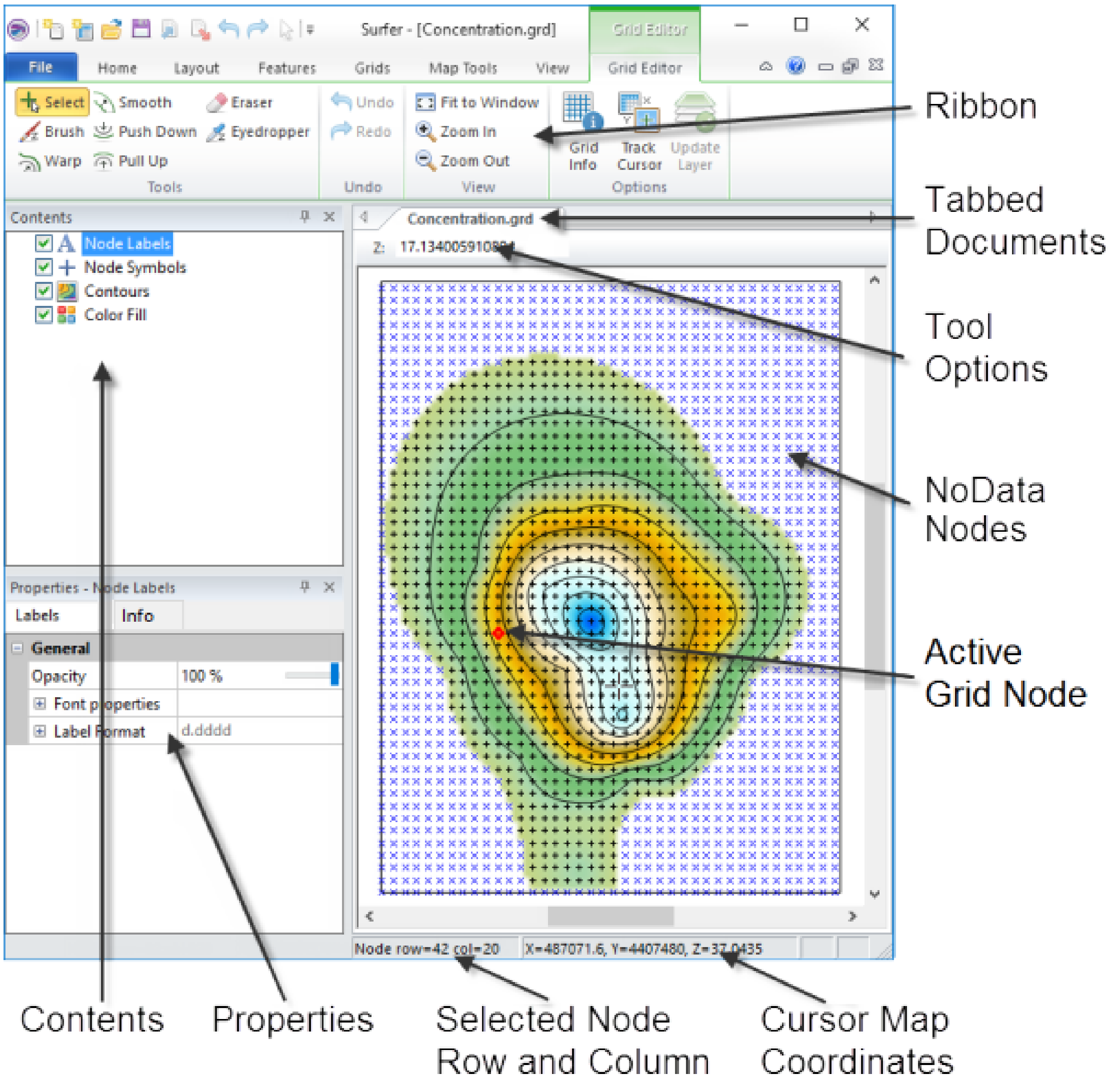
Grayscale Image

Grayscale images are imported directly. Grayscale images have a single color value and do not need to use the intensity equation. Surfer does not normalize the grayscale value. The value is used exactly as specified in the image.

For example, consider a grayscale image with a pixel that contains a value of 55. The grid node value would be set to 55.

Grid Editor Window

The following image and table explain the purpose of the grid editor window components.



*This is the **Surfer** grid editor with the **Contents** and **Properties** windows on the left and grid editor window on the right.*

Component Name	Component Definition
Ribbon	The ribbon contains the **Grid Editor** commands.
Contents	Toggle the display of the *Node Labels*, *Node Symbols*, *Contours*, *Context Layers*, and *Color Fill* with the **Contents** window.
Properties	Edit *Node Labels*, *Node Symbols*, *Contours*, *Context Layers*, and *Color Fill* **Labels** properties and view **Info** properties in the **Properties** window.
Tabbed Documents	Plot windows, worksheet windows, and grid editor windows are displayed as tabbed documents.
Tool Options	The tool options bar contains the *Z value box*, *Brush size*, *Density*, and/or *Pressure* depending on the selected tool mode.

Active Node	The node that is currently selected. The active node is highlighted with a red diamond.
Grid Node	Each grid node is indicated with a black "+" in the grid editor window by default. NoData nodes are indicated with a blue "x".
Status Bar	The status bar includes information about the selected property, active node grid coordinates, and cursor map coordinates.

Grid Editor Commands

The **Grid Editor** ribbon tab includes the following commands:

Select	Select a grid node to edit the grid Z values one node at a time
Brush	Apply a specific Z value to one or more nodes
Warp	Drag grid values from one region into another
Smooth	Apply weighted averaging to grid nodes
Push Down	Decrease grid node values
Pull Up	Increase grid node values
Eraser	Assign the NoData value to grid nodes
Eyedropper	Acquire a grid node value by clicking on the grid
Undo	Undo the last operation
Redo	Redo the last undone operation
Fit to Window	Fits the entire grid in the grid editor window
Zoom In	Increase the grid editor window magnification
Zoom Out	Decrease the grid editor window magnification
Zoom Rectangle	Zoom in to an area of interest
Grid Info	Display information about the grid in a report window
Track Cursor	Track cursor location across plot, worksheet, and grid editor windows for maps, data files, and grids.
Update Layer	Updates the associated map layer with the edited grid

Using the Grid Editor

The grid editor can be used on existing map layers or on grid files without first creating a map.

To edit a map layer's grid:

1. Select the map layer created from a grid file to edit in the plot document **Contents** window. Only the grid for this map layer will be edited even when multiple layers, such as contour or color relief, use the same grid file.

2. Click **Map Tools | Edit Layer | Grid** in the plot window. The grid file is opened and is represented by a filled contour map. The location of each grid node in the file is marked with a black "+". NoData nodes are marked with a blue "x".
3. Use the **Grid Editor | Tools** commands to make the desired adjustments to the grid.
4. When you are done editing the grid, click the **Grid Editor | Options | Update Layer** command to update the map layer in the plot document with your grid.
5. Click the plot document tab to view the changes to the map layer. If you wish to revert the changes to the map layer, click the **Undo** command while viewing the plot window. If you are satisfied with the changes to the map layer, you may wish to save the edited grid to a file.
6. If you wish to save your edits to a file, click **File | Save As** to create a new grid file. Click **File | Save** to overwrite the existing grid file. It is necessary to save your edits to a file with **Save** or **Save As** if you wish to update all layers in your map to use the edited grid.
7. To close the grid editor window, click the **File | Close** command or click the X in the grid editor document tab. To view an existing window and keep the grid editor window open, click on another document tab.

To edit a grid file:

1. Click the **Grids | Editor | Grid Editor** command and select the grid file in the **Open Grid** dialog. The grid file is opened and is represented by a filled contour map. The location of each grid node in the file is marked with a black "+". NoData nodes are marked with a blue "x".
2. Use the **Grid Editor | Tools** commands to make the desired adjustments to the grid.
3. When you are done editing the grid, click **File | Save As** to create a new grid file. Click **File | Save** to overwrite the existing grid file. It is necessary to save your edited grid to a file with **Save** or **Save As** if you wish to create map layers with the grid.
4. To close the grid editor window, click the **File | Close** command or click the X in the grid editor document tab. To view an existing window and keep the grid editor window open, click on another document tab.

File Types

Surfer uses four basic file types: data, grid, base map, and **Surfer** .SRF files.

Data Files

Various types of data files are used to produce grid files, point cloud maps, and drillhole maps or to post data points on a map. These files are generally referred to as XYZ data files or data files throughout the help. Data can be read from various file types. Most data files contain numeric XY location coordinates and optional Z values. The Z values contain the variable to be modeled, such as elevation, concentration, rainfall, or similar types of values.

XYZ data files contain raw data that **Surfer** interprets to produce a grid file. To create a grid file, you must start with an XYZ data file. XYZ data files are organized in column and row format. Surfer requires the X, Y, and Z data to be in three separate columns.

Grid Files

Grid files, also known as raster files, produce several different types of grid-based maps, are used to perform grid calculations, and to carry out grid operations. Grid files are a regularly spaced rectangular array of Z values in columns and rows. Grid files can be created in **Surfer** using the **Home | Grid Data | Grid Data** command or can be imported from a wide variety of sources such as WCS servers or other applications.

Base Map Files

Base map files contain XY location data such as aerial photography, state boundaries, rivers, or point locations. Base map files can be used to create layers overlaid on other map types, or to specify the limits for assigning NoData values, faults, breaklines, or slice calculations. Base map files can be created from a wide variety of vector and image formats. Base map files may be referred to as vector data files, raster data files, and images or image files in the help, depending on the type of data in the base map file.

Surfer Files

Surfer .SRF files preserve all the objects and object settings contained in a plot window. These files are called **Surfer** .SRF files throughout the documentation. **Surfer** can open .SRF files from previous versions as far back as **Surfer 7**. **Surfer** can save files to previous .SRF formats for sharing with other users. For example, the *Surfer 15 Document .SRF* file type can be opened in **Surfer 15**, but does not contain features that are in later **Surfer** versions. Beginning with version 16, the *Surfer Plot (*.srf)* file type is backwards compatible with all **Surfer** versions **16** and newer.

Gridding Overview

A grid, also known as a raster in other software products, is a rectangular region comprised of evenly spaced rows and columns. The intersection of a row and column is called a grid node. Rows contain grid nodes with the same Y coordinate. Columns contain grid nodes with the same X coordinate. Contour, color relief, grid values, peaks and depressions, vector, viewshed, watershed, 3D surface, 3D wireframe map layers all require grids in **Surfer**.

What is Gridding?

Gridding is the process of taking irregularly or regularly spaced XYZ data and generating a regularly spaced grid of Z values at each grid node by interpolating or extrapolating the data values. In addition to gridding data, **Surfer** can also use a variety of other grid files directly. For a list of these, refer to the **File Format Chart** in the online *Help*.

Gridding Methods

Gridding the data produces a regularly spaced, rectangular array of grid nodes, with a calculated Z value at each node, from regularly or irregularly spaced XYZ data. The term "irregularly spaced" means that the distance between data points varies in the X or Y direction, or both. Irregularly spaced data often has many holes where data are missing. Gridding calculates the Z values for grid nodes where data exists, and can also calculate the Z values for grid nodes in the holes where no data exists, by extrapolating or interpolating the Z values in the data. The gridding method determines the mathematical algorithms used to compute the Z value at each grid node. Each method results in a different representation of your data. It is advantageous to test each method with a typical data set to determine the gridding method that provides you with the most satisfying interpretation of your data.

When your XYZ data is regularly spaced, meaning the distance between data points does not change in the X and Y directions, you may produce a grid file that uses the Z values directly and does not interpolate values for the grid nodes. See the *Producing a grid file from a regular array of XYZ data* help topic for more information.

General Gridding Options

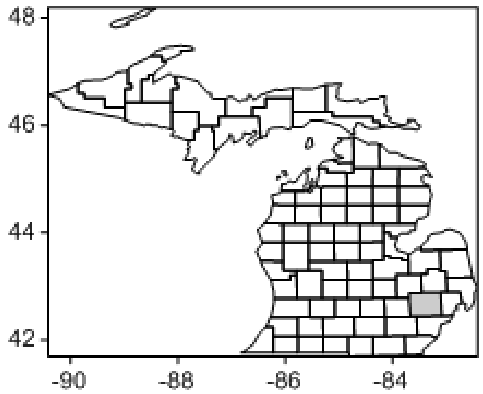
Each gridding method has its own set of gridding options. Some of the options are the same or similar for the different gridding methods, while other options are specific to particular gridding methods. Some options that are available to multiple gridding methods include: *Search*, *Anisotropy*, *Breaklines*, and *Faults*.

Map Types

Several different map types can be created, modified, and displayed with **Surfer**. These map types include base, contour, post, classed post, 3D surface, 3D wireframe, color relief, grid values, drillhole, peaks and depressions, point cloud, watershed, viewshed, 1-grid vector, and 2-grid vector maps. A brief description and example of each map is listed below.

Base Map

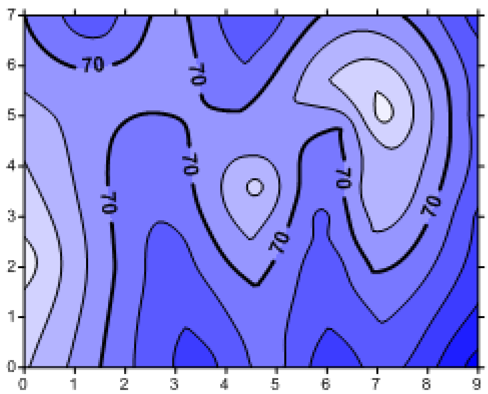
Base maps display boundaries on a map and can contain polygons, polylines, 3D polygons, 3D polylines, 3D polymesh objects, points, text, images, or metafiles. Base maps can be overlaid with other map layers to provide details such as roads, buildings, streams, city locations, areas of no data, and so on. Base maps can be produced from vector files, images, and data files. Individual base map objects can be edited, moved, reshaped, or deleted. Symbology can be added to a base map to communicate statistical information about the map features. Empty base maps can be created and used for drawing objects on other maps. Raster (image) and vector base maps can be downloaded from online WMS, OSM, and WFS mapping servers.



This is a base map of Michigan with county polygons. One of the individual polygons has fill.

Contour Map

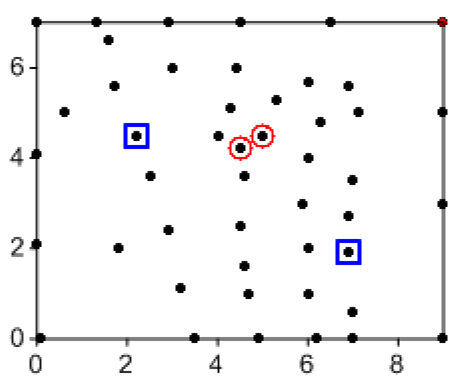
Contour maps are two-dimensional representations of three-dimensional data. Contours define lines of equal Z values across the map extents. The shape of the surface is shown by the contour lines. Contour maps can display the contour lines and colors or patterns between the contour lines. Contours can be linearly or logarithmically spaced, or a custom spacing can be set between each set of lines.



This is a contour map consisting of contour lines representing elevation.

Post Map

Post maps and classed post maps show data locations on a map. You can customize the symbols and text associated with each data location on the map. Each location can have multiple labels. Classed post maps allow you to specify classes and change symbol properties for each class. Classes can be saved and loaded for future maps.



The post map layer has black symbols. The classed post map layer has red *circles and blue squares. Only a sample of the data set is displayed in the classed post map.*

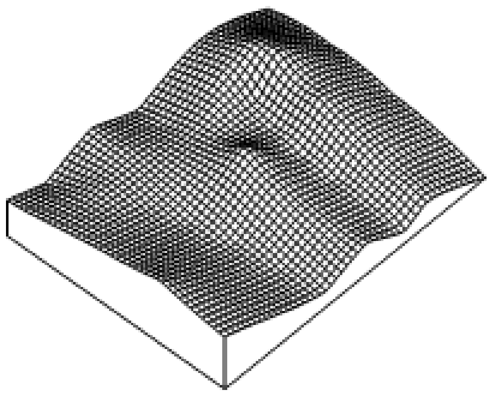
3D Surface Map

3D surface maps are color three-dimensional representations of a grid file. The colors, lighting, overlays, and mesh can be altered on a surface. Multiple 3D surface maps can be layered to create a block diagram.

This is a 3D surface map of the Telluride, Colorado USGS SDTS grid file.

3D Wireframe Map

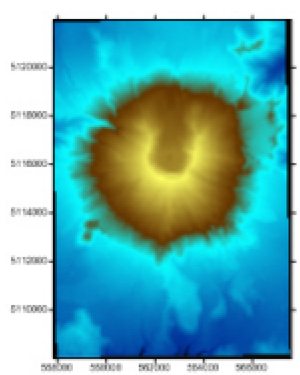
3D wireframe maps are three-dimensional representations of a grid file. Wireframes are created by connecting Z values along lines of constant X and Y.



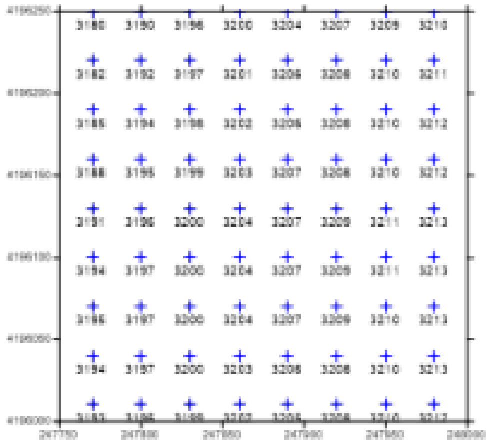
This is a 3D wireframe map with a custom rotation (47°), tilt (49°), and field of view (112°).

Color Relief Map

Color relief maps are raster images based on grid files. Color relief maps assign colors based on Z values from a grid file. NoData regions on the color relief map are shown as a separate color or as a transparent fill. Pixels can be interpolated to create a smooth image. Hill shading or reflectance shading can be applied to the color relief map to enhance its depth and appearance.



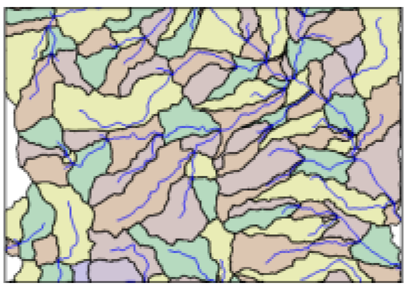
This is a color relief map of the Helens2.grd sample file.



The grid values layer posts symbols, labels, and/or lines along grid rows and columns at a specified density.

Grid Values Map

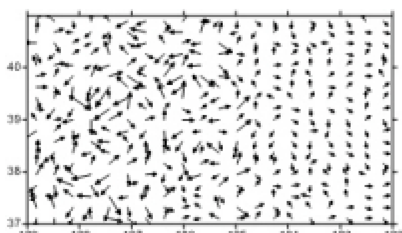
Grid values maps show symbols labels at grid node locations across the map. The density of the labels and symbols is controlled in the X and Y directions independently. Symbol color can vary by value across a colormap, and symbols and labels can be displayed for only a specific range of values. Grid lines can be added to the map.



This is a watershed map of a USGS SDTS grid file.

Watershed Map

Watershed maps display the direction that water flows across the grid. The watershed map breaks the grid into drainage basins and streams. Colors can be assigned to the basins and line properties can be associated with the streams. In addition, depressions can be removed by filling the depression.



This is a 1-grid vector map of Colorado elevations.

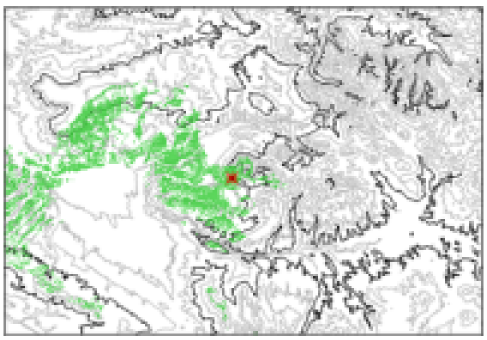
Vector Map

1-grid and 2-grid vector maps display direction and magnitude data using individually oriented arrows. At any grid node on the map, the arrow points in the downhill direction of the steepest descent and the arrow length is proportional to the slope magnitude. Vector maps can be created using information in one grid file (i.e. a numerically computed gradient) or two different grid files (i.e. each grid giving a component of the vectors).

This is a point cloud of the Golden, CO area from USGS Elevation Source Data

Point Cloud Map

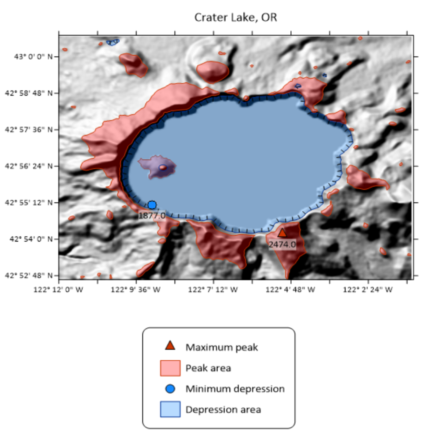
Point cloud maps display LAS/LAZ data as points at XY locations. LAS/LAZ data can be combined from multiple files and filtered with various criteria when creating a point cloud map. Color is assigned to the points by elevation, intensity, return number, or classification. Surfer includes commands for modifying, classifying, and exporting points in a point cloud layer. A grid can be created from the point cloud layer. Point cloud layers are displayed in the 3D View as three-dimensional points.



This is a viewshed layer overlaid on a contour map.

Viewshed Layer

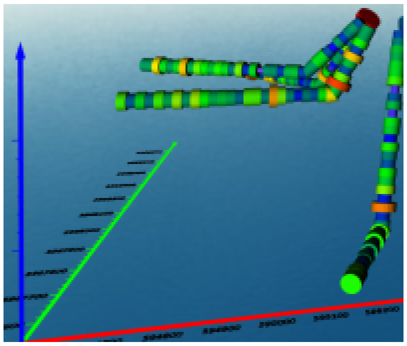
Viewshed layers highlight the regions of a map that are visible (or invisible) from a transmitter location. The transmitter, receiver, and obstruction height above the surface can be specified. The viewshed analysis radius and angle can also be specified. Viewsheds can be added to any 2D grid based map. A viewshed can also be added to a 3D surface map that is displayed with no tilt (90 degrees) and in the orthographic view.



The legend for this peaks and depressions map of the Crater Lake area explains the shaded areas and markings.

Peaks and Depressions Map

Volumes of surface water and ground water are informed by topographic data, Karst topography, and geographic data, which are captured in Surfer grid files and mapped in peaks and depression maps. Boundaries can be drawn around peaks where water flows from and depressions which capture water to create unique areas for statistical analysis.



This is a 3D view of a drillhole layer

Drillhole Map

A drillhole map can be created from collars, survey, intervals, and point data. The 2D drillhole layer shows the location, deviation, and path of each hole, core, or well. The drillhole map can also be viewed in 3D. Separate symbols can be defined to create a legend of the drillholes by hole id or any other attribute in the data.

Map Wizard

Click the **Home | Wizard | Map Wizard** command or the  button to create a map with the **Map Wizard**. The **Map Wizard** steps through the map creation process from raw data to a map with one or more layers. If necessary, data is gridded to create a grid-based map layers.

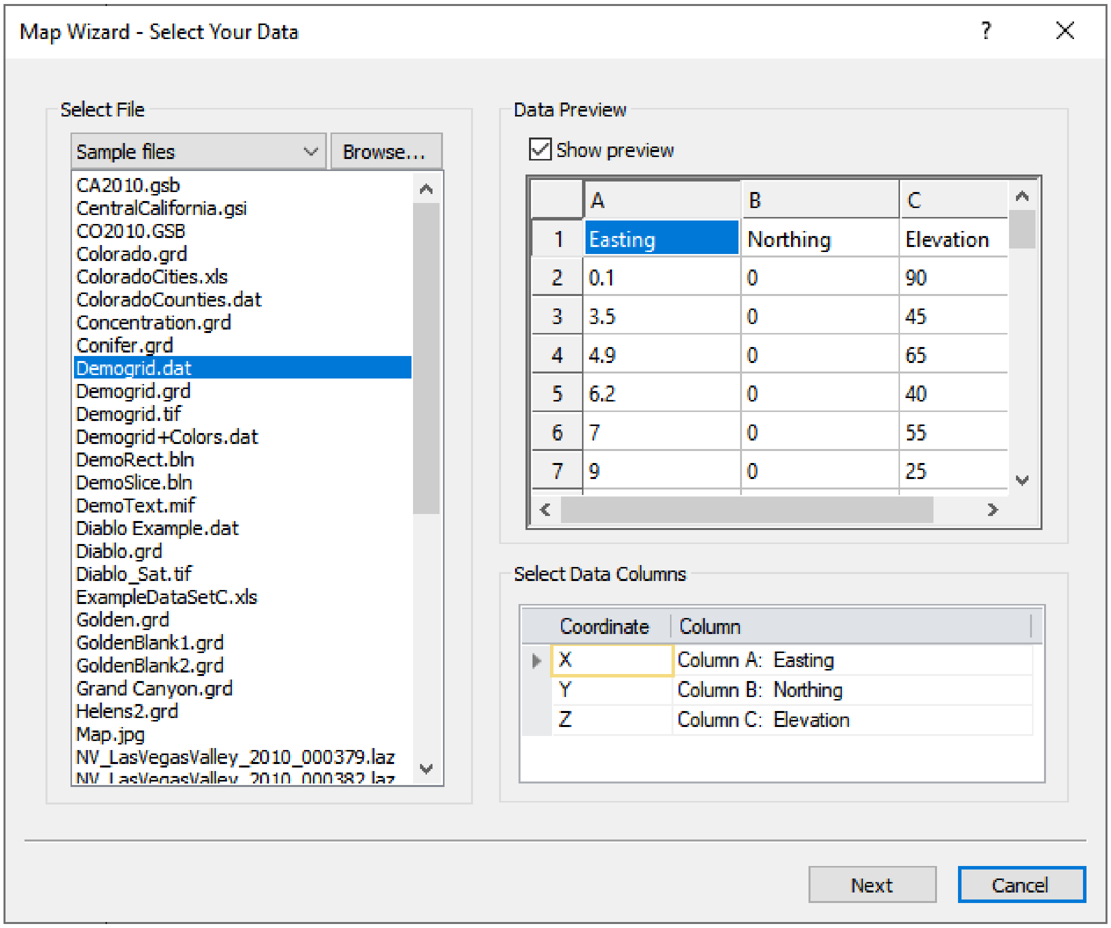
The **Map Wizard** consists of two or three pages, depending on the input data type. Click the following links for more information on each of the wizard pages.

1. Select Your Data - Select the data, boundary, or grid file you wish to use to create your map.
2. Select Your Map Type - Select one or more map layer types and decide to create a new map or place the new layers in an existing map.
3. Select Gridding Parameters - Create a grid file if you selected a data file and a grid-based layer in the first two steps.

A map is created, or layers are added to a map, in the plot window after proceeding through the **Map Wizard**.

Map Wizard - Select Your Data

Click the **Home | Wizard | Map Wizard** command to open the **Map Wizard**. The first page is the **Select Your Data** page. Here you select the data, grid, boundary, or image file you wish to use to create your map. Once you have selected your data source, click *Next* to proceed to the Select Your Map Type page.



Specify the data file and coordinate columns or grid file you wish to use to create a map.

Select File

Select your data, grid, vector, or image file in the *Select File* list. The *Select File* list includes a list of *Recent files*, *Sample files*, *Project files*, or files in a folder you select with *Browse*. To change the *Select File* list, click the current selection at the top of the list and select *Recent files* , *Sample files*, *Project files*, or *Browse*.

Recent Files

When *Recent files* is selected in the *Select File* list, the most recently opened files are displayed in the *Select File* list. By default the number of files in the *Recent files* list is 10. Change the number of files in the *Recent files* list in the Options dialog User Interface page.

Sample Files

When *Sample files* is selected in the *Select File* list, the included with **Surfer** are displayed in the *Select File* list.

Project Files

When *Project files* is selected in the *Select File* list, the files located in the user defined project folder are displayed in the *Select File* list. The *Project folder* option can be set in the Options dialog on the General page or in the Welcome to Surfer dialog.

Browse

Click *Browse* near the *Select File* list to open a file with the Open dialog. After you select a file in the **Open** dialog, the file preview is loaded in the **Select Your Data** page and the folder contents are visible in the *Select Files* list.

Import or Load Options

Depending on the data file that is selected, an additional dialog may appear that requires further selection. For example, if selecting a multi-tabbed spreadsheet, you may be prompted to select one of the sheets of the spreadsheet data. If selecting a LiDAR data file, you are prompted to load the file as *Point cloud* or *Data points* data. When loading LiDAR data as a point cloud, only the point cloud layer type can be created. When loading LiDAR data as data points, the data is treated like any other data file, and any layer type, except points cloud layers, can be created.

Data Preview

The *Data Preview* section contains a preview of the selected file. Check the *Show preview* check box to see a preview of the selected data, grid, or boundary file. When a file is selected in the *Select File* list, the file is loaded and a preview is displayed. The **Surfer** status bar displays the loading progress for large data files.

- Data files are displayed in a table.
- Grid files are displayed as a color relief map.
- Boundary files are displayed as a base map.

Clear the *Show preview* check box to hide the preview.

Select Data Columns

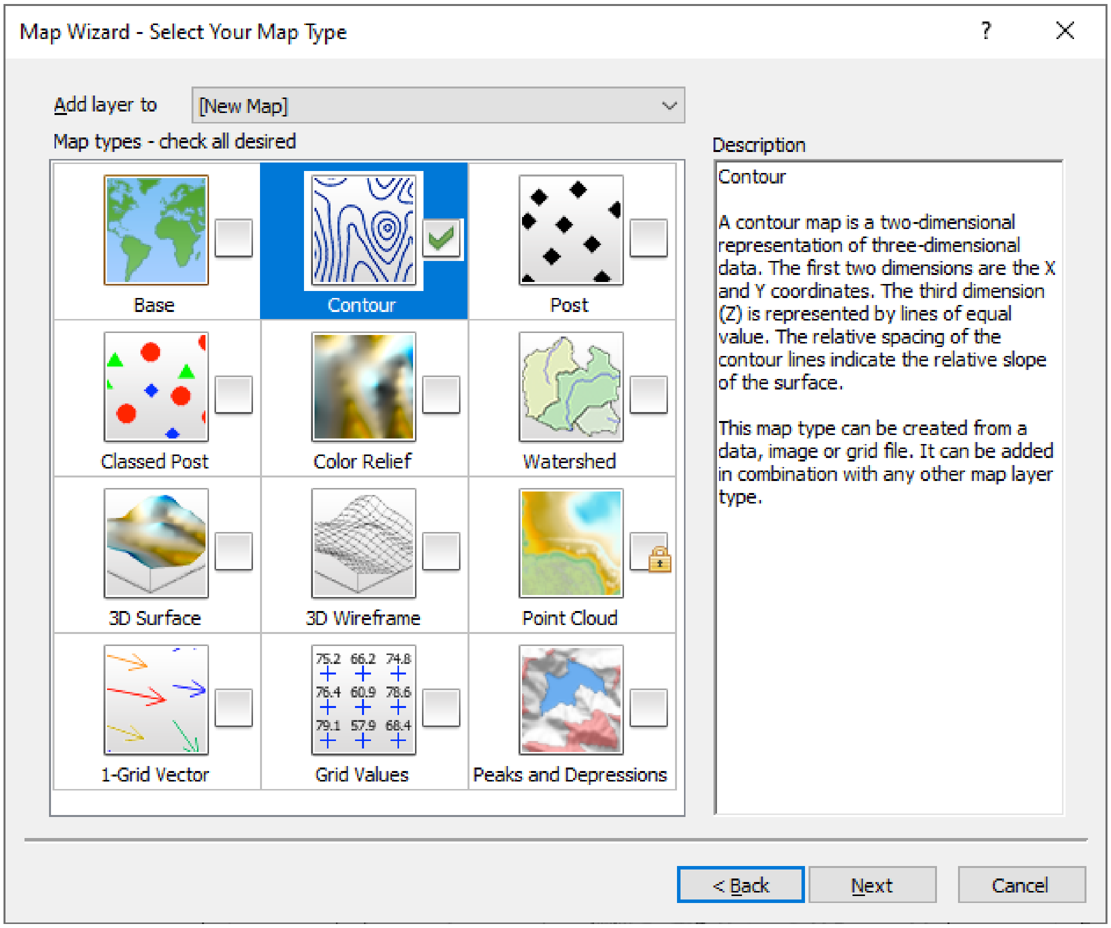
When a data file is selected in the *Select File* list, the *Select Data Columns* section is displayed. Specify the columns that contain the X, Y, and Z data in the *Select Data Columns* table. The *Coordinate* column indicates the row for the X, Y, and Z selections. The *Column* column includes the column letter and, if a header row is present, the column header. To change a selection, click the column you wish to changed and select the desired column from the list.

Next and Cancel

Click *Next* to proceed to the Select Your Map Type page. Click *Cancel* to close the **Map Wizard** without creating a map.

Map Wizard - Select Your Map Type

Click the **Home | Wizard | Map Wizard** command to open the **Map Wizard**. The second page is the **Select Your Map Type** page. Here you select the layer types you would like to create in your map and whether to create a new map or add the layers to an existing map. Click *Next* to proceed to the Select Gridding Parameters or click *Finish* to create your map. Which button is displayed depends on your selections in the Select Your Data and **Select Your Map Type** pages.



Specify the layer types you wish to create and select whether to create a new map or add the layer(s) to an existing map.

Add Layer To

The *Add layer to* option determines if the new map layer or layers will be added to a new map or an existing map. A new map is created by default. Select *[New Map]* to create a new map in the plot document. Alternatively, click the *Add layer to* field and select an existing map from the list to add the layers to an existing map.

Map Types

Select the desired layer types by clicking the map type. A check mark indicates a map layer will be included. Click on a selected layer type to clear the check box. A lock indicates the map type is not compatible with the source file selected on the Select Your Data page.

- Data files (including LiDAR data files) can be used to create any of the map layer types: Base, Contour, Post, Classed Post, Color Relief, 3D Surface, 3D Wireframe, Watershed, 1-Grid Vector, and Grid Values.
- Grid files can be used to create the grid-based map layers: Contour, Color Relief, 3D Surface, 3D Wireframe, Watershed, 1-Grid Vector, Peaks and Depressions, and Grid Values.
- LiDAR data files can be used to create the Point Cloud map layer. Select *Point Cloud* from the **LiDAR Load Options** dialog that appears after selecting a LiDAR data file.
- Boundary files can only be used to create a Base map. Try the **Home | New Map | Base** command to quickly create base maps without the **Map Wizard**.

The 3D wireframe layer type cannot be overlaid with a 3D surface or color relief layer. When a new selection cannot overlay with a previous selection, the previous selection is cleared.

Creating Grid Based Maps from Data Files

When a data file is selected on the Select Your Data page and a grid-based layer is selected on the **Select Your Map Type** page, a grid must be created before creating the map. The *Finish* button changes to *Next*. Click *Next* to proceed to the Select Gridding Parameters page and create the grid.

When the data source is a grid file or boundary file, the map can be created by clicking *Finish*.

Description

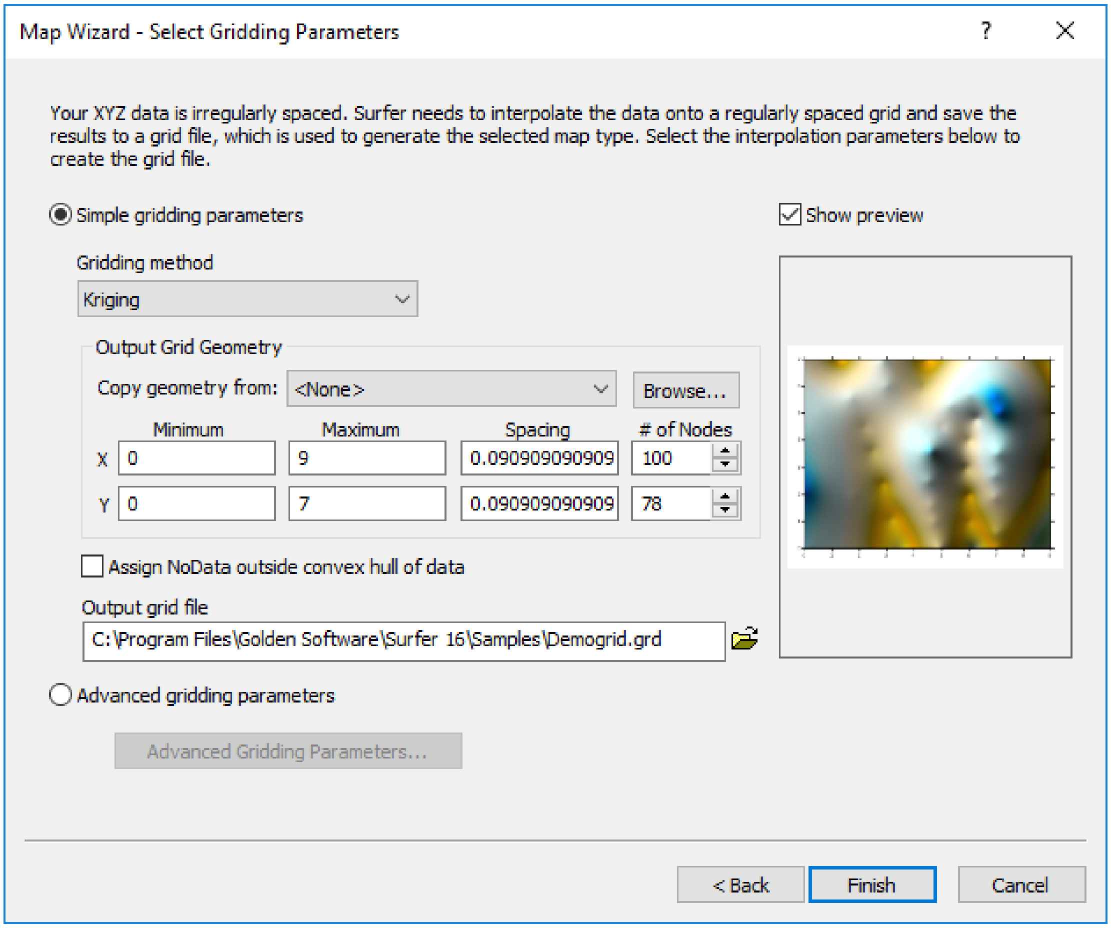
The *Description* section includes a brief explanation of the map type, as well as which data types are used to create the layer type and any conflicts with other layer types.

Back, Finish, Next, or Cancel

Click *Back* to return to the Select Your Data page if you wish to change the data source. Click *Finish* to create your map. When gridding is necessary, click *Next* to proceed to the Select Gridding Parameters page. Click *Cancel* to close the **Map Wizard** without creating a map.

Map Wizard - Select Gridding Parameters

Click the **Home | Wizard | Map Wizard** command to open the **Map Wizard**. The third page is the **Select Gridding Parameters** page. Here you specify the gridding parameters for creating the grid for the grid-based layer or layers. Once you have specified the gridding parameters, click *Finish* to create your grid file and map.



Specify gridding parameters for creating a grid from the source data file.

Simple Gridding Parameters

Select *Simple gridding parameters* to quickly generate a grid with the default gridding options. When *Simple gridding parameters* is selected, specify the Gridding method and output grid geometry.

Gridding Method

Surfer has several different gridding methods. These gridding methods define the way in which the XYZ data are interpolated when producing a grid file. Change the *Gridding method* by clicking the current selection and selecting the desired method from the list. Refer to the gridding methods for more information on the options.

Output Grid Geometry

The *Output Grid Geometry* section defines the grid limits and grid density. The *Output Grid Geometry* section also controls whether grid nodes outside the data are automatically assigned the NoData value. See the *Grid Data* topic for a grid geometry example.

Copy Geometry

The *Copy geometry from* option copies the grid geometry from an existing map layer or grid file. This option is useful when creating grids that will become overlaid map layers, processed with the Grid Math command, or used to calculate a volume between two surfaces. The **Math** and **Volume** commands require the input grids to have the same geometry.

To copy the geometry from an existing layer, select the layer in the *Copy geometry from* list. To copy the geometry from a grid file, click *Browse* and select the file in the Open Grid dialog. Select <*None*> to return the *Output Grid Geometry* options to their default values and to manually edit the grid geometry.

Minimum and Maximum X and Y Coordinate (Grid Limits)

Grid limits are the minimum and maximum X and Y coordinates for the grid. **Surfer** computes the minimum and maximum X and Y values from the XYZ data file. These values are used as the default minimum and maximum coordinates for the grid.

Grid limits define the X and Y extent of contour maps, color relief maps, vector maps, 3D wireframes, and 3D surfaces created from grid files. When creating a grid file, you can set the grid limits to the X and Y extents you want to use for your map. Once a grid file is created, you cannot produce a grid-based map larger than the extent of the grid file. If you find you need larger grid limits, you must regrid the data. You can, however, read in a subset of the grid file to produce a map smaller than the extent of the grid file.

When either the X, Y, or Z value is in a date/time format, the date/time values are converted and stored in the grid as numbers.

Spacing and # of Nodes (Grid Density)

Grid density is usually defined by the number of columns and rows in the grid, and is a measure of the number of grid nodes in the grid. The *# of Nodes* in the *X Direction* is the number of grid columns, and the *# of Nodes* in the *Y Direction* is the number of grid rows. The direction (*X Direction* or *Y Direction*) that covers the greater extent (the greater number of data units) is assigned 100 grid nodes by default. The number of grid nodes in the other direction is computed so that the grid nodes *Spacing* in the two directions are as close to one another as possible.

By defining the grid limits and the number of rows and columns, the *Spacing* values are automatically determined as the distance in data units between adjacent rows and adjacent columns.

Assign NoData Outside Convex Hull

Check the box next to the *Assign NoData outside convex hull of data* to automatically assign the to the grid nodes outside the convex hull of the data. Leave the box unchecked to extrapolate the data to the minimum and maximum grid limits, regardless of whether data exists in these areas.

Output Grid File

Choose a path and file name for the grid file in the *Output grid file* option. You can type a path and file name, or click the  button to browse to a new path and enter a file name in the **Save Grid As** dialog.

Advanced Gridding Parameters

Select *Advanced gridding parameters* for full control over the gridding method, advanced options, and grid geometry. After selecting *Advanced gridding parameters*, click the *Advanced Gridding Parameters* button to set the gridding options in the Grid Data dialog. See the *Grid Data* topic for more information on the advanced gridding parameters.

Back, Finish, or Cancel

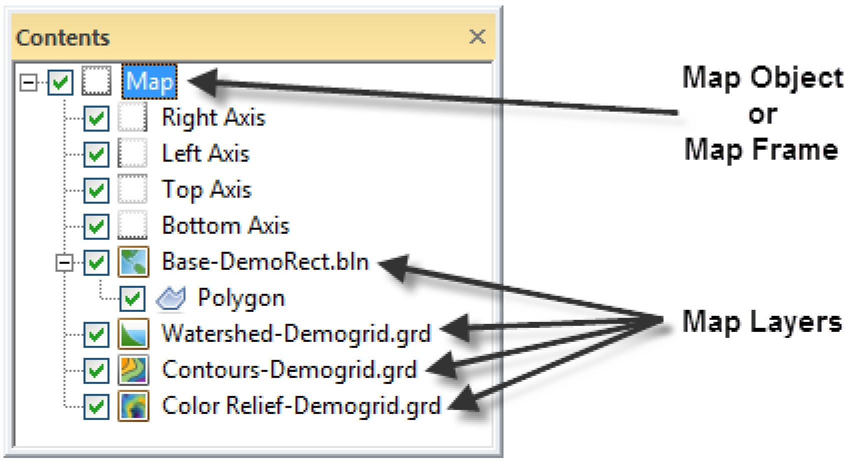
Click *Back* to return to the Select Your Map Type page. Click *Finish* to create the map or add the layers to the existing map. Click *Cancel* to close the dialog without creating a map.

Introduction to Map Layers

A map layer is a single map type contained in a larger map object. The map layer may be a contour layer, a post layer, a base layer, or any other layer type that **Surfer** can create. The larger map object contains all of the individual map layers and axes used to create the entire map. Map layers can be created as separate maps or added to a single map object.

It is possible to combine several maps created from related data to create one map object with multiple map layers. You can add any combination and number of contour, base, post, color relief, grid values, point cloud, drillhole, vector, watershed, viewshed, or 3D surface layers to a single map. However, a map can contain only one 3D wireframe layer, and the wireframe cannot be combined with 3D surface or color relief layers.

There are multiple ways to overlay map layers in **Surfer**. If you have multiple maps and wish to move only one layer, you can drag a map layer from one map object to another map object in the **Contents** window. If you wish to combine all the layers from multiple maps, you can select all of the maps and use the **Map Tools | Map Tools | Overlay Maps** command. This moves all of the map layers to a single map object. If you have already created a map and need to add map layers to it, you can select the map and use one of the **Home | Add to Map | Layer** commands to add a map layer to the existing map. Grid-based map layers can be changed to other layer types by selecting the object in the **Contents** window and using the **Map Tools | Layer Tools | Convert Layer** command.



This one map object contains four map layers. There is a base layer, watershed layer, contour layer, and a color relief layer.

Using Map Layers

When you use map layers, the layers use a single set of X, Y, and Z axes and the maps are positioned according to the map object coordinate system. If two or more maps use the same limits, they will overlay on top of one another. If maps cover adjacent areas, adding a map layer places them in the correct position relative to one another and creates a single set of axes that span the entire range. Layered maps become a single object and are moved and scaled as a single entity.

Consider a contour map and a base map that displays the outline of a lake on the contour map. The limits of the base map are the X, Y extents of the lake and are not the same as the contour map limits. If you create both the base map and the contour map in a single plot window as separate maps by using the **Home | New Map** commands for both maps, they do not overlay correctly because the maps have different scaling. In addition, each map uses a different set of X, Y axes. The two maps can be overlaid to correctly position the lake on the contour map by dragging the base map layer to the other map object that has the contour layer. The result will be a map object with a base map layer and contour map layer. Alternatively, if you create the contour map and then added a base map layer with the **Home | Add to Map | Layer | Base** command, the two maps are automatically scaled and combined into a single map using a single set of axes. The lake is correctly positioned on the contour map.

Sharing Data Between Layers

Surfer maintains a list of files currently being used along with the data that is shared from those files among the various layers. Grids, images and data files are recognized as being open in the background, thus making opening and using file information quicker. The **Open** dialogs for grids, images, and data files will have their respective *Open Grid, Open images, Open worksheet* field that shows the list of open files that can be shared.

Shared Files

Surfer tracks the files being used by maintaining a list of opened files for new maps and the files used in saved maps. The file(s) used to build maps are shown in a compiled list in the *Open Grids* field of the Open Grid dialog. The only exception is that vector data between base layers will not be shown.

Layers and 3D Wireframes

When you layer a contour, post, or base map on a 3D wireframe, the maps are draped over the wireframe. The wireframe is drawn in the usual fashion but the base, vector, or contour maps are "molded" over the top of the wireframe lines. Hidden lines are not removed from maps layered on wireframes. For example, contour lines are not hidden when the contour map lies over a wireframe.

Layers and 3D Surfaces

When you layer maps on top of 3D surface maps, hidden lines are removed and the maps are "molded" on the surface. Surface maps and images, vector files, and even other surface maps can be overlaid onto a single map object. The Overlays page in the surface properties dialog contains options for handling color in these cases.

Layer Exceptions

The **Map Tools | Add to Map | Layer** commands add a map layer to the selected map. Most combinations of map types can be layered. The exceptions are combining a 3D wireframe and 3D surface map, adding a raster map layer or point cloud to a wireframe, and adding multiple wireframe layers. Raster maps include color relief maps, surfaces, and base maps containing an image. The options under the **Add** command change to fit the existing map. For example, if a 3D wireframe map is selected, the **Map Tools | Add to Map | Layer | 3D Surface** command is grayed out.

Method 1: Adding a Map Layer to an Existing Map Frame

1. Create a new map with a **Home | New Map** command. For example, you can choose **Home | New Map | Contour** to create a contour map.

2. Select the map and click the **Map Tools | Add to Map | Layer** command to add a map layer. Select the map layer type to add to your existing map. For example, select the contour map and use the **Map Tools | Add to Map | Post** to add a post map layer to the contour map.
3. The maps are combined in the correct position based on their coordinates and limits. For example, in the **Contents** window, you will see one map object with a contour map layer and a post map layer.

Method 2: Overlaying Two Existing Maps

1. Create a new map with a **Home | New Map** command. For example, you can choose **Home | New Map | Contour** to create a contour map.
2. Create a second map with the **Home | New Map** command. You could create a post map with the **Home | New Map | Post | Post** command.
3. Note that each map has an independent set of axes.
4. Click **Home | Selection | Select All** to select both the contour and post maps.
5. Click **Map Tools | Map Tools | Overlay Maps**. The two maps are combined onto a single map object with a single set of axes. The empty map object is automatically deleted.

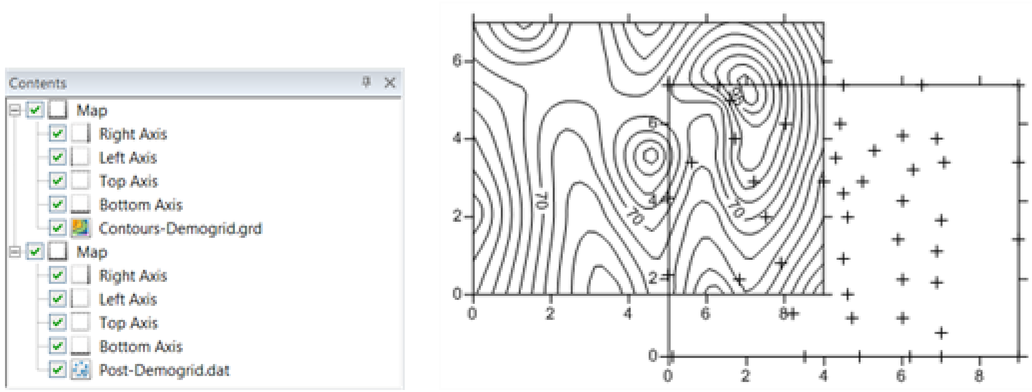
This method works well when you have multiple map layers that you want to combine.

Method 3: Combining Two Existing Maps in the Contents Window

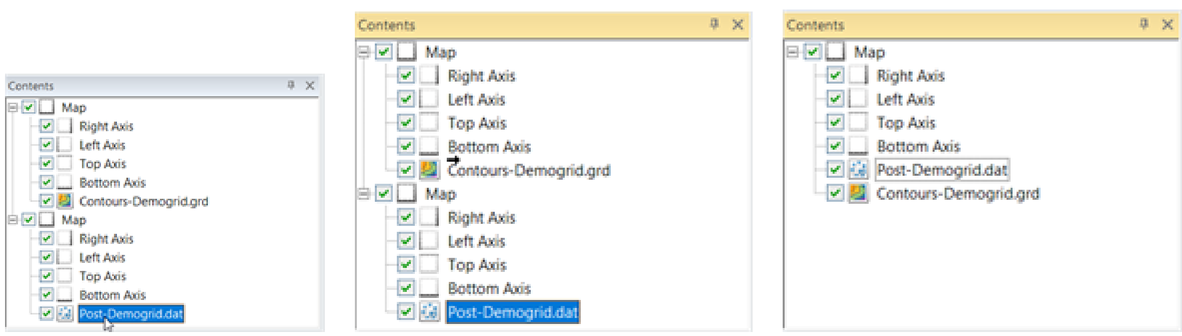
If two maps already exist, you can move (or overlay) a map layer from one map frame into the other map frame by dragging and dropping in the Contents window.

1. Create a new map with a **Home | New Map** command. For example, you can choose **Home | New Map | Contour** to create a contour map.
2. Create a second map with the **Home | New Map** command. You could create a post map with the **Home | New Map | Post | Post** command.
3. Note that each map has an independent set of axes.
4. Select the post map layer in the **Contents** window and drag it to the contour map object. To do this, left-click and hold the left mouse button while you drag the map layer to a new map frame. When the cursor changes to a horizontal arrow, release the left mouse button and the map layer is added to the contour map's map frame. The post map will now be overlaid on the contour map. An empty map frame may remain after removing the last map layer from the map object, depending on your options.
5. If an empty map frame exists, select the empty map frame and press DELETE on the keyboard to remove the empty map frame. The end result is a single map object with two map layers: a post map layer and a contour map layer.

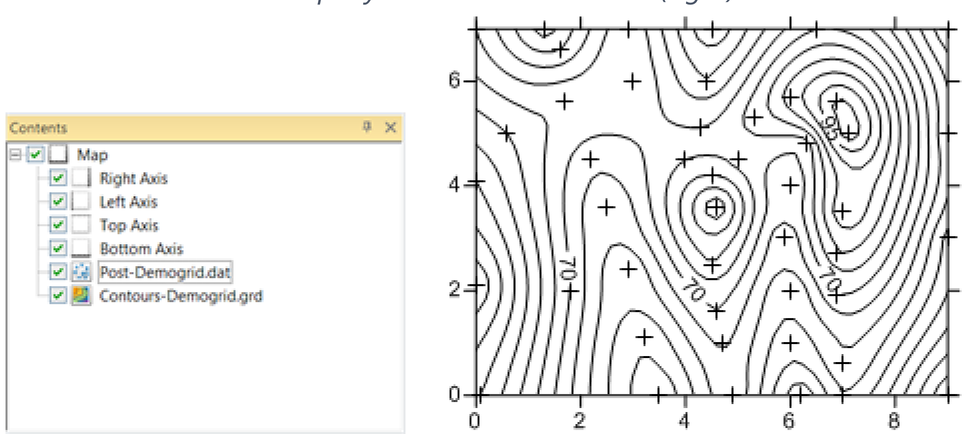
Additional map layers can be added with the **Map Tools | Add to Map** commands.



*The contour map layer and the post map layer are displayed in separate map objects in the **Contents** window and the plot window.*



First left-click and select the post map layer (left), then drag the post map layer to the other map object. When the cursor is a horizontal arrow (middle), release the mouse button to drop the map layer in the new location (right).



The result of this method is one Map object with two map layers.

Layer Map Limits

If a map layer is added to a map frame and the map layer exceeds the current map limits, a **Surfer** warning message will be displayed allowing you to adjust the map limits to include all layers. Select *Yes* to adjust the map to include all layers. Select *No* to leave the current map limits.

Editing a Map Layer

To edit individual layers in a multi-layer map, select the map layer (i.e. *Contours*) in the plot window or Contents window and use the **Properties** window to edit the properties. Make the desired changes in the map layer properties, and the map layer is redrawn with the specified changes.

Hiding a Map Layer

After adding map layers, it is possible to hide one or more of the layers. To temporarily hide a map layer, uncheck the visibility box next to the map layer name (i.e. *Contours*) in the Contents window. The map is redrawn without the selected overlay. To make the overlay visible again, recheck the visibility box. Note that if a surface is made invisible, the overlays are also made invisible.

Removing a Map Layer

Select the map layer and use the Break Apart Layer command to remove a map layer from a map object. Alternatively, right-click on the map layer and select **Break Apart Layer**.

Deleting a Map Layer

To delete a map layer from a map frame, select the map layer in the **Contents** window and press the DELETE key on the keyboard. Alternatively, you can select the map layer and use the **Home | Clipboard | Delete** command, or right-click the map layer and select **Delete**.

Coordinate Systems

A coordinate system is method of defining how a file's point locations display on a map. Different types of coordinate systems exist that control how the coordinates are shown on the map. In **Surfer**, a map can be unreferenced in local coordinates, referenced to a geographic latitude and longitude coordinate system, or referenced to a known projection and datum. Each data set, grid, map layer, and the map frame can have an associated coordinate system. All coordinate systems for individual layers are converted “on the fly” to the map’s target coordinate system. This allows maps with different coordinate systems to be easily combined in **Surfer**.

A local coordinate system generally is considered unreferenced. A local system has a location that begins numbering at an arbitrary location and increments numbers equidistant in the X and Y directions from this location. This is frequently referred to as a *Cartesian coordinate system*. The distance units can be specified for an unreferenced local system in the **Assign Coordinate System** dialog.

A *Geographic* coordinate system uses a spherical surface to define locations on the earth. Geographic coordinate systems are commonly called unprojected lat/long. **Surfer** has several predefined geographic coordinate systems available. Each system has a different datum. The same latitude and longitude value will plot in different locations depending on the datum.

A *Projected* coordinate system consists of a projection and a datum. Each projection distorts some portion of the map, based on the ellipsoid and datum specified. Coordinates can be lat/long, meters, feet, or other units. Different projections cause different types of distortion. It is recommended that you do not use projected coordinate systems if you do not need to convert between coordinate systems or if all your data are in the same coordinate system.

Map Coordinate System Overview

In **Surfer**, data, grids, map layers, and maps can have an associated coordinate system. All coordinate systems defined by the data, grids, and map layers are converted "on the fly" to the map's target coordinate system. This allows maps with different coordinate systems to be easily combined in **Surfer**.

The standard procedure for creating maps in a specific coordinate system is as follows:

1. Create the map by clicking on the appropriate **Home | New Map** command.
2. In the **Open Grid(s)** dialog, select the file to open and click *Open*.
3. In the **Contents** window, click on the map layer to select it.
4. In the **Properties** window, click on the **Coordinate System** page.
5. If the *Coordinate system* is not correct, click the *Set* button next to *Coordinate System*. The **Assign Coordinate System** dialog opens. This is the initial coordinate system for the map layer, i.e. the coordinate system for the source data. Select the correct coordinate system in the dialog. When finished making changes, click *OK*.
6. To change the target coordinate system for the map, click on the *Map* object in the **Contents** window. In the **Properties** window, click on the **Coordinate System** tab. This is the coordinate system in which you want the map to display.
7. Click on the *Change* button next to *Coordinate System* to set the desired target coordinate system. When finished, click *OK*.
8. All of the map layers are converted on the fly to the target coordinate system. The entire map is now displayed in the desired coordinate system.

Surfer does not require a map coordinate system be defined. Maps can be created from unreferenced data, grid, and map layers. As long as all map layers have the same X and Y ranges, coordinate systems do not need to be specified. If you do not specify a source coordinate system for each map layer, it is highly recommended that you do not change the target coordinate system. Changes to the target coordinate system for the map can cause the unreferenced map layers to appear incorrectly or to not appear.

3D surface maps and wireframe maps cannot be converted to a new coordinate system.

File Menu Commands

The **File** menu in the plot document has the following commands:

New	Creates a new plot window or new worksheet window
Open	Opens a file in a plot document
Open in Worksheet	Opens a data file in a worksheet
Close	Closes the active document
Close All	Closes all open documents
Save	Saves the active document
Save As	Saves the active document window to a new file name or location
Import	Imports vector or graphics files
Export	Exports vector or graphics files
Reload Map Data	Reloads external data used to generate maps
Reload Layer Data	Reloads only external data used to generate the selected layer
Page Setup	Set the page options for the plot, or set the page and printing options for the worksheet
Print	Prints the active plot document or worksheet
Options	Set **Surfer** options and default properties in the plot window
Defaults	Set advanced default settings
Customize Ribbon	Customize the ribbon, quick access toolbar, and keyboard commands
Online	Check for Update or open the Golden Software Home Page, Surfer Product Page, or Frequently Asked Questions
Feedback	Send a Problem Report, Suggestion, or Information Request
Recent Documents	Open a recent plot, worksheet, or grid file
License Info	Activate a product key, connect to a license server, or view license information

About Surfer	View version information, system information, and contact information
Exit	Closes **Surfer**

New Plot

You can create a new plot document or worksheet with the **File | New** command or click the button. Click the **File | New | Plot** command, or click the button in the Quick Access Toolbar, to create a new plot document. Maps are created in a plot document.

New Worksheet

You can create a new plot document or worksheet with the **File | New** command. The worksheet document is used to manage data. Click the **File | New | Worksheet** command, or click the button in the Quick Access Toolbar, to create a new worksheet window.

Open

The **File | Open** command opens a file as a map into a new plot window. You can also click on the button on the Quick Access Toolbar, or press CTRL + O on the keyboard to open files.

Grid files open grid-based maps versus data files which open data-based maps. The default for grid files is set to the Color Relief Map whereas for data files the default is the Post Map. The default map types for grid files and data files can be changed by the user by clicking on the File | Options command and navigating to the General page.

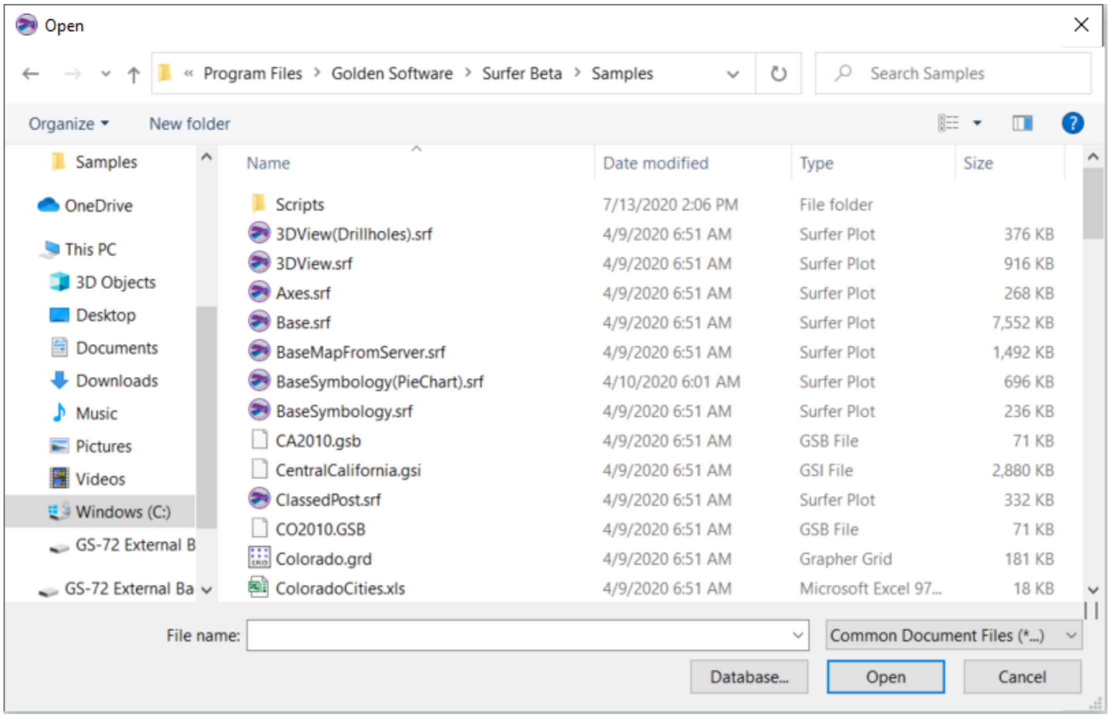
Other file formats open as specific map types:

- **Surfer** .SRF files open in a new plot window.
- Image files open as Base maps.
- LAS files open as a point cloud layer map.
- Vector files open as Base (vector) layer maps.

Instead of opening data files as maps, they may be opened as worksheets by clicking on the File | Open in Worksheet command. In the **Open** dialog, the file types shown are limited to *Common Data Files (*...)*.

Open Dialog

Click the **File | Open** command in the plot document, worksheet document, or grid node editor to open the **Open** dialog. Hold down the CTRL key while clicking on files to select multiple files, or hold down the SHIFT key to select adjacent files. Then click *Open* and the maps will open in a single plot window.



Select the file to open in the ***Open*** *dialog.*

Look In

The *Look in* field shows the current directory. Click the down arrow to see the directory structure and click on the folders to change directories.

Creating New Folders and Changing the View

The buttons to the right of the *Look in* field allow you to create new folders and change the view of the file list.

File List

The file list displays files in the current directory. The current directory is listed in the *Look in* field. The *Files of type* field controls the display of the file list. For example, if *Golden Software Data (*.DAT)* is listed in the *Files of type* field only [*.DAT] files appear in the files list.

Specify a File Name

The *File name* field shows the name of the selected file, or type a path and file name into the box to open a file.

Specify a File Type

The *Files of type* field shows the file format to be opened. To change the file format, click the down arrow and select the file type from the list. *All Files (*.*)* display all files in a directory.

The *Common Document Files (*...)* format type is selected when clicking the **File | Open** command. This displays all the common file formats in the navigation pane. If a different format type is selected, **Surfer** will remember the setting until the end of the current session. When **Surfer** is restarted, the default format type will be used.

The *Common Data Files (*...)* format type is selected when clicking the **File | Open in Worksheet** command.

To see all files in the directory, choose *All Files (*.*)* from the *Files of type* list. Double-click on a file to open it or single click the file and then click the *Open* button. The *All Files* shows all of the file formats even if the file type is not appropriate for the action chosen (i.e. displaying a data file when creating a grid based map that requires a grid file).

Import Database

Click the *Database* button in the **Open** dialog to open the Data Link Properties dialog and import a database.

Data Link Properties

You can open database files in **Surfer** by clicking *Database* in the Open and Open Data dialogs or *Load Database* in the Import Data dialog. The **Data Link Properties** dialog will open. Imported databases appear in a new worksheet window. Once the worksheet is saved, the link to the database is removed.

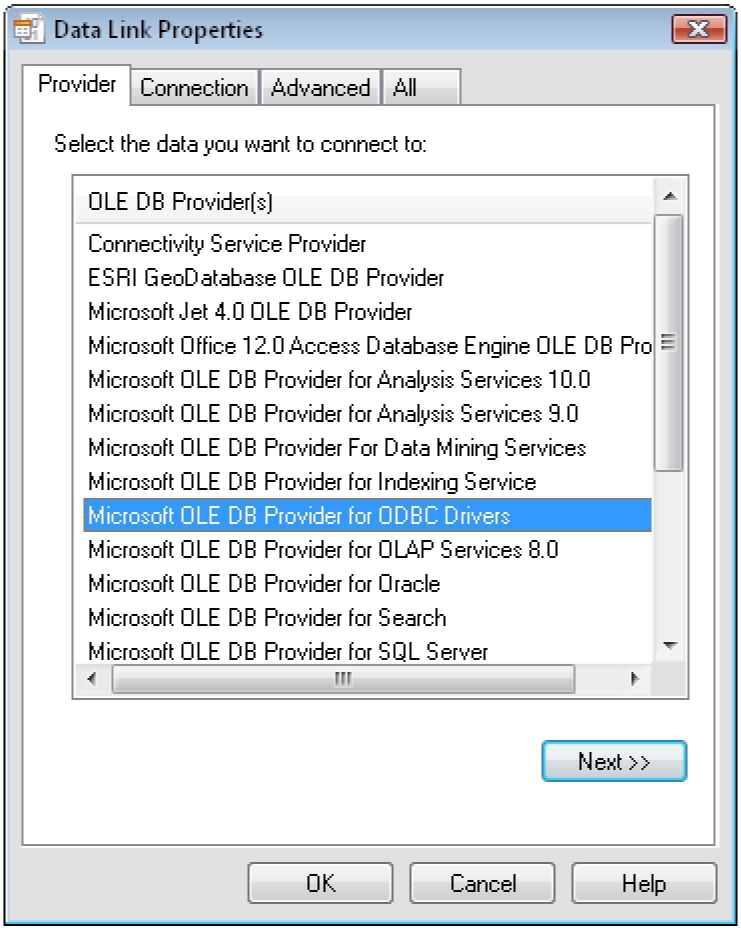
The data linking provides a method to link virtually any database supported by Microsoft via an OLE DB Provider, ODBC, or some other supported format. Because the data link provides access to many types of databases that vary by computer, and since this link is provided by Microsoft, only general information is provided here. Click *Help* in the **Data Link Properties** dialog to access Microsoft's help file for this dialog. If you need additional information on specific connection information, contact your network administrator.

ODBC Drivers

Note that ODBC drivers are installed by the database program, not by **Surfer**. If a database program is not installed on the computer, a driver may still be able to be located to import the data from the database. For example, the Microsoft Access database engine can be downloaded from Microsoft's website.

Data Link Properties Dialog

Data linking is used to define links to many types of databases. The **Data Link Properties** dialog is accessed by clicking the *Database* button in the **Open** or **Open Data** dialogs or the *Load Database* button in the **Import Data** dialogs. The **Open** dialog appears when you use the File | Open command. The **Import Data** dialog appears when you use the worksheet File | Import command. The **Open Data** dialog appears when you use the Grid Data, New Variogram, Classed Post Map, or Post Map command.



*Use the **Data Link Properties** dialog to define links to a database.*

Help Button

Click the *Help* button in the **Data Link Properties** dialog to access Microsoft's help file for this dialog. If you need additional information on specific connection information, contact your network administrator.

Data Linking

To use data linking,

1. Click the File | Open command or one of the other commands listed above.
2. Click *Database* in the Open and Open Data dialogs or *Load Database* in the Import Data dialog.
3. The **Data Link Properties** dialog opens, with the *Provider* tab active.
4. Step through the dialogs to import the database data.

Provider Tab

Use the **Provider** tab to select the appropriate OLE DB provider for the type of data you want to access. Not all applications allow you to specify a provider or modify the current selection; this tab is displayed only if your application allows the OLE DB provider selection to be edited. You can save a data link with the application itself or as a separate file. For more information about creating a data link, refer to the Microsoft help.

Option	Description
OLE DB Provider(s)	Lists all OLE DB providers detected on your computer. For more information about providers, see "Microsoft OLE DB Providers Overview" in the MDAC SDK.
Next	Opens the Connection tab for the selected OLE DB provider.

Note You can navigate directly to the **Connection** tab by double-clicking the desired provider. For more information about Data Links, see the Data Link API Reference.

Click **Next** or click the **Connection** tab to specify how to connect to ODBC data.

Connection Tab

This **Connection** tab is provider-specific and displays only the connection properties that are required by the Microsoft OLE DB Provider for ODBC.

Option	Description
Use data source name	Select from the list, or type the ODBC data source name (DSN) you want to access. You can add more sources through the ODBC Data Source Administrator. Refresh the list by clicking Refresh.
Use connection string	Allows you to type or build an ODBC connection string instead of using an existing DSN.
Build	Opens the Select Data Source dialog box. Once you select a data source the connection string in that data source will appear in the Connection tab of the Data Link Properties dialog. If you select a File DSN, the resulting ODBC connection string is not based on a DSN. The ODBC connection string is persisted in the data link (.udl) file and does not rely on the selected file DSN. If you select a Machine DSN, the resulting ODBC connection string is based on a DSN. The ODBC connection string references the selected machine DSN. If a user on a different system attempts to access the data link (.udl) file, the user must also have the machine DSN installed.

User name	Type the User ID to use for authentication when you log on to the data source.
Password	Type the password to use for authentication when you log on to the data source.
Blank password	Enables the specified provider to return a blank password in the connection string.
Allow saving password	Allows the password to be saved with the connection string. Whether the password is included in the connection string depends on the functionality of the calling application. Note If saved, the password is returned and saved unmasked and unencrypted.
Enter the initial catalog to use	Type in the name of the catalog (or database), or select from the drop-down list.
Test Connection	Click to attempt a connection to the specified data source. If the connection fails, ensure that the settings are correct. For example, spelling errors and case sensitivity can cause failed connections.

Note: For more Data Link connection information, see the Data Link API Reference.

Advanced Tab

Use the **Advanced** tab to view and set other initialization properties for your data.

The **Advanced** tab of the **Data Link Properties** dialog box is provider-specific and displays only the initialization properties required by the selected OLE DB provider. For more information about advanced initialization properties, see the documentation provided with each specific OLE DB provider.

The following table describes most initialization options.

Option	**Description**

Impersonation level	The level of impersonation that the server is allowed to use when impersonating the client. This property applies only to network connections other than Remote Procedure Call (RPC) connections; these impersonation levels are similar to those provided by RPCs. The values of this property correspond directly to the levels of impersonation that can be specified for authenticated RPC connections, but can be applied to connections other than authenticated RPCs. Select from the following levels: • Anonymous—The client is anonymous to the server. The server process cannot obtain identification information about the client and cannot impersonate the client. • Delegate—The process can impersonate the client's security context while acting on behalf of the client. The server process can also make outgoing calls to other servers while acting on behalf of the client. • Identify—The server can obtain the client's identity. The server can impersonate the client for Access Control List (ACL) checking but cannot access system objects as the client. • Impersonate—The server process can impersonate the client's security context while acting on behalf of the client. This information is obtained when the connection is established, not on every call.

Protection level	The level of data protection sent between client and server. This property applies only to network connections other than RPC connections; these protection levels are similar to those provided by RPCs. The values of this property correspond directly to the levels of protection that can be specified for authenticated RPC connections, but can be applied to connections other than authenticated RPCs. Select from the following levels: • Call—Authenticates the source of the data at the beginning of each request from the client to the server. • Connect—Authenticates only when the client establishes the connection with the server. • None—Performs no authentication of data sent to the server. • Pkt—Authenticates that all data received is from the client. • Pkt Integrity—Authenticates that all data received is from the client and that it has not been changed in transit. • Pkt Privacy—Authenticates that all data received is from the client, that it has not been changed in transit, and protects the privacy of the data by encrypting it.
Connect timeout	Specifies the amount of time (in seconds) that the OLE DB provider waits for initialization to complete. If initialization times out, an error is returned and the connection is not created.
Access permissions	Select one or more of the following permissions: • **Read** - Read only. • **ReadWrite** - Read and write. • **Share Deny None** - Neither read nor write access can be denied to others. • **Share Deny Read** - Prevents others from opening in read mode. • **Share Deny Write** - Prevents others from opening in write mode. • **Share Exclusive** - Prevents others from opening in read/write mode. • **Write** - Write only.

Note For more Data Link initialization information, see the Data Link API Reference.

All Tab

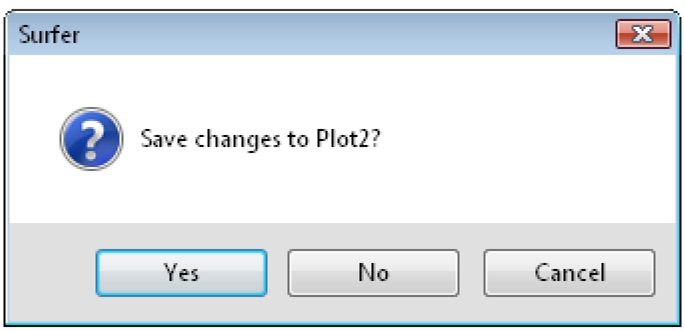
Use the **All** tab to view and edit all OLE DB initialization properties available for your OLE DB provider. Properties can vary depending on the OLE DB provider you are using. For more information about the initialization properties, refer to the documentation provided with each specific OLE DB provider.

Option	Description
Initialization properties list	Lists all properties and their current values.
Edit Value	Opens the Edit Property Value dialog box for the selected property.

Note: You can open the **Edit Property Value** dialog box by double-clicking the desired property. For more Data Link initialization information, see the Data Link API Reference.

Close

Clicking the **File | Close** command or the button, right-clicking on a document tab and selecting **Close**, or clicking the X next to the tab name closes the active window. If you have not saved the current changes, you are prompted to save changes before the window closes.



Specify if you would like changes saved in the ***Surfer*** *dialog.*

Yes

Click *Yes* to save changes and then close the window. If the file has not been previously saved the Save As dialog appears.

No

Click *No* to close the document without saving changes.

Cancel

Click *Cancel* to return to the active document window.

Open In Worksheet and Open Data

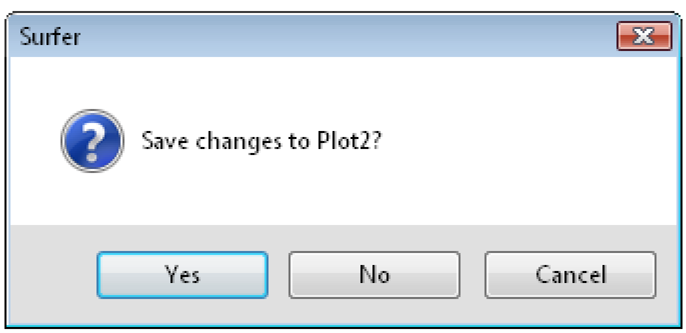
The **File | Open in Worksheet** command or the button opens a data file in a worksheet window. In the Open Data dialog, the file types shown are limited to *Common Data Files (*...)*. Compare this with the File | Open command and its **Open** dialog which opens maps, not worksheets, regardless of the file type. Hold down the CTRL key while clicking on files to select multiple files, or hold down the SHIFT key to select adjacent files. Then click *Open* and the data files will open in separate worksheet windows.

When a worksheet is already open, you can click the **Data | Edit | Open Data** command to open a data file. The **Open** dialog that opens for **Data | Edit | Open Data** is same as that for the **File | Open in Worksheet** dialog.

Close All

Click the **File | Close All** command or the button to close all plot, worksheet, and grid documents in **Surfer** without exiting the application. You are prompted to save or discard any changes in each document before the window closes.

Click the application control close button or File | Exit to close all documents and exit **Surfer**.



*Specify if you would like changes saved in the **Surfer** dialog.*

Yes

Click *Yes* to save changes and then close the window. If the file has not been previously saved the Save As dialog appears.

No

Click *No* to close the document without saving changes.

Cancel

Click *Cancel* to return to the active document window.

Save

The **File | Save** command writes information to disk using the current file name and type. You can also click the button on the quick access toolbar to save a file. If the file has not yet been saved, the Save As dialog is displayed so you can give the file a name and select the file type. If you would like to save an existing file to a new file name or change the file type, choose File | Save As instead of **File | Save**.

In the plot document, the **Save** command saves information in the **Surfer** .SRF file format that is not recognized by other applications. When you want to use **Surfer** information in other applications, you can use the File | Export command to create files in several useful formats.

Use Caution when Saving Excel Files!

A file can be saved in an Excel format from **Surfer**, but only one worksheet can be saved. **Surfer** does not allow for saving multiple worksheets in a single Excel document. If a multi-worksheet Excel document is opened and saved as an .XLS file from **Surfer**, be aware that only the single worksheet will be saved in the document. All the unused worksheets will be lost. In this case, a warning message is issued.

File Names, Formats, and File Extensions

When a worksheet file is saved, the file format can be specified by typing the appropriate extension on the file name. If the needed file is an ASCII DAT file, type a file name such as MYDATA.DAT. The .DAT extension tells the worksheet to save the file as an ASCII DAT file.

If the extension is not included in the file name the format is determined by the *Save as type* field. For example, if the name MYDATA is typed into the file name field and theSave as typefield is set toExcel Spreadsheet (*.XLS), the file is saved as

The file can be saved with any extension by enclosing the file name in double quotes. The file is saved with the name and extension typed in the file name box, but it is saved in the format specified in the *Save as type* field. For example, type the name (with quotes) "MYDATA.ABC in the file name box. If theSave as typefield is set toComma Separated Variables (*., the file is saved as

Surfer Files

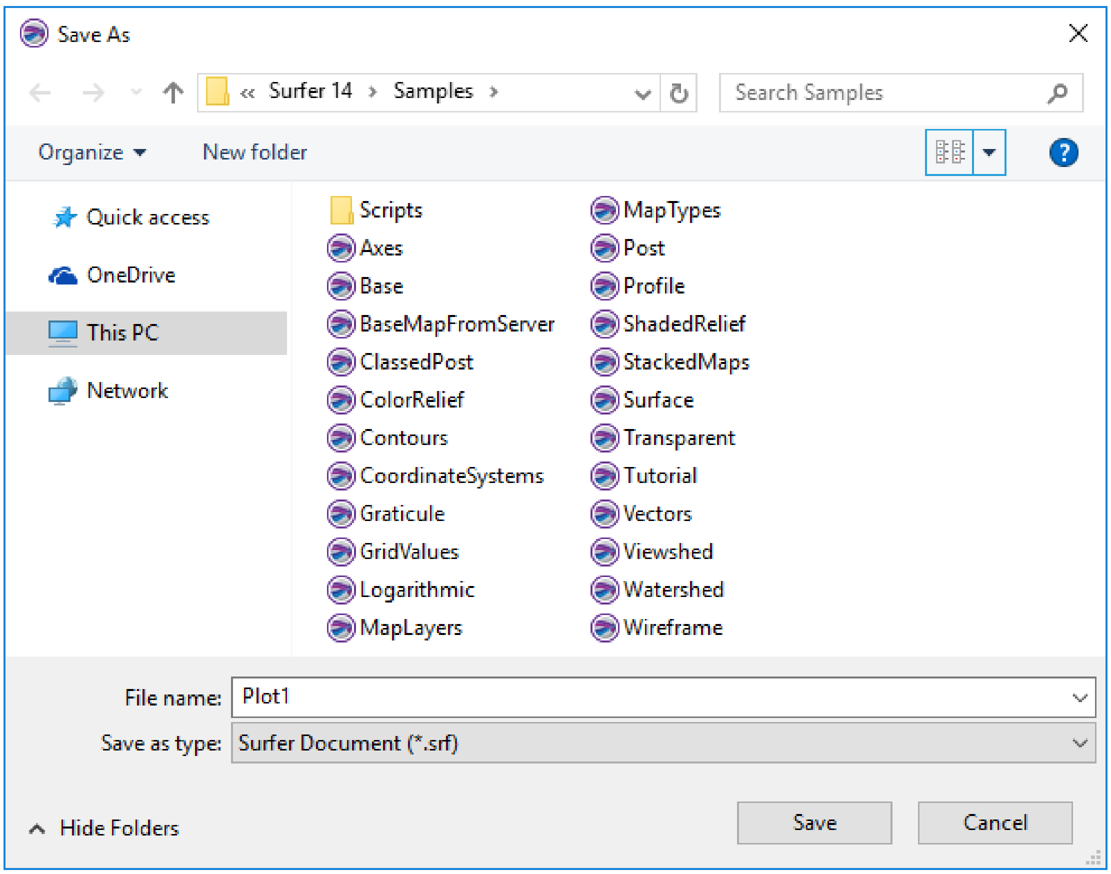
Surfer .SRF files preserve all the objects and object settings contained in a plot window. These files are called **Surfer** .SRF files throughout the documentation. **Surfer** can open .SRF files from previous versions as far back as **Surfer 7**. **Surfer** can save files to previous .SRF formats for sharing with other users. For example, the *Surfer 15 Document .SRF* file type can be opened in **Surfer 15**, but does not contain features that are in later **Surfer** versions. Beginning with version 16, the *Surfer Plot (*.srf)* file type is backwards compatible with all **Surfer** versions **16** and newer.

Save As

The **File | Save As** command or the button saves a new document or saves a modified document with a new file name. The **File | Save As** command in the plot document and worksheet document opens the **Save As** dialog. The **File | Save As** command in the grid editor opens the Save Grid As dialog.

Save As Dialog

The **Save As** dialog is displayed when saving a document with the **File | Save As** command.



*Specify the save location, file name, and file type in the **Save As** dialog. This graphic may look different, depending on the operating system.*

Save In

The *Save In* field shows the current directory. Click the down arrow to see the directory structure and click on the folders to change directories.

Button Shortcuts

The buttons to the right of the *Save in* field allow you to create new folders and change the view of the file list.

File List

The *File list* displays the files using the extension specified in the *Save as type* box. A file can be overwritten by selecting it from the file list.

File Name

The *File name* box displays the name of the selected file, or type in the path and file name of the file to be saved.

Save As Type

Select the file format in the *Save as type* list.

File Types

The available file types to save as or export depend on the location from you are exporting.

- Save **Surfer** files (.SRF) in the plot document with **File | Save As**. *Surfer Plot (*.srf)* files created in **Surfer** can be opened in 16 and later versions. The *Surfer Plot (*.srf)* file type will be backward compatible with every Surfer version back to **Surfer 16** as new versions are released. It is only necessary to save back to a specific file version if you wish to open the file in **Surfer 15** or older versions.
- Save version-specific backward compatible **Surfer** files for versions prior to **Surfer 16** by selecting *Surfer 11 Plot (*.srf)*, *Surfer 12 Plot (*.srf)*, *Surfer 13 Plot (*.srf)*, *Surfer 14 Plot (*.srf)*, and *Surfer 15 Plot (*.srf)* files can be opened in previous versions of **Surfer**. For example *Surfer 11 Plot (*.srf)* files can be opened by **Surfer 11** and newer versions. New features are removed from the file when saving in a previous **Surfer** version format.
- Save BLN, BNA, CSV, DAT, TXT, SLK, XLS, XLSX files in the Worksheet Document with **File | Save As**.

File Names, Formats, and File Extensions

When a worksheet file is saved, the file format can be specified by typing the appropriate extension on the file name. If the needed file is an ASCII DAT file, type a file name such as MYDATA.DAT. The .DAT extension tells the worksheet to save the file as an ASCII DAT file.

If the extension is not included in the file name the format is determined by the *Save as type* field. For example, if the name MYDATA is typed into the file name field and theSave as typefield is set toExcel Spreadsheet (*.XLS), the file is saved as

The file can be saved with any extension by enclosing the file name in double quotes. The file is saved with the name and extension typed in the file name box, but it is saved in the format specified in the *Save as type* field. For example, type the name (with quotes) "MYDATA.ABC in the file name box. If theSave as typefield is set toComma Separated Variables (*., the file is saved as

Use Caution when Saving Excel Files!

A file can be saved in an Excel format from **Surfer**, but only one worksheet can be saved. **Surfer** does not allow for saving multiple worksheets in a single Excel document. If a multi-worksheet Excel document is opened and saved as an .XLS file from **Surfer**, be aware that only the single worksheet will be saved in the document. If the existing file is overwritten all the unused worksheets will be lost. In this case, a warning message is issued.

Check for Update

Click the **File | Online | Check for Update** command or the button to check for the most recent version of **Surfer**. If there is an update available, you can follow the directions to download and install the free update. An update contains minor changes to the program. Normally new features are added in updates. See the Golden Software website version information page for a list of changes. Surfer can be updated to the latest minor version regardless of maintenance status.

Before using this command, make sure your computer is connected to the internet. Follow the directions in the dialog to complete the update if an update is available. If you have any difficulties with the update, please contact technical support.

Automatic Update

Surfer automatically checks for available updates and notifications are presented through the application. This preference can be adjusted at any time using the **File | Options** command. The user can prompt Surfer manually to check for updates from the **File | Options** command.

Check for Internet Update

- Click the **File | Online | Check for Update** command, the **Internet Update** dialog appears. If you are running in Windows 7 as a user, a dialog appears asking for Administrator permissions. Click the *Allow* button.
- Click the *Next* button to proceed. **Surfer** will attempt to connect to the Golden Software server and check if an update exists for your version of the product.

- If no update exists and/or you are already running the latest version, a dialog will appear letting you know there are no updates for your current version of **Surfer**. Click the *OK* button and the **Internet Update** dialog will close.
- If an update is available, the dialog will inform you about the specifics of the update. Click *Next* to download the update file. A progress gauge displays. If you choose not to download the update at this time, click *Cancel* . It is highly advised that updates be installed when available as updates contain corrections to known issues in the software.
- When the download is complete, the **Install Updates** dialog will appear.
- Save any changes to your work and exit the **Surfer** program by clicking the **File | Exit** command. Click *Install* to proceed with the update.
- After the update is installed successfully, you can open **Surfer** and continue working.

Upgrade

If your maintenance license is active, you can upgrade to the next major version (e.g. **Surfer** 23 to **Surfer** 24) by clicking the **File | Online | Check for Update** command. If your maintenance has expired and you'd like to renew your maintenance, contact Golden Software.

About Surfer

Use the **File | About Surfer** command or click the  button to see detailed information about **Surfer** such as: version number, copyright date, serial number, system information, and Golden Software, LLC contact information.

About Dialog

Use the **File | About Surfer** command to open the **About** dialog.

*Find your **Surfer** version number in the **About** dialog.*

Surfer Version

The version of **Surfer** you are currently using is listed in the **About** dialog. Use the File | Online | Check for Update command to ensure you are running the most recent version of **Surfer**. Updates (e.g. **Surfer** 15.0 to **Surfer** 15.1) are free.

Company Contact Information

The Golden Software mailing address, sales phone number, and technical support phone number are listed in the **About** dialog. You can also use the **File | Feedback** commands to contact technical support.

Golden Software Website

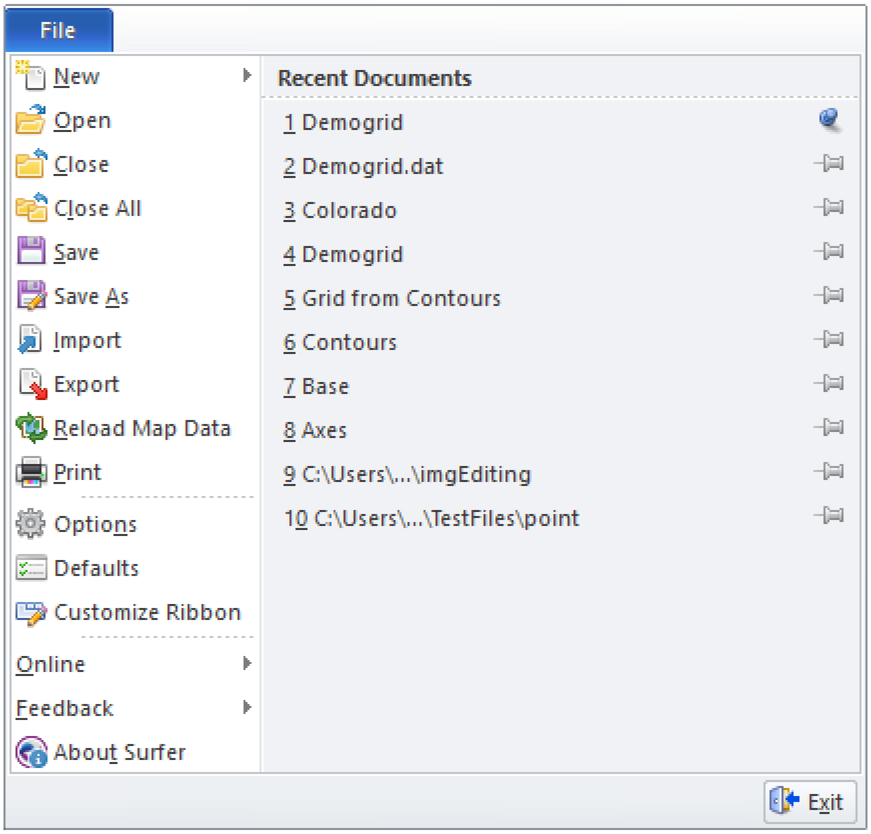
Click the link to open the Golden Software website in a new internet browser window.

System Info

Click the *System Info* button to open the **System File Information** dialog. Detailed file information (*Name, Version, Date, Size*) is available for files *In Directory, In Memory,* or for *All* system files. You can view the system file information, or copy the information by clicking *Copy to Clipboard* button.

Recent Documents

Use the numbers and file names listed on the right side of the **File** menu to open the most recently used files. You can type a number that corresponds with the document or click on the document name to open it.



Click on any of the document names listed in the Recent Documents list to open that file.

You can pin documents to the *Recent Documents* list. Pinned files will be moved to the top of the *Recent Documents* list and will not be removed as new files are added to the list.

To pin a file, click the gray pin to the right of the file name. The pin is displayed as , and the file is pinned to the top of the *Recent Documents* list.

To unpin a file from the *Recent Documents* list, click the blue pin to the right of the file name. The pin is displayed as , and the file is unpinned.

Exit

Use the **File | Exit** command or click the Exit button to close **Surfer**. If changes have been made to any open documents there will be a prompt to save the changes.

Home Tab Commands

The ribbon **Home** tab contains the following commands:

Paste	Insert clipboard contents into the document
Paste Special	Insert clipboard contents into the plot document or worksheet with formatting options
Copy	Copy the selected object to the clipboard
Cut	Cut the selected object to the clipboard
Delete	Delete the selected objects
Duplicate	Duplicate the selected object
Copy Format	Copy the selected object's format to the clipboard
Paste Format	Paste the previously copied format to the selected object
Move/Copy to Layer	Move or copy the selected object to another base (vector) layer or the plot document
Undo	Undo the last action
Redo	Redo the previously undone action
Previous View	Reverses the last operation that changed the view in the plot window
Select	Changes the cursor to the select tool
Select All	Select all objects in the document
Deselect All	Deselect all selected objects
Block Select	Select objects within a rectangle
Invert	Reverse the selection
Rename	Change the name of the selected object
Transform	Moves, scales, rotates objects, or clears transforms
Grid Data	Create a grid from irregularly spaced XYZ data
Map Wizard	Creates a map with one or more layers from a boundary, data, or grid file
Base	Create a new base map from a boundary file, data file, or server or create a new empty base map
Contour	Create a new contour map
Post	Create a new post map or classed post map
3D Surface	Create a new surface or wireframe
Color Relief	Create a new color relief map
Speciality	Create a new grid values, watershed, 1-grid vector, 2-grid vector, point cloud or peaks and depressions map
Layer	Add a base, base from server, base from data, empty base, contour, post, classed post, surface, wireframe, color relief, grid values, watershed, viewshed, 1-grid vector, 2-grid vector, or point cloud layer to the selected map frame
Graphic	Imports a foreign file format such as an image or vector graphics
Text	Creates a text block

Polyline	Create a polyline
Polygon	Creates a polygon
Point	Creates a point
Spline Polyline	Creates a spline polyline
Range Ring	Creates a range ring
Rectangle	Creates a rectangle or square
Rounded Rect-angle	Creates a rectangle with rounded corners
Ellipse	Creates an ellipse or circle
Help	Open the help file
Knowledge Base	Open the Knowledge Base in the default browser

Welcome to Surfer Help

There are several ways to obtain help in **Surfer**:

Getting Help

Within **Surfer**, the help file is opened by clicking the **Home | Help | Help** command or the *Help* button in the upper right corner of the ribbon. You can also quickly search the *Help* by typing a term in the **Search commands and Help** box above the ribbon and clicking *Search help file* in the results. Alternatively, press F1 at any time to open the *Help*.

Context-Sensitive Help

To obtain context-sensitive help about dialogs or highlighted commands:

- Find the function of commands by hovering the cursor over the command and press Fn and F1.
- Click the ? button, the *Help* button, or press F1 in dialogs to open the help topic pertaining to that dialog.
- Press SHIFT + F1 on your keyboard, then click a command or screen region to view information regarding that item.

Internet Help Resources

There are several Internet help resources:

- Use the **File | Feedback** commands to send an **Information Request**, **Problem Report**, or **Suggestion** by email.
- Use the **File | Online** commands to access additional help, including the Golden Software home page, Surfer product page, and Frequently Asked Question (FAQ's) page.

- Search goldensoftware.com/products/surfer to find upcoming webinars, read the Golden Software Blog, and download the latest **Surfer** *Quick Start Guide* and *User Guide*.
- From support.goldensoftware.com download new versions of **Surfer**, access **Surfer** Knowledge Base articles and recordings of webinars and training videos, and with active maintenance submit a request for support.
- The web help can be viewed by navigating to surferhelp.goldensoftware.com.

Complete the Surfer Tutorial

The Surfer tutorial is a great way to get started in **Surfer**. Tutorial lessons one through six will teach you the basics of creating and editing a map. There are also additional optional advanced tutorial lessons available. If you are using the demo version of **Surfer**, you will not be able to complete some of the tutorial steps that require saving or exporting. The demo version is a fully functional read-only version of the program. When this is a factor it is noted in the text and you are directed to proceed to the next step that can be accomplished with the demo.

Automation Help

The *Surfer Automation* help book in the table of contents is designed to help you work with Scripter. Each object, method, and property has a help topic in **Surfer**. Use the object hierarchy to determine how to access each object. Also, each method and property contains some sample code lines with the command. To find out how a particular method or property is accessed click the object name in the *Used by* list. In some cases you may need to change some words to work with the particular object if the sample was not specifically written for the object. Sample scripts are also available in the Samples folder (C:\Program Files\Golden Software\Surfer\Samples\Scripts) to help get you started.

A Note about the Documentation

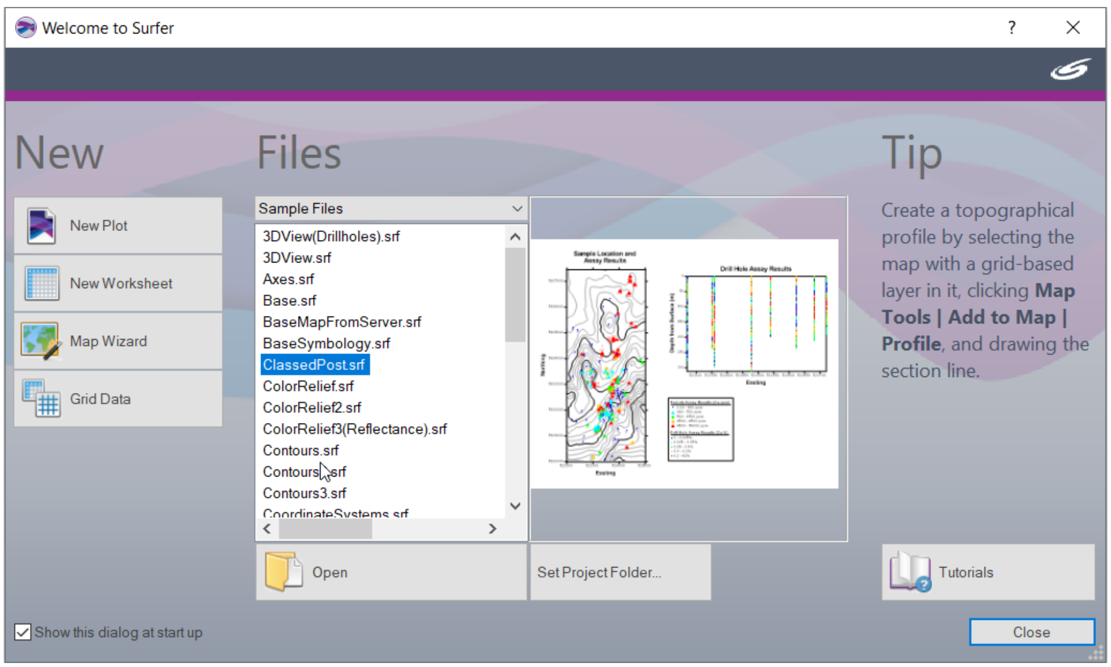
The **Surfer** documentation includes the online help and the quick start guide. Use the **Help | Contents** command in the program to access the detailed online help. Information about each command and feature in **Surfer** is included in the online help. In the event the information cannot be located in the online help, other sources of **Surfer** help include our FAQs, knowledge base, and contacting our technical support.

Various font styles are used throughout the **Surfer** documentation. **Bold** text indicates menu commands, dialog names, and page names. *Italic* text indicates items within a dialog such as group names, options, and field names. For example, the **Save As** dialog contains a *Save as type* list. Bold and italic text occasionally may be used for emphasis.

In addition, menu commands appear as **File | Open**. This means, "click on the **File** menu at the top of the document, then click on the **Open** command within the **File** menu list." The first word is always the menu name, followed by the commands within the menu list.

Welcome to Surfer Dialog

The **Welcome to Surfer** dialog is displayed when you first start **Surfer**. The **Welcome to Surfer** dialog provides immediate access to the File | New | Plot, File | New | Worksheet, Map Wizard, and **Grid Data | Grid Data** commands; sample files, recent files, and project files; and the tutorial. The **Welcome to Surfer** dialog also displays a tip about using **Surfer**. Resize the **Welcome to Surfer** dialog by clicking and dragging any side or corner of the dialog.



*The **Welcome to Surfer** dialog helps you get started quickly in **Surfer**.*

New Plot

Click the *New Plot* button to start **Surfer** with a new plot open in the plot window.

New Worksheet

Click the *New Worksheet* button to start **Surfer** with a new worksheet open in the worksheet window.

Map Wizard

Click *Map Wizard* to create a new plot and get straight to creating your map with the Map Wizard.

Grid Data

Click the *Grid Data* button to start **Surfer** with the Grid Data dialog **Select Data** page.

Open Files and File Preview

The *Files* section of the **Welcome to Surfer** dialog displays Surfer SRF files in one of three categories. Click the current selection in the *Files Type* list, and select *Recent Files*, *Sample Files*, or *Project Files* to change which files are displayed in the *Files* section. The *Files Type* selection is remembered the next time **Surfer** is started.

Click a file name to see a preview image in the **Welcome to Surfer** dialog. The preview is only available for **Surfer 13** and newer .SRF files. Click on another file or press the UP ARROW or DOWN ARROW keys to change the file selection. Once a file is selected, the *Open* button is enabled. Click the *Open* button to start **Surfer** with the selected file open in the plot window. Alternatively, double-click on a file name to start **Surfer** with the selected file open in the plot window.

If the file you wish to open is not in the *Recent Files* , *Sample Files* , or *Project Files* list, select *Browse* in the *Files Type* list. Next, select the file to open in the Open dialog.

Recent Files

When *Recent Files* is selected in the *Files Type* list, the most recently opened **Surfer** files are displayed in the *Files* list. By default the number of files in the *Recent Files* list is 10. Change the number of files in the *Recent Files* list in the Options dialog User Interface page.

Sample Files

When *Sample Files* is selected in the *Files Type* list, the sample files included with **Surfer** are displayed in the *Files* list.

Project Files

When *Project Files* is selected in the *Files Type* list, the **Surfer** files located in the user defined project folder are displayed in the *Files* list. Change the project folder by clicking the *Set Project Folder* button and selecting the appropriate folder in the **Select Folder** dialog. Alternatively, the *Project folder* option can be set in the Options dialog on the General page. However, if the project folder is set in the **Options** dialog, **Surfer** must be closed and reopened for the changes to take effect.

Browse

Select *Browse* in the *Files Type* list to open a file with the Open dialog.

Tip

The *Tip* is a useful statement regarding a **Surfer** command or process.

Open

Click the *Open* button to start **Surfer** with the selected file in the *Files* list in the plot window. The *Open* button is enabled when a file is selected in the *Files* list.

Set Project Folder

Change the project folder by clicking the *Set Project Folder* button and selecting the appropriate folder in the **Select Folder** dialog. The *Files Type* selection is changed to *Project Files* after changing the project folder. If you click *Cancel* in the **Select Folder** dialog, the *Files Type* selection is not changed.

Tutorials

Click the *Tutorials* button to open the online help *Tutorial Introduction* help topic. The tutorial is a useful starting place for users who are new to **Surfer**.

Hiding the Welcome to Surfer Dialog

Uncheck the *Show this dialog at start up* check box to start **Surfer** without displaying the **Welcome to Surfer** dialog in the future. The **Welcome to Surfer** dialog can be enabled or disabled by checking or unchecking the *Show welcome screen at startup* check box in the Options dialog on the User Interface page.

Close

Click the *Close* button, the button, or press ESC to close the **Welcome to Surfer** dialog and start **Surfer** with a new plot in the plot window.

Technical Support

Golden Software's technical support is free to registered users of Golden Software products. Our technical support staff is trained to help you find answers to your questions quickly and accurately. We are happy to answer all of your questions about any of our products, both before and after your purchase. We also welcome suggestions for improvements to our software and encourage you to contact us with any ideas you may have for adding new features and capabilities to our programs.

When contacting us with your question please have the following information available:

- Your **Surfer** product key
- Your **Surfer** version number, found in **File | About Surfer**
- The operating system you are using (Windows 7, 8, 10 or higher)
- The steps taken to produce your problem
- The exact wording of the first error message that appears (if any)

If you cannot find the answer to your question in online help, the quick start guide, or on our web page FAQs or Knowledge Base, please do not hesitate to contact us:

Phone: 303-279-1021

Fax: 303-279-0909

Email: surfersupport@goldensoftware.com

Web: www.goldensoftware.com

Mail: Golden Software, LLC, PO BOX 281, Golden, Colorado, 80402-0281, USA

Register Product Key

Please remember to register your software by filling out the registration form online. Registering your product key entitles you to free technical support, announcements, and training resources. Our database is confidential, and Golden Software will not share or sell your contact information.

To register your product key,

1. Navigate to http://myaccount.goldensoftware.com.
2. Log in to your account.
3. Click *Register Software*.
4. Fill out the registration form.
5. Click *Submit Registration*.

Your product key is located in your email download instructions. Please take a minute to register your copy of **Surfer** with us.

What's New in Surfer?

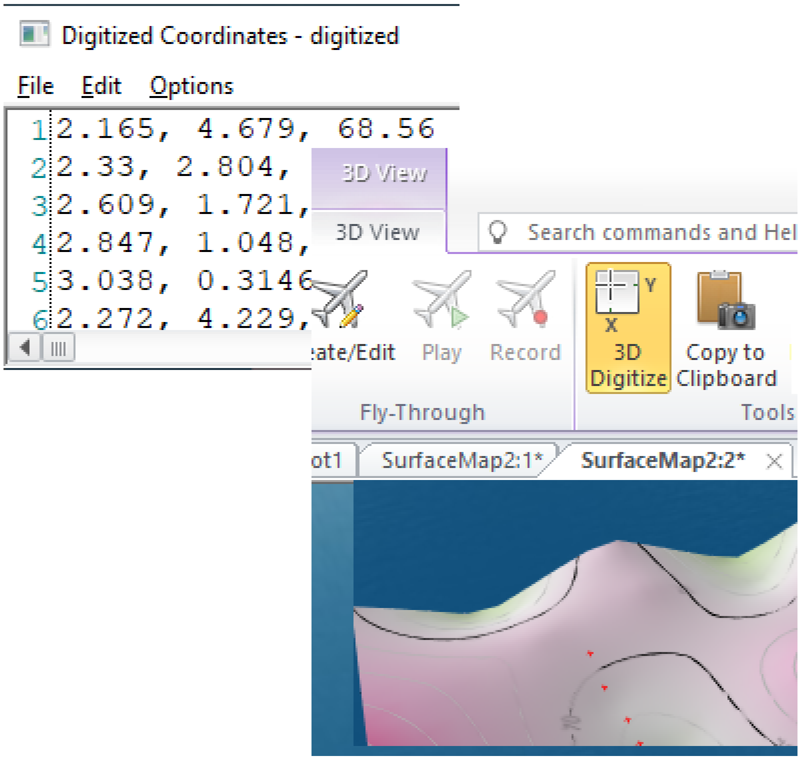
There are several new, exciting features in the latest release of **Surfer Beta**! We focused on making it easier to create great looking maps by adding some visual enhancements and some time saving improvements. You can now digitize in 3D, save a TIFF image as a grid, and get instant grid information. The top new features are listed below. Full help topics for these new features can be found in the help contained in the **Surfer Beta** application. A more complete list of the new features can be found in the Surfer Beta Version History page.

Improved 3D Surface Maps

- New methods have been added to show the base on surface maps. You can now select three different Z value settings to define how the base is shown. Previously, the 3D surface map base filled the sides of the map under the surface layer from the surface down to the map minimum Z value. Now you can select to show the base from the surface down to the minimum Z value in the map, to the minimum Z value in the selected grid, or to a Z value that you specify.

Digitize in 3D

- Before this new beta release, you could digitize your XY coordinates in 2D maps. Many of you use this feature to create blanking data. You can now perform the same type of digitization of data in the 3D view using the new **3D View | Tools | 3D Digitize** command. 3D digitization collects XY and Z coordinates. Although you can use this new feature in a variety of functions, one function could be to obtain the XYZ coordinates of specific locations, which could be points on a surface or subsurface points.
- This new feature works well with our new drillhole map. For example, you can digitize specific locations on each drillhole (e.g. the top of a unit or stratigraphic layer using your drillhole data). You can then go back to the 2D plot view, grid the digitized XYZ data, and create a contour map from the digitized data. Then, you can add that contour map to the 3D view as a surface displayed with the drillholes.



3D Digitize

New Frames for Color Scale Bars and Map Scale Bars

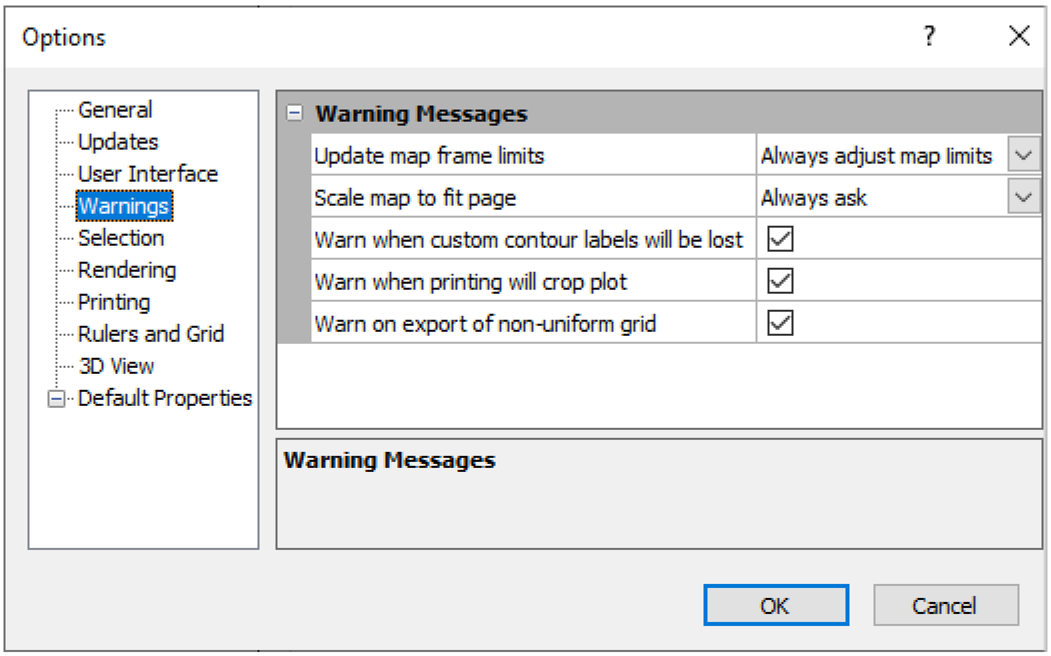
- We have added two important enhancements to the visual appeal and aesthetics of maps that you need to present to clients or peers. You can now add and customize a frame around color scale bars and map scale bars. You can set the margin between the frame and text in the scale bars and you can define its fill and line properties.

More Help with Color Scale Bars

- We know that many of you often build maps that have several layers and thus have several color scale bars. A new enhancement in **Surfer Beta** makes it easy for you to know which layer a color scale bar belongs to. In the **Contents** window, click on one of the color scales to show the layer it belongs to. The name of the layer will show in the *Linked layer* property in the **General** property tab.

Manage Warning Messages and Warning Default Behavior

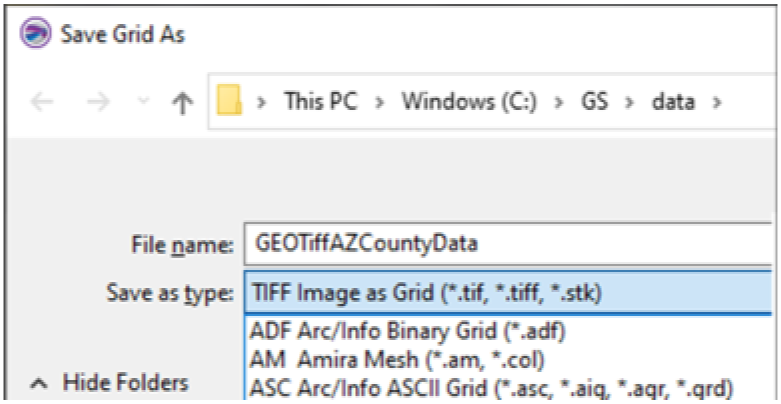
- Because of **Surfer's** flexibility, the application helps prevent some changes to maps that you may not be aware would occur without the warning. For example **Surfer** has a warning when increasing the limits of the map or when changing a map would make it larger than the printed page. These warnings give you the option of cancelling out of the change or accepting the change and continuing. Because some of you perform routine tasks that trigger these types of warnings, we have made several of these warnings optional. You can use the **File | Options** command to open the **Options** dialog and define the options to suppress these warnings. Two of the warnings also let you define default behavior.



Manage Warning Messages

Save a Grid as a GEOTiff

- You can now save a grid as a GEOTiff instead of just as an image TIFF. These files can be used as a grid data file. You could always export a grid as an image TIF, but our new features allows you to save a grid as a GeoTIFF so that the Z values are saved as actual Z values instead of as colors in an image. This feature is found in the **Save Grid As** dialog's new *TIFF Image as Grid* option. You can select this option from all of the tools and commands you regularly use with grid files: **Grid Data, Function, Filter, Convert, Spline Smooth, Calculus**, etc. Saving a TIFF as a grid also works in **Surfer** automation.



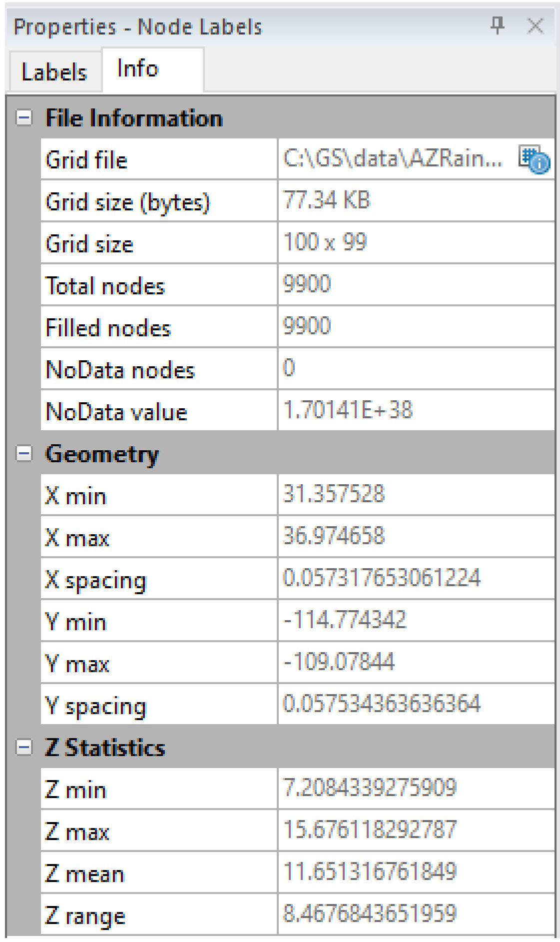
Save Grid as TIFF

Enhancements to the Map Wizard

- The **Map Wizard** has some new wizardry! We have added point cloud and peaks and depressions maps to this wizard. These enhancements work the same way as creating other maps in the **Map Wizard**. Simply select the **Home | Wizard | Map Wizard** command and then select your data file. Note that if you select a LiDAR data file, you are prompted to select to use it as either point cloud or data points data.

Quick access to Grid Info

- We created a quicker way to access grid information by expanding a grid's **Info** properties page and adding an **Info** properties page to the **Grid Editor**. These new **Info** pages contain detailed information about a grid's geometry, statistical, and file information. The **Info** properties tab is a quick way to see information for any grid in the plot window or the active grid in the **Grid Editor**. Simply click on a grid layer and select the Info properties page. At the bottom of the **Info** page for grid layers, click the **Copy** button to copy and then paste this information into another application. This grid information, and more, is also available from the **Grids | Grid Info** command or the **Grid Editor | Options | Grid Info** command from the **Grid Editor**.


Properties - Node Labels
Labels
Info
File Information
Grid file
C:\GS\data\AZRain...
Grid size (bytes)
77.34 KB
Grid size
100 x 99
Total nodes
9900
Filled nodes
9900
NoData nodes
0
NoData value
1.70141E+38
Geometry
X min
31.357528
X max
36.974658
X spacing
0.057317653061224
Y min
-114.774342
Y max
-109.07844
Y spacing
0.057534363636364
Z Statistics
Z min
7.2084339275909
Z max
15.676118292787
Z mean
11.651316761849
Z range
8.4676843651959

Grid Info

Chapter 2 - Tutorial

Tutorial

The tutorial is designed to introduce basic **Surfer** features and should take less than an hour to complete. After you have completed the tutorial, you will have the skills needed to create maps in **Surfer** using your own data. The tutorial can be accessed in the program by clicking the ? button and navigating to the *Tutorial* book or by clicking *Tutorials* in the **Welcome to Surfer** dialog.

If you find you still have questions after you have completed the tutorial, you should consider reviewing the material in **Surfer's** extensive in-program help. The help is also available on the web. Golden Software's website contains a knowledge base of questions and answers and training videos. Usually, the answers to your questions are found in one of these locations. However, if you find you still have questions, do not hesitate to contact Golden Software’s technical support team. We are happy to answer your questions before they become problems.

Tutorial Overview

The following is an overview of lessons included in the tutorial.

Starting Surfer	Open **Surfer** and a new plot window
Creating a Map	Create a map with multiple layers
Changing Map Properties	Change the map coordinate system and scaling and adjust the appearance of the map layers
Viewing Maps in 3D	View the map in the 3D view and show 3D vector contours
Saving and Exporting	Export the map in various file formats and save the project

A Note about the Documentation

Various font styles are used throughout the **Surfer** quick start guide and online help. **Bold** text indicates commands, dialog names, tab names, and page names. *Italic* text indicates items within a dialog or the **Contents** or **Properties** windows such as section names, options, and field names. For example, the **Save As** dialog contains a *Save as type* list. Bold and italic text may occasionally be used for emphasis.

Also, commands appear as **Home | New Map | Contour**. This means, "click or scroll to the **Home** tab at the top of the plot window, then click on the **Contour** command within the **New Map** command group." The first word is always the menu or ribbon tab name, followed by the command group, and finally the command name within the menu list or on the ribbon.

Sample File Location

The sample files used in the tutorial lessons are located in the **Surfer** Samples folder. The Samples folder is located by default at C:\Program Files\Golden Software\Surfer\Samples. Note, if you are running the 32-bit version of **Surfer** on a 64-bit version of Windows, the Samples folder is located at C:\Program Files (x86)\Golden Software\Surfer\Samples, by default.

Getting Started

Open **Surfer** by:

- double-clicking the **Surfer** icon in the desktop, or
- selecting **Surfer** in the Start menu, or
- searching for **Surfer** in the Task Bar search field.

The **Welcome to Surfer** dialog is displayed each time you start **Surfer**. To prepare for the tutorial, click *New Plot* to create a new plot in the plot window. The plot window is the main component of the Surfer user interface.

If you opened **Surfer** before starting the tutorial, click **File | New | Plot Document** to open a new plot document if you don't already have an empty plot document.

Each plot document is a new **Surfer** project and is saved to its own SRF file. Multiple plot windows can be open at once. In **Surfer**, data is imported while creating new maps and features. In the next lesson we'll start with creating a map.

Creating a Map

First, we'll create a map with multiple layers. A map frame or map object is a frame with four axes and a coordinate system. Data is visualized in one or more layers that have been added to the map frame. Maps are created by selecting a layer type and choosing the data, grid, image, or vector file. This action adds a new map with a single layer to the plot window.

Add a contour map to the empty plot.

1. Click the **Home | New Map | Contour** command.
2. In the **Open Grid(s)** dialog, navigate to the sample files folder (C:\Program Files\Golden Software\Surfer\Samples by default).

3. Select the *Diablo.grd* grid file.
4. Click *Open*.

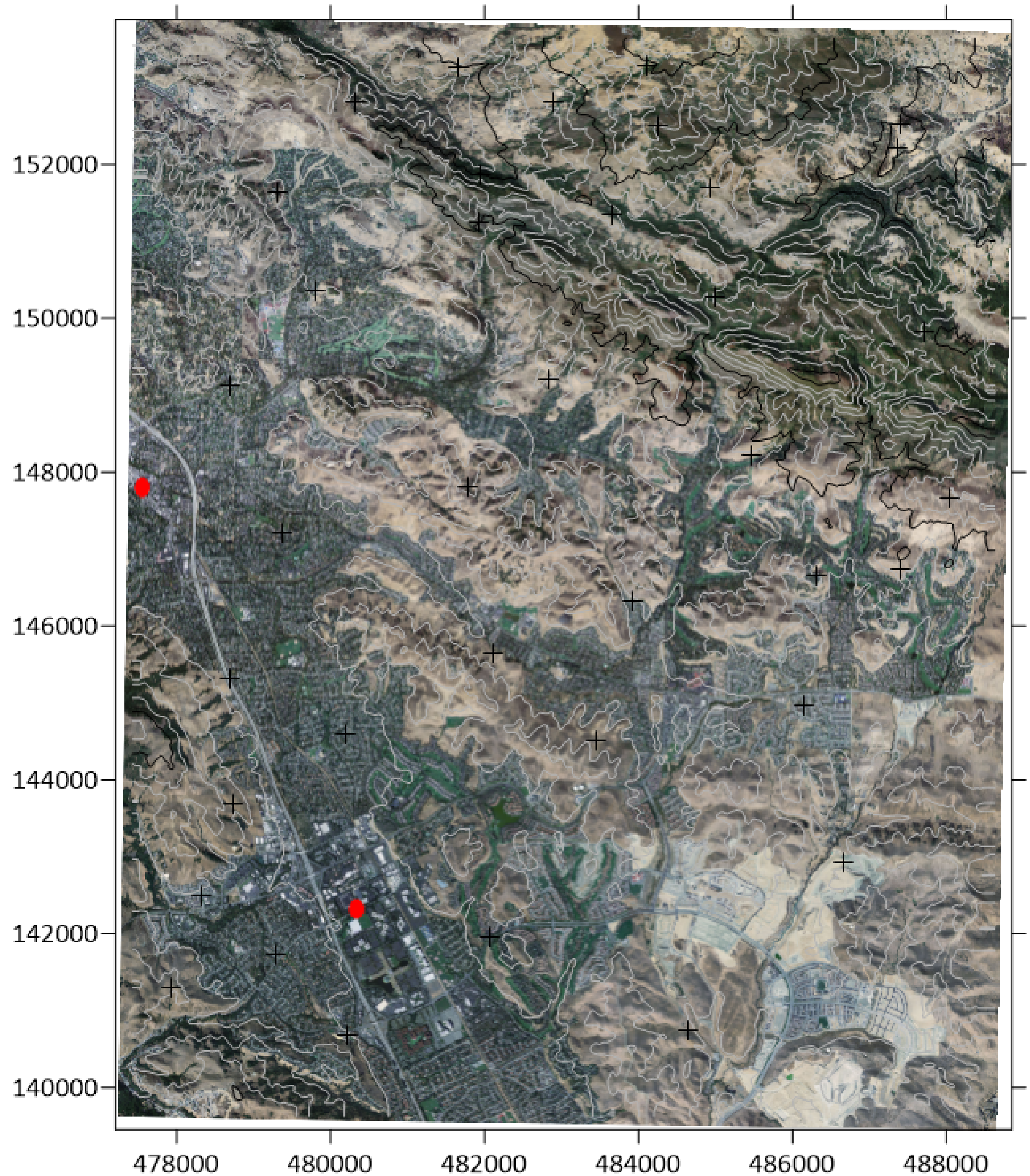
The *Diablo.grd* sample file is a grid file with the topography of the southern face of Mt. Diablo in California. Most maps in **Surfer** use grids for input data. However, we'll add two more layers that use different data files to the map frame. Next, add a post layer.

1. Select the *Map* frame in the **Contents** window or click the map in the plot window.
2. Click the **Home | Add to Map | Layer | Post** command.
3. Select the *Diablo Example.dat* sample data file.
4. Click *Open*.

The *Diablo Example.dat* sample file is a tabular data file with XYZ coordinates for various points in the Mt. Diablo region. Many different data file types are supported, and the most common are CSV and XLSX files. Finally, we'll add an image as a base raster layer.

1. Select the *Map* frame in the **Contents** window or click the map in the plot window.
2. Click the **Home | Add to Map | Layer | Base** command.
3. Select the *Diablo_Sat.tif* sample image file.
4. Click *Open*.
5. Click *OK* in the **TIFF Import Options** dialog to accept the default options.
6. Click *Yes* in the **Surfer Warning** dialog to automatically extend the map limits to include the entire image.

Images are usually imported into Surfer for use as base maps. However, the **Home | Insert | Graphic** command can be used to add images directly into the plot window without creating a map.



Multiple map layers are added to the map.

It is difficult to see the contours and post map symbols with the image base map. Next we will change the map properties to improve its appearance.

Changing Map Properties

There are primarily two ways to change how the map looks: by changing the source data or by changing the properties. You can change the data you use to create the map layers by creating and editing grids with the **Grids** ribbon commands or by creating and editing features with the **Features** ribbon commands.

You can change how maps appear with the map and layer *properties*. In this tutorial, we will focus on changing properties.

Surfer automatically converts source data from different coordinate systems into the coordinate system specified by the *Map* frame. In this case,

- Diablo.grd (contour layer) uses State Plane 1927 California III (meters),
- Diablo Example.dat (post layer) uses NAD27 UTM Zone 10N,
- and Diablo_Sat.tif (base raster layer) uses WGS1984.

We'll change the *map target coordinate system* to WGS1984.

1. The target coordinate system is part of the *Map* frame. Select the *Map* frame in the **Contents** window. For this action, it is not recommended to click in the plot window to select the *Map* frame.
2. Click the **Coordinate System** tab in the **Properties** window.
3. Click *Change* in the *Coordinate system* field.
4. Use the search bar at the top of the **Assign Coordinate System** dialog to locate the *World Geodetic System 1984* coordinate system in the structured list. Select *World Geodetic System 1984*.
5. Click *OK*.

Next, because WGS1984 is a geographic (unprojected) coordinate system we will adjust the scaling to reduce distortion in the map.

1. Click the **Scale** tab in the **Properties** window.
2. Clear the *Proportional XY scaling* option.
3. Set the *Length (page units)* in the *X Scale* section to 6in (15.2cm) if it is not already.
4. Set the *Length (page units)* in the *Y Scale* section to 7.6in (19.3cm).

Now the map frame has been modified. Next we will modify the map layers to improve contrast between the contours, post symbols, and base map image.

1. Select the *Contours-Diablo.grd* layer in the **Contents** window.
2. Click the **Levels** tab in the **Properties** window.
3. In the *Major Contours | Line properties* section,
    1. Set the line *Color* property to White
    2. Set the line *Width* property to 0.02 in (0.05cm).
4. In the *Minor Contours | Line properties* section, set the line *Color* property to White.
5. Select the *Post-Diablo Example.dat* layer in the **Contents** window.
6. Click the **Symbol** tab in the **Properties** window.
7. In the *Symbol | Symbol properties* section, set the *Symbol* property to the filled diamond (Number 6) and *Fill color* property to Cyan.

Tip: Click the **Home | Undo | Undo** command or press CTRL+Z on the keyboard to reverse the last operation performed. Click the **Home | Undo | Redo** command or press CTRL+Y on the keyboard to reverse the last **Undo** command.

We have completed the basics of creating a map and changing some properties. Next we'll see the map in the 3D view.

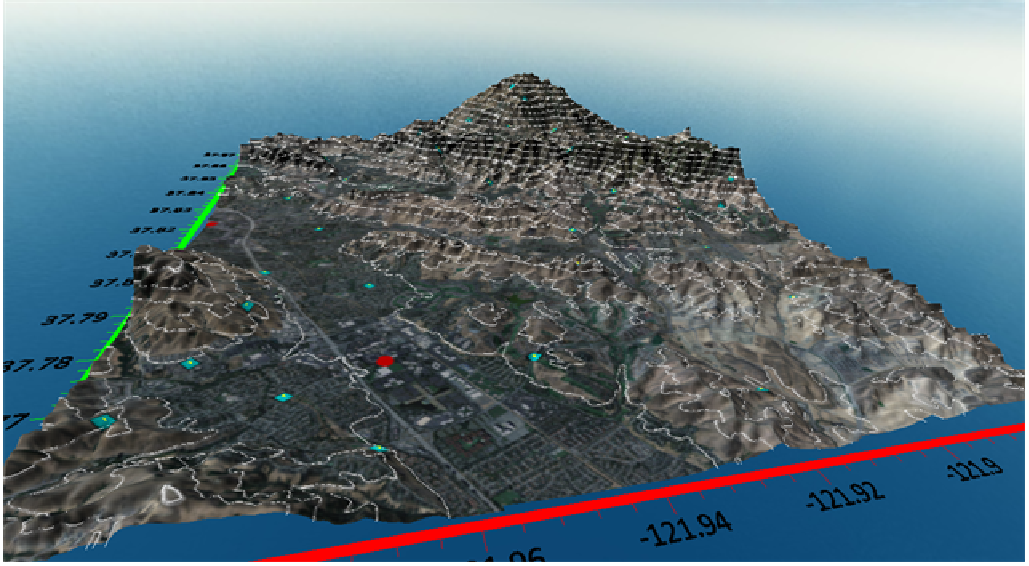
Viewing a Map in 3D

The **3D view window** can help provide context and understanding when looking at complex information. You can pan, tilt, and zoom the view, create fly-throughs, "walk" around surfaces, and export images and 3D PDF models. Surfaces are created from grid-based layers in the map; vector objects are created from contours, post layers, vector base layers, and drillhole layers; 3D point clouds are created from point cloud layers. All layers from the plot window can be overlaid on each surface.

1. Select the *Map* frame in the **Contents** window.
2. Click the **Map Tools | View | 3D View** command.

We can change properties in the 3D view window in the same manner as the last lesson.

1. Select the *Environment* in the **Contents** window.
2. Click the **General** tab in the **Properties** window.
3. Set the *Vertical exaggeration* to 3e-05.
4. Select the *Surfaces* in the **Contents** window.
5. In the **Properties** window, set the *Surface quality* to 100 by using the slider or by typing *100*. The lower the surface quality the more likely it is vector objects can appear to intersect the surface, even in places they should not. If your PC's performance becomes slow, reduce the *Surface quality* until acceptable speeds are restored.
6. Select the *Diablo.grd* surface in the **Contents** window.
7. In the *Textures to Display* section of the **Properties** window, clear the *Contours-Diablo.grd* option.
8. Check the box next to *Contours-Diablo.grd* in the **Contents** window to display 3D vector contour lines.



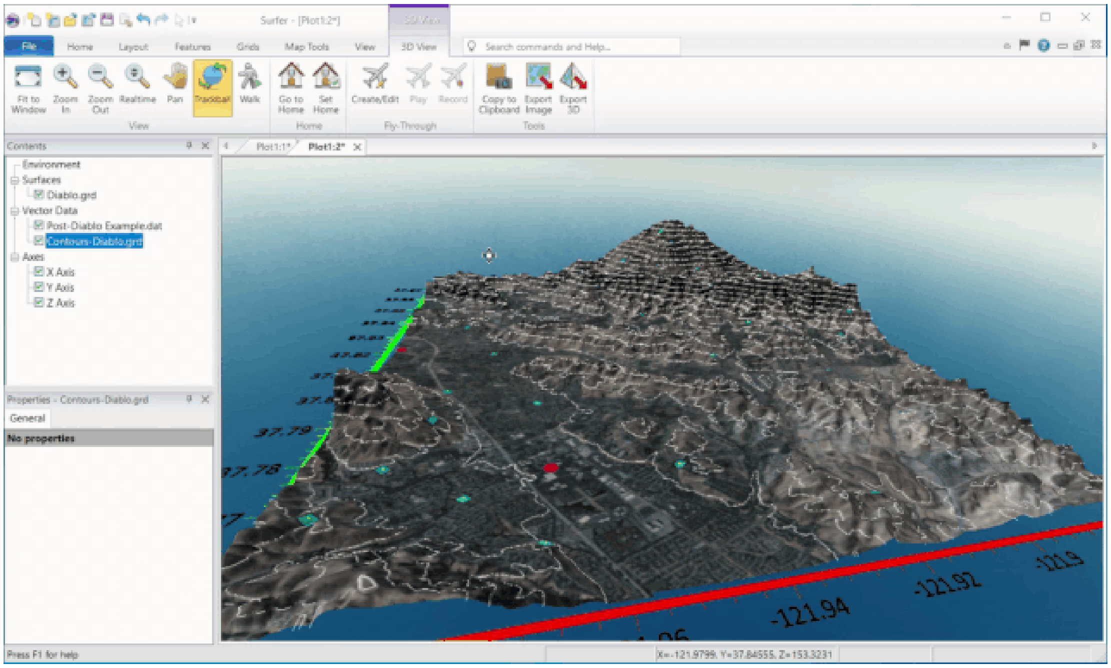
Use the 3D view to explore for meaning in your data.

The **Properties** window includes many options to change the appearance in both the plot window and 3D view. Next, we will export the map.

Saving and Exporting

Surfer is not usually the final stop for the maps users create. Most users incorporate the visualizations into reports and use the contours, features, and other components in other programs. Since we are already in the 3D view window, we'll start with exporting an image from the 3D view.

1. Click the **3D View | Tools | Export Image** command.
2. In the **Export Image** dialog, set the *Save as type* to PNG.
3. Specify a location and name for the image file, e.g. your Documents folder. The Samples folder will be displayed by default, but most users cannot save to the Samples folder.
4. Click *Save*.
5. Review the export options in the **Export Options** dialog, and click *OK*.

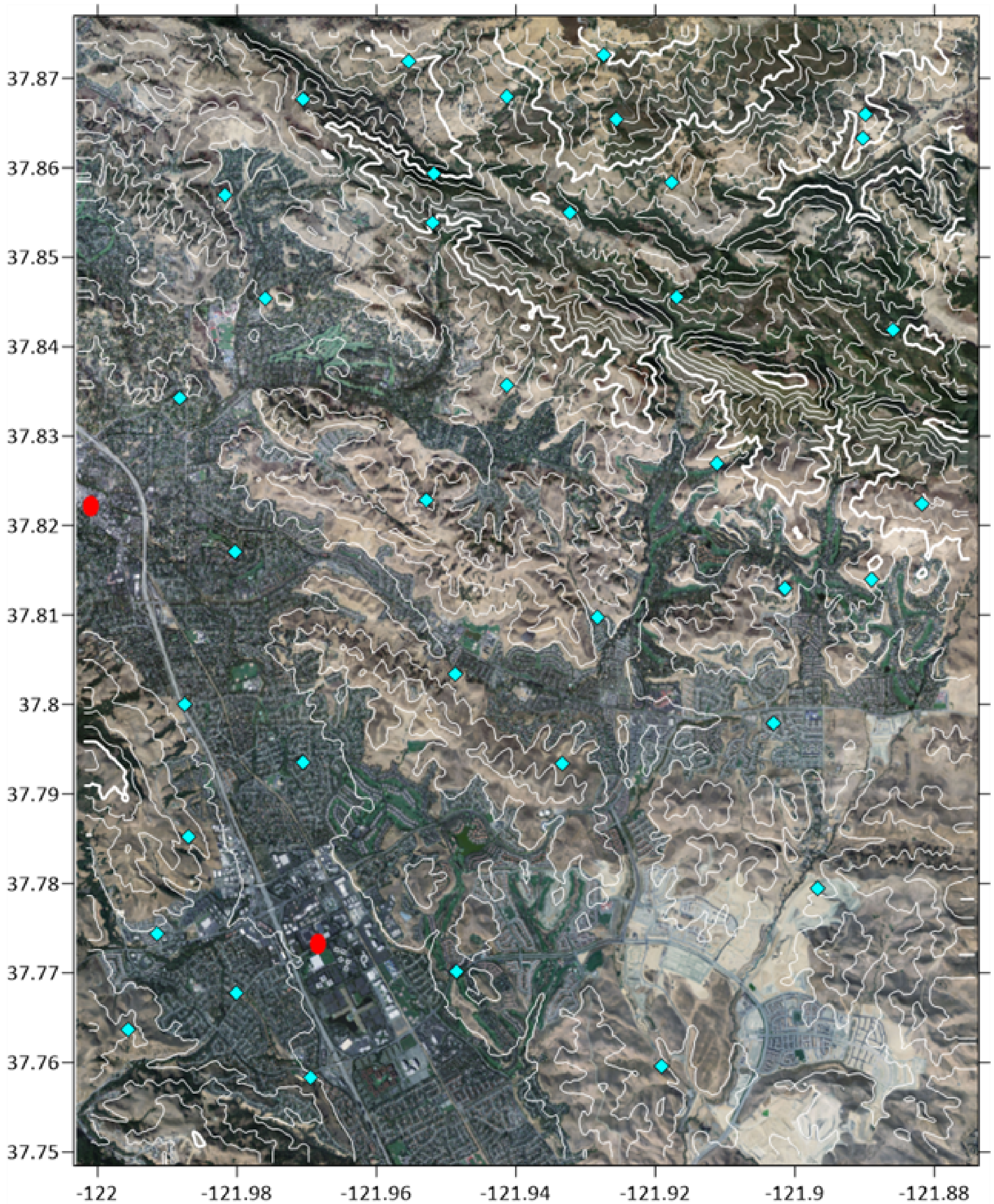


Use tabs to navigate between plot windows, 3D view windows, worksheets, and grid editor windows.

Next we'll export a .KML file. **Surfer** supports many export formats for moving data in to and out of other programs.

1. Switch back to the plot window by clicking the *Plot1:1* tab.
2. Click the **File | Export** command.
3. Set the *Save as type* to KML.
4. Specify a path and file name for the KML file.
5. Click *Save*.
6. Review the options in the **Export Options** dialog and click *OK*.

Now you have a .PNG image and a .KML file with the tutorial map for use in other mediums.



The completed tutorial map includes three overlaid layers with visible contour lines and post symbols.

This concludes the tutorial. If you wish, save the tutorial map by clicking the **File | Save** command. Feel free to experiment with the tutorial map. We also recommend you review the **Surfer** sample files. Check out the in-program help, the online KB articles, or contact support if you need more information.

Chapter 3 - Data Files and the Worksheet

Data Files

Data files contain the raw information used to create a grid file, perform residual calculations, or produce post maps. Each record in a data file occupies a single row and is comprised of at least two values (X, Y) for post maps and at least three values for gridding (X, Y, Z). The X, Y, and Z values are each placed in separate columns. The X and Y coordinates define the position of the point on the map, and the Z value defines the value assigned to the specific X, Y location. Common examples of X, Y coordinates include longitude and latitude, easting and northing, or UTM (Universal Transverse Mercator) coordinates. The Z data might be topographic elevation, water depth, chemical concentration, temperature, or any other quantity amenable to mapping.

Data files can be created in the **Surfer** worksheet, a text editor, or any program that can produce files in one of the supported file formats. Regardless of the program used to create your data files, you must save the file on disk prior to performing any **Surfer** operation requiring a data file, including the gridding operation. **Surfer** reads data only from a data file in one of the recognized formats.

It is not necessary to open a data file in the worksheet in order to use the data file for a command (e.g. **Grid | Data**). If you want to view or alter the data in a data file, you can use the File | Open in Worksheet command to gain access to the worksheet data.

Surfer requires the use of decimal degree values when using Latitude and Longitude data.

XYZ Data Files

XYZ data files contain the raw data **Surfer** interprets to produce a grid file. Before you create a grid file in **Surfer**, you must create an XYZ data file. XYZ data files must be organized in column and row format. By default, **Surfer** expects the X data to be contained in column A, the Y data in column B, and the Z data in column C. However, the data can be placed in any order in any column.

Portions of two simple data files are shown below. The order of the data in the file is not important. These examples contain descriptive headers in Row 1 of each column. Such information is helpful but not required by **Surfer** to create a grid file. When text appears in Row 1 of a column, this text appears in list boxes in various **Surfer** dialogs as column titles. If a number resides in Row 1, it is not incorporated into the dialogs, and instead, the column heading (such as column B) is displayed.

	A	B	C
1	0.1	0	90
2	3.5	0	45
3	4.9	0	65
4	6.2	0	40
5	7	0	55
6	9	0	25
7	9	5	55
8	9	3	48

This is a simple XYZ data file.

	A	B	C
1	Longitude	Latitude	Elevation
2	-109	39.205	1464
3	-108.965	39.337	1524
4	-108.93	39.389	1385
5	-108.895	39.526	1583
6	-108.86	39.588	1445
7	-108.825	39.795	1371
8	-108.79	40.003	1371

This is another example of an XYZ data file with header information in row 1 of each column in the data file.

Missing Entries

Rows with non-numeric entries (empty cells or text) in any of the X, Y, or Z columns are excluded when performing various tasks, including gridding or transforming data in the worksheet. If there is no Z information for a particular XY location, you can leave the Z cell blank for that row. In the example shown here, there are two data records without Z values. These records are not considered during the gridding operation.

	A	B	C
1	X Data	Y Data	Z Data
2	0.1	0	90
3	9	3	48
4	1.3	7	
5	4.7	1	66
6	1.7	5.6	75
7	6	1	
8	2.5	3.6	60

Blank Z column cells are ignored when gridding a data file.

Multiple Columns of Information for Additional Maps

Data files can contain up to one billion columns. Since you can specify the columns to be gridded, your X, Y, and Z values can occupy any three columns. This allows you to have columns containing other information particular to each point. The data file can contain several Z columns, so you can produce several contour maps using the same XY coordinates. For example, you might have concentrations of different contaminants at each sample location. All the contaminant concentration data can be placed in the same data file.

	A	B	C	D	E	F	G	H
1	X Coord	Y Coord	SS-A	SH-A	SLT-B	SH-B	SS-B	SA-C
2	4345	14005	1432	1478	1503	1593	1627	1645
3	4355	14015	1437	1483	1501	1589	1625	1640
4	4365	14025	1445	1479	1498	1592	1637	1638
5	4375	14035	1434	1466	1515	1586	1626	1635
6	4385	14045	1435	1476	1510	1597	1616	1637
7	4389	14055	1448	1475	1512	1601	1603	1638

This is an example of an XYZ data file containing several columns of Z data. You could use this file to create several different grid files, where each uses the same XY coordinates, but different Z data.

Additional Information in Data Files

Data files may contain information in addition to the X, Y, and Z values. For example, when posting data with the **Home | New Map | Post** command, additional columns can be used to specify the symbol, the rotation angle, the symbol color, labels, etc. The following is an example of such a data file. Columns A, B, and C contain the X, Y, and Z data used to produce a contour map of depth to the water table. Columns D, E, and F contain information used to create an overlaying post map.

Plot1* | Sheet1.dat

B10 | 7

	A x	B y	C z	D	E	F
1	Easting	Northing	Elevation	Symbol set: index	Color	Angle
2	0.1	0	90	Arial:65	Red	45
3	3.5	0	45	Arial:66	green	30
4	4.9	0	65	Arial:67	blue	170
5	6.2	0	40	GSI Default Symbols:4	purple	18
6	7	0	55	GSI Default Symbols:8	majestic pu	145
7	9	0	25	GSI Default Symbols:14	Red	22

A data file used to create a post map or a classed post map can contain several columns of data. Each column can have a different effect on the posted data points.

Data as Numbers or Text

Worksheet data are in one of two forms: numbers or text. Grid file creation, statistics, post maps, and other operations require data as numbers. Text data (even if it contains numeric digits) can be used for labels in **Surfer**, but it cannot be used to create grids or in any operation requiring numbers.

Numbers can consist of digits (0 - 9), decimal points (.), the letters "e," "d," "E," or "D" (indicating an exponent), and the plus (+) or minus (-) sign. If you type any characters other than these when entering a number (or type any of the special characters more than once), **Surfer** automatically converts the cell entry to text. For example, if your longitude data appears as 104.5 W in a worksheet cell, it is interpreted as text and cannot be used to grid data. To successfully read this data, use the -104.5 format to indicate a location 104.5 degrees west of the prime meridian. If a number has if formatted as text and should be formatted as a number, highlight the cell or group of cells to select them and click the Text to Number command.

You can also convert numeric data to text by typing a single quotation mark (') in front of the number. **Surfer** does not place the single quotation in the worksheet cell, however the single quotation is visible in the Active Cell Edit Box.

By default, numeric data is right justified in a cell, and text is left justified. Cell entries, whether numeric or text, can be justified by specifying the desired alignment using the options on the **Alignment** page of the Format Cells dialog. Use the Text to Number command to remove text formatting.

B25	'1.6		
	A	B	C
25	4.6	1.6	70
26	4.5	2.5	80
27	4.6	3.6	95
28	4.5	4.2	80
29	4.3	5.1	70
30	4.4	6	60
31	5.3	5.3	78
32	6	5.7	88

Notice that column B is left aligned. This means the numbers are formatted as text. When a cell is highlighted, an apostrophe appears in the active cell edit box, also indicating that the number is formatted as text.

Data File Formats

Surfer can import and export data in several data file formats. A variety of commands in the plot document, worksheet document, and grid node editor can be used to import and export data. The commands are summarized below:

Import Data File Formats

- Data | Edit | Merge in the worksheet document
- File | Open in Worksheet in the plot, worksheet, or grid editor

Export Data File Formats

- File | Export in the plot document
- File | Save As in the plot document
- Grids | Edit | Convert in the plot document
- File | Save As in the grid editor
- File | Save As in the worksheet document

Date/Time Formatting

In addition to numbers and text, dates and times are format types in **Surfer**. Dates and times can be used to create a grid, as axis and plot labels, and to set axis limits.

Using Date/Time Formatting

To use dates and times in **Surfer**, the data need to be formatted as dates and times. One way to format data in **Surfer** is to use the worksheet. The worksheet can be accessed with the File | New | Worksheet or File | Open command. Highlight the column containing dates and times and select Data | Format | Format Cells to set the column as date/time in the worksheet. On the **Number** tab, select *Date/time* as the *Type*. Next, type the appropriate *Date/Time format* option, or click the  button and select or create a date/time format in the Date/Time Format Builder dialog.

Once the formatting is set to date/time, you can use the date/time information just as you would use numbers in **Surfer**:

- you can create a post map of the data using date/time values

- you can set the map limits using date/time values

- you can grid date/time values

Date/time information can also be used as labels anywhere in the map layer or as axis tick labels.

Date/Time formats are made of combinations of year, month, day, hours, minutes, seconds, BC/AD or BCE/CE designation,and AM/PM designation. Years are shown as yy or yyyy. Months are shown as M, MM, MMM, MMMM, or MMMMM. Days are shown as d, dd, ddd, or dddd. Hours are shown as h, hh, H, HH, or [h]. Minutes are shown as m, mm, or [mm]. Seconds are shown as ss, ss.0, ss.00, ss.000, ss.0000, or [ss]. AM/PM designation is shown as tt or TT. BC/AD designation is shown as gg or GG. BCE/CE designation is shown as g, G, ggg, or GGG. See the Date Time Formats help topic for examples of date/time formats.

Date/Time Formatting Tips

- In the worksheet, save data files containing date/time formatting as Excel files to preserve the date time formatting as seen in the worksheet.
- You can save date/time-formatted data files as ASCII files (.DAT, .CSV, .TXT, .BNA, or BLN). Sometimes this is necessary if you exceed the Excel row or column limits. When opening the file in **Surfer**'s worksheet, you can make the serial numbers appear as dates by using **Data | Format | Format Cells**.
- If you have formatted the data as date/time in another spreadsheet program such as Excel, the data are formatted as date/time in **Surfer**.
- Whenever possible, enter and display dates and times in one of the many calendar formats, e.g., "6/14/2009" or "14-June-2009", and let the software handle converting to/from internal numeric representations.
- When the recognized format is ambiguous (i.e. 10/7/12), the month, day, and year order is determined by the Windows locale. In some countries, this will be recognized as M/d/yy, in others as d/M/yy, and in others as YY/M/d. It is important to use non-ambiguous date/time formats when the Windows locale may change.
- The year 0 is defined, according to the ISO 8601:2004 standard.
- If dates/times occur before 1/1/0000, use the BC or BCE suffix after the date. So, Alexander III of Macedon's birthday would be listed as 20-July-356 BCE in the worksheet. Using AD or CE is not necessary and the worksheet will automatically remove these in dates after 1/1/0000.
- When a two digit year is input in the worksheet (00 to 99), it means the year in the current century. For instance, inputting 11/4/13, indicates that the year is 2013, not 0013. In order to have the year 0013, the full four digits (0013) must be input for the date. So, the date would be input as 11/4/0013 CE for November 4, 0013 CE or 11/4/0013 BCE for November 4, 0013 BCE.

Working with Date/Time Values

Date/time values can be displayed as labels on axes, map layers, and used in setting limits on maps. Below are some methods available to work with date/time formats.

Formatting Data as Date/Time

To format cells in the worksheet as date/time, open the worksheet and select all of the cells that should be date/time format. Click the **Data | Format | Format Cells** command. In the Format Cells dialog, select *Date/time* as the *Type* and type the date/time format string into the *Date/Time format* field. Click OK and the selected cell is formatted as date/time. Alternatively, click the ... button to create the date/time format in the Date/Time Format Builder dialog. Save to a format, such as an Excel file, that accepts date/time formats to retain the date/-time format.

Gridding Date/Time Values

Any worksheet column containing numbers, dates, or times can be used for gridding. When using date/time formats for any of the *Data Columns*, the values are stored in the grid as numbers, not in date/time format. To display date/time formats on the map, select the appropriate map part (axis, map layer, or map) and set the date/time label format.

Formatting Axes to Display Date/Time

Any axis can be changed to display dates or times for axis labels. To display date or time labels, click on the axis to select it. In the **Properties** window, click on the General tab. In the *Labels* section in the *Label Format* section, change the *Type* to *Date/time*. Then, set the *Date/Time format* to the desired label formats.

Formatting Contour Maps to Display Date/Time Labels

Any contour map label can be changed to display dates or times for axis labels. To display date or time labels, click on the contour map layer to select it. In the **Properties** window, click on the Levels tab.

For simple or logarithmic level methods: In the *Labels* section in the *Label Format* section, change the *Type* to *Date/time*. Then, set the *Date/Time Format* to the desired label formats.

For advanced level methods: Click the *Edit Levels* button next to the *Contour levels* command. In the **Levels for Map** dialog, click the *Label* button. Click the *Format* button to open the Label Format dialog. Change the *Type* to *Date/Time*. Then, set the *Date/Time Format* to the desired label formats. Click *OK* in all dialogs and the labels update.

Formatting Post or Classed Post Maps to Display Date/Time Labels

Any post map or classed post map label can be changed to display dates or times for axis labels. To display date or time labels, click on the post map layer or classed post map layer to select it. In the **Properties** window, click on the Labels tab. In the *Label Set 1* section, set the *Worksheet column* to the column that contains the date/time values. In the *Label Format* section, change the *Type* to *Date/time*. Then, set the *Date/Time Format* to the desired label formats.

Setting Map Limits with Date/Time

When using date/time formats for any of the axis labels, the minimum and maximum on the **Limits** tab are entered in date/time format. To change the map limits, click on the *Map* object to select it. In the **Properties** window, click on the Limits tab. Highlight the existing date/time value in any of the *xMin, xMax, yMin,* or *yMax* boxes and enter the minimum or maximum date/time value. For instance, 02/02/2014 12:00:00 AM can be entered into the *xMin* option. The map limits must be entered in M/d/yyyy hh:mm:ss TT format.

Grid Residuals

The Grids | Calculate | Residuals command takes an existing grid and an X,Y,Z column from a data file and computes the residuals at the locations specified in the data file. The residual value is written to a new column in the worksheet. If the input Z values in the worksheet are in date/time format, then the residuals are the difference between the Z grid value and the input date/time Z value. This is not a date/time format, but is rather the difference between the times, signifying a time duration. The units are days.

Date Time Formats

Date and time formats can be set from the worksheet, from labels, and from axes. In addition, date and time formats can be used for data columns when creating post maps or when gridding data. Date and time options are case sensitive.

When dates are parsed during input/import, the month and day of week names must match those of the local language as set in the Windows Control Panel, otherwise the entry will not be recognized as a valid date and will be treated as a text string.

Date/Time formats are made of combinations of locale, year, month, day, hours, minutes, seconds, BC/AD or BCE/CE designation, and AM/PM designation. Years are shown as yy or yyyy. Months are shown as M, MM, MMM, MMMM, or MMMMM. Days are shown as d, dd, ddd, or dddd. Hours are shown as h, hh, H, HH, or [h]. Minutes are shown as m, mm, or [mm]. Seconds are shown as ss, ss.0, ss.00, ss.000, ss.0000, or [ss]. AM/PM designation is shown as tt or TT. BC/AD designation is shown as gg or GG. BCE/CE designation is shown as g, G, ggg or GGG.

To add new date/time designations, use any combination of the following codes:

d	9	Single digit day, excluding leading zero
dd	09	Double digit day, including leading zero
ddd	Wed	Shortened day of week name
dddd	Wednesday	Full day of week name
M	7	Single digit month, excluding leading zero
MM	07	Double digit month, including leading zero
MMM	Jul	Shortened month name
MMMM	July	Full month name
MMMMM	J	First letter of month name
yy	98	Two digit year
yyyy	1998	Full year
g		Before Common Era designator - Includes space and bce or nothing if ce, lower case
gg	ad	BC/AD designator - Includes space and bc or ad, lower case
ggg	ce	Before Common Era designator - Includes space and bce or ce, lower case
G		Before Common Era designator - Includes space and BCE or nothing if CE, upper case
GG	AD	BC/AD designator - Includes space and BC or AD, upper case
GGG	CE	Before Common Era designator - Includes space and BCE or CE, upper case
h	6	Single digit hours - 1-12, excluding leading zero
hh	06	Double digit hours - 01-12, including leading zero
H	18	Hours - 0-23 military, excluding leading zero
HH	18	Hours - 00-23 military, including leading zero
[h]	1003914	Hours portion of total time, excludes leading zeros
m	45	Minutes - 0-60, excluding leading zero
mm	45	Minutes - 00 to 60, including leading zero
[mm]	45	Minutes portion of total time, includes leading zeros
ss	44	Seconds - 0-60, rounded to the nearest second
ss.0	44.1	Seconds - 0-60, rounded to the nearest tenth of a second
ss.00	44.12	Seconds - 0-60, rounded to the nearest hundredth of a second
ss.000	44.123	Seconds - 0-60, rounded to the nearest millisecond
ss.0000	44.12345	Seconds - 0-60, maximum precision

[ss]	44	Seconds portion of total time, includes leading zeros
tt	pm	am or pm designator, lower case
TT	PM	AM or PM designator, upper case
\		escape character - output next character verbatim
'...'		output ALL characters between single quotes verbatim, including escape character
[$-xxxx]	[$-409]	xxxx is an up to four hex digit representation of a locale ID

Custom Date/Time Example

mm/dd/yy h:mm:ss tt	Month double digits, Day double digits, Year double digits, Hour in standard format, Minutes, Seconds and AM/PM designation	04/14/09 6:45:44 PM

When dates are parsed during input/import, the month and day of week names must match those of the local language as set in the Windows Control Panel, otherwise the entry will not be recognized as a valid date and will be treated as a text string.

When the recognized format is ambiguous (i.e. 10/7/12), the month, day, and year order is determined by the Windows locale. In some countries, this will be recognized as M/d/yy, in others as d/M/yy, and in others as YY/M/d. It is important to use non-ambiguous date/time formats when the Windows locale may change.

The tables below show many examples of date/time format strings.

Date Formats

All rows below use the date September 7, 1998 for the *Example*.

Date/Time Code	**Example**	**Description**
(None)		Date not displayed
M/d/yy	9/7/98	Single digit month and day, two digit year, separated with /
MM/dd/yy	09/07/98	Double digit month, day, and year, separated with /
M/d/yyyy	9/7/1998	Single digit month and day, full year, separated with /
MMM dd, yyyy	Sep 07, 1998	Shortened month name, double digit day, full year, separated with spaces and comma
MMMM dd, yyyy	September 07, 1998	Full month name, double digit day, full year, separated with spaces and comma

MMMM-d-yyyy	September-7-1998	Full month name, single digit day, full year, separated with -
d MMMM yyyy	7 September 1998	Single digit day, full month name, full year, separated with spaces
d-MMM-yy	7-Sep-98	Single digit day, shortened month name, two digit year, separated with -
dd-MMM-yy	07-Sep-98	Double digit day, shortened month name, two digit year, separated with -
d-MMM-yyyy	7-Sep-1998	Single digit day, shortened month name, full year, separated with -
d-MMM	7-Sep	Single digit day, shortened month name, separated with -
MMM-yy	Sep-98	Shortened month name, two digit year, separated with -
MMM-yyyy	Sep-1998	Shortened month name, full year, separated with -
MMMM-yy	September-98	Full month name, two digit year, separated with -
MMMM-yyyy	September-1998	Full month name, full year, separated with -
MM-dd-yy	09-07-98	Double digit month and day, two digit year, separated with -
yyyy	1998	Full year
yyyy gg	1998 ad	Full year with lowercase bc/ad designation
yyyy GGG	1998 CE	Full year with uppercase BCE/CE designation
yy	98	Two digit year
MMMMM	S	First letter of month name
MMMM	September	Full month name
MMM	Sep	Shortened month name
MM	09	Double digit month
M	9	Single digit month
MMMMM-yy	S-98	First letter of month name, two digit year, separated with -
MMM-d	Sep-7	Shortened month name, single digit day, separated with -
M/d	9/7	Single digit month and day, separated with /
dddd	Monday	Full day of week name
ddd	Mon	Shortened day of week name
dd	07	Double digit day
d	7	Single digit day
d/M/yy	7/9/98	Single digit day and month, two digit year, separated with /
d.M.yy	7.9.98	Single digit day and month, two digit year, separated with .

dd/MM/yy	07/09/98	Double digit day and month, two digit year, separated with /
dd/MM/yyyy	07/09/1998	Double digit day and month, full year, separated with /
yy/MM/dd	98/09/07	Two digit year, double digit month and day, separated with /
yyyy-MM-dd	1998-09-07	Full year, double digit month and day, separated with -

Time Formats

All rows below use the time 2:45:44.12 PM for the *Example*.

Date/Time Code	**Example**	**Description**
(None)		Time not displayed
h:mm tt	2:45 PM	Hour in 0-12 (standard format), two digit Minutes 00 to 60, then a space and AM or PM
h:mm	14:45	Hour in 0-23 (military time), two digit Minutes 00 to 60
hh:mm	14:45	Two digit Hour 00-23 (military time), two digit Minutes 00 to 60
h:mm:ss tt	2:45:44 PM	Hour in 0-12 (standard format), two digit Minutes 00 to 60
h:mm:ss	14:45:44	Hour in 0-23 (military time), two digit Minutes 00 to 60, two digit Seconds 00 to 60
hh:mm:ss	14:45:44	Two digit Hour 00-23 (military time), two digit Minutes 00 to 60, two digit Seconds 00 to 60
m:ss	45:44	Single digit Minutes 0 to 60, two digit Seconds 00 to 60
mm:ss	45:44	Two digit Minutes 00 to 60, two digit Seconds 00 to 60
m:ss.0	45:44.1	Single digit Minutes 0 to 60, two digit Seconds 00 to 60, fractional seconds rounded to the nearest tenth of a second
mm:ss.0	45:44.1	Two digit Minutes 00 to 60, two digit Seconds 00 to 60, fractional seconds rounded to the nearest tenth of a second
h:mm:ss.000	14:45:44.12	Hour in 0-23 (military time), two digit Minutes 00 to 60, two digit Seconds, 00 to 60, fractional seconds with full precision

m:ss.000	45:44.12	Single digit Minutes 0 to 60, two digit Seconds 00 to 60, fractional seconds with full precision
mm:ss.000	45:44.12	Two digit Minutes 00 to 60, two digit Seconds 00 to 60, fractional seconds with full precision
[h]:mm:ss	865094:45:44	Total hours (day value plus hour value), two digit Minutes 00 to 60, two digit Seconds 00 to 60. Example Explanation: Date value 865080 = September 7, 1998 Hour value = 14, added to 865080 = 865094

Opening a Worksheet Window

You can view, enter, or modify data in the worksheet document.

To open a blank worksheet window:

- Click the File | New | Worksheet command in the plot document, grid editor, or worksheet document.
- Click the  button in the toolbar.
- Press the CTRL + W keyboard command.

To view worksheet data:

- Click the File | Open in Worksheet command in the plot document, grid node editor, or worksheet document, click on a data file and click *Open*. To open multiple data files in separate windows, hold down the CTRL key while clicking on files to select multiple files, or hold down the SHIFT key to select adjacent files.
- Click Data | Edit | Open Data command in a worksheet document. In the Open dialog, select a data file and click *OK*.

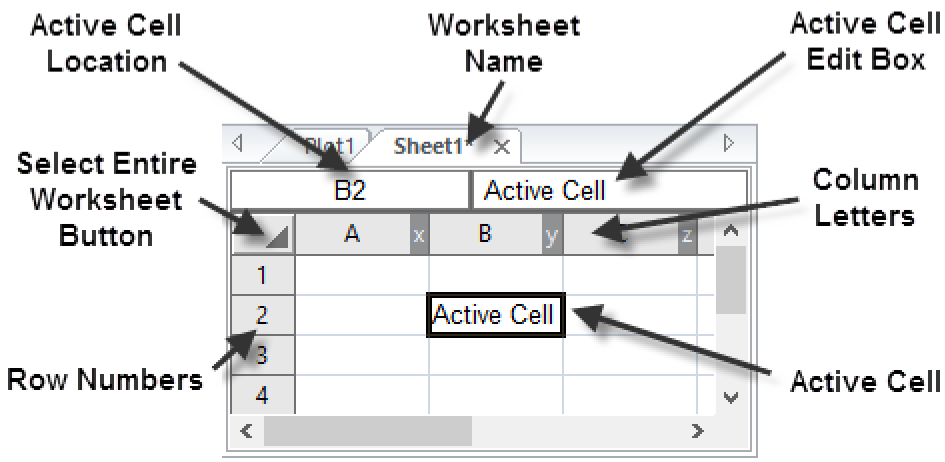
  Click the  button on the **Data** tab Ribbon. In the **Open** dialog, select a data file.
- Select the Data | Edit | Merge command in the worksheet and then select a data file.
- If there is an open worksheet window, return to it at any time by clicking the desired worksheet tab.

To enter and modify worksheet data:

See Working with Worksheet Data for more information.

Worksheet Window

To enter data in a worksheet, click the **File | Open** command to open an existing data file or click the **File | New | Worksheet** command to create a blank worksheet. The components of the worksheet window are discussed below.

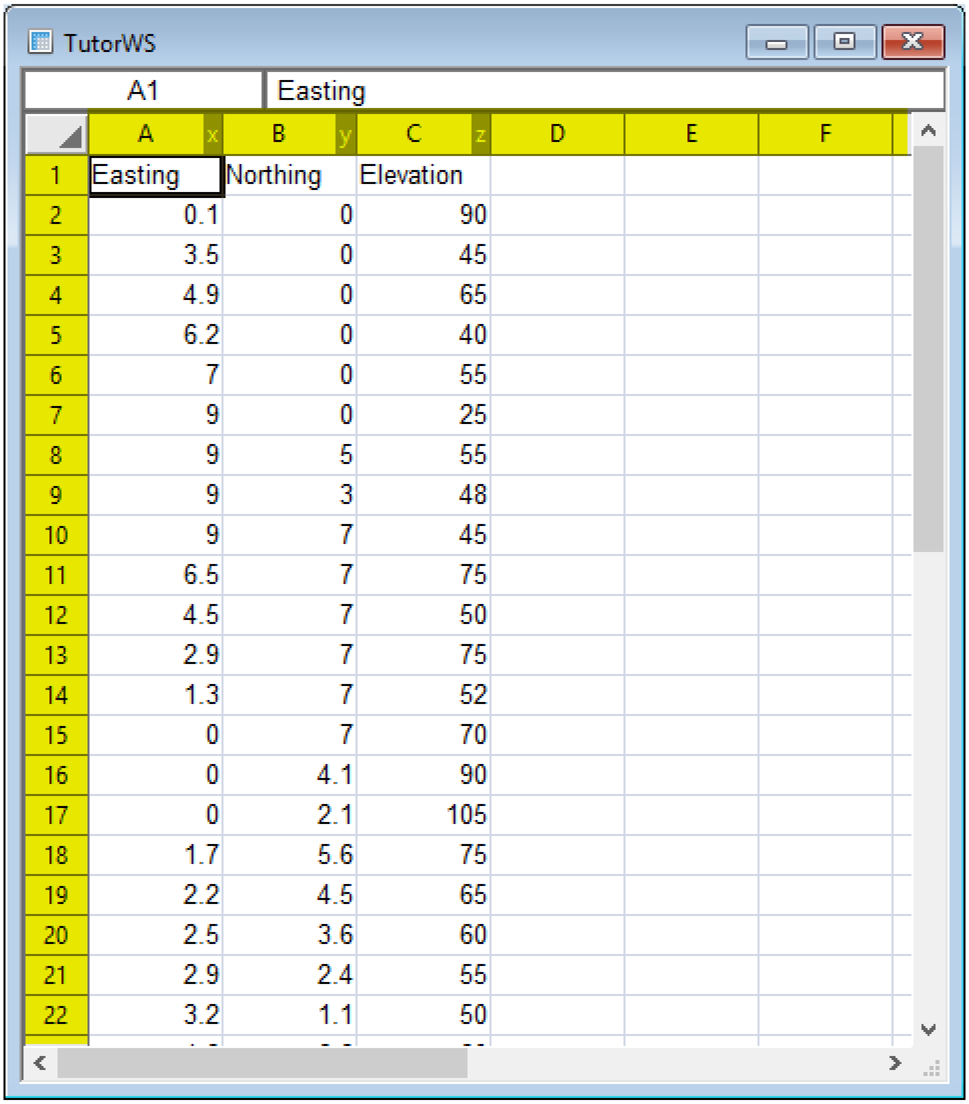


The components of a worksheet window shown above are described in the table below.

Component	Function
Column Letters	The letter that identifies a column of the worksheet.
Row Numbers	The number that identifies a row of the worksheet.
Active Cell	The cell highlighted with a bold outline. The active cell receives data input (numeric values or text strings) from the keyboard. Only one cell is active at a time.
Active Cell Location	The location of the active cell, specified by column letter and row number.
Active Cell Edit Box	The box displaying the data or text contained in the active cell. Data typed into an empty cell appears in both the edit box and the active cell.
Worksheet Name	The name of the data file displayed in the worksheet or the worksheet number prior to saving.
Select Entire Worksheet Button	The button used to select all cells in the worksheet. Located in the top left corner of the worksheet.

Row and Column Label Bars

The worksheet cells are located by column label bars (A, B, C...) or row label bars (1,2,3...). Click the label to select entire rows or columns, to change row height, to change column width, or to hide or unhide rows and columns. To select multiple rows or columns, drag the mouse over several adjacent labels.



The column and row label bars are highlighted in this example.

Active Cell

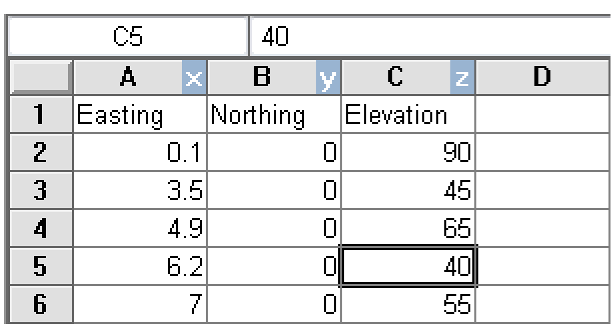
The active cell is displayed with a heavy border surrounding the cell. The contents of this cell are displayed in the cell edit box. You can enter or edit data in the active cell. To edit existing data, activate the desired cell and press the F2 key or highlight the information in the cell edit box.

Special Key Functions when editing the active cell include the following:

Keyboard Command	**Action**
ESC	ESC cancels edit mode and restores the original contents of the active cell.
ENTER	ENTER stores the contents of the cell edit box and then moves the active cell down one cell.
CTRL+ENTER	CTRL+ENTER completes the entry and keeps the current cell active.
ARROWS (left and right)	Left and right ARROWS move within the cell's text if the F2 key has been pressed. Otherwise, these keys store the contents of the cell edit box and then move the active cell to the left or right.
ARROWS (up and down)	Up and down ARROWS store the contents of the cell edit box in the active cell and move the active cell above or below.
DELETE	DELETE deletes the character to the right of the cursor if the F2 key has been pressed. Otherwise, pressing the delete key deletes the entire contents of the cell.
BACKSPACE	BACKSPACE deletes the character to the left of the cursor if the F2 key has been pressed. Otherwise, pressing the backspace key deletes the entire contents of the cell.
PAGE UP and PAGE DOWN	PAGE UP and PAGE DOWN store the contents of the cell edit box in the active cell and move one page up or down.
TAB and SHIFT+TAB	TAB and SHIFT+TAB store the contents of the cell edit box in the active cell and move the active cell to the right or left.

Active Cell Location Box

The active cell location box shows the location of the active cell in the worksheet. Letters are the column labels and numbers are the row labels.


C5 | 40

	A x	B y	C z	D
1	Easting	Northing	Elevation	
2	0.1	0	90	
3	3.5	0	45	
4	4.9	0	65	
5	6.2	0	40	
6	7	0	55	

This example shows the active cell as cell C5. The name of the active cell "C5" is listed in the active cell location box in the upper left portion of the worksheet.

Active Cell Edit Box

The cell edit box is located at the top of the worksheet window just above the column letter bar. The cell edit box shows the contents of the active cell and is used for editing cells. Use the cell edit box to see the contents of a worksheet cell when the column is too narrow to display all of the cell contents.

To begin editing the selected cell, press the F2 key. Alternatively, highlight the contents of the cell edit box to edit the cell. To overwrite the current cell contents, simply begin typing without pressing F2. If the mouse is clicked on a new cell, the new cell becomes the active cell.

Right-click in the active cell edit box to access the following commands in the context menu:

Right to left Reading order	Toggles right to left reading order on or off.
Show Unicode control characters	Toggles the display of Unicode control characters on or off.
Insert Unicode control character	Select a Unicode control character from the list, and it is inserted in the active cell edit box at the cursor location.
Open/Close IME	When a user types a phonetic representation of a word, the IME displays a candidate list on the screen. The user can select the intended word or phrase from among several different possible representations in the candidate list, and the user's selection then replaces the phonetic representation in the document. This command toggles the IME on or off.
Reconversion	IME reconversion allows users who are typing in Japanese to convert back and forth between the phonetic spelling of a word (using the standard Western keyboard) and the Japanese character that represents the word.

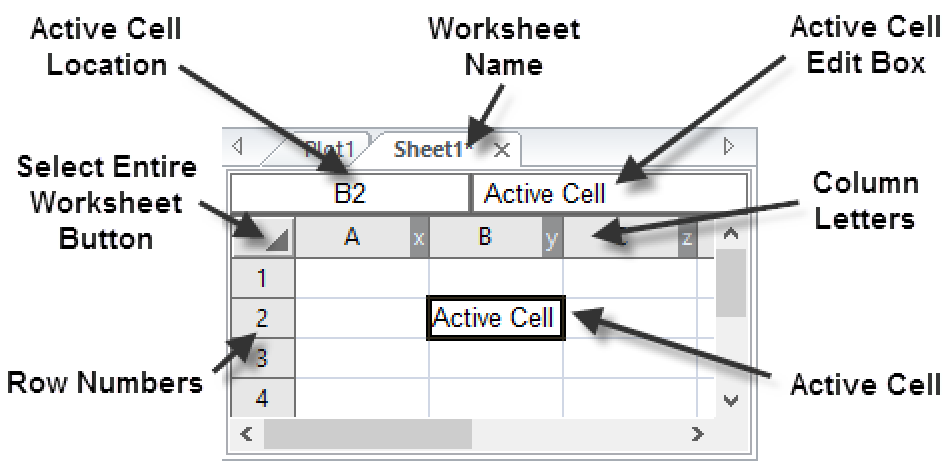
Special Key Functions when Editing the Active Cell:

Keyboard Command	**Action**
ESC	ESC cancels edit mode and restores the original contents of the active cell.
ENTER	ENTER stores the contents of the cell edit box and then moves the active cell down one cell.
CTRL+ENTER	CTRL+ENTER completes the entry and keeps the current cell active.

ARROWS (left and right)	Left and right ARROWS move within the cell's text if the F2 key has been pressed. Otherwise, these keys store the contents of the cell edit box and then move the active cell to the left or right.
ARROWS (up and down)	Up and down ARROWS store the contents of the cell edit box in the active cell and move the active cell above or below.
DELETE	DELETE deletes the character to the right of the cursor if the F2 key has been pressed. Otherwise, pressing the delete key deletes the entire contents of the cell.
BACKSPACE	BACKSPACE deletes the character to the left of the cursor if the F2 key has been pressed. Otherwise, pressing the backspace key deletes the entire contents of the cell.
PAGE UP and PAGE DOWN	PAGE UP and PAGE DOWN store the contents of the cell edit box in the active cell and move one page up or down.
TAB and SHIFT+TAB	TAB and SHIFT+TAB store the contents of the cell edit box in the active cell and move the active cell to the right or left.

Select Entire Worksheet

Clicking on the small box above the row labels and to the left of the column labels selects the entire worksheet.



The Select Entire Worksheet *button is located to the left of column A and above row 1.*

Working with Worksheet Data

There are three ways to enter data into the worksheet. Data are entered into the worksheet by using the File | Open in Worksheet command and opening a data file, by typing data directly into the worksheet, or by copying the data from another application and pasting it into the worksheet. Use the Data menu commands to sort the data, filter the data, view statistics, transform the data using mathematical functions, assign default columns for coordinate data, assign a coordinate system to the data, and project coordinates.

There are two basic modes in the worksheet. Normal mode is when the active cell can be moved throughout the worksheet, and edit mode allows the contents of a single cell to be edited in the active cell edit box. Only one mode may be active at a given time. ESC, ENTER, or clicking on another cell can be used to exit edit mode and return to normal mode.

Entering Data Into a Cell

Edit the contents of a cell by making it the active cell. The active cell is positioned by clicking on a cell with the mouse, by using the ARROW keys, PAGE UP, PAGE DOWN, TAB, HOME, END, and SHIFT+TAB. Press the F2 key or highlight the contents of the cell edit box to edit the contents of the cell.

To enter new data and delete the old, position the active cell and begin typing. Edit mode is entered automatically and the old data is deleted. Pressing the ENTER, Up or Down ARROWS, TAB, SHIFT+TAB, PAGE UP, or PAGE DOWN keys causes the edit changes to be recorded permanently in the cell. After pressing F2 or highlighting the cell edit box use the HOME, END, BACKSPACE, DEL, and ARROW keys to edit the cell. Pressing ESC while editing a cell cancels the changes and restores the original data.

Moving the Active Cell

You can designate any worksheet cell as the active cell by left-clicking on it with the mouse. The active cell can also be repositioned by using keyboard commands. The active cell is the cell with a thick border drawn around it.

Keyboard Command	Action
ARROW keys (Up, Down, Left, Right)	The ARROWS move the active cell to an adjacent cell.
PAGE UP/PAGE DOWN	Press the PAGE UP or PAGE DOWN to move the active cell up or down by the number of rows visible in the window.
HOME	Press HOME to move the active cell to the first occupied cell in the current column. Press HOME again to move the active cell to the top row in the current column.
END	Press END to move the active cell to the last occupied row in the current column. Press END again to move the active cell to the bottom row of the worksheet.
ENTER	Press ENTER to move the active cell down one row and end "edit mode."
TAB	Press TAB to move the active cell right one column and end "edit mode."

SHIFT + ENTER	Press SHIFT+ENTER to move the active cell up one row and end "edit mode."
SHIFT + TAB	Press SHIFT+TAB to move the active cell left one column and end "edit mode."
CTRL+HOME	Press CTRL+HOME to move the active cell to the top cell of the left most column in the worksheet (A1).
CTRL+END	Press CTRL+END to move the active cell to the bottom occupied row of the last occupied column in the work-sheet.
CTRL+LEFT ARROW	The CTRL+LEFT ARROW behavior depends on the position of the active cell. If the active cell is to the right of the last occupied column in the current row, it moves the active cell to the last occupied column in the current row. If the active cell is in or to the left of the last occupied column in the current row, but to the right of the first occupied column in the current row, it moves the active cell to the first occupied column in the current row. Otherwise,
CTRL+RIGHT ARROW	The CTRL+RIGHT ARROW behavior depends on the pos-ition of the active cell. If the active cell is to the left of the first occupied column in the current row, it moves the act-ive cell to the first occupied column in the current row. If the active cell is in or to the right of the first occupied column in the current row, but to the left of the last occu-pied column in the current row, it moves the active cell to the last occupied column. Otherwise,
CTRL+UP ARROW	The CTRL+UP ARROW behavior depends on the position of the active cell. If the active cell is below the bottom occu-pied row in the current column, it moves the active cell to the bottom occupied row in the current column. If the act-ive cell is below the top occupied row in the current column, but in or above the bottom occupied row in the current column, it moves the active cell to the top occu-pied row in the current column. Otherwise,
CTRL+DOWN ARROW	The CTRL+DOWN ARROW behavior depends on the pos-ition of the active cell. If the active cell is above the top occupied row in the current column, it moves the active cell to the top occupied row in the current column. If the active cell is above the bottom occupied row in the current column, but below the top occupied row in the current column, it moves the active cell to the bottom occupied row in the current column. Otherwise,
ENTER, TAB, SHIFT+ENTER, and SHIFT+TAB	If a block of cells is selected, the ENTER, TAB, SHIFT+ENTER, and SHIFT+TAB keys move the active cell within a group of selected cells without canceling the selec-tion.

Moving the Active Cell Within Selections

The ENTER, TAB, SHIFT+ENTER, and SHIFT+TAB keys move the active cell within a group of selected cells without canceling the selection.

Pasting Data

If data are copied to the clipboard from another software application, the contents of the clipboard can be pasted into the worksheet. If the source application is Microsoft Excel, some formatting information is retained. When pasting data into the worksheet, select a cell and use Home | Clipboard | Paste (CTRL+V). Any data to the right or below the active cell is overwritten, so be sure to locate the active cell carefully. When data are copied to the clipboard, special formatting information is also copied. The Home | Clipboard | Paste | Paste Special command determines the format in which the contents are pasted into the worksheet.

Opening Data Files

When you create a grid file or use another command that requires data, you do not need to open the data into the worksheet first. However, the worksheet is available if you would like to view or edit your data. The File | Open in Worksheet command loads a data file into a new worksheet.

If the worksheet already contains data, additional data can be imported into the worksheet using the Data | Edit | Merge command. The contents of the new file are merged into the worksheet at the active cell so it is imperative that the cell be positioned at the edge of the existing data. Any cells in the existing worksheet that lie to the right and below the active cell will be overwritten with the contents of the merging file.

Multiple files can be opened at one time into the same worksheet with **Data | Edit | Merge** using the SHIFT or CONTROL keyboard keys while selecting files in the Import Data dialog.

Worksheet Input Modes

The worksheet has several special input modes that tracks the mouse position:

- Drag-Select Mode - for selecting cells with the mouse
- Drag-Row-Height Mode - for adjusting row heights with the mouse
- Drag-Column-Width Mode - for adjusting column widths with the mouse

Pressing the ESC key before releasing the mouse button cancels the mouse-tracking mode.

Selecting Cells

The keyboard and the mouse may be used to select cells. Selected cells are indicated by reverse video (white background becomes black, etc.). Hidden cells are selected if their columns or rows are within a selected block of cells. Single cells, a rectangular block of cells, one or more rows, one or more columns, or the entire worksheet can be selected.

Selecting Cells

Cells may be selected to:

- perform editing and clipboard functions,
- perform a transform function,
- sort the selected cells,
- compute statistics for selected cells, or to
- set column properties for several columns via the column width, row height, and cell format commands.

There are several ways to select cells:

- Clicking on the small box above the row labels and to the left of the column label bar selects the entire worksheet.
- To deselect all selected cells, click the left mouse button anywhere within the worksheet, or move the active cell with an ARROW key. Alternatively, the PAGE UP, PAGE DOWN, HOME, and END keys may also be used to deselect the cells.
- To rapidly select a large block, first select one corner of the block, and then use the scroll bars to scroll to the opposite corner. Hold down the SHIFT key and click on the cell at the opposite corner. The PAGE UP, PAGE DOWN, HOME, and END keys may also be used, but the SHIFT key must be held down while these keys are pressed. The SHIFT key is not needed while using the scroll bars.
- To select all cells in a column or row, click the column letter or row number. To select several adjacent columns or rows, press and hold the left mouse button and drag the pointer on the column letters or row numbers. To deselect a single row or column from a multiple row or column selection, hold CTRL and click the row or column label.
- While holding down the CTRL key, the active cell may be repositioned for selecting a new, discontinuous block.

- The CTRL key is used to select multiple blocks and the SHIFT key is used to resize the last selected block. Details and exceptions are given in separate help sections for selecting with the mouse and selecting with the keyboard.
- If entire rows or columns are selected by clicking on the headers, some operations, such as statistics, can take a long time. Rather than clicking on the headers, only select the cells containing data.
- Clicking and holding the left mouse button while dragging the mouse in the worksheet selects a block. Similarly, using the SHIFT key plus the ARROW keys selects a block.
- The keys used with SHIFT for selecting cells are the ARROW keys, PAGE UP, PAGE DOWN, HOME, and END. TAB and SHIFT+TAB cannot be used.
- While holding down the SHIFT key, the last selected block may be resized. Use the SHIFT key and the mouse or the SHIFT key and ARROW keys.
- The active cell is at one corner (or edge) of a selected block and must first be positioned before selecting multiple cells.
- The last block cannot be resized if the active cell has been moved.

Selecting Cells with the Keyboard

The keyboard may be used to select cells. Selected cells are indicated by reverse video (white background becomes black, etc.).

To Select	**Process**
Single cells	Click in the cell to select it, or use the arrow keys to select a cell. The selected cell will have a thick outline around it.
A rect-angular block of cells	Move the active cell to one corner of the block. While holding down the SHIFT key, use the movement keys to position the opposite corner of the block. The movement keys include the ARROW keys, PAGE UP, PAGE DOWN, HOME, and END, but not TAB and SHIFT+TAB. When the block has been sized, release the SHIFT key. To resize the block, see the instructions below.
Several adjacent rows	Select the first or last row. Then, while holding down the SHIFT key, use the vertical movement keys. These include up ARROW, down ARROW, page up, page down, HOME, and END.
Several adjacent columns	Select the first or last column. Then, while holding down the SHIFT key, use the right and left ARROW keys.

Resize the Last Selected Block

To resize the last selected block, hold down the SHIFT key while using the movement keys (as appropriate to the type of block). The last block cannot be resized if the active cell has been moved.

Deselect All Selected Cells

To deselect all selected cells, left-click anywhere within the worksheet or move the active cell with an ARROW key or other movement key.

Selecting Cells with the Mouse

The mouse may be used to select cells. Selected cells are indicated by reverse video (white background becomes black, etc.).

To Select	Process
Single cells	Click the left mouse button on the cell. The cell will have a thick outline around it.
A rectangular block of cells	Move the active cell to one corner of the block. Click and hold the left mouse button, and drag it to the opposite corner of the block. Then release the mouse button.
An entire row	Click the mouse on the row label.
Several adjacent rows	Click and hold the mouse on the first row label and drag it to the last row. Make sure the cursor is a normal arrow cursor not the double arrow cursor used for selecting column dividing lines. To deselect a single row from a multiple row selection, hold CTRL and click the row label.
An entire column	Click the mouse on the column label.
Several adjacent columns	Click and hold the mouse on the first column label and drag it to the last column. Make sure the cursor is a normal arrow cursor not the double arrow cursor used for selecting column dividing lines. To deselect a column from a multiple column selection, hold CTRL and click the column label.
The entire worksheet	Click on the small box above the row labels and to the left of the column label bar.

The worksheet will scroll automatically if the mouse is dragged past the visible limits of the worksheet.

Select Additional Blocks

To select additional blocks, hold down the CTRL key while clicking.

Resize the Last Selected Block

To resize the last selected block, hold down the SHIFT key while clicking and holding the left mouse button. Then, drag the edge of the last selected block to the new position. The last block cannot be resized if the active cell has been moved.

Deselect All Selected Cells

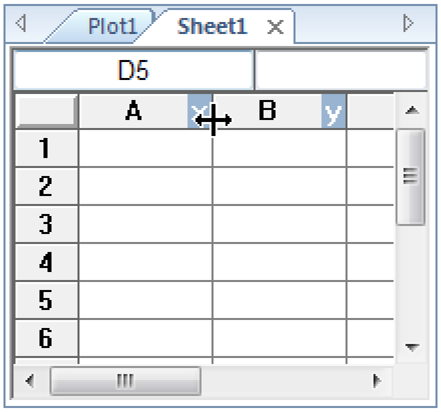
To deselect all selected cells, left-click anywhere within the worksheet or move the active cell with an arrow key or other movement key.

Selecting a Column or Row Dividing Line

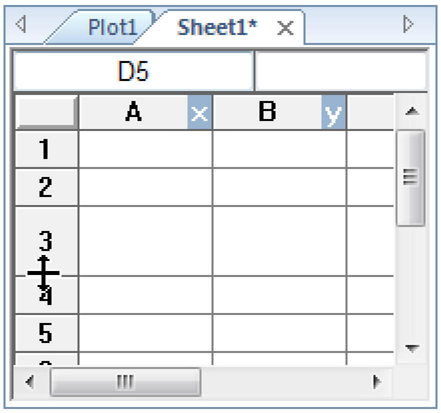
The column or row dividing lines are the lines between the column letter labels and row number labels along the borders of the worksheet. These lines divide the columns or rows. When selecting a dividing line, the cursor must be within appoximately a characther's width of the dividing line and it must be on the label bar.

Change the column width or row height by dragging the dividing line. Rows or columns can be hidden or unhidden by using the mouse. The Data | Format | Column Width or Data | Format | Row Height commands can also be used to set column widths or row heights.

Move the cursor to the label bar near the dividing line until the cursor changes to a  between columns, and a  between rows. The cursor must be within approximately a character's width of the dividing line and it must be on the label bar. Click and hold the left mouse button and drag the dividing line.



This example shows the cursor being used to change the width of column A.



This example shows the cursor being used to change the height of row 3.

Hiding Columns or Rows

The mouse may be used to hide columns or rows.

To hide a column, first click on the vertical dividing line to the right of the column. Drag the vertical dividing line to the left as far as it will go and then release the mouse button. If there are hidden columns to the right of this column, grab the left side of the vertical dividing line. If the right side of the vertical dividing line is selected, the vertical dividing line for the adjacent hidden column is selected.

To hide a row, first click on the horizontal dividing line at the bottom of the row. Drag the horizontal dividing line up as far as it will go and then release the mouse button. If there are hidden rows above this row, grab horizontal dividing line just below the line. If the top side of the horizontal dividing line is selected, the horizontal dividing line for the adjacent hidden row is selected.

With the Format Menu

Columns and rows can also be hidden with the Data | Format | Column Width and Data | Format | Row Height commands. Select the columns or rows to hide, click the **Data | Format | Column Width** or **Data | Format | Row Height** command, and then set the *Column Width* or *Row Height* to zero.

Displaying Hidden Columns or Rows

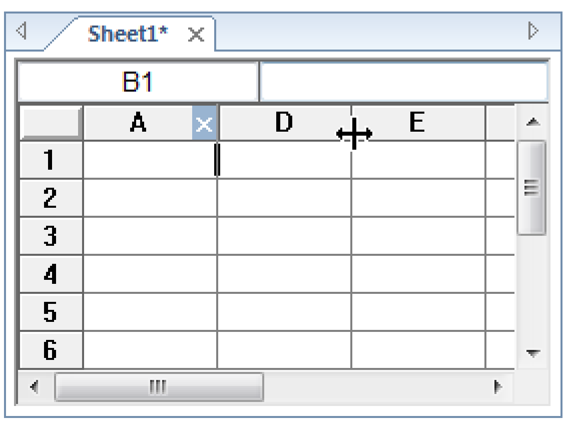
The mouse may be used to display hidden columns or rows.

To display a hidden column, first click on the vertical dividing line at the **right** of the hidden column and then drag the vertical dividing line to the new position. If several adjacent columns are hidden, only the far right column is displayed after dragging the dividing line. If the cursor is to the left of the vertical dividing line when the line is selected, then the selected vertical dividing line is for the visible column to the left and not for the hidden column.

To display a hidden row, first click on the horizontal dividing line below the hidden row and then drag the horizontal dividing line to the new position. If several adjacent rows are hidden, only the bottom row is displayed after dragging the dividing line. If the cursor is above the horizontal dividing line when the line is selected, then the horizontal dividing line will be for the visible row above the hidden rows, and not for the hidden rows.

Example

If columns B, C, and D are hidden and columns A and E are visible, then one vertical dividing line appears between columns A and



This example shows the vertical line being dragged to display hidden column D.

With the Format Menu

Hidden columns and rows can also be displayed with the Data | Format | Column Width and Data | Format | Row Height commands. To display hidden rows or columns, select the columns or rows on both sides of the hidden columns or rows, click the **Data | Format | Column Width** or **Data | Format | Row Height** command, and then set the *Column Width* or *Row Height* to a number greater than zero.

Worksheet Error Codes and Special Numeric Values

There are a few different error codes and special numeric values that can appear in a worksheet cell depending on the type and nature of the data that appears.

Codes	Explanation
#########	number will not fit in the column - the column must be wider for the number to be shown
#N/A	value cannot be computed (for example, not enough data to calculate a statistic)
#DIV/0!	an attempt to divide-by-zero was made in performing a calculation
#ERROR	a value could not be computed (for example, square root of a negative number)
#OVERFLOW	the value is too large for the worksheet (largest absolute value is about 1.797E+308)
1.#INF	the value is too large for the worksheet (i.e., "infinite" value)
1.#IND	numeric value is indefinite (usually the result of performing a calculation with an infinite value or attempting to divide by zero)

Worksheet Specifications

The following technical specifications for the worksheet include the number of cells allowed in the worksheet and the nature of the numbers allowed in the worksheet.

- Maximum number of rows in a worksheet: 1 billion
- Maximum number of columns in a worksheet: 1 billion
- Maximum numeric precision: 15 digits (Counting the digits before and after the decimal place)

- Maximum numeric resolution: 2.22E-16 (The smallest detectable difference between two numbers)
- Maximum absolute value: 1.79769E+308 (The largest value that can be represented)
- Minimum absolute value: 2.22507E-308 (The smallest value that is different from zero)
- Double precision floating-point numbers can only represent approximately 15 significant decimal digits.
- Approximate memory requirements for unformatted numeric data: 10.5 bytes per cell + 24 bytes per column

Example 1

10,000 rows of numbers in 3 columns

30,000 cells x 10.5 bytes/cell = 315,000 bytes (308 Kbytes)

3 columns x 24 bytes/column = 72 bytes

TOTAL MEMORY NEEDED (in addition to memory needed to run the program): 308 Kbytes

Example 2

3 rows of numbers in 10,000 columns

30,000 cells x 10.5 bytes/cell = 315,000 bytes (308 Kbytes)

10,000 columns x 24 bytes/column = 240,000 bytes (234 Kbytes)

TOTAL MEMORY NEEDED (in addition to memory needed to run the program): 542 Kbytes

Worksheet Commands

The following commands can be used to create and modify data in the worksheet.

Paste

Click the **Home | Clipboard | Paste** command or the  button, or press CTRL+V on the keyboard to paste the clipboard contents into the current document. The objects to be pasted must first be placed in the clipboard using the Cut or Copy commands of **Surfer** or some other application. The clipboard contents remain on the clipboard until something new is cut or copied to the clipboard.

Worksheet

In the worksheet, the upper left corner of the pasted data is placed in the active cell. Any cells in the existing worksheet that lie to the right of and below the active cell will be overwritten with the contents of the pasted data. The following rules are used to paste into the worksheet:

- Only the TAB character is recognized as a column separator. Spaces, commas, semi-colons, etc. are included in a text cell.
- The RETURN character is recognized as the row separator.
- Numbers paste as number values. The period can always be used as a decimal separator, and the system locale determines any other decimal separator. For example if the system locale uses a comma as the decimal separator, then both 123.456 and 123,456 are pasted as the number 123,456.
- Mixed text and numbers paste as text.
- Dates and/or times will paste as date values in a wide variety of date/time formats. Ambiguous dates are determined by the system locale setting. If dates are not pasted correctly, consider using Paste Special and the *Locale* settings in the **Data Import Options** dialog.

Pasting a single cell's contents across multiple cells

A single cell's contents can be pasted into multiple cells by coping the cell, selecting a block of cells, and using the **Paste** command. Each cell in the selection is populated with the clipboard contents when this operation is performed. If more than one cell is copied, then the copied cells are only pasted once.

The multiple paste operation will not be performed for an entire row, entire column, or across multiple selections. When an entire row or column is selected, only the first cell in the row or column will receive the pasted content. If you attempt to paste in multiple selections an error message will be displayed.

Plot

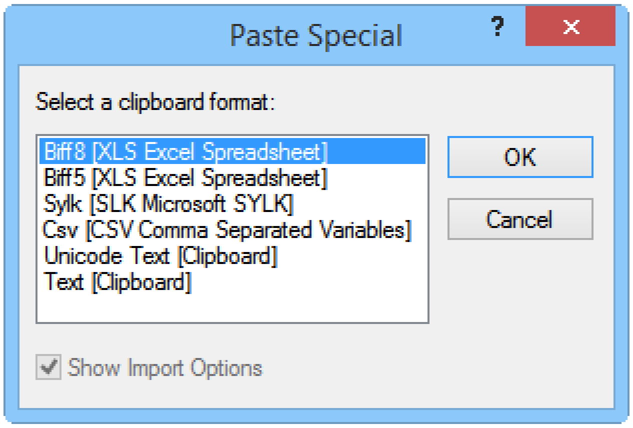
In the plot window, the clipboard contents are pasted in the center of the window. The pasted contents are automatically selected, and can be dragged to a new location. Alternatively, change the position or size of the selected object with the Position/Size group.

Paste Special - Worksheet

When data are copied to the clipboard, special formatting information is also copied. The **Paste Special** dialog determines the format in which the contents are pasted into the worksheet. The **Paste Special** command can also be used to remove text formatting.

Paste Special Dialog

Click the **Home | Clipboard | Paste | Paste Special** command in the worksheet to open the **Paste Special** dialog.



*Select a paste special format in the **Paste Special** dialog. This example shows options after copying **Surfer** worksheet data and using paste special.*

The clipboard formats displayed may vary depending the original location of the information being copied. For example, data copied from the **Surfer** worksheet may yield different options than data copied from Excel.

Biff8 [Excel Spreadsheet]

The *Biff8 [Excel Spreadsheet]* format is a Microsoft Excel Binary Interchange File Format (BIFF) version 8.

Biff5 [Excel Spreadsheet]

The *Biff5 [Excel Spreadsheet]* format is a Microsoft Excel Binary Interchange File Format (BIFF) version 5.

Biff4 [Excel Spreadsheet]

The *Biff4 [Excel Spreadsheet]* format is a Microsoft Excel Binary Interchange File Format (BIFF) version 4.

Biff3 [Excel Spreadsheet]

The *Biff3 [Excel Spreadsheet]* format is a Microsoft Excel Binary Interchange File Format (BIFF) version 3.

Biff [Excel Spreadsheet]

The *Biff [Excel Spreadsheet]* format is a Microsoft Excel Binary Interchange File Format (BIFF).

Sylk [Microsoft SYLK]

The *Sylk [Microsoft SYLK]* format is a symbolic link Microsoft file format typically used to exchange data between applications, specifically spreadsheets. The *Sylk* file format is composed of only displayable ANSI characters, allowing it to be easily created and processed by other applications, such as databases.

Unicode Text [Clipboard Text]

The *Unicode Text [Clipboard Text]* format is unformatted text.

Text [Clipboard Text]

The *Text [Clipboard Text]* format is unformatted text.

Show Import Options

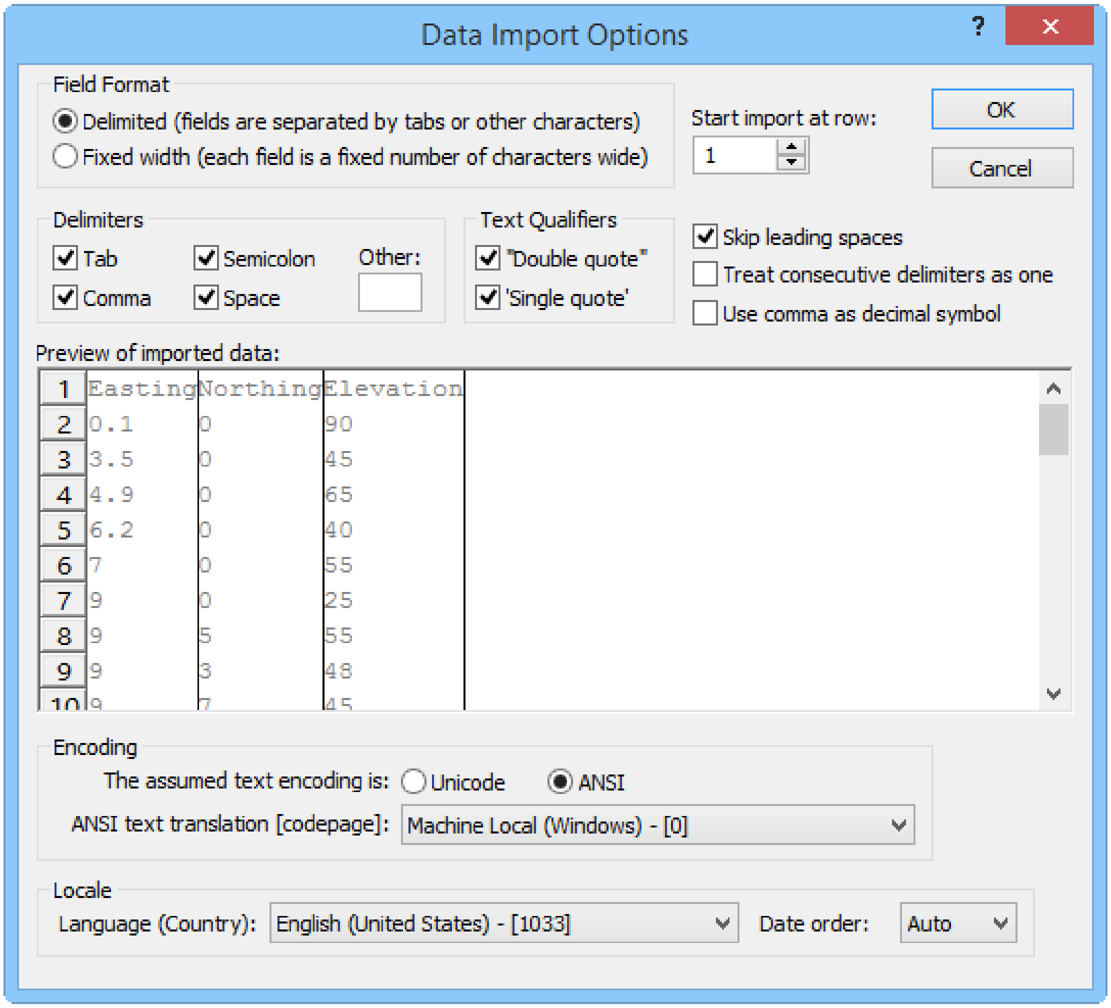
When *Unicode Text [Clipboard Text]* or *Text [Clipboard Text]* is selected, the *Show Import Options* option is available. Check the box to open the Data Import Options dialog before importing the data. The **Data Import Options** dialog is useful when pasting data with comma, period, semicolon, or other column delimiters; fixed-width column data; ambiguous date/time formatted data; and data from different locales.

Data Import Options

If a file is in an ASCII text format with an unrecognized file extension, the **Data Import Options** dialog appears when opening the file. Choose the *Delimiters* used in the file (*Tab*, *Comma*, *Semicolon*, *Space*, or *Other*), and the *Text Qualifiers* used in the file (*Double Quote* or *Single Quote*).

Data Import Options Dialog

The **Data Import Options** dialog may appear when importing tabular data from delimited text files (i.e. .DAT, .CSV, .TXT). These file formats are assumed to have one record per line in which each record contains a fixed number of numeric data fields.



*The **Data Import Options** dialog allows you to specify import options.*

Field Format

Specify the format of the input fields in the *Field Format* group. The options are *Delimited* or *Fixed Width*.

Delimited

Choose *Delimited (fields are separated by tabs or other characters)* if the imported data uses delimiters (tab, semicolon, comma, space, other) to separate data fields. The *Delimiters* group is used to specify how the fields are separated if *Delimited (fields are separated by tabs or other characters)* is the selected *Field Format*.

Fixed Width

Choose *Fixed Width (each field is a fixed number of characters wide)* if the imported data uses a fixed width to separate data fields.

Start Import at Row

Specify the row number at which to start the data import in the *Start import at row* box. To change the first row to import, highlight the existing value and type a new value or click the buttons to increase or decrease the value. For example, a value of one will start the data import at row one. A value of five will start the data import at row five and ignore the data in rows one through four.

Delimiters

Choose the desired delimiters to be used during the import process by checking the box next to *Tab*, *Comma*, *Semicolon*, or *Space*. You may also enter a custom delimiter in the *Other* box. More than one delimiter may be checked.

Text Qualifiers

Check the box next to *Double Quote* or *Single Quote* in the *Text Qualifiers* group to indicate the correct qualifier to identify text values in the data file. Everything between the selected characters will be interpreted as a single value, and any delimiter characters between text qualifiers are ignored and treated as part of the text.

Double Quote

Check the box next to *"Double Quote"* to specify that everything between those marks should be interpreted as a single value, and any delimiter characters between any two quote characters are not treated as a delimiter.

For example, if *Space* is chosen as the delimiter and *Double Quote* is chosen as the text qualifier, the string "Aspen Park" is treated as a single data value due to the double quotes surrounding it, and the space delimiter between the words is treated as part of the value.

Single Quote

Check the box next to *'Single Quote'* to specify that everything between those marks should be interpreted as a single value, and any delimiter characters between any two quote characters are not treated as a delimiter.

For example, if *Space* is chosen as the delimiter and *Single Quote* is chosen as the text qualifier, the string 'Aspen Park' is treated as a single data value due to the single quotes surrounding it, and the space delimiter between the words is treated as part of the value.

Skip Leading Spaces

Check the box next to *Skip leading spaces* to tell the software to ignore spaces that appear before initial text.

Treat Consecutive Delimiters as One

Check the box next to *Treat consecutive delimiters as one* to instruct the software to interpret any consecutive delimiters into a single delimiter rather than breaking to a new column for each consecutive delimiter.

Use Comma as Decimal Symbol

Check the box next to *Use comma as decimal symbol* to interpret every comma as the decimal symbol. The number 123,45 in the file would be displayed as 123.45 in the program worksheet with this option checked.

Preview

The parsed data are shown in the *Preview* section.

Encoding

The *Encoding* section allows the choice of *Unicode* data or *ANSI* data when importing or opening an ASCII data file. Unicode data is often referred to as international data. It would include character sets from Russia, Israel, China, Greece, Hungary, among others. After selecting *Unicode*, select the *ANSI text translation [codepage]* option that will read the data correctly. If the data does not appear correctly in the *Preview* window, the *Encoding* may be specified incorrectly.

ANSI encoding contains characters within the first 256 characters of a font. These are normally in English.

Locale

The locale section of the dialog contains options for determining date/time values. The *Language (Country)* setting determines which month names are interpreted as part of a date. For example, if *German* (*Germany)* is selected, "Oktober" will be recognized as a valid month name. English month names are always recognized as valid month names. The default *Language (Country)* is determined by the user locale set in the Windows Control Panel. To change the *Language (Country)*, click the current option and select a language from the list.

The *Date order* option specifies the order in which dates are written in the data file. The date 02/03/04 is ambiguous and could be Month-Day-Year, Day-Month-Year, Year-Month-Day, etc. The *Date order* option ensures dates in the data file are imported correctly into **Surfer**. The default *Date order* is *Auto*. The standard date order for the *Language (Country)* setting is used when *Date order* is set to *Auto*. Specify the *Date order* for the data file import by clicking the current *Date order* selection and then selecting the desired *Date order* from the list. All six combinations of Day (*D*), Month (*M*), and Year (*Y*) are included in the *Date order* list.

OK or Cancel

Click *OK* to proceed with the import process. Click *Cancel* to close the dialog without importing the data set.

Copy

Click the **Home | Clipboard | Copy** command or the button, or press CTRL+C on the keyboard to copy the selected objects to the clipboard. The original objects remain in the window. Use this command to duplicate objects in a different location in the same window, or copy the objects into a different window or application. The copied objects can later be pasted with the Paste or Paste Special commands.

Only one set of data may be placed in the clipboard at a time. The next Cut or **Copy** command replaces the contents of the clipboard.

Cut

Click the **Home | Clipboard | Cut** command or the button, or press CTRL+X on the keyboard to move the selected objects to the clipboard. This deletes the selected objects from the file after copying them to the clipboard. Cut objects can later be pasted with the Paste or Paste Special commands.

Only one set of data may be placed in the clipboard at a time. The next **Cut** or Copy command replaces the contents of the clipboard.

Data Tab Commands

The **Data** tab contains commands for editing the cell and worksheet, finding and replacing cell contents, formatting the cells, manipulating data, and assigning or projecting coordinate systems. The **Data** tab is only available when viewing a worksheet.

The worksheet ribbon **Data** tab has the following commands:

Open Data	Open data file in a new worksheet
Clear	Remove the selected cells
Merge	Combines a worksheet file with the existing worksheet
Insert	Displace selected cells and insert new cells into the worksheet
Delete	Delete selected worksheet cells and move rows or columns
Find	Find a particular word or phrase in the worksheet
Find Next	Find the next occurrence of the word or phrase
Replace	Replace the word or phrase with alternate text
Format Cells	Sets the numeric format, alignment, and background color for the selected cells
Column Width	Sets column widths for selected cells
Row Height	Sets row height for selected cells
Transform	Applies a mathematical transform to columns

Sort	Sorts selected cells
Spatial Filter	Spatially filters X, Y and optional Z coordinates
Statistics	Computes statistics on selected cells
Text to Number	Convert the text in selected cells to numbers
Transpose	Convert columns to rows and rows to columns
Assign XYZ Columns	Specifies the default columns for X, Y, and Z coordinate data
Assign Coordinate System	Specifies the existing coordinate system
New Projected Coordinates	Specifies the columns containing the source X, Y coordinates and target X, Y columns for a new coordinate system
DMS to DD	Convert DMS latitudes and longitudes to decimal degrees
Track Cursor	Track cursor location across plot, worksheet, and grid editor windows for a maps, data files, and grids.

Clear - Worksheet

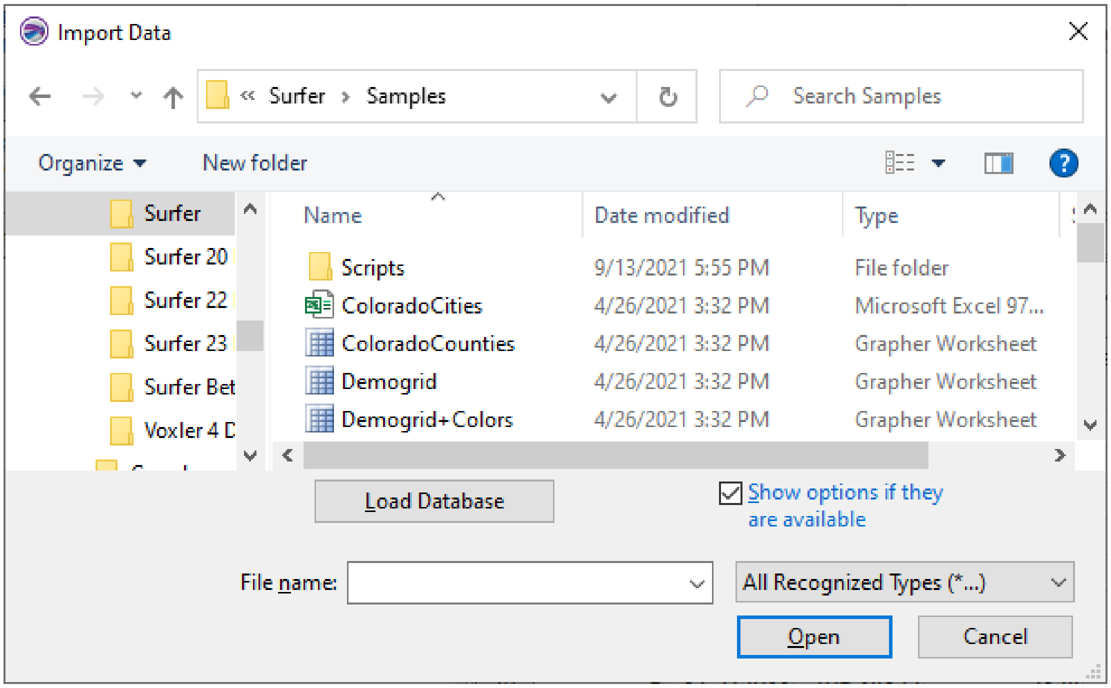
Click the **Data | Edit | Clear** command or the button, or press the DELETE key on the keyboard, to remove data from selected worksheet cells. The cells become empty when the data are removed. To shift the data from unselected cells into the selected cell locations, use the Delete command instead.

Merge - Worksheet Document

The **Data | Edit | Merge** command in the worksheet loads the contents of a data file into the existing worksheet. Select a file to import into the existing file in the **Import Data** dialog. The contents of the new file are merged into the worksheet at the active cell so be sure to position the cell at the edge of the existing data. Any cells in the existing worksheet that lie to the right of and below the active cell are overwritten with the contents of the merging file.

Import Data Dialog

Click the **Data | Edit | Merge** command or the button to open the **Import Data** dialog.



*Specify files to import into the **Surfer** worksheet using the **Import Data** dialog.*

Look In

The *Look in* field shows the current directory. Click the down arrow to see the directory structure and click on the folders to change directories.

<u>Creating New Folders and Changing the View</u>

The buttons to the right of the *Look in* field allow you to create new folders and change the view of the file list.

File List

The *File list* displays files in the current directory. The current directory is listed in the *Look in* field. The *Files of type* field controls the display of the file list. For example, if *DAT Data (*.dat)* is listed in the *Files of type* field only *.DAT files appear in the files list.

Specify a File Name

The *File name* field shows the name of the selected file. Alternatively, type a path and file name into the box to open a file.

Files of Type

The *Files of type* field controls the display of the file list. For example, if *DAT Data (*.dat)* is listed in the *Files of type* field only *.DAT files appear in the files list.

The *All Recognized Types (*...)* format type is selected by default. This displays all the common file formats in the navigation pane. If a different format type is selected, **Surfer** will remember the setting until the end of the current session. When **Surfer** is restarted, the default format type will be used.

To see all files in the directory, choose *All Files (*.*)* from the *Files of type* list. Double-click on a file to open it or single-click the file and then click the *Open* button. The *All Files (*.*)* option shows all of the file formats in the current directory, even if the file type is not appropriate for the action chosen. For example, a .GRD file may be displayed, even though a .GRD file cannot be imported into the worksheet.

Select a file type from the *Files of type* list. The following formats are supported:

- ACCDB Microsoft Access .ACCDB
- BLN Golden Software Blanking .BLN
- BNA Atlas Boundary .BNA
- CSV Comma Separated Variables .CSV
- DAT Data .DAT
- DBF Database .DBF
- DXF AutoCAD Drawing Data .DXF
- LAS LiDAR Data .LAS
- MDB Microsoft Access 97-2003 Database .MDB
- SEG-P1 Data Exchange Format .SEG
- P1 Data Exchange Format .SP1
- SLK Sylk Spreadsheet .SLK
- TXT Text Data .TXT
- XLS Excel Spreadsheet .XLS
- XLSX Excel 2007 Spreadsheet .XLSX
- XLSM Excel 2007 Spreadsheet .XLSM

Load Database

Click the *Load Database* button in the **Import Data** dialog to open the Data Link Properties dialog and import a database.

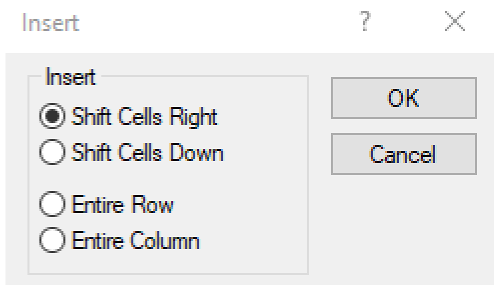
Show Options If They Are Available

If *Show options if they are available* is checked, then opening .TXT files or ASCII text files with unsupported file extensions will bring up the Data Import Options dialog where you can specify the import options. The **Data Import Options** dialog can be used to import data from different locales correctly, for example when the data uses ambiguous or different date order or when the data uses different decimal and column delimiters than the local PC.

Insert - Worksheet

The **Data | Edit | Insert** command inserts a single blank cell or a block of blank cells in the worksheet. Select cells in the area in which you wish to insert cells and then click **Data | Edit | Insert**, or right-click and select *Insert* from the context menu. The **Insert** dialog appears. Specify how you want the original displaced contents moved when the blank cells are inserted.

Click the **Data | Edit | Insert** command or the button, or press CTRL+R on the keyboard, to open the **Insert** dialog.



*When using **Insert**, you can shift cells to the right or down to make room for the new cells.*

Shift Cells Right or Shift Cells Down

Click the *Shift Cells Down* or *Shift Cells Right* option to insert blank cells and displace the original contents either down or to the right.

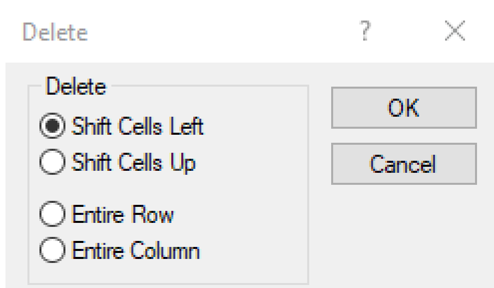
Entire Row or Entire Column

Click the *Entire Row* or *Entire Column* option to insert an entire row or column in the area that contains highlighted cells.

Delete - Worksheet

The **Data | Edit | Delete** command deletes the selected worksheet cells and shifts cells up or to the left to fill in the gap. After selecting **Data | Edit | Delete**, the **Delete** dialog appears. Specify the desired behavior of the cells and click the *OK* button. The selected cells are deleted and the contents of cells below or to the right are moved to fill the deleted block.

Click the **Data | Edit | Delete** command or the button, or press CTRL+D on the keyboard, to open the **Delete** dialog.



*When using **Data | Edit | Delete**, you can shift cells to the left or up to fill in the gap.*

Shift Cells Left or Shift Cells Up

Click the *Shift Cells Up* or *Shift Cells Left* option to specify if cells will be shifted to the left or up to fill in the gap after deleting the selected cells.

Delete Entire Row or Entire Column

Click *Entire Row* or *Entire Column* to delete the entire row or column that contains highlighted cells.

Leave Deleted Cells Empty

To leave the selected cells empty when the data are removed, use the Clear command, press the DEL key, or use the Cut command.

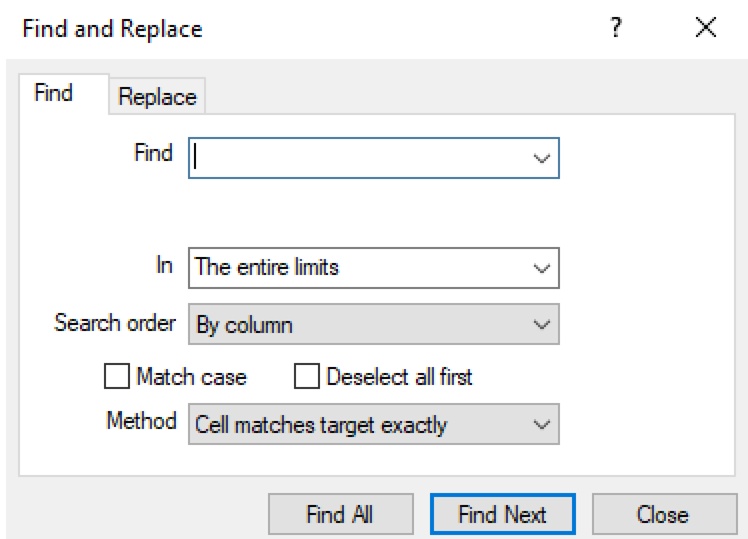
Find

The **Find and Replace** dialog displays when the **Data | Find | Find** or **Data | Find | Replace** commands are selected. The **Find and Replace** dialog is used to locate and replace specific numbers or text in the worksheet.

The **Data | Find | Find Next** command is used to find the next instance of a particular number, word, or phrase in the worksheet. If the **Data | Find | Find** command was not used initially, the **Find and Replace** dialog opens so that you can define your search criteria.

Find Page

Clicking the **Data | Find | Find** command or the  button, pressing CTRL+F on the keyboard, or clicking the F3 key on the keyboard opens the **Find** page of the **Find and Replace** dialog.



*Search for and replace specific text in the worksheet with the **Find and Replace** dialog.*

Find

To find objects, type the text you want to search for in the *Find* field. Click the arrow at the right to select from a list of the most recently used text strings. The asterisk (*) and question mark (?) wildcards can be used in the *Find* box. Click the arrow at the right to select from a list of the most recently used criteria.

- A question mark ? finds a single character in the specified location. For example, 200? finds 2009, 2008, 200a, etc.
- An asterisk * finds any number of characters at the specified location. For example, *01 finds 601, 1201, cd01, etc.

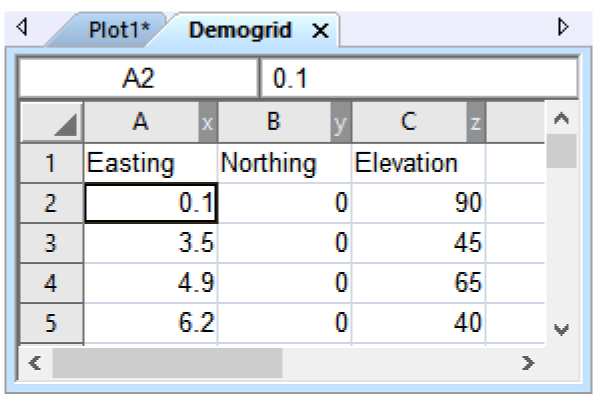
In

Next to *In*, choose the parameters of the search from the list. Choices include *The column where active cell is*, *The row where active cell is*, and *The entire limits*.

- Select *The column where the active cell is* to search only the column (e.g. column B) of the active cell (e.g. cell B2) for the information listed in the *Find* field.
- Select *The row where active cell is* to search only the row (e.g. row _2) of the active cell (e.g. cell B2) for the information listed in the *Find* field.
- Select *The entire limits* to search the entire worksheet for the information listed in the *Find* field.

Search Order

The *Search order* controls the direction of the search: down through columns by selecting *By column* or to the right across rows by selecting *By row*.



In this example, cell A2 is selected. If the Find criteria is "9", and By column is the search order, cell A4 is found first. If By row is the search order, cell C2 is found first.

Match Case

If you have case sensitive characters in the *Find* text string, check the *Match case* check box. Selecting *Match case* distinguishes between uppercase and lowercase characters. For example, a search for "Elevation" with the *Match case* option selected will not find entries for "elevation", but will find entries for "Elevation".

Deselect All First

Check the *Deselect all first* box to deselect all selected cells before performing the search. All previously selected cells will be deselected prior to the search when the *Deselect all first* check box is checked. If the *Deselect all first* box is cleared, the results of a previous search will remain highlighted when performing the next search.

Method

Choose the search *Method* from the list to determine how the search is performed. The examples assume "Golden, CO" is in the *Find* field.

- Select *Cell matches target exactly* to require that the exact criteria in the *Search* box is present in a cell before it is selected. For example, only cells that have exactly "Golden, CO" will be selected.
- Select *Cell contains target phrase* to require that the phrase in the *Search* box is present in a cell before it is selected. For example, cells that has "Golden CO", "Golden Company", or "Golden Colorado" will be selected.
- Select *Cell contains all of the target words* to require that all of the *Search* criteria words are present in a cell before it is selected. For example, cells that have "Golden" and "CO" somewhere in the cell (i.e. "Golden is the best city in Colorado" will be selected).
- Select *Cell contains any of the target words* to require that any of the *Search* criteria words are present in a cell before it is selected. For example, cells that have "Golden is a city" or "CO is a state" will be selected.

Find All Button

Click the *Find All* button to find all occurrences of the *Find* criteria in the worksheet. All of the cells that contain the *Find* criteria will be highlighted.

Find Next Button

Click the *Find Next* button to find the next occurrence of the characters specified in the *Find* box. This allows you to meet the criteria one at a time. The next instance of the *Find* criteria will be highlighted.

Close Button

Click the *Close* button to exit the **Find and Replace** dialog.

Find Next

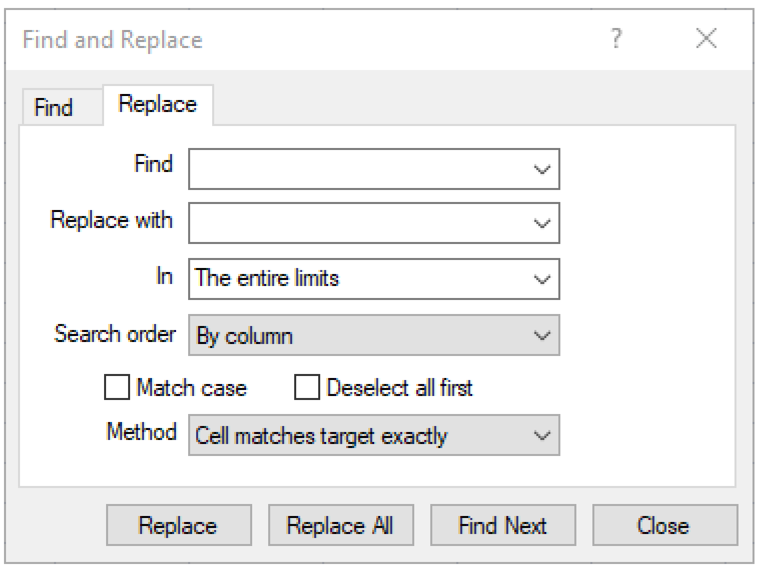
Clicking the **Data | Find | Find Next** command or the button or pressing the F3 key on the keyboard finds the next instance of a particular number, word, or phrase in the worksheet. Each cell matching the search parameters remains selected. If the **Data | Find | Find** command was not used initially, the [Find and Replace dialog] opens so that you can define your search criteria.

Replace

The **Find and Replace** dialog displays when the **Data | Find | Find** or **Data | Find | Replace** commands are selected. The **Find and Replace** dialog is used to search and replace specific numbers or text in the worksheet.

Replace Page

Clicking the **Data | Find | Replace** command or the button or pressing CTRL+H on the keyboard opens the **Replace** page of the **Find and Replace** dialog. The **Replace** page has all of the **Find** page fields, with the addition of the *Replace with* field. The **Replace** page *Method* field has only two options.



*Use the **Find and Replace** dialog to replace numbers or text in the worksheet.*

Replace With

Type the text you want to replace in the *Find* box. To delete the characters in the *Find* box from your worksheet, leave the *Replace with* box blank. Click the arrow at the right to select from a list of the most recently searched items.

Method

Choose the search *Method* from the list to determine how the search is performed. The examples assume "Golden, CO" is in the *Find* field.

- Select *Cell matches target exactly* to require that the exact criteria in the *Search* box is present in a cell before it is selected. For example, only cells that have exactly "Golden, CO" will be selected.
- Select *Cell contains target phrase* to require that the phrase in the *Search* box is present in a cell before it is selected. For example, cells that has "Golden CO", "Golden Company", or "Golden Colorado" will be selected.

Replace Button

Click the *Replace* button to replace the selected occurrence of the criteria in the *Find* box with the criteria in the *Replace with* box, find the next occurrence of the criteria in the *Find* box, and then stop. If you want to automatically replace all occurrences of the search criteria in the worksheet, click the *Replace All* button.

Replace All Button

Click the *Replace All* button to replace all occurrences of the *Find* criteria in your document with the *Replace with* criteria. If you want to review and selectively replace each occurrence, click the *Replace* button.

Format Cells

Cell numbers, alignment, or background color can be formatted through the **Format Cells** dialog. To format a cell, select the cells to be formatted (see Selecting Worksheet Cells), and click the **Data | Format | Format Cells** command or the button. The **Format Cells** dialog opens.

The **Format Cells** dialog has three pages: **Number**, **Alignment**, and **Background**.

Number Page

Use the Number page to change the numeric data display in the worksheet.

Alignment Page

Use the Alignment page to set the cell alignment.

Background Page

Use the Background page to set cell background color.

Text String

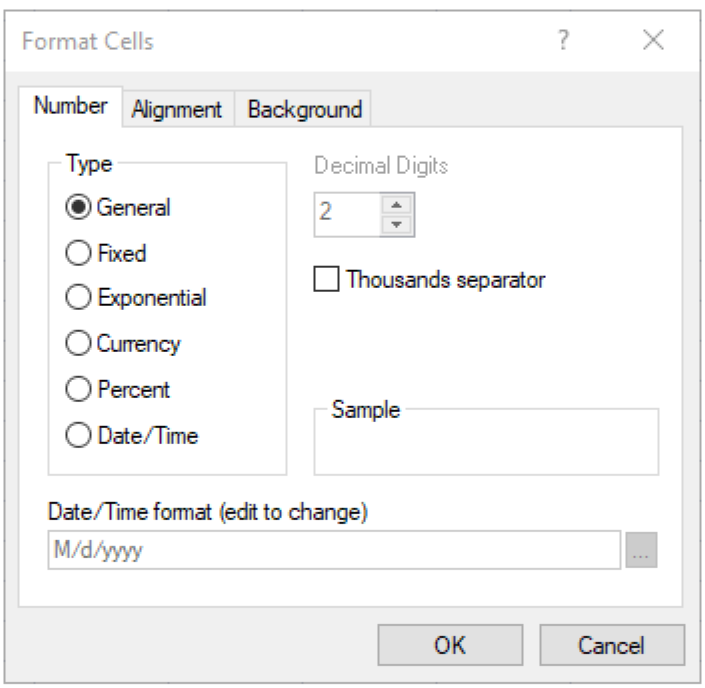
Number formatting has no effect on a numeric text string (numbers entered as text). A number with an apostrophe in front of it ('8123) is a text string. The apostrophe only shows in the active cell edit box. For example, an ASCII data file might contain the digits "8123" (digits surrounded by quotes), '8123 (digits preceded with an apostrophe), numbers with letters, or numbers with symbols (i.e. a blackslash "\"). These "numbers" are read as text and not as a number. The Transform command can be used to perform a mathematical function, such as ATOI(X), to convert a text string to an integer value. Alternatively, the cell can be converted to numbers with the Text to Number command.

Preserving Cell Format

The only formats that preserve cell-formatting information are Excel or SYLK SLK. ASCII file formats (.CSV, .TXT, .DAT, .BNA, .BLN) do not preserve file format information.

Format Cells - Number

Cell numbers, alignment, or background color can be formatted through the **Format Cells** dialog. To format a cell, select the cells to be formatted (see Selecting Worksheet Cells), and then click the **Data | Format | Format Cells** command. Use the **Number** page to change the numeric data display in the worksheet. Number formatting has no effect on a numeric text string (numbers entered as text). For example, an ASCII data file might contain the numbers "8123" (numbers surrounded by quotes) which are read as text and not as a number. The Transform command can be used to perform a mathematical function, such as ATOI(X), to convert a text string to an integer value. Alternatively, the cell can be converted to numbers with the Text to Number command.



*Select the number format options on the **Number** page of the **Format Cells** dialog.*

Type

The *Type* section contains the numeric format for the selected cells. Available options are *General, Fixed, Exponential, Currency, Percent,* and *Date/Time*. Click on the desired option.

- *General* displays numbers as fixed or exponential, whichever is shorter.
- *Fixed* displays numbers as d.ddd. The number to the left of the decimal can vary. Set the number to the right of the decimal in theDecimal Digitsbox.
- *Exponential* displays numbers as d.ddde+dd. Set the number of digits to the right of the decimal in the *Decimal Digits* box.
- *Currency* displays fixed numbers with a currency symbol such as the dollar sign ($).
- *Percent* displays numeric values (such as 0.13) as percentages with a percent symbol suffix (13%).
- *Date/Time* formats the cells as date and/or time. Select *Date/Time* and then type or select the *Date/Time format*.

Decimal Digits

The *Decimal Digits* controls the number of digits to the right of the decimal when the *Type* is set to *Fixed, Exponential, Currency,* or *Percent*. To change the *Decimal Digits*, highlight the existing value and type a new value. Alternatively, click the to increase or decrease the value.

Thousands Separator

The *Thousands separator* option controls whether a comma appears in the number, indicating thousands. When checked, a comma appears every three digits to the left of the decimal point. When unchecked, the number appears without the comma. Do not type a comma when entering data as this causes the number to be read as text.

Sample

The *Sample* box displays the current number format.

Date/Time Format

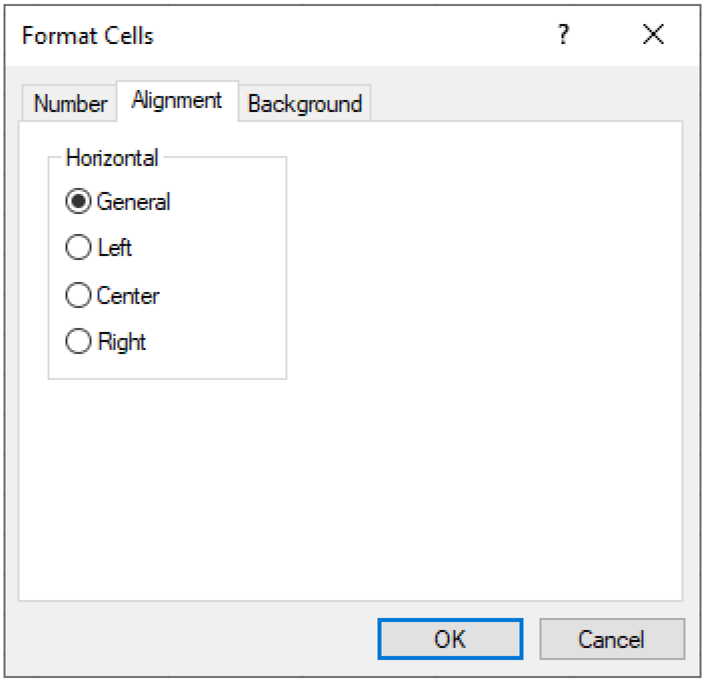
When the *Type* is set to *Date/Time*, the *Date/Time format* option becomes available. Type the desired format into the *Date/Time format* field, or click the button to insert a date/time format with the Date/Time Format Builder dialog. Available formats are made of combinations of day, month, year, AD/BC or CE/BCE designation, hours, minutes, seconds, and AM/PM designation. Days are shown as d or dd. Months are shown as M, MM, MMM, MMMM, or MMMMM. Years are shown as yy or yyyy. Hours are shown as h or hh. Minutes are shown as m or mm. Seconds are shown as ss. Sub-seconds are displayed as ss.0 or ss.000. AM/PM designation are shown as tt or TT. BC/AD designation is shown as gg or GG. BCE/CE designation is shown as g, ggg, G, or GGG. Total elapsed time is shown as [h]. After clicking on the *Date/Time format*, the *Sample* updates to show a value in the selected format.

OK or Cancel

Click *OK* to make the change to the cell format. Click *Cancel* to return to the worksheet without making the change.

Format Cells - Alignment

Cell numbers, alignment, or background color can be formatted through the **Format Cells** dialog. To format a cell, select the cells to be formatted (see Selecting Worksheet Cells), and then click the **Data | Format | Format Cells** command. Use the **Alignment** page to align the cell in one of four ways. By default, imported ASCII files automatically align numbers to the right and text to the left.



*Select the alignment options on the **Alignment** page of the **Format Cells** dialog.*

General

General aligns text on the left side of the cell and numbers on the right side of the cell.

Left

Left aligns text and numbers with the left side of the cell.

Center

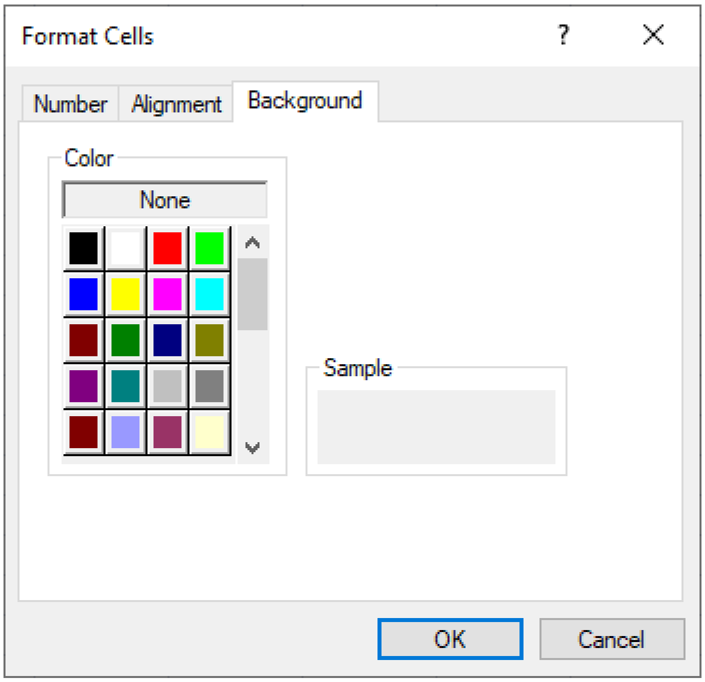
Center aligns text and numbers with the center of the cell.

Right

Right aligns text and numbers with the right side of the cell.

Format Cells - Background

Cell numbers, alignment, or background color can be formatted through the **Format Cells** dialog. To format a cell, select the cells to be formatted (see Selecting Worksheet Cells), and then click the **Data | Format | Format Cells** command. You can set cell background color on the **Background** page. Save the worksheet in Excel format to save background color in the file.



Select the cell background color the ***Background*** *page of the* ***Format Cells*** *dialog.*

None

Click the *None* button to remove any previously assigned background colors.

Color Palette

Select a cell background color from the color palette.

Sample

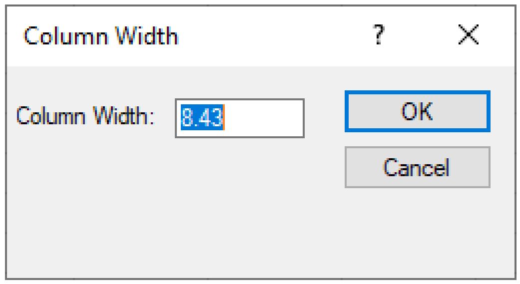
A sample of the color is displayed in the *Sample* box.

Column Width

You can change the column width of selected cells by clicking the **Data | Format | Column Width** command or by the  or by using the mouse to resize the column. The Excel XLS or SYLK SLK file format must be used to save the column width in the file since ASCII file formats (.CSV, .TXT, .DAT, .BNA, .BLN) do not preserve file format information.

Column Width Dialog

To set column widths or to hide columns, select either the entire column or individual cells within the columns (see Selecting Worksheet Cells), and then click the **Data | Format | Column Width** command. Enter the width for the selected column or cells into the **Column Width** dialog. Columns can range from zero to 512 characters wide. The value zero (0) hides the column.



*Change the column width by selecting columns, clicking **Column Width**, and then entering a number into the **Column Width** dialog.*

Changing Column Widths with the Mouse

Column width can also be changed using the mouse. When the cursor is moved to the line that defines the right boundary of the column header, the cursor changes to a ✛. Press and hold the left mouse button and move the cursor to the left or right to change the width of the column. You can double-click the column line to automatically set the column width. When automatically setting the column width, the column narrows or widens to the smallest size necessary to completely display the data and column name.

Hide a Column with the Mouse

You can hide a column by moving the cursor to the left until the next dividing line is reached.

Display Hidden Columns with the Mouse

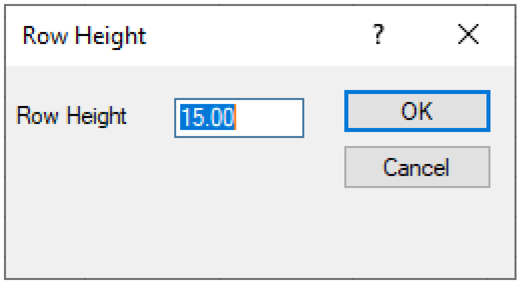
To display hidden columns, press and hold the left mouse button at the right edge of the hidden column and move the cursor to the right to widen the column.

Row Height

You can change the **Row Height** of selected cells by clicking the **Data | Format | Row Height** command or the button or by using the mouse to size the row. The Excel XLS or SYLK SLK file format must be used to save the row height and numeric format information with the file since ASCII file formats (.CSV, .TXT, .DAT, .BNA, .BLN) do not preserve file format information.

Row Height Dialog

To set the row height or hide rows, select either the entire row or individual cells within the rows (see Selecting Worksheet Cells), and click the **Data | Format | Row Height** command. Enter the height for the selected row or cells in the **Row Height** dialog. Rows can range from zero to 512 points in height. A value of zero (0) hides the row.



*Change the row height by selecting rows, clicking **Data | Format | Row Height** , and then entering a number into the **Row Height** dialog.*

Changing Row Heights with the Mouse

Row height can also be changed using the mouse. When the cursor is moved to the line that defines the lower boundary of the row header, the cursor changes to a line with two arrows. Press and hold the left mouse button and move the cursor up or down to change the height of the row.

Hide a Row with the Mouse

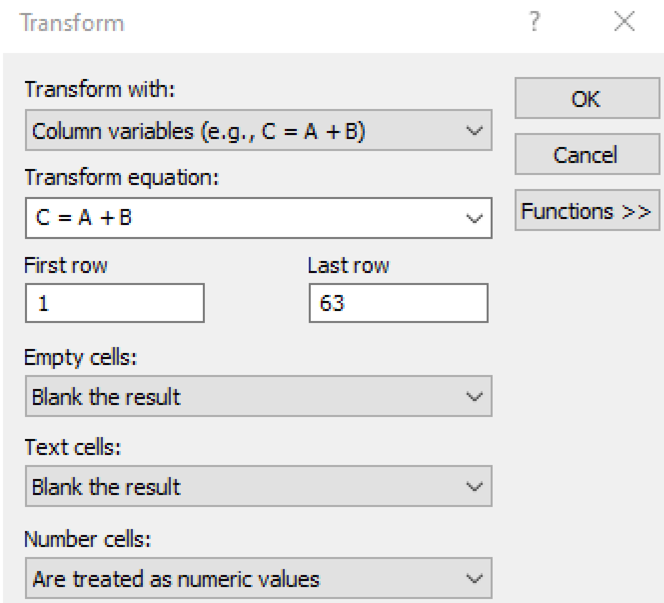
You can hide a row by moving the cursor up until the next dividing line is reached.

Display Hidden Row with the Mouse

To display hidden rows, press and hold the left mouse button at the bottom of the hidden row and move the cursor down to stretch the row height.

Transform - Worksheet

Click the **Data | Data | Transform** command or the *fx* button to open the **Transform** dialog, where you can apply mathematical transformations to the columns, rows, or cells. Valid math operators include addition (+), subtraction (-), multiplication (*), and division (/) as well as a large library of built-in mathematical functions. Parentheses should be used to override precedence or for clarification.



*Use the **Transform** dialog to apply math functions to data. The dialog options update to reflect the option selected for Transform with field.*

Transform With

Select the type of transform from the *Transform with* list. *Column variables (e.g., C = A + B)* applies the transform equation to the specified rows in the *Transform equation* column. *Row variables (i.e., _3 = _1 + _2)* applies the transform equation to the specified columns in the *Transform equation* row. *Cell variables (i.e., C3 = A1 + B2)* applies the transform equation only to the cell specified in the *Transform equation*.

Transform Equation

Type the formula into the *Transform equation* box. Formulas consist of a destination column, row, or cell on the left side of the equation and a mathematical manipulation on the right side of the equation. Use the column label letters, row numbers, or cell locations on both sides of the equation. Click the down arrow to use previously entered equations. For columns, a sample equation may be C = A + B. For rows, a sample equation is _4=_1+_2. For cells, a sample equation would look like C2=A1+B1-C1.

If the transform method is by column, the range functions (sum, avg, std, rowmin and rowmax) take column indices only, i.e., sum(A...C). If transform method is by variable rows, the range functions take row indices only, i.e., sum (_1..._3). If transform method is by variable cells, the range functions are not supported.

The last ten functions are stored in the *Transform equation* field. After ten functions are included in the list, the oldest function is replaced when a new function is added. The *Transform equations* are stored between sessions. To use a stored function, click the in the *Transform equation* box and select a function from the list. Note that the *First row* and *Last row* or *First col* and *Last col* values are not saved with the stored *Transform equations*.

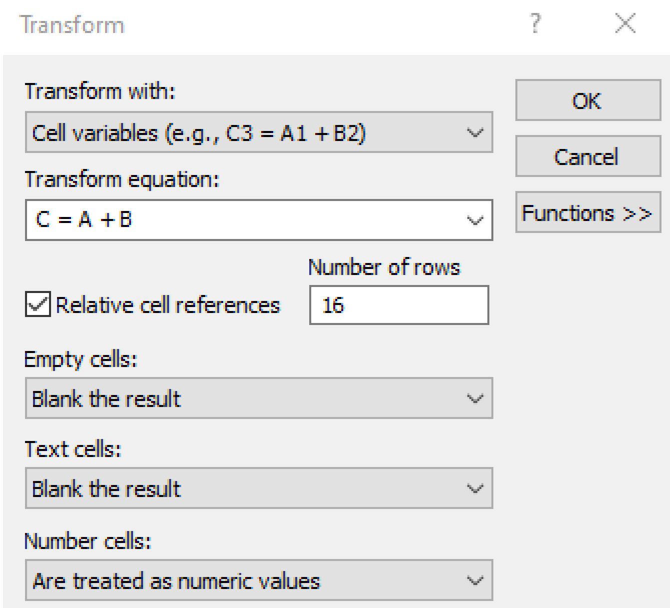
First and Last Columns and Rows

When calculating transformations on columns, enter the *First row* and the *Last row* to limit the calculation to the specified rows. When calculating transformations on rows, enter the *First col* and *Last col* to limit the calculation to the specified columns. When calculating transformations on cells, the *First row, Last row, First col,* and *Last col* options are not available.

By default, these are set to the first row and last row (or first column and last column) with text or numbers entered into a cell for the entire worksheet.

Relative Cell References

Select the *Relative cell references* option to apply the *Transform equation* to multiple cells. The *Relative cell references* option is only available when *Transform with* is set to *Cell variables (e.g., C3 = A1 + B2)*. The *Transform equation* is only applied relatively by incrementing the row numbers. The columns do not change when *Relative cell references* is selected.



The Relative cell references *option applies the* Transform equation *to multiple rows.*

Specify the number of rows to which the *Transform equation* is applied in the *Number of rows* field. Note this value is not the row number. By default the *Number of rows* value is the total number of rows in the worksheet.

Empty Cells

The *Empty cells* option controls how empty cells are treated in the calculations of formulas. Available options are *Blank the result, Are treated as the number zero (0),* and *Are treated as empty text ("").* The default option is *Blank the result*, which results in the formula not being calculated for any row that contains a blank cell in any of transform equation rows or columns.

- Setting the *Empty cells* option to *Blank the result* results in a blank cell for the transform when the cells on the right side of the equation are empty.
- Setting the *Empty cells* option to *Are treated as the number zero (0)* results in the transform creating a number when all of the cells on the right side of the equation are empty or numeric. When the right side of the equation combines text and blank cells, the equation is blank.
- Setting the *Empty cells* option to *Are treated as empty text ("")* results in the transform creating a text string when all of the cells on the right side of the equation are empty or text. When the right side of the equation combines numeric and blank cells, the equation is blank.

Text Cells

The *Text cells* option controls how text cells are treated in the calculations of formulas. Available options are *Blank the result, Are treated as text, Are converted to numbers (if possible),* and *Are treated as the number zero (0).* The default option is *Blank the result*, which results in the formula not being calculated for any row that contains a text cell in any of transform equation rows or columns.

- Setting the *Text cells* option to *Blank the result* results in a blank cell for the transform when any of the cells on the right side of the equation contain text strings (including numbers formatted as text).
- Setting the *Text cells* option to *Are treated as text* results in the transform creating a text string when all of the cells on the right side of the equation are text (or treated as text). If a mix of text cells and numbers or empty cells (that are not treated as text) are in the cells on the right side of the equation, the transform results in a blank cell. This option allows text strings to be concatenated.
- Setting the *Text cells* option to *Are converted to numbers (if possible)* results in the transform creating a number when all of the cells on the right side of the equation are numeric or treated as numbers. Any cells with numbers formatted as text are treated as the number. For example, the text string '05 would be treated as the number 5 if this option is selected.
- Setting the *Text cells* option to *Are treated as the number zero (0)* results in the transform creating a number when all of the cells on the right side of the equation are numeric or treated as numbers. Any cells with text are replaced with the value zero for the transform. For example, if you are using the

equation C=A+B and A has *Colorado* and B has *45*, the value in cell C will be 45.

Number Cells

The *Number cells* option controls how numeric cells are treated in the calculations of formulas. Available options are *Blank the result, Are treated as numeric values, Are converted to text,* and *Are treated as empty text ("").* The default option is *Are treated as numeric values*, which results in the formula being calculated for any row that contains numbers in any of transform equation rows or columns.

- Setting the *Number cells* option to *Blank the result* results in a blank cell for the transform when any of the cells on the right side of the equation contain numbers. This option is useful when you only want to combine text cells or blank cells.
- Setting the *Number cells* option to *Are treated as numeric values* results in the transform creating a number when all of the cells on the right side of the equation are number (or treated as numbers). If a mix of text cells and numbers or empty cells (that are not treated as numbers) are in the cells on the right side of the equation, the transform results in a blank cell.
- Setting the *Number cells* option to *Are converted to text* results in the transform creating a text string when all of the cells on the right side of the equation are text or treated as text. Any cells with numbers are treated as the text string of the number. For example, number 5 is in the cell, so the text string would appear as '5 if this option is selected.
- Setting the *Number cells* option to *Are treated as empty text ("")* results in the transform creating a text string when all of the cells on the right side of the equation are text or treated as text. Any cells with numbers are replaced with "" for the transform. For example, if you are using the equation C=A+B and A has *Colorado* and B has *45*, the value in cell C will be *Colorado*.

Combining Text, Numbers, and Empty Cells

Many possible combinations of the *Empty cells, Text cells,* and *Number cells* exist to allow combining these different types of cells in a *Transform equation* . If the transform result is not what you expect, check the settings for these options and adjust if necessary.

Functions

Click the *Functions >>* button to display a list of predefined mathematical functions. Click the *Functions <<* button again to hide the list of predefined mathematical functions.

To use a function, place the cursor in the location to add a function, select a function from the list, click the *Insert* button, and then replace the X in the function with a column letter (A); underscore and row number (_1); or cell location (A1). Also, be sure to use proper mathematical operators (+_*/) between the function and the rest of the equation. The definition of the function is listed below the *Function name* list when a function is selected.

The IF function is not included in the *Functions* list. See the *Mathematical Functions* topic for the syntax of the IF function.

Insert

When the *Functions* are expanded, the *Insert* button is visible. Click the *Insert* button to add a function to the *Transform equation* box. In the *Transform equation* box, manually change the variable (i.e. X or Y) in the listed functions to a column letter, row number (_1), or cell location.

Examples

An example of a column formula is C = A + B. Columns A and B are added and inserted into column C with this equation. The formula adds the contents of A and B in each row and places the results in column C for that row.

An example of a row formula is _4=_1+_2. Rows 1 and 2 are added and inserted into row 4 with this equation. The formula adds the contents of the 1 and 2 in each column listed between the *First col* and *Last col* values and places the results in row 4 for that column.

An example of a cell formula is C2=A1+B1-C1. The value in C1 is subtracted from the sum of the values in cells A1 and B1. The result is inserted into cell C2 with this equation.

Example Functions

This example shows how to use the built in functions. Consider, for example, taking the cosine of data in column C. Column D is the first empty column, so we will use column D as the destination column.

1. Click the **Data | Data | Transform** command to open the **Transform** dialog. You do not need to highlight any columns before selecting **Transform**.
2. In the *Transform equation* box, type "D = " without the quotes.
3. Click the *Functions* button.
4. Double-click on the function name COS(X) in the *Function name* group. Alternatively, you could select a *Function name* and click the *Insert* button.
5. COS(X) is automatically placed in the equation as "D = COS(X)" without the quotes.
6. Replace the X in the function with the column letter containing the data to be transformed (column C). The equation will be "D = COS(C)" without the quotes.
7. Change the *First row* and *Last row* if you wish.
8. Make sure that *Empty cells* and *Text cells* are set to *Blank the result* to only calculate values with numbers.
9. Click *OK* to create a new data column with column C's data transformed with the cosine.

Sort - Worksheet

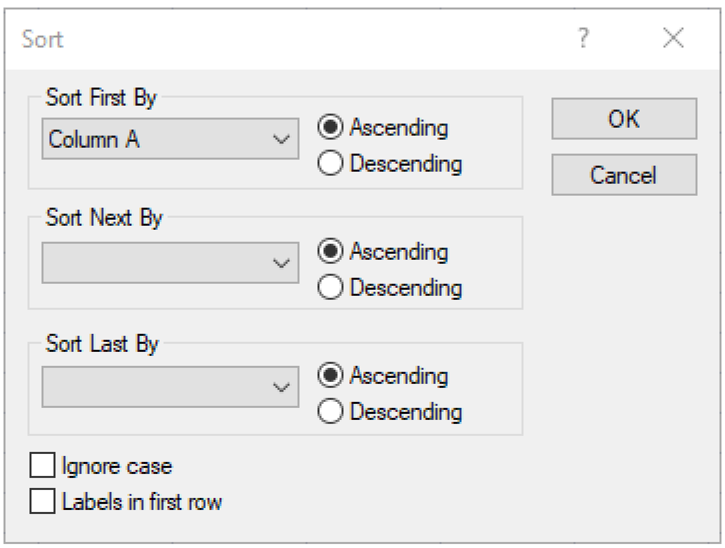
The **Data | Data | Sort** command arranges data according to rank in user-specified sort columns. Sorting rank is based on numbers, ASCII characters, and punctuation. Sort numeric data, text, or mixed columns. Sorting specifications are made in the **Sort** dialog.

Selecting Cells to Sort

Sorting is performed only on the selected columns (see Selecting Cells). If only one column is selected, only that column is sorted. To keep records (rows of data) together, select all columns containing data even if only one column is sorted.

Sort Dialog

Click the **Data | Data | Sort** command or the button in the worksheet to open the **Sort** dialog.



*Click the **Data | Data | Sort** command to sort data on multiple columns.*

Sort Order

The *Sort First By* option defines the primary column on which the rows are sorted. The positions of the sorted rows are determined by the *Ascending* or *Descending* rank in the *Sort First By* column.

Secondary Sort

When two or more rows have identical entries in the *Sort First By* column, the *Sort Next By* column can further organize the data set. Duplicates in the *Sort First By Column* are then sorted according to the rank in the *Sort Next By* column.

Final Sort

The *Sort Last By* column can be used when the *Sort Next By* column contains duplicates.

Ascending or Descending Sort

The sort order in an *Ascending* sort is based on the ASCII table. Numeric values are placed first, followed in order by cells starting with a space character, common punctuation, numeric text (numbers entered as text), uppercase letters, less common punctuation, lower case letters, uncommon punctuation, and blank cells. *Descending* order is the opposite of ascending order although blank cells are still listed last.

0	1	2	3	4	5	6	7	8	9
space	!	"	#	$	%	&	'	(	)
*	+	,	-	.	/	"0"	"1"	"2"	"3"
"4"	"5"	"6"	"7"	"8"	"9"	:	;	<	=
>	?	@	A	B	C	D	E	F	G
H	I	J	K	L	M	N	O	P	Q
R	S	T	U	V	W	X	Y	Z	[
\	]	^	_	`	a	b	c	d	e
f	g	h	i	j	k	l	m	n	o
p	q	r	s	t	u	v	w	x	y
z	{	|	}	~	blank				

This ASCII table shows the sort order in the worksheet.

Ignore Case

Because sorting is based on an ASCII table, upper and lowercase letters are treated differently. For example, "A" is sorted separately from "a." If the letters are to be treated as the same during the sort, check the *Ignore case* option. When this check box is activated, "A" is considered identical to "a" in the sorting rank.

Labels in First Row

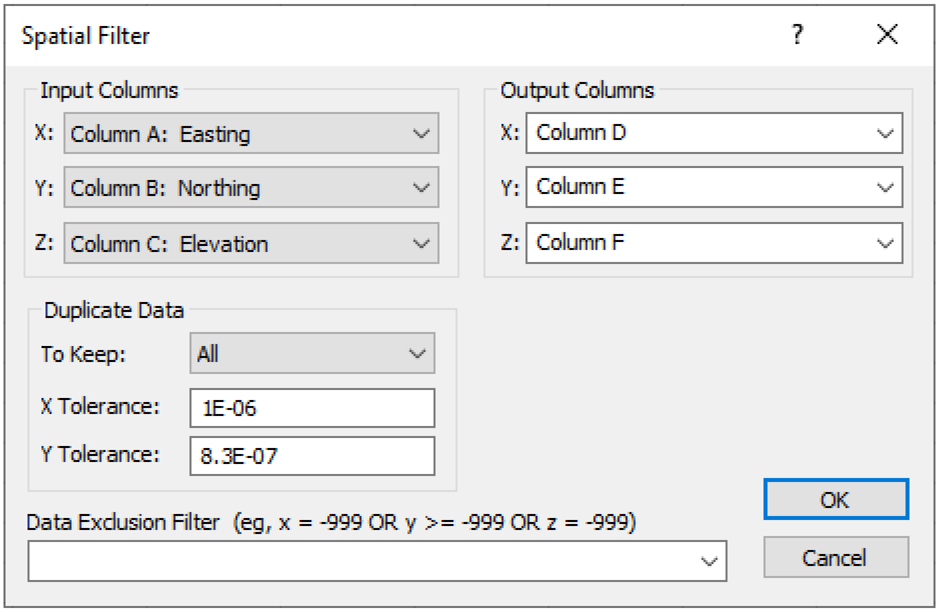
The data set may contain text identifying the data in the column (header information) in Row 1. In this case, click the *Labels in first row* option to exclude the label row from the sort process.

Spatial Filter

The **Data | Data | Spatial Filter** command can be used to spatially filter the X, Y and Z coordinates. This can be useful when pre-filtering data before creating a post map, variogram, or grid file and saving the data for use in other programs in the future. The spatial filter can be used to allow duplicate removal and arbitrary exclusion with a mathematical expression. Spatial filter specifications are made in the **Spatial Filter** dialog.

Spatial Filter Dialog

Click the **Data | Data | Spatial Filter** command or the button in the worksheet to open the **Spatial Filter** dialog.



Specify custom options to spatially filter data in the ***Spatial Filter*** *dialog.*

Input Columns

The *Input Columns* section indicates the original X, Y and Z columns. Select the *Input Columns* in the X, Y, and Z lists. Click the arrow button to see the list. The Z column is optional and can be set to *None* if the operation is for 2D (X, Y) coordinates only. If the Z *Input Columns* is set to *None*, the Z *Output Columns* is not available.

Output Columns

The *Output Columns* section indicates the location where the filtered results of the X, Y and Z columns are written. Select the *Output Columns* in the X, Y, and Z lists. Click the arrow button to see the list. If the Z *Input Columns* is set to *None* , the Z *Output Columns* is not available.

Filtered results are written to the original row and new worksheet columns as specified in the *Output Columns* group. New data are added to the bottom of the worksheet to ensure that the source and target points stay aligned on the same row.

Duplicate Data

The *Duplicate Data* section contains methods for defining and handling duplicate data points. Duplicate data are two or more data points having nearly identical X, Y coordinates (Z values may vary for these X, Y coordinates). Select the action for duplicate data in the *To Keep* list. Enter the *X Tolerance* and *Y Tolerance*. If a point is deleted (due to duplication or exclusion), the output cell is left blank.

To Keep

Duplicates are determined by moving from the lowest X value to the highest X value. A datum only belongs to one set of duplicates. *The To Keep* options specify which duplicate data points to keep and which to delete in each set of duplicate points. Specify *All, None, First, Last, Minimum X, Maximum X, Median X, Minimum Y, Maximum Y, Median Y, Minimum Z, Maximum Z, Median Z, Sum, Average, Midrange,* or *Random* from the *To Keep* list.

All	Do not delete any duplicates.
None	Eliminate all of the duplicates.
First	Keep the first point, as defined by the order in the data file, from each set of duplicates.
Last	Keep the last point, as defined by the order in the data file, from each set of duplicates.
Minimum X	Keep the point with the minimum X coordinate.
Maximum X	Keep the point with the maximum X coordinate.
Median X	Keep the point with the median X coordinate.
Minimum Y	Keep the point with the minimum Y coordinate.
Maximum Y	Keep the point with the maximum Y coordinate.
Median Y	Keep the point with the median Y coordinate.
Minimum Z	Keep the point with the minimum Z value.
Maximum Z	Keep the point with the maximum Z value.
Median Z	Keep the point with the median Z value.
Sum	Create an artificial datum at the centroid of the duplicate points with a Z value equal to the sum of the duplicate set's Z values.
Average	Create an artificial datum at the centroid of the duplicate points with a Z value equal to the average of the duplicate set's Z values.
Midrange	Create an artificial datum at the centroid of the duplicate points with a Z value equal to the midrange of the duplicate observations' Z values halfway between the minimum Z and the maximum Z.
Random	Keep a single randomly selected representative point.

X and Y Tolerance

In addition to the *To Keep* options there are *X Tolerance* and *Y Tolerance* options. For example, two points, A and B are duplicates if:

$$|X_A - X_B| < X\,Tolerance$$
and
$$|Y_A - Y_B| < Y\,Tolerance$$

Using this definition, it is possible for points A and B to be "duplicates," for point B and C to be "duplicates," but for A and C to not be "duplicates."

Data Exclusion Filter

The *Data Exclusion Filter* allows a Boolean expression to specify how to exclude data. The *Data Exclusion Filter* can be used with any column in the worksheet that contains numbers. Columns in the worksheet that contain text or columns that are empty will not be excluded by the *Data Exclusion Filter*.

To use one of the X, Y, or Z columns, use X, Y, or Z in the *Data Exclusion Filter* . To use another column from the worksheet, use _A, _B, _C, etc. The underscore is required when specifying a worksheet column.

For example:

X=-999 or Y=-999 or Z=-999	Excludes any data with a -999 value in either the X, Y, or Z columns.
X<10 or X>20 or Y<10 or Y>20	Excludes all data except for points in the range 10 to 20 for both the X and Y directions.
Z < 0.0	Excludes any triplet with Z value less than 0.0.
_A > 10	Excludes any row in the worksheet that contains a value greater than 10 in column A.
Z < 0 AND _D = -999	Excludes any triplet with Z value less than 0.0 and whose row in the worksheet contains a value in column D equal to -999.

Boolean expressions, used by Grids | New Grid | Function, Grids | Calculate | Math, Grid | Data, and Grid | Variogram, include:

- logical operators (AND, OR, XOR, NOT)
- comparison operators (=, <>, <, >, <=, >=)
- the IF function - for example IF(condition,

The words AND, OR, XOR, NOT, and IF are reserved keywords and may not be used as variable names.

To use a stored function, click the next to the current function. This will display the ten most recent functions used. The functions are stored in the registry, so the equations are stored between **Surfer** sessions. You can also start typing the function in the function box. If the function is in the ten function history, the entire function will auto-complete.

For example, consider the case of ignoring data outside of a grid. The original grid *X Maximum* is 50, but the grid *X Maximum* is reset to 40. To limit the search to data with X values less than 40, use the *Data Exclusion Filter* by entering X > 40 into the *Data Exclusion Filter* text box. This tells **Surfer** to exclude all data with X values greater than 40.

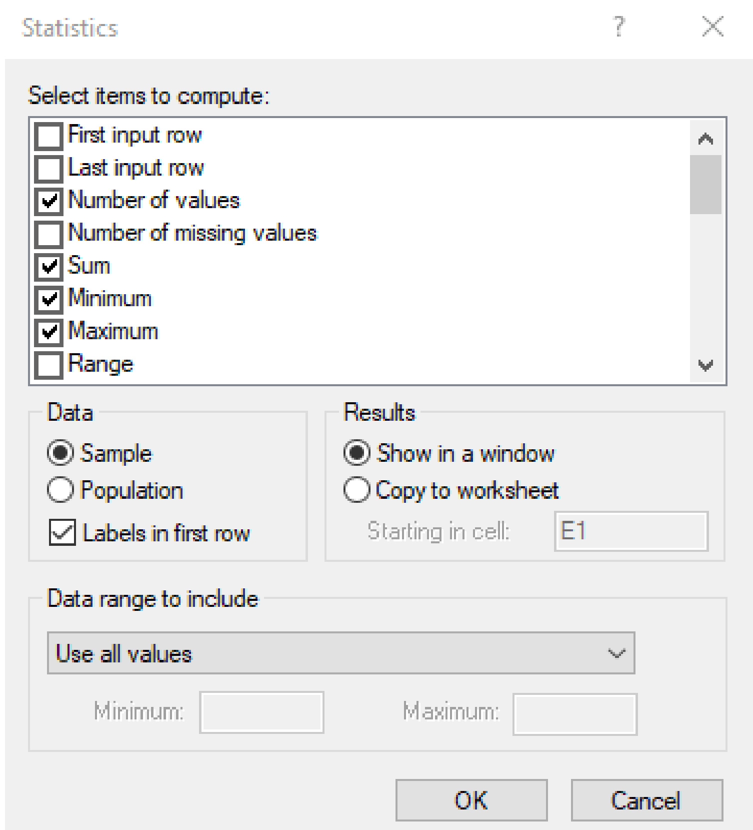
Consider a second case where data contains a numerical identifier in column D. When the value in this column is equal to -999, the data point is considered inaccurate and should not be used when gridding. To grid only those data where column D is not equal to -999, exclude column D with the *Data Exclusion Filter* by entering _D = -999 into the *Data Exclusion Filter* text box. This excludes all rows of data where column D contains the value -999.

Statistics - Worksheet

The **Data | Data | Statistics** command calculates statistical values for a group of selected numeric cells (see Selecting Cells). Select an entire column or a continuous group of cells in a column to use the **Statistics** command. If a rectangular block of rows and columns are selected, the **Statistics** command calculates the statistics for each column separately. A warning message appears if a group of cells cannot be used with the **Statistics** command. Non-numeric cell entries (empty cells or text) are ignored in statistics calculations.

Statistics Dialog

Use the **Data | Data | Statistics** command or the Σ button in the worksheet to open the **Statistics** dialog.



Click in the box adjacent to the statistics name to compute the statistics for the selected columns.

Select Items to Compute

Click in the check boxes next to the statistics options to calculate the statistics for the selected data:

- *First input row* reports the first row number in the selection. If the *Labels in first row* option is checked, the *First input row* is the second row in the selection.
- *Last input row* reports the last row number containing data in the column.
- *Number of values* indicates the number of numeric cells in the column.
- *Number of missing values* indicates the number of non-numeric cells in the selection. If columns are selected by clicking the column letters, the number of missing values includes blank values up to the last used row in the worksheet, which may be different than the last used row in the selected column. If cells are selected by highlighting specific cells, then only the blank cells within the selection are counted.
- *Sum* is the sum of all numeric cells in the column.
- *Minimum* indicates the minimum value in the column.
- *Maximum* indicates the maximum value in the column.
- *Range* indicates the range of the numeric values in the column (*Maximum* – *Minimum*).

- Mean is the arithmetic average of the data values. It is the sum of the data values divided by the number of data values.
- *Median* is the middle value among the data values. Half of the data values are larger than the median and half are smaller than the median. When there are an even number of data values the median is the average of the two middle values.
- *Mode* is the value that appears most often in a data set. If the data set contains multiple modes, the modes will be displayed separated by a comma. #N/A will be displayed if no mode exists for the data set, i.e. there is an equal number of each data value.
- *First quartile (25thpercentile)* is the value such that one-fourth of the data values are smaller than the quartile and three-fourths of the data values are larger than the first quartile.
- *Third quartile (75thpercentile)* is the value such that three-fourths of the data values are smaller than the quartile and one-fourth of the data values are larger than the third quartile.
- Standard error of the mean
- 95% confidence interval for the mean
- 99% confidence interval for the mean
- Variance
- Average deviation
- Standard deviation
- Coefficient of variation
- Coefficient of skewness
- Coefficient of kurtosis
- Kolmogorov-Smirnov goodness of fit for normal distribution
- Critical value of K-S statistic at 90% significance level
- Critical value of K-S statistic at 95% significance level
- Critical value of K-S statistic at 99% significance level

Data Group

The *Data* group is used to select *Sample* or *Population* statistics. Select *Sample* or *Population* statistics, depending on whether the data represent a statistical sample or the complete set of all possible members of a population.

The *Labels in first row* option is also specified in the *Data* group. Check the *Labels in first row* box if the first row of the selection contains descriptive labels. If this box is checked the label appears at the top of the statistics report for each column.

Results Group

The *Results* group is used to show the statistics report in a window or copy the results to a new location of the worksheet. Select *Show in a window* to write the statistics results to a Statistics Results dialog. The results in this dialog can be copied to the clipboard to paste to other locations.

Select *Copy to worksheet* to write the statics report to a new location in the worksheet. Use the *Starting in cell* box to specify the cell for the upper left corner of the statistics report. If the destination cells contain data, a warning is displayed that data will be overwritten. Click *OK* to overwrite the data, or click *Cancel* to set a new *Starting in cell* location.

Data Range to Include Group

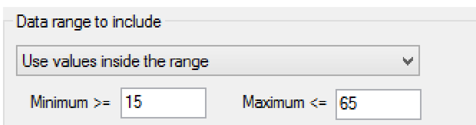
The *Data range to include* contains options to limit the values where the statistics are calculated. Available options are *Use all values*, *Use values inside the range*, *Use values outside the range*, and *Use all values except*.

Use all values

When the *Data range to include* is set to *Use all values*, all of the values in the highlighted section are used to calculate the statistics.

Use values inside the range

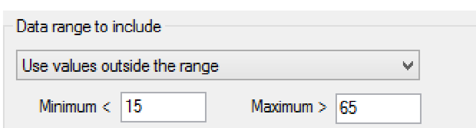
When the *Data range to include* is set to *Use values inside the range*, the *Minimum >=* and *Maximum <=* options are available. Type in the data values that bracket the range of values where the statistics should be calculated. For instance, if the *Minimum >=* is set to 15 and the *Maximum <=* is set to 65, only data points between (and including) 15 and 65 are used for calculating the statistics.



Only the values that are inside the range are included in the calculated statistics.

Use values outside the range

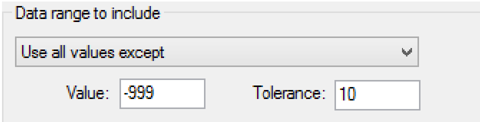
When the *Data range to include* is set to *Use values outside the range*, the *Minimum <* and *Maximum >* options are available. Type in the data values that bracket the range of values where the statistics should be calculated. For instance, if the *Minimum <* is set to 15 and the *Maximum >* is set to 65, only data points below 15 or greater than 65 (and excluding 15 and 65) are used for calculating the statistics.



Only the values that are outside the range are included in the calculated statistics.

Use all values except

When the *Data range to include* is set to *Use all values except*, the *Value* and *Tolerance* options are available. Type in the data value that should be excluded in the *Value* box. The *Tolerance* value gives a range on either side of the *Value*. Everything in the range *Value-Tolerance* to *Value+Tolerance* is excluded from the statistics calculation. For instance, if the *Value* is set to -999 and the *Tolerance* is set to 10, all values between -1009 and -989 are excluded from the statistics. This means that all values less than -1009 and greater than -989 are included in the statistics calculations.

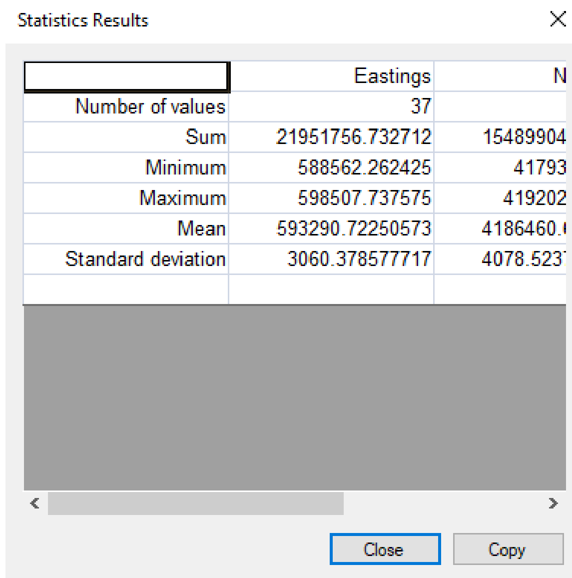


Only the values that are outside the range Value-Tolerance *to* Value+Tolerance *are included in the calculated statistics.*

The *Use all values except* option can be used to ignore NoData values. For example, the NoData value for the *Concentration.grd* sample grid file is the Surfer default NoData value, 1.70141E+38. Two methods exist for ignoring the NoData value. One method is to enter the *NoData Value* displayed in the Grid | Info report exactly into the *Value* field of the **Statistics** dialog. Using *Concentration.grd* as an example, 1.70141E+38 would be entered into the *Value* field. Another method is to enter an approximation of the NoData value and a tolerance that will include the actual NoData value while excluding the desired data. Using this method with the previous example, 1.70E+38 is entered into the *Value* field and 1E+37 is entered into the *Tolerance* field. Both of these methods result in the **Statistics** command returning results for the non-NoData values only.

Statistics Results Dialog

The **Statistics Results** dialog is displayed when the *Show in a window* option is selected in the *Results* section of the Statistics dialog.



*The **Statistics Results** dialog displays the results from the **Statistics** command.*

Click *Copy* to copy the results to the clipboard. The results can be pasted in another program. If you wish to paste the results into a **Surfer** worksheet, close the **Statistics Results** dialog first.

Click *Close* to close the **Statistics Results** dialog.

95% and 99% Confidence Interval for the Mean

If CI is the value of the confidence interval reported by the worksheet, the range of values between the sample mean minus

95% Confidence Interval for the Mean

$$\pm t_{(n-1),\alpha=.05}(SE)$$

99% Confidence Interval for the Mean

$$\pm t_{(n-1),\alpha=.01}(SE)$$

where

tv,a=the value of the Student’s

SE = the standard error of the mean

Average Deviation

The average deviation is the average of the difference between the absolute values of data points and the mean.

Population Mean Deviation (MD)

$$MD = \frac{1}{N}\sum_{i=1}^{N}\left|(x_i - \mu)\right|$$

Sample Mean Deviation (MD)

$$MD = \frac{1}{n}\sum_{i=1}^{n}\left|(x_i - \bar{x})\right|$$

Definition of Population Mean (μ)

Definition of Sample Mean ($\bar{x}$)

where

N	= number of data values for a population
n	= number of data values for a sample
xi	= ithdata value

Coefficient of Kurtosis

Kurtosis is a measure of how sharp the data peak is. Traditionally the value of this coefficient is compared to a value of 0.0, which is the coefficient of kurtosis for a normal distribution (i.e. the bell-shaped curve). A value greater than 0 indicates a peaked distribution and a value less than 0 indicates a flat distribution. Without a very large sample size, the use of this coefficient is of questionable value.

Univariate Data Kurtosis (γ_2)

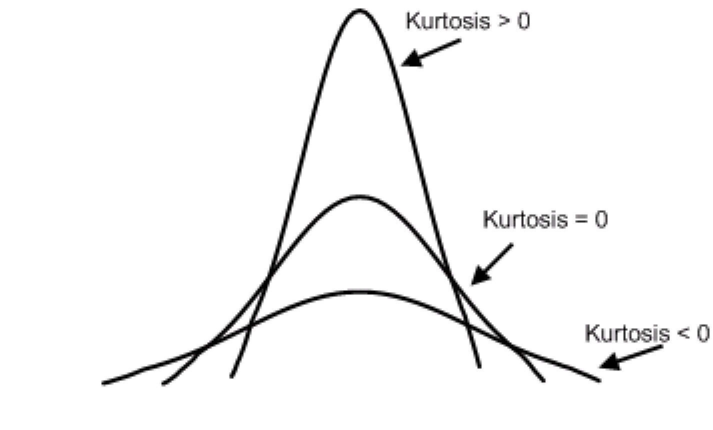
$$\gamma_2 = \left(\frac{1}{N\,\sigma^4}\sum_{i=1}^{N}(x_i - \mu)^4\right)$$

Population Kurtosis (γ_2)

$$\gamma_2 = \left(\frac{1}{N\,\sigma^4}\sum_{i=1}^{N}(x_i - \mu)^4\right) - 3$$

Sample Kurtosis (g_2)

$$g_2 = \left\{ \frac{n\,(n+1)}{(n-1)(n-2)(n-3)} \sum \left(\frac{x_i - \bar{x}}{s} \right)^4 \right\} - \frac{3\,(n-1)^2}{(n-2)(n-3)}$$



(adapted from King and Julstrom, 1982)

where

σ	= Population Standard Deviation
s	= Sample Standard Deviation
μ	= Population Mean
$\bar{x}$	= Sample Mean
N	= number of data values for a population
n	= number of data values for a sample
xi	= ithdata value

Coefficient of Skewness

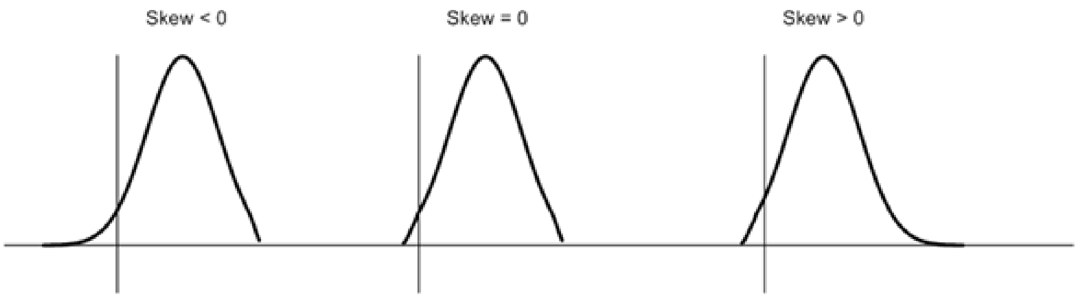
Skew is a measure of asymmetry in the distribution. A positive skew indicates a longer tail to the right, while a negative skew indicates a longer tail to the left. A perfectly symmetric distribution, like the normal distribution, has a skew equal to 0. For small data sets this measure is unreliable.

Population Skew (γ_1)

$$\gamma_1 = \frac{1}{N\,\sigma^3} \sum_{i=1}^{N} (x_i - \mu)^3$$

Sample Skew (g_1)

$$g_1 = \frac{n}{(n-1)(n-2)} \sum \left(\frac{x_i - \bar{x}}{s} \right)^3$$



(adapted from King and Julstrom, 1982 and Hildebrand, 1986)

where

σ	= Population Standard Deviation
s	= Sample Standard Deviation
μ	= Population Mean
$\bar{x}$	= Sample Mean
N	= number of data values for a population
n	= number of data values for a sample
xi	= ithdata value

Coefficient of Variation

The coefficient of variation is the standard deviation divided by the mean. The worksheet reports the quotient: it does not convert the value to a percentage. The coefficient of variation is a dimensionless measure of variation. This statistic is not defined for the case of a zero mean and this measure is only useful when dealing with strictly positive data.

Population Coefficient of Variation (V)

$$V = \sigma / \mu$$

Sample Coefficient of Variation (V)

$$V = s / \bar{x}$$

where

σ	= Population Standard Deviation
s	= Sample Standard Deviation

μ	= Population Mean
$\bar{x}$	= Sample Mean

Critical Value of K-S Statistic at 90%, 95%, and 99% Significance Level

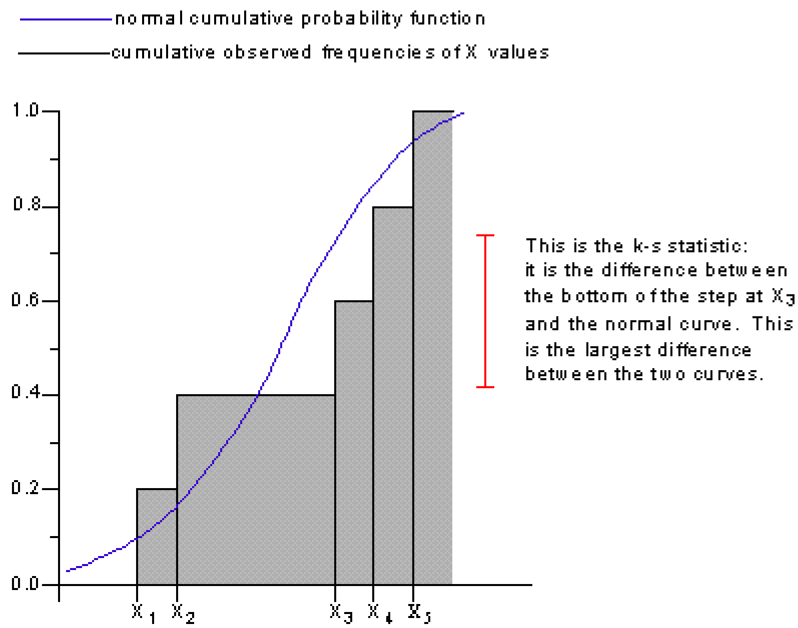
The critical value of K-S statistic at 90%, 95%, or 99% significance level are indicators of normal distributions. For example, if a sample collected from a population has a normal frequency distribution, the K-S statistic for that sample is less than the critical value 90, 95, or 99 percent of the time. If the K-S statistic is larger than the critical value, the hypothesis that the underlying population is distributed normally with a mean of $\bar{x}$ and a standard deviation of s should be rejected.

Kolmogorov-Smirnov Goodness of Fit for Normal Distribution

The Kolmogorov-Smirnov statistic is the largest difference between an expected cumulative probability distribution and an observed frequency distribution. The expected distribution used here is the normal probability distribution with mean and variance equal to the mean and variance of the sample data. The observed frequency distribution is a stepped function that increases by 1/n with each step, where n is the number of values in the data set.

For example, suppose that there are five values in a data set. The observed frequency distribution is 0 to the left of the first data point. At the first data point the observed distribution function jumps to 0.2 (since there are five data values, the size of the step at each value is one divided by five). At each successive data value the observed frequency distribution jumps by 0.2.

The K-S statistic is calculated as the largest difference (in absolute value) between the normal cumulative probability function and the observed frequency distribution, as shown below. Note that at each step it's necessary to compute the difference between bottom of the step and the normal curve and also between the top of the step and the normal curve.



Mean

The mean is the arithmetic average of the data values. It is the sum of the data values divided by the number of data values.

Population Mean (μ)

$$\mu = \frac{1}{N}\sum_{i=1}^{N} x_i$$

Sample Mean ($\bar{x}$)

$$\bar{x} = \frac{1}{n}\sum_{i=1}^{n} x_i$$

where

N	= number of data values for a population
n	= number of data values for a sample
xi	= ithdata value

Standard Deviation

The standard deviation is the square root of the variance.

Population Standard Deviation (σ)

$$\sigma = \sqrt{\sigma^2}$$

Sample Standard Deviation (*s*)

$$s = \sqrt{s^2}$$

where

σ^2	= Population Variance
s^2	= Sample Variance

Standard Error of the Mean

The standard error of the mean is an estimate of the standard deviation of means that would be found if many samples of *n* items were repeatedly collected from the same population.

An alternate description: Suppose many samples of size *n* were repeatedly collected from the same population and the means of these many samples were calculated. The means of the samples would themselves form a data set. The standard error of the mean is an estimate of the standard deviation of this theoretical sample of means.

Standard Error of the Mean (SE)

$$SE = s / \sqrt{n}$$

where

S	= Sample Standard Deviation
n	= number of data values for a sample

Variance

The population variance is the average of the squared deviations from the mean. The sample variance is the sum of the squared deviations from the mean divided by one less than the number of data values.

Population Variance (σ^2)

$$\sigma^2 = \frac{1}{N}\sum_{i=1}^{N}(x_i - \mu)^2$$

Sample Variance (S^2)

$$s^2 = \frac{1}{(n-1)}\sum_{i=1}^{n}(x_i - \bar{x})^2$$

where

μ	= Population Mean
$\bar{x}$	= Sample Mean
N	= number of data values for a population
n	= number of data values for a sample
xi	= ithdata value

Statistics References

Clark, P. and F. Evans. (1954) Distance to nearest neighbor as a measure of spatial relationships in populations. Ecology, 35:445–453.

Cressie, N. A. C. (1991) *Statistics for Spatial Data.* New York: John Wiley & Sons, Inc.

Gilbert, Richard O. (1987)Statistical Methods For Environmental Pollution Monitoring. New York: Van Nostrand Reinhold.

Hildebrand, David K. (1986) *Statistical Thinking for Behavioral Scientists*. Duxbury Press.

King, Ronald S. and Bryant Julstrom. (1982) Applied Statistics Using the Computer. Sherman Oaks, California: Alfred Publishing Company.

Press, William H. et al. (1992) *Numerical Recipes in C: The Art of Scientific Computing*. New York: Cambridge University Press.

Montgomery, D. C. and G. C. Runger. (1994) Applied Statistics and Probability for Engineers. New York: John Wiley & Sons, Inc.

Ryan, T. (1997) Modern Regression Methods. New York: John Wiley Sons, Inc.

Sheskin, D.J. (2000) *Handbook of Parametric and Nonparametric Statistical Procedures, Second Edition*. Boca Raton, Florida: Chapman & Hall/CRC.

Skellam, J. (1952) Studies in Statistical Ecology. I: Spatial Pattern., 39:346–362.

Sokal, Robert R. and F. James Rohlf. (1981) Biometry: Principles and Practices of Statistics in Biological Research. New York: Freeman and Co.

Zar, Jerrold H. (1974) Biostatistical Analysis. Englewood Cliffs, New Jersey: Prentice Hall.

Text To Number

Click the **Data | Data | Text to Number** command or the button to convert text strings in selected cells to numbers. This command will eliminate extraneous zeros and convert text to simplified numeric format.

To determine if the cell contains a number formatted as text, click on the cell to select it. The cell edit box displays the number with a ' before the number. For instance, in the image below, the number 40 appears as '40, if the number if formatted as text.

To convert a cell:

1. Click on the cell to select it.

C5 '40

	A x	B y	C z	D
1	Easting	Northing	Elevation	
2	0.1	0	90	
3	3.5	0	45	
4	4.9	0	65	
5	6.2	0	40	
6	7	0	55	

Click on the cell to select it. Notice the cell edit box contains an apostrophe (') before the number.

2. Click the **Data | Data | Text to Number** command.
3. The cell converts to numeric format, if the cell had been a text number. The cell edit box now displays the number without the ' before it.

C5	40			
	A x	B y	C z	D
1	Easting	Northing	Elevation	
2	0.1	0	90	
3	3.5	0	45	
4	4.9	0	65	
5	6.2	0	40	
6	7	0	55	

The cell now shows the number right aligned and the cell edit box does not contain an apostrophe (') before the number.

Transpose

The **Data | Data | Transpose** command rearranges data from columns to rows or from rows to columns. To quickly switch the layout of your data, highlight the data that should be flipped. Click the **Data | Data | Transpose** command or the button and the columns become rows and the rows become columns.

For example, consider the following data:

	A	B	C	D	E
1	Category	Spring	Summer	Fall	Winter
2	A	12	14	15	21
3	B	13	5	23	12
4	C	51	21	12	32

Categories A, B, and C are displayed with each category in a row.

Highlight the rows 1-4. Click the **Data | Data | Transpose** command and the data appears in columns:

	A	B	C	D
1	Category	A	B	C
2	Spring	12	13	51
3	Summer	14	5	21
4	Fall	15	23	12
5	Winter	21	12	32

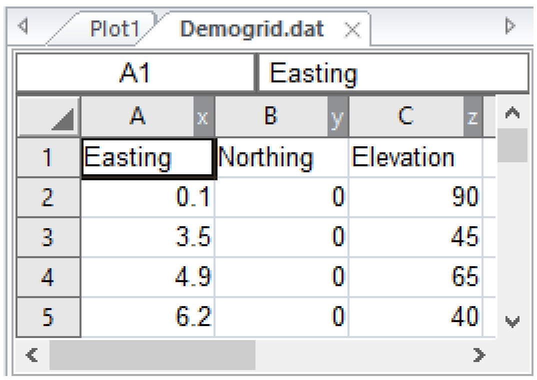
Categories A, B, and C are now displayed with each category in a separate column.

Assign XYZ Columns - Worksheet

The **Data | Data | Assign XYZ Columns** command allows you to specify the X, Y, and Z columns containing your coordinate data.

The assigned columns are used as the default X, Y, and Z columns for commands that require column input, such as the **Grids | New Grid | Grid Data** and **Home | New Map | Post** commands. The track cursor command requires that the X, Y, and Z columns be specified properly in order to display the current coordinates when a point is clicked in another window.

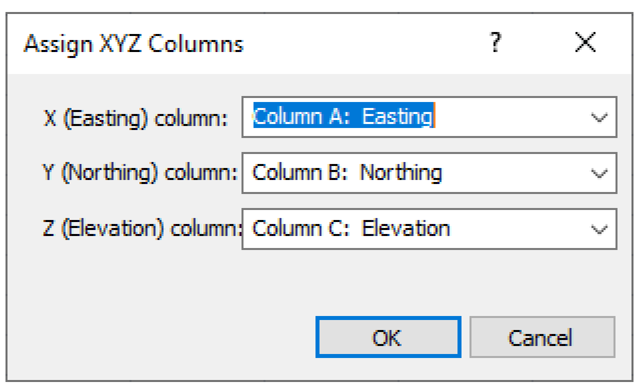
Indicators in the worksheet column headers display the current columns. When the coordinates are changed in the **Assign XYZ Columns** dialog, the X, Y, and Z indicators move to the specified columns. If there is no actual data in the user assigned columns, **Surfer** will assign columns that contain data to the coordinate columns.



*Indicators in the worksheet column headers display the X, Y, and Z columns specified in the **Assign XYZ Columns** dialog.*

The Assign XYZ Columns Dialog

The **Data | Data | Assign XYZ Columns** command or the  button opens the **Assign XYZ Columns** dialog.



*Use the **Assign XYZ Columns** dialog to specify the worksheet columns where the X, Y, and Z values are located.*

X (Easting) Column

Select a worksheet column from the *X (Easting) column* list.

Y (Northing) Column

Select a worksheet column from the *Y (Northing) column* list.

Z (Elevation) Column

Select a worksheet column from the *Z (Elevation) column* list.

Click *OK* to save the assigned X, Y, and Z columns. The X, Y, and Z indicators will move to the assigned columns. Click *Cancel* to cancel the change in the column definitions. The default columns are used.

Assign Coordinate System - Worksheet

The **Data | Coordinate System | Assign Coordinate System** command or the button links a data file to a specific coordinate system. Once the coordinate system is defined for the data file, a Golden Software Georeference .GSR2 file is created. This file contains all the relevant projection information that **Surfer** needs to load the data in the proper projection.

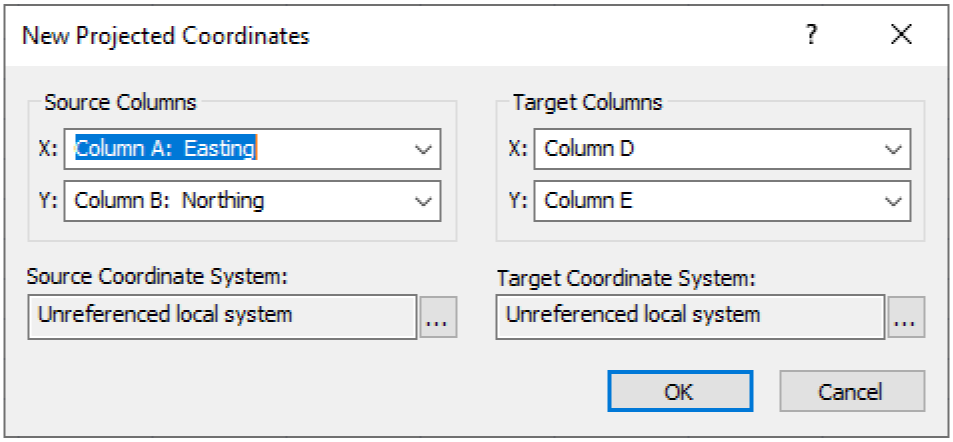
When a .GRD file is created using the Grids | New Grid | Grid Data command, the .GSR2 file for the data is read. The projection information can be saved with the grid file using the Spatial References options. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in **Surfer**, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original data file, but the .GSR2 is required to define the coordinate system. When a map is created from either the data file or the .GRD file, the .GSR2 file is read and the map layer automatically has the correct coordinate system.

New Projected Coordinates - Worksheet

The **Data | Coordinate System | New Projected Coordinates** command specifies a *Source Coordinate System* and projects the X and Y coordinate data to a new *Target Coordinate System*. The input *Source Columns* and output *Target Columns* allow you to either overwrite the original coordinate columns, or write the new coordinates to new columns in the worksheet. For example, this command can be useful if you have coordinate data in latitude and longitude and need to project the coordinates to UTM.

New Projected Coordinates Dialog

The **Data | Coordinate System | New Projected Coordinates** command or the button in the worksheet opens the **New Projected Coordinates** dialog.



*Change coordinates in the **New Projected Coordinates** dialog.*

Source Columns

Specify the columns containing the X and Y coordinates in the *Source Columns X* and *Y*. Click the arrow to see a list of the data columns in your worksheet. The *Source Columns* contain the X and Y values in the existing coordinate system.

Source Coordinate System

Assign the *Source Coordinate System* by clicking the ... button to open the Assign Coordinate System dialog. The *Source Coordinate System* is the system that your X, Y coordinate data are currently using.

Target Columns

Specify the X and Y destination columns in the *Target Columns X* and *Y*. The projected output coordinates will be located in the target columns specified. Click the arrow to see a list of the data columns in your worksheet.

Target Coordinate System

Assign the *Target Coordinate System* by clicking the ... button to open the Assign Coordinate System dialog. The *Target Coordinate System* is the new coordinate system that you want to use to project your X, Y coordinate data.

OK and Cancel

Click *OK* to write the new projected X and Y coordinates to the *Target Columns*. Click *Cancel* to close the **New Projected Coordinates** dialog without making any changes to the worksheet.

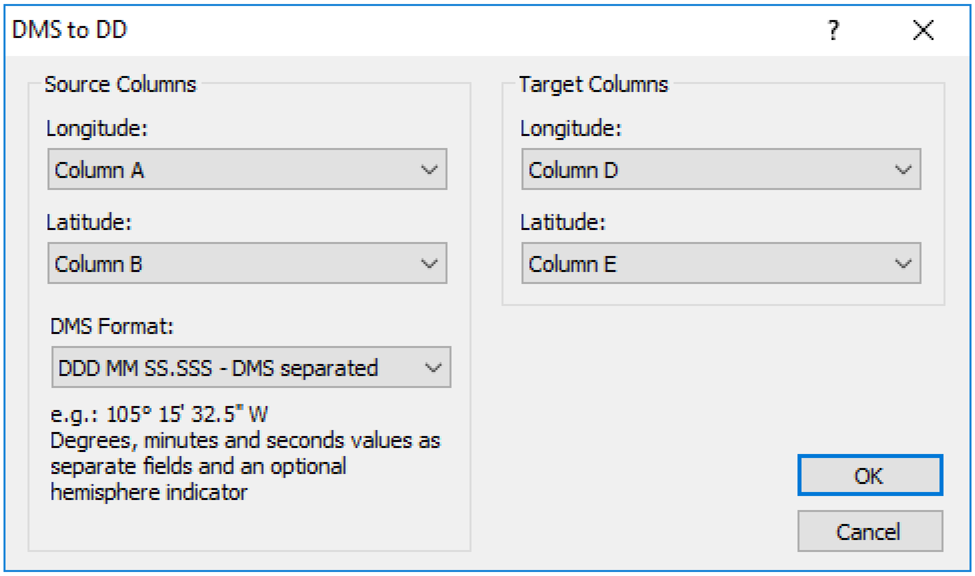
DMS to DD

Click the **Data | Coordinate System | DMS to DD** command or the button to convert DMS longitude and latitude values to decimal degrees. The **DMS to**

DD command converts two source columns of data and writes the new values in two target columns.

DMS to DD Dialog

The **DMS to DD** dialog is opened when the **DMS to DD** command is clicked. Specify the source and target columns and the source DMS format in the **DMS to DD** dialog.



Specify the source and target columns and the DMS format.

Source Columns

The *Source Columns* group contains the *Longitude* and *Latitude* columns and the *DMS Format*.

Longitude and Latitude

Select the column containing DMS longitude values in the *Longitude* field. Select the column containing DMS latitude values in the *Latitude* field. Click the current selection to change the column. The *Longitude* and *Latitude* column lists only include columns with data. *Latitude* column values must be between -90 and 90 degrees.

DMS Format

Select the source data format in the *DMS Format* field. Click the current selection and select the desired format from the list. DMS source data must be formatted in one of the following data formats, and all of the source data must use the same format:

- DDD MM SS.SSS - DMS separated - this format includes degrees, minutes, and seconds separated by spaces. Optionally, this format can include symbols and a hemisphere indicator, e.g 105° 6' 32.5" W.

- DDD MM.MMM - DM separated - this format includes degrees and minutes separated by a space. Optionally, this format can include symbols and a hemisphere indicator, e.g. 105° 6.54167' W.
- DDMMSS.SSS - DMS packed - this format includes degrees, minutes, and seconds without any spaces. The minutes and seconds values must include two digits. Optionally, this format can include a hemisphere indicator, e.g. 1050632.5 W.
- DDMM.MMM - DM packed - this format includes degrees and minutes without any spaces. The minutes value must include two digits. Optionally, this format can include a hemisphere indicator, e.g. 10506.54167 W
- DD.DDD H - Degrees w/ hemisphere - this format is decimal degrees with a hemisphere indicator, e.g. 105.109 W

Target Columns

The *Target Columns* group contains the *Longitude* and *Latitude* columns. Select the column where the converted longitude values should be created in the *Longitude* field. Select the column where the converted latitude values should be created in the *Latitude* field. Click the current selection to change the column. Select the same columns as the *Source Columns* if you wish to overwrite the source data. The default *Longitude* and *Latitude* columns are the first empty columns in the worksheet.

OK and Cancel

Click *OK* to create the decimal degrees longitude and latitude columns. Click *Cancel* to close the **DMS to DD** dialog without converting the values to decimal degrees.

Chapter 4 - Creating Grid Files

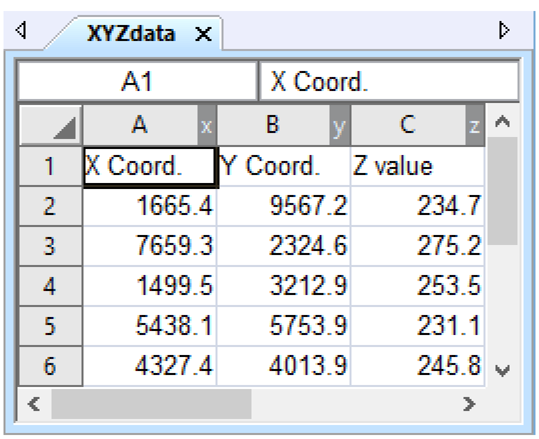
Grid Files

Contour maps, color relief maps, vector maps, watershed maps, 3D surfaces, and 3D wireframes all require grids for their creation in **Surfer**. A grid is a regular, rectangular array of values. The Home | Grid Data | Grid Data command provides you with several methods for generating a **Surfer** .GRD grid file from your XYZ data. In addition to the grid files that **Surfer** creates, it can also read many common grid file formats directly. A grid with all NoData nodes cannot be saved.

A Gridding Example

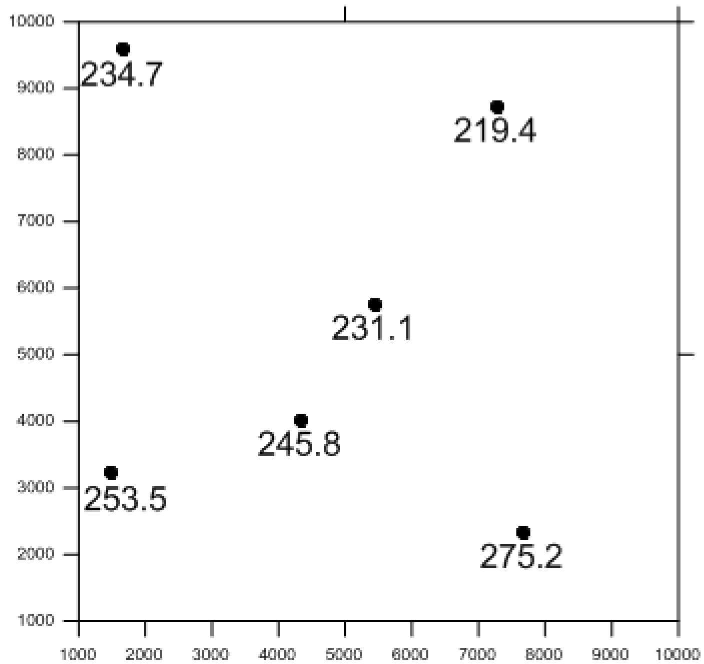
Consider the scenario of producing a contour map of water table depth given well data collected over a region. The well locations are not regularly spaced over the area of interest. If you provide **Surfer** with the locations of the wells (the XY coordinates) and the depth to the water table (the Z value) in the form of an XYZ data file, **Surfer** can produce a grid file from the original data and a grid-based map from the gridded data. The following series of figures show the normal progression from a data file, to a grid file, to a contour map.

1. In a worksheet window, define well locations (X and Y coordinates) and water table depth (Z value) at each location in an XYZ data file.



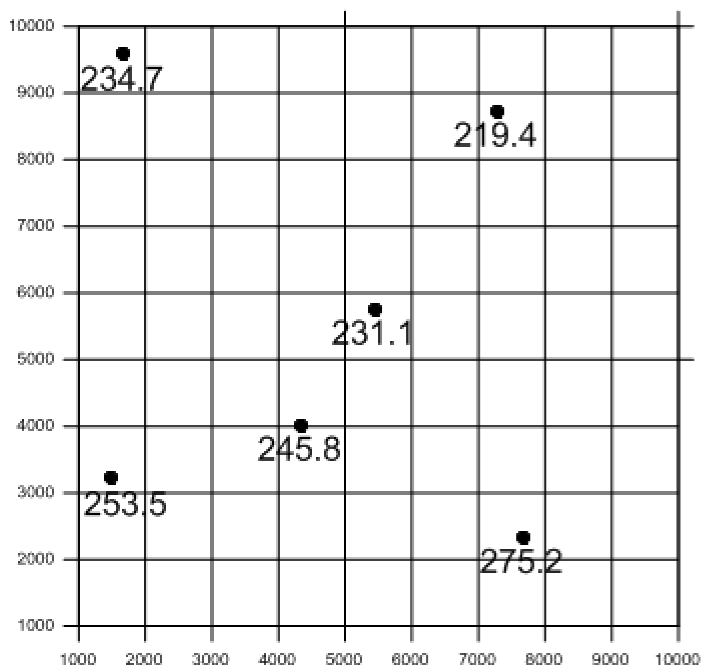
This is the XYZ data file that defines the well locations and water table depth at each location.

2. In the plot window, click the **Home | New Map | Post** command. Select the data file created in step 1and click *Open* to create a post map displaying the data locations with Z value labels. This step is to show the irregularly spaced data across the map.



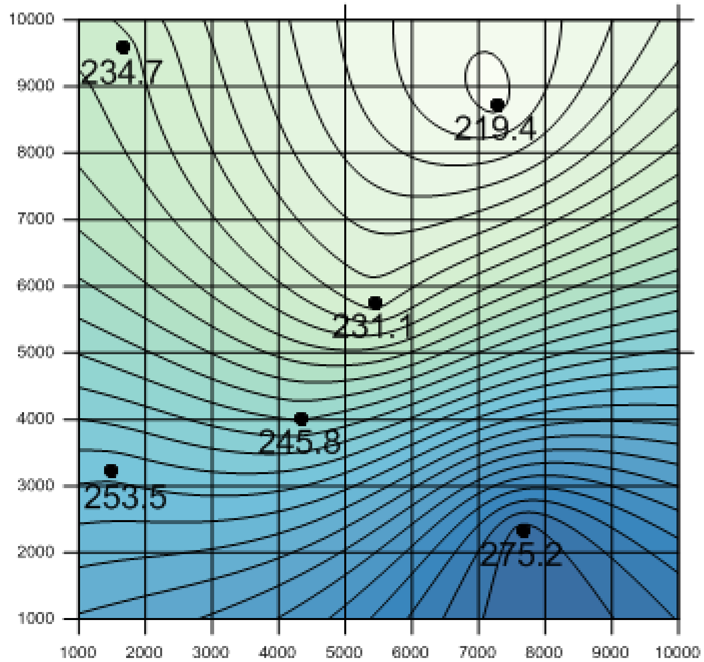
This is a post map displaying the data locations. The well locations are irregularly spaced over the map in this example. There are many "holes" where no data exists.

3. Click the **Grids | New Grid | Grid Data** command to create a regularly spaced grid .GRD file from the irregularly spaced XYZ data file. Use the default values in the Grid Data dialog and click *OK* to create the .GRD file.



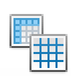
Gridding interpolates a Z value at the intersection of each row and column in the grid file, thereby filling holes in the data. Here the rows and columns are represented by grid lines drawn across the map.

4. Click once on the post map to select it. Click the **Home | Add to Map | Layer | Contour** command to add a contour map of the grid file to the post map of the data file.



The irregularly spaced data points are used to interpolate grid node values. These interpolated values are written to a grid file. The grid file is used to produce the contour map. This figure shows the filled contour map, the posted data points, and the layout of the grid.

Grid Data

Grid files are necessary in **Surfer** to create grid-based maps types. Data files are typically randomly spaced files, and this data must be converted into an evenly spaced grid before using many of **Surfer's** features. Grid files are produced from XYZ data using the **Home | Grid Data | Grid Data**, the  button, or the **Grids | New Grid | Grid Data** command. With this command, you can specify the parameters for the particular gridding method and the extents of the grid. The gridding methods define the way in which the XYZ data are interpolated when producing a grid file.

Data files should be arranged with all X data in one column, all Y data in another column, and all Z data in a third column. Each data column can be any column in the data file and in any order. The columns are selected in the **Select Data** page.

When creating a grid file you can usually accept all of the default gridding parameters and generate a grid file that represents your data well. Under most circumstances, the recommended gridding method is kriging with the default linear variogram. This is the selected default gridding method because it gives good results for most XYZ data sets.

There are several gridding parameters you can set when producing a grid file. Refer to the gridding method for more information on specific parameters. Most gridding methods require at least three non-collinear data points. The *Inverse Distance*, *Nearest Neighbor*, *Moving Average*, and *Data Metrics* methods require at least three data points, collinear or not. Some methods require more data points. For example, a higher-order polynomial fit needs more than three data points; there must be at least as many data as there are degrees of freedom. When the *Z Transform* is set to *Log, save as log* or *Log, save as linear*, at least three data points must contain Z values that are non-negative and non-zero. Click the **Grids | New Grid | Grid Data** or **Home | Grid Data | Grid Data** command to choose the data to be used in the gridding process.

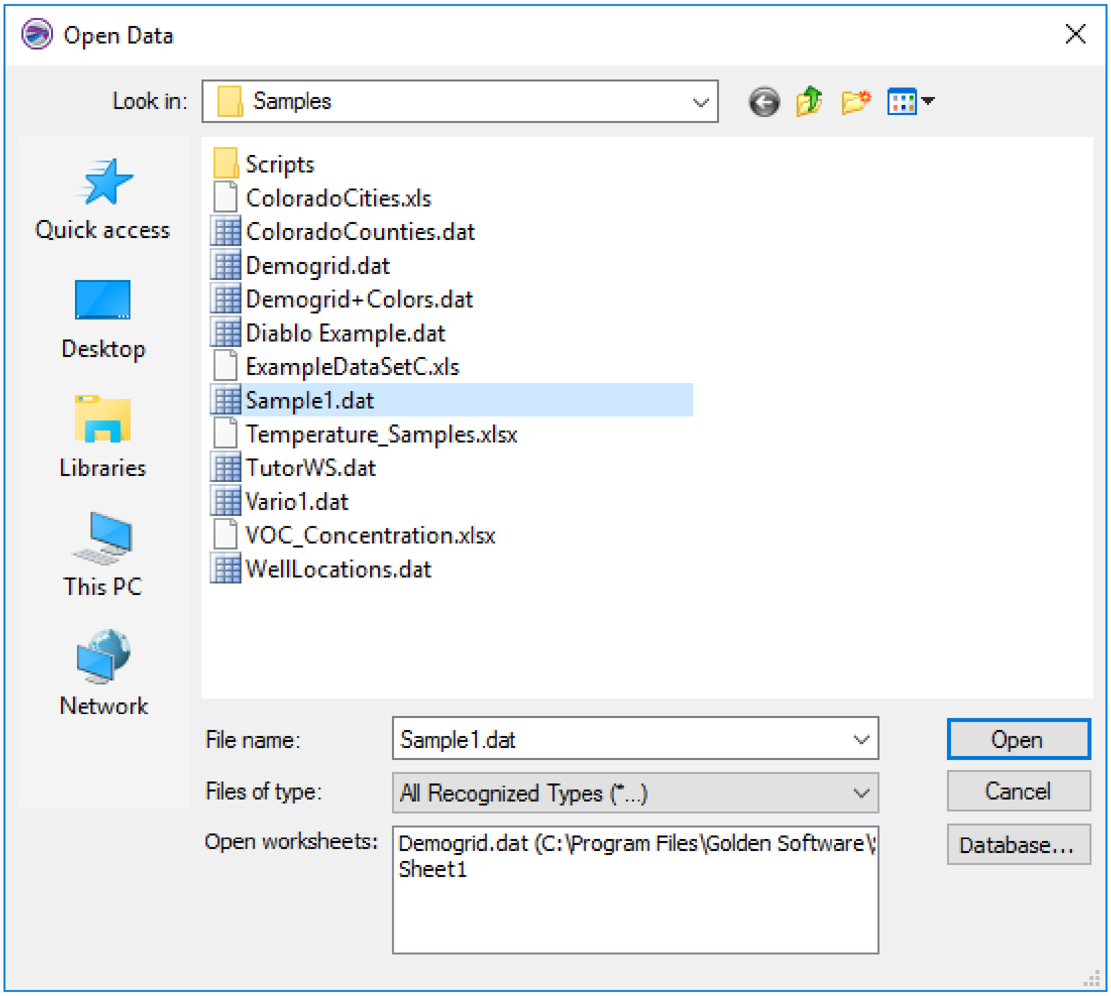
Grid Data Dialog

Click the **Grids | New Grid | Grid Data** or **Home | Grid Data | Grid Data** command to display the **Grid Data** dialog. The **Grid Data** dialog includes multiple pages:

- Select Data
- Variogram
- Options
- Cross Validation
- Output

Open Data Dialog

The **Open Data** dialog is used by some plot window grid commands (i.e. **Grid Data**).



*Select a data file in the **Open Data** dialog.*

Look In

The *Look in* field shows the current directory. Click the down arrow to see the directory structure and click on the folders to change directories.

Creating New Folders and Changing the View

The buttons to the right of the *Look in* field allow you to create new folders and change the view of the file list.

File List

The *File list* displays files in the current directory. The current directory is listed in the *Look in* field. The *Files of type* field controls the display of the file list. For example, if *DAT Data (*.dat)* is listed in the *Files of type* field only *.DAT files appear in the files list.

Open Worksheets

Surfer tracks the files being used by maintaining a list of opened files for new maps and the files used in saved maps. The file(s) used to build maps are shown in a compiled list in the *Open Worksheets* field.

Specify a File Name

The *File name* field shows the name of the selected file. Alternatively, type a path and file name into the box to open a file.

Files of Type

The *Files of type* field controls the display of the file list. For example, if *DAT Data (*.dat)* is listed in the *Files of type* field only *.DAT files appear in the files list.

The *All Recognized Types (*...)* format type is selected by default. This displays all the common file formats in the navigation pane. If a different format type is selected, **Surfer** will remember the setting until the end of the current session. When **Surfer** is restarted, the default format type will be used.

To see all files in the directory, choose *All Files (*.*)* from the *Files of type* list. Double-click on a file to open it or single-click the file and then click the *Open* button. The *All Files (*.*)* option shows all of the file formats in the current directory, even if the file type is not appropriate for the action chosen. For example, a GRD file may be displayed, even though a GRD file cannot be imported into the worksheet.

Select a file type from the *Files of type* list. The following formats are supported:

- ACCDB Access Database (*.accdb)
- BLN Golden Software Blanking .BLN
- BNA Atlas Boundary .BNA
- CSV Comma Separated Variables .CSV
- DAT Data .DAT
- DBF Database .DBF
- DXF AutoCAD Drawing Data (*.dxf)
- LAS LiDAR Data (*.las)
- MDB Microsoft Access .MDB
- SEG-P1 Data Exchange Format (*.seg)
- P1 Data Exchange Format (*.sp1)
- SLK Sylk Spreadsheet .SLK
- TXT Text Data .TXT
- XLS Excel Spreadsheet .XLS
- XLSX Excel 2007 Spreadsheet .XLSX
- XLSM Excel 2007 Spreadsheet .XLSM
- ACCDB Access Database (*.accdb)
- BLN Golden Software Blanking .BLN
- BNA Atlas Boundary .BNA

- CSV Comma Separated Variables .CSV
- DAT Data .DAT
- DBF Database .DBF
- DXF AutoCAD Drawing Data (*.dxf)
- LAS LiDAR Data (*.las)
- MDB Microsoft Access .MDB
- SEG-P1 Data Exchange Format (*.seg)
- P1 Data Exchange Format (*.sp1)
- SLK Sylk Spreadsheet .SLK
- TXT Text Data .TXT
- XLS Excel Spreadsheet .XLS
- XLSX Excel 2007 Spreadsheet .XLSX
- XLSM Excel 2007 Spreadsheet .XLSM

Open Worksheets

The *Open worksheets* group displays worksheets that are currently open to be selected. If a worksheet is selected from the *Open worksheets* group that has assigned XYZ coordinates, the following dialogs (i.e. Grid Data dialog) will have the X, Y, and Z columns populated to match the assigned XYZ coordinates.

Surfer tracks the files being used by maintaining a list of opened files for new maps and the files used in saved maps. The file(s) used in existing maps are shown in a compiled list in the *Open worksheets* field of the **Open Data** dialog.

Load Database

Click the **Database** button in the **Open Data** dialog to open the Data Link Properties dialog and import a database.

Cross Validation

Generally, cross validation can be considered an objective method of assessing the quality of a gridding method, or to compare the relative quality of two or more candidate gridding methods. In **Surfer**, cross validation can be used with all gridding methods. While cross validation can be used to select a gridding method, the results can also be used to assess the spatial variation in gridding quality and to guide data sampling. Cross validation is always performed on the linear Z values, not the transformed Z values.

A generalized discussion of cross validation is given here. Refer to one of the many geostatistics books for more information. In the listed references below, much of the discussion concerns kriging, but the generalized discussion applies to all of the gridding methods in **Surfer**.

Cross Validation Process

Given the known values at N observation locations in the original data set, cross validation allows you to assess the relative quality of the grid by computing and investigating the gridding errors. In **Surfer**, these errors are calculated by removing the first observation from the data set, and using the remaining data and the specified algorithm to interpolate a value at the first observation location. Using the known observation value at this location, the interpolation error is computed as:

error = interpolated value − observed value

Then, the first observation is put back into the data set and the second observation is removed from the data set. Using the remaining data (including the first observation), and the specified algorithm, a value is interpolated at the second observation location. Using the known observation value at this location, the interpolation error is computed as before.

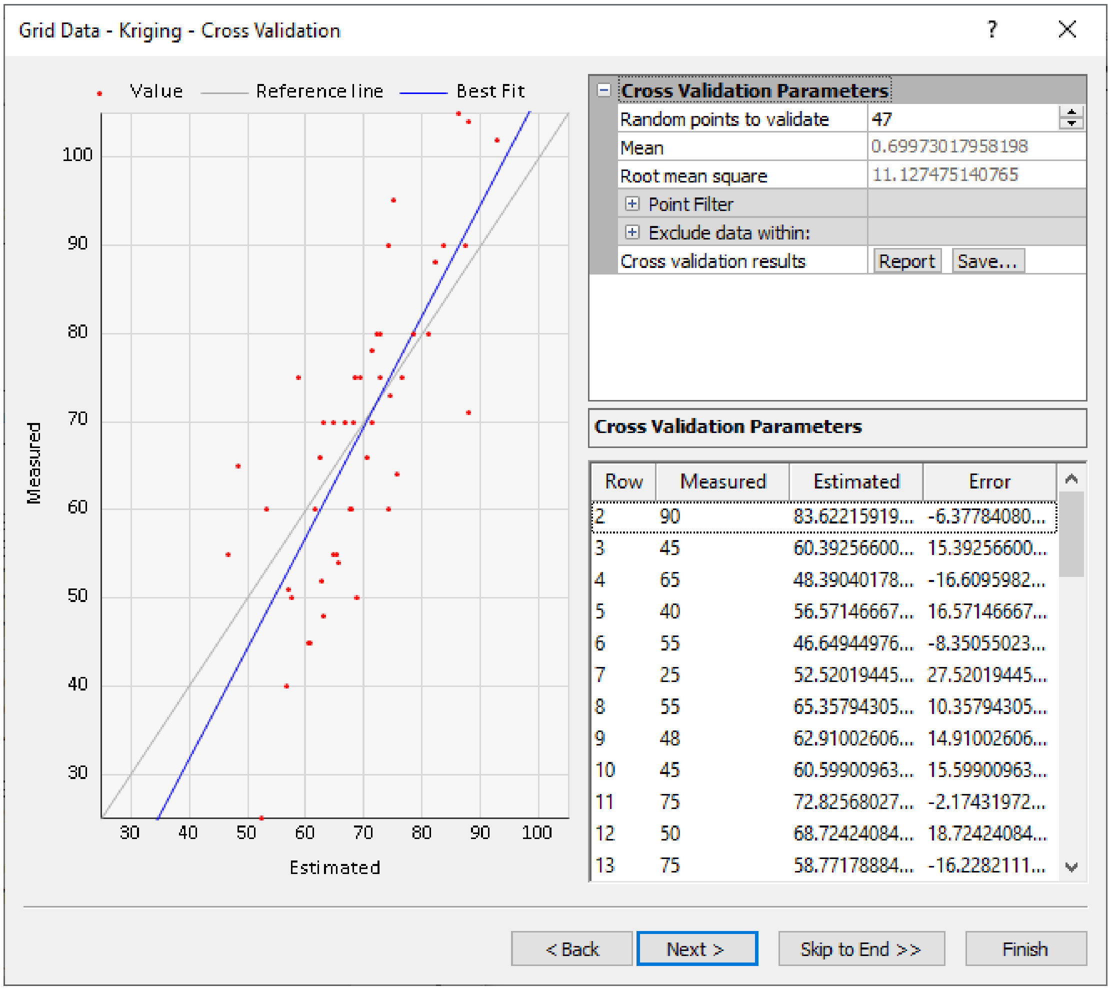
The second observation is put back into the data set and the process is continued in this fashion for the third, fourth, fifth observations, etc., all the way through up to and including observation N, the last observation in the data file. This process generates N interpolation errors. Various statistics computed for the errors can be used as a quantitative, objective measure of quality for the gridding method.

Thus, cross validation involves four steps:

1. Select a gridding method, along with all of the defining parameters.
2. For each observation location, interpolate the value using the neighboring data, but not the observation itself.
3. Compute the resulting interpolation errors.
4. Assess the quality of the selected gridding method using various summary statistics of the errors.

Grid Data - Cross Validation

The **Cross Validation** page is displayed after the Options page in the Grid Data dialog.



Evaluate gridding methods with cross validation.

Number of Random Points to Validate

For large and very large data sets the cross validation process of sweeping through each and every observation point can take a great deal of time and computational effort. A quicker alternative is to carry-out the cross validation procedure at a random subset of the observation locations.

Number of random points to validate allows you to specify the number of cross validation points. By default, this value is equal to the total number of observations. For large and very large data sets this values should be 1,000 or more to ensure a relatively stable set of error statistics. Note, the entire data set is used through-out the cross validation process, not just the random subset. The random subset merely identifies the locations at which cross validation errors are computed. Also, note that the random subset is determined without replacement, using a random number generator that is randomly initialized for every execution; thus, the random subset may be different every time cross validation is run.

Points Filter

Set the X, Y, and Z direction values for *Minimum* and *Maximum* in the *Point Filter* section. This restricts the cross validation to a subarea of the data extent. Data falling outside of these limits may be used during the interpolation, but they are not used as cross validation points. The limits in Z direction are useful to exclude cross validation at known anomalous observation locations.

Exclude Data Within

Often observations come in relatively homogeneous clusters. In these circumstances, the standard cross validation approach may not generate useful results as the interpolated values are merely the values of the close-by adjacent observations. To counter this potential problem, the *X tolerance* and *Y tolerance* fields define a centered rectangular buffer zone around each of the observations at which cross validation is carried out. This buffer zone is two times the *X tolerance* in the X direction, and two times the *Y tolerance* in the Y direction. Any observations falling within a validation point's buffer zone are not used in the interpolation of that validation point. When these values are zero, all points in the validation limits are used.

Cross Validation Report

Click *Report* to generate a cross validation report.

Cross Validation Results File

Click *Save* to save a cross validation results file. The results are presented in an ASCII data file, which can be used as a gridding data file. The first line in the file contains column titles. Each subsequent line is associated with a validation point. There are seven numeric values per line. The first three values are the X, Y, and Z values from the original data file of each validation point. The fourth column, titled *ID* is the line number from the original data file of the validation point. The next two columns are the estimated and residual values. The *nData* column contains the total number of original data points.

Statistics Note

Under most interesting statistical models, the cross validation errors are not statistically independent: the estimate at one observation location uses much of the same information as the estimate at a neighboring observation location. As such, standard hypothesis testing and test of statistical significance are not valid. A discussion of this point can be found in the cross validation reference, Kitanidis (1997).

Cross Validation References

The following references include extensive discussion of the theory and application of cross validation. Isaaks and Srivastava [1989], Kitanidis [1997], Olea [1999], and Chiles and Delfiner [1999] focus on cross validation with kriging, but the general discussion applies to all of the supported gridding methods inSurfer.

Chiles, J. P. and P. Delfiner. (1999)Geostatistics: Modeling Spatial Uncertainty. John Wiley and Sons, New York, 695 pp.

Isaaks, E. H. and R. M. Srivastava. (1989) Applied Geostatistics. Oxford University Press, Inc., New York, 561 pp.

Kitanidis, P. K. (1997)Introduction to Geostatistics: Applications inCambridge University Press, Cambridge, 249 pp.

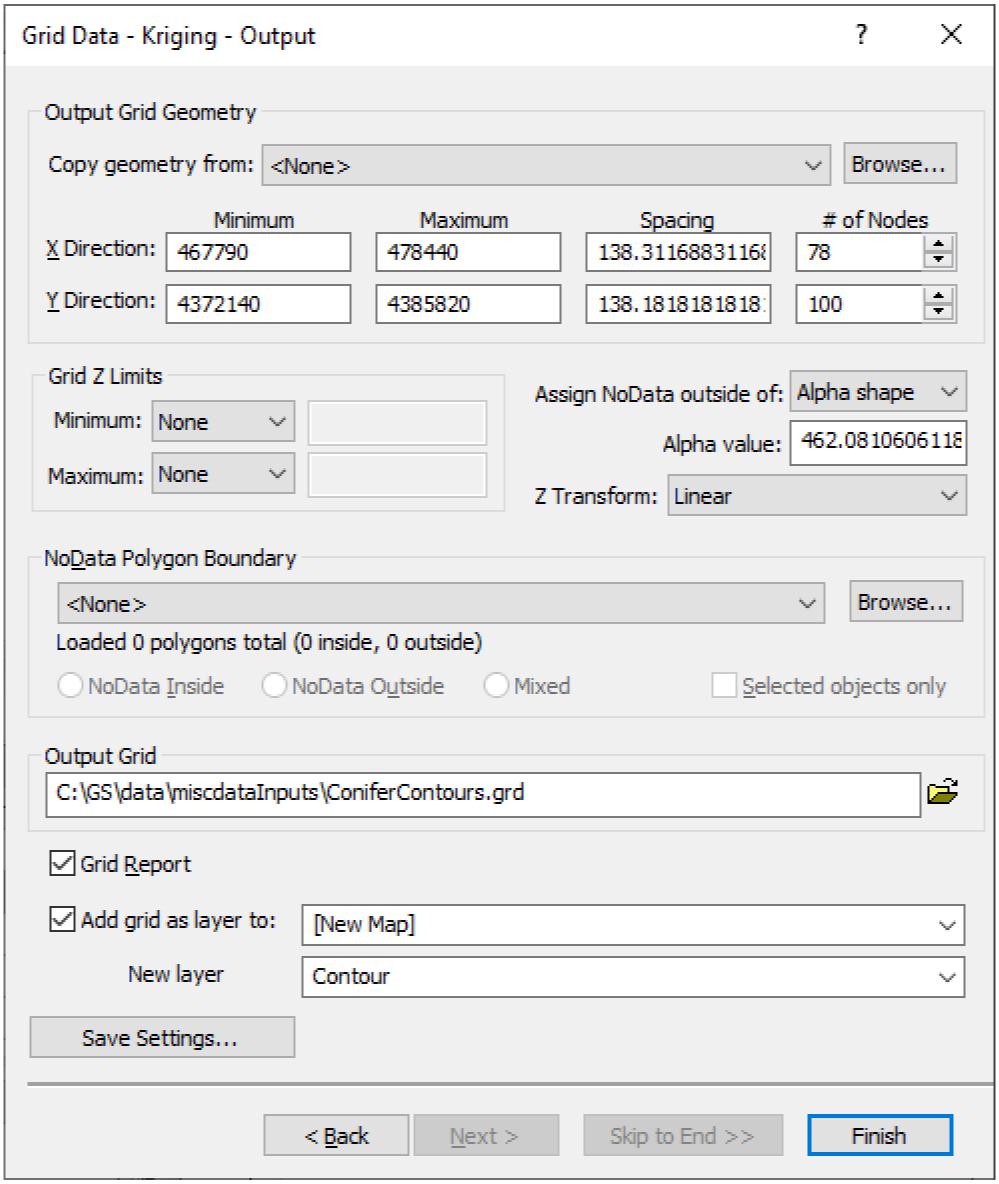
Olea, R. (1999) *Geostatistics for Engineers and Earth Scientists*. Kluwer Academic Publishers, Boston, 303 pp.

What's Next?

Click *Next* or *Skip to End* to proceed to the Output page. Click *Finish* to create the grid file.

Grid Data - Output

The **Output** page in the Grid Data dialog includes output options for the grid file, including geometry, data limits and transforms, assigning NoData, and generating a report.



*Specify grid file output options in the **Output** page.*

Output Grid Geometry

The *Output Grid Geometry* section defines the grid limits and grid density.

Copy Geometry

The *Copy geometry from* option copies the grid geometry from an existing map layer or grid file. This option is useful when creating grids that will become overlaid map layers, processed with the Grid Math command, or used to calculate a volume between two surfaces. The **Math** and **Volume** commands require the input grids to have the same geometry.

To copy the geometry from an existing layer, select the layer in the *Copy geometry from* list. To copy the geometry from a grid file, click *Browse* and select the file in the Open Grid dialog. Select *<None>* to return the *Output Grid Geometry* options to their default values and to manually edit the grid geometry.

Minimum and Maximum X and Y Coordinate (Grid Limits)

Grid limits are the minimum and maximum X and Y coordinates for the grid. **Surfer** computes the minimum and maximum X and Y values from the XYZ data file. These values are used as the default minimum and maximum coordinates for the grid.

Grid limits define the X and Y extent of the output grid. The extents of the grid define the extents of contour maps, color relief maps, shaded relief maps, vector maps, 3D wireframes, and 3D surfaces created from grid files. When creating a grid file, you can set the grid limits to the X and Y extents you want to use for your map. Once a grid file is created, you cannot produce a grid-based map larger than the extent of the grid file. If you find you need larger grid limits, you must regrid the data. You can, however, read in a subset of the grid file to produce a map smaller than the extent of the grid file.

When either the X, Y, or Z value is in a date/time format, the date/time values are converted and stored in the grid as numbers.

Spacing and # of Nodes (Grid Density)

Grid density is usually defined by the number of columns and rows in the grid, and is a measure of the number of grid nodes in the grid. The *# of Nodes* in the *X Direction* is the number of grid columns, and the *# of Nodes* in the *Y Direction* is the number of grid rows. The direction (*X Direction* or *Y Direction*) that covers the greater extent (the greater number of data units) is assigned 100 grid nodes by default. The number of grid nodes in the other direction is computed so that the grid nodes *Spacing* in the two directions are as close to one another as possible.

By defining the grid limits and the number of rows and columns, the *Spacing* values are automatically determined as the distance in data units between adjacent rows and adjacent columns.

Note on High Density Grid Files

Higher grid densities (smaller *Spacing* and a larger *# of Nodes)* increase the smoothness in grid-based maps. However, an increase in the number of grid nodes proportionally increases the gridding time, drawing time, and the grid file size. You can have up to 2,147,483,647 rows and columns in a grid file. It is likely your computer will run out of memory before reaching the maximum grid size. The primary use for the large grid size maximum is to allow grids with extreme aspect ratios to be created.

The larger the density of grid nodes in the grid, the smoother the map that is created from the grid. Contour lines and XY lines defining a wireframe are a series of straight-line segments. More X and Y grid nodes in a grid file result in shorter line segments for contours or wireframe maps. This provides a smoother appearance to contour lines on a contour map or smoother appearing wireframe.

Although highly dense grid files can be created, time and space are practical limits to the number of grid nodes you may want to create in a grid file. The grid density limit is based on the amount of available memory in your computer and the size of the data file used to create the grid. Limited memory, very large data files, very dense grids, or any combination of these factors can greatly increase gridding time. When gridding begins, the status bar provides you with information about the estimated gridding time to complete the task. If gridding time is excessive, click in the plot window to cancel the gridding operation.

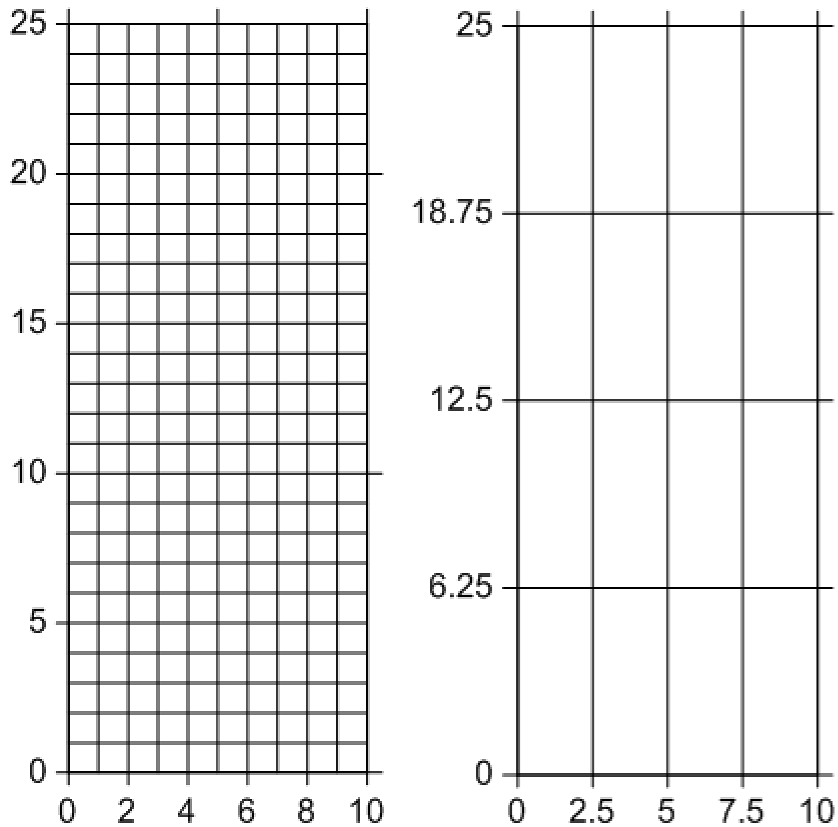
Some examples of the amount of memory needed to grid large files:

- A 10,000 x 10,000 grid requires 10000*10000*8 = 763MB.
- A 15,000 x 15,000 grid requires 1.7GB.
- A 20,000 x 20,000 grid requires 3GB which is more than a 32-bit OS can address (although it is possible on a 64 bit OS)
- A 2,147,483,647 x 2 grid requires 32GB of contiguous RAM (most computers contain a maximum of 16GB RAM stored noncontiguously)

You can also increase or decrease the grid density by using the Grid | Spline Smooth, Grid | Extract, or Grids | Resize | Mosaic commands.

Output Grid Geometry Example

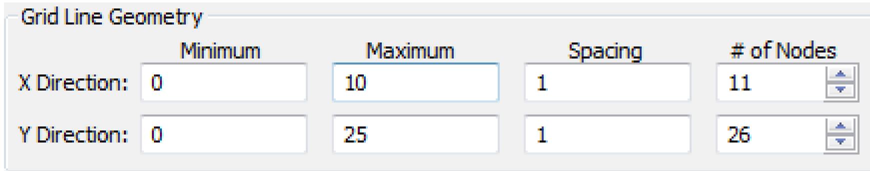
Consider these examples. The data range from 0 to 25 in the Y dimension and 0 to 10 in the X dimension. The two examples use different numbers of grid nodes, or grid spacing, during gridding.



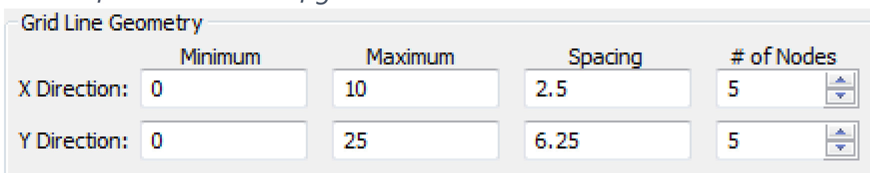
Two different Grid Line Geometry examples are shown here. These are based on the same data file. The coordinates range from zero to 10 in the X direction and zero to 25 in the Y dimension.

In the example on the left above, the grid *Spacing* is set approximately equal in the X and Y dimensions (one unit each). This results in a different number of grid nodes in the X and Y dimensions. In the example on the right above, the same *# of Nodes* are specified in the two dimensions. This results in an unequal spacing in data units in the two dimensions.

The *Output Grid Geometry* information specified in the **Grid Data** dialog for each of the examples is displayed below.



This shows the Output Grid Geometry information for the 11 by 26 grid. The grid node spacing values are set to one, resulting in a different number of grid nodes in the X and Y dimensions.



This shows the Output Grid Geometry information for the 5 by 5 grid. The number of nodes is equal, resulting in different spacing in the X and Y dimensions.

Grid Z Limits

In some cases, the gridding interpolation and extrapolation can result in undesired values, for example negative numbers in cases where negative values are physically impossible. The *Grid Z Limits* options clamp the grid output to specific minimum and maximum values.

The *Grid Z Limits* are applied after the interpolation operation. After the grid interpolation is performed, **Surfer** locates any grid values less than the *Minimum* and replaces them with the *Data min* or *Custom* value. **Surfer** locates any grid values greater than the *Maximum* and replaces them with the *Data max* or *Custom* value.

To clamp the output to a specific minimum value, click the current selection next to *Minimum*, and select *None*, *Data min*, or *Custom* from the list. If *Data min* is selected, the data minimum will be displayed in the field to the right of the *Minimum* list. Select *Custom* and type a value in the input box to use a user-defined *Minimum*.

To clamp the output to a specific maximum value, click the current selection next to *Maximum*, and select *None*, *Data max*, or *Custom* from the list. If *Data max* is selected, the data maximum will be displayed in the field to the right of the *Maximum* list. Select *Custom* and type a value in the input box to use a user-defined *Maximum*.

Assign NoData

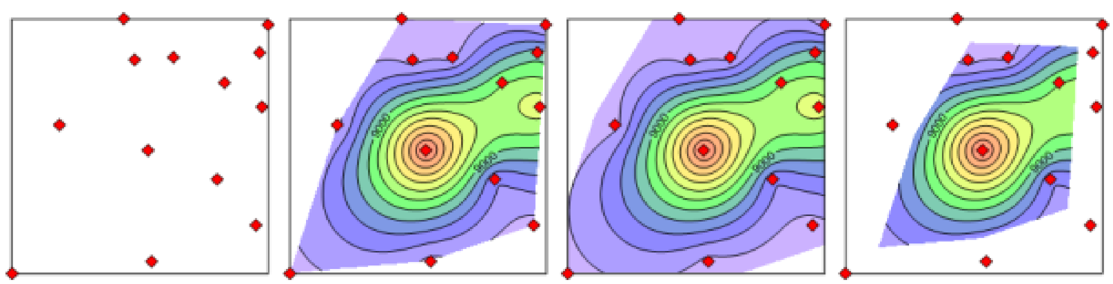
Convex Hull

Select *Convex hull* from the *Assign NoData outside of* list to assign the NoData value to the grid nodes outside the convex hull of the data. The convex hull of a data set is the smallest convex polygon containing all the data. The convex hull can be thought of as a rubber band that encompasses all data points. The rubber band only touches the outside points. Areas inside the convex hull without data are still gridded. See the *Grid Data* topic for an example of assigning NoData outside the convex hull of the data.

If *Convex hull* is selected from the *Assign NoData outside of* list, the **Grid Contours** dialog displays the *Inflate convex hull by* box. Enter a value to expand or contract the convex hull. When set to zero, the boundary connects the outside data points exactly. When set to a positive value, the area assigned the NoData value is moved outside the convex hull boundary by the number of map units specified. When set to a negative value, the area assigned the NoData value is moved inside the convex hull boundary by the number of map units specified. Values are in horizontal (X) map units. If the value is set to a large positive value, the grid values may extend all the way to the minimum and maximum X and Y limits of the grid, essentially overriding the *Assign NoData outside convex hull of data* option. If the value is set to a large negative value, the entire grid may be assigned the NoData value, resulting in no grid file being created.

Convex Hull Example

This example, with a data set such as the points on the left below, checking the *Assign NoData outside convex hull of data* option will leave the outer edges of the map blank due to the NoData values in these areas. The three contour maps display the resulting grid file when the *Inflate convex hull by* option is set to 0, inflated by 1.5, and deflated by -1.5. Contours always extend to the same minimum and maximum X and Y coordinates. Contour lines align in all three situations. The only difference is how far the contours extend from the convex hull.



This data has an area near the top left corner with no data. When the Assign NoData outside convex hull of data option is selected, no contours appear in this area. Setting the Inflate convex hull by value to a positive value (3rd image) creates a buffer around the outside of the data points. Setting the Inflate convex hull by value to a negative value (last image) brings the contours inside the convex hull of data.

Alpha Shape

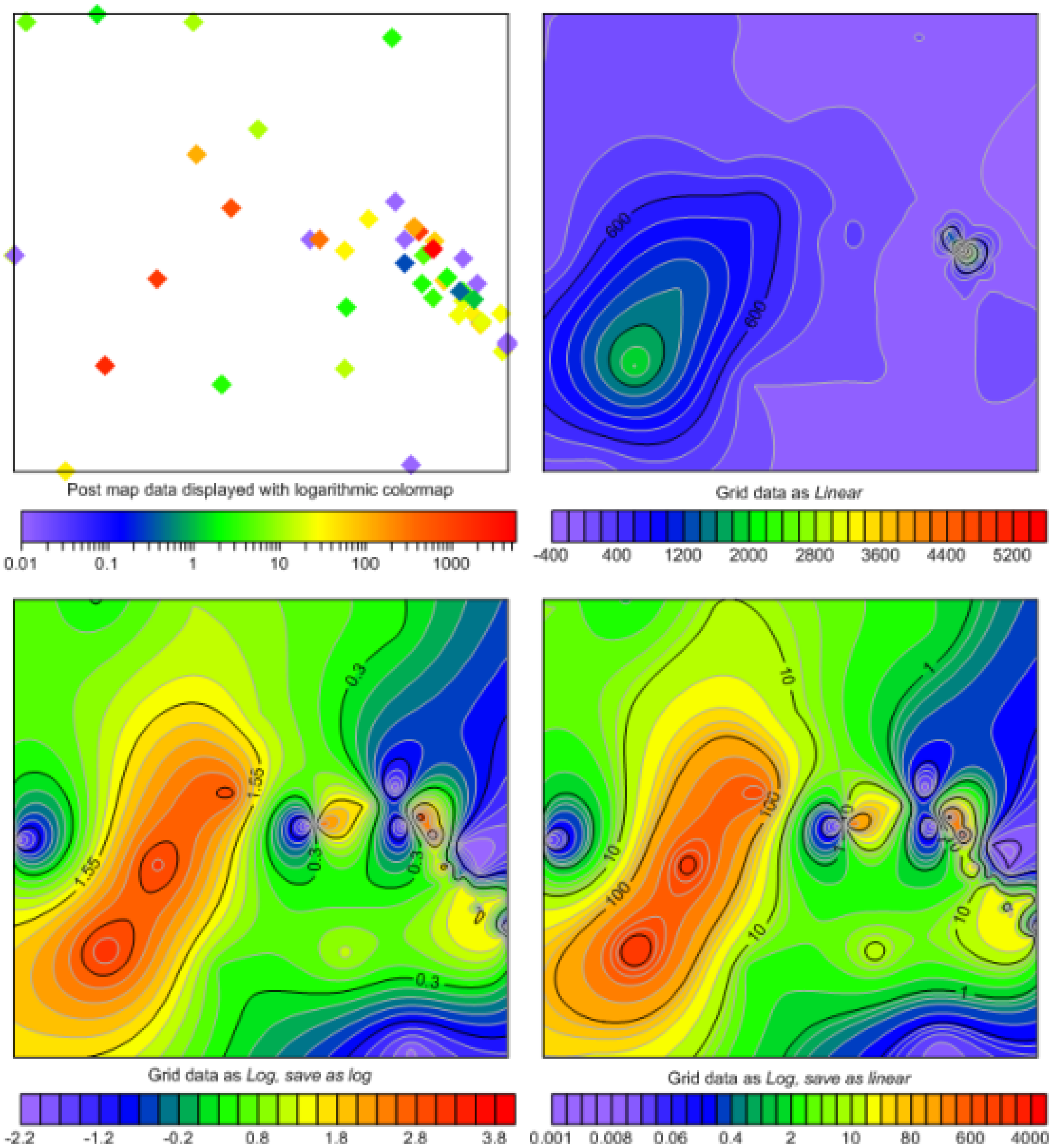
Select *Alpha shape* from the *Assign NoData outside of* list to assign the NoData value to the grid nodes outside the alpha value of the data. Select *Alpha shape* instead of *Convex hull* to have a tighter boundary, especially if a boundary could form concave areas. If *Alpha shape* is selected from the *Assign NoData outside of* list, the **Grid Contours** dialog displays the *Alpha value* box. Enter a value to define the radius of the circles that are created by the edges of the triangulation of the points. Larger alpha values create a more convex boundary and smaller alpha numbers create a tighter, concave boundary. If too low or too high of an alpha value was entered, a message will appear stating that the grid could not be assigned NoData because of the alpha value. For more information on alpha shapes or to determine an appropriate alpha value, see the Alpha Shape help topic.

Z Transform

The *Z Transform* option changes how the Z values are gridded. Available options are *Linear; Log, save as log;* and *Log, save as linear.* To change the *Z Transform* option, click on the existing option and select the desired option from the list.

Linear uses the Z values in the worksheet for gridding. No transformation is applied to the Z values. The *Linear* method is a good option for data that gradually increases over space. This is the default *Z Transform*.

Both *Log* options use take the log (base 10) of the Z values before gridding. The log (base 10) of the Z value is then used for gridding. The *Log, save as log* takes the log (base 10) of the Z values and uses the log value for gridding. The grid is then saved with the log (base 10) values. The *Log, save as linear* takes the log (base 10) of the Z values and uses the log value for gridding. The grid is then converted back to the linear Z values by taking the antilog of the gridded results. When *Log, save as log* or *Log, save as linear* is selected, at least three data points must be positive Z values. Negative values are ignored for gridding. Both *Log* methods are good options when the data changes very quickly over a small area or when very high and very low values occur very closely to each other. This can be common with concentration values in ground water or geochemical data.



The images above display the difference in gridding the posted data with linear (top right) and log (bottom contours). The log contours on bottom show the difference in Z values between the grid when Log, save as log *(bottom left) and* Log, save as linear *(bottom right) are selected.*

Polygon Boundary

Specify the region or regions to be assigned the NoData value and whether or not to use only selected objects in the *NoData Polygon Boundary* section. Select either a map layer or vector file in the *NoData Polygon Boundary* section:

- Click the current selection and select a base layer from the list. Only base layers that contain at least one polygon, polyline, 3D polygon or 3D polyline will be included in the list. The base layer must use the same source

coordinate system as the grid.
- Click *Browse* to load a vector file with the Open dialog. The file must use the same coordinate system as the grid.

The number of polygons and 3D polygons and vertices is displayed below the *NoData polygon boundary* once a file or map layer has been selected. If the boundaries have blanking flags or BLN_Flag attributes, the total number of *inside* and *outside* flags is displayed.

Polyline Boundaries

Polylines and 3D polylines can be used for NoData polygon boundaries. The polylines in the base layer or vector file will be treated as polygons and 3D polylines treated as 3D polygons while assigning NoData values. The **Assign NoData** command is not recommended with open polylines as unexpected results may occur. Before clicking **Grids | Edit | Assign NoData**, consider converting the object to another type better suited to the operation you wish to perform with one of the Change To commands, and edit features with the Reshape command.

If the layer you wish to use contains both polygons and polylines, but you only wish to use some or all of the polygons, select the objects you wish to use before clicking **Grids | Edit | Assign NoData** and select the *Selected objects only* option. If the file you wish to use contains both polylines and polygons, first load the file as a base layer, and then use the **Assign NoData** command with the *Selected objects only* option.

Inside, Outside, or Mixed

Select *Inside* to assign the NoData value to the region inside the NoData polygon/3D polygon boundary or boundaries. Select *Outside* to assign the NoData value to the region outside the NoData polygon/3D polygon boundary or boundaries. Select *Mixed* to use the blanking flag or BLN_Flag attribute values from the file or layer. The *Mixed* option is only available when the layer or file contains both blanking flags or BLN_Flag attributes: assign NoData inside (1) and assign NoData outside (0). If all blanking flags or BLN_Flag attributes are the same, the *NoData Inside* or *NoData Outside* option is selected automatically, and the *Mixed* option is not available.

Selected Objects Only

Select the *Selected objects only* option to use only the selected objects in the base layer to assign NoData values to the grid. When the *Selected objects only* box is checked, the *Loaded polygons*and*vertices* values are updated. Select a base layer in the *NoData Polygon Boundary* field to use the *Selected objects only* option. The *Selected objects only* option is not available when the *NoData Polygon Boundary* is a vector file. The polygon or polygons must be selected before clicking the **Grids | Edit | Assign NoData** command.

Supported Vector Formats

The following file formats can be used to assign NoData values to a grid:

BLN Golden Software Blanking

BNA Atlas Boundary
DDF SDTS TVP
DLG USGS Digital Line Graph
DXF AutoCAD Drawing
E00 Esri AcrInfo Export Format
EMF Windows Enhanced Metafile
GML Geo Markup Language
GSB Golden Software Boundary
GSI Golden Software Interchange
KML Google Earth Keyhole Markup Language
MIF MapInfo Interchange Format
PLT Golden Software PlotCall
PLY Stanford Polygon
SHP Esri Shapefile
STL Stereo Lithograph
WMF Windows Metafile

Output Grid

Choose a path and file name for the grid file in the *Output Grid* group. You can type a path and file name, or click the button to browse to a new path and enter a file name in the **Save Grid As** dialog.

Grid Report

Check the box next to the *Grid Report* option to create a gridding report that includes all the gridding parameters used to generate a grid. This report also includes statistics about the grid. You can also access the grid statistics by creating a grid information report. Create a grid information report in the Grid Editor by clicking the Grid Editor | Options | Grid Info command or by clicking the **Grids | Info | Grid Info** command from any document window.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

Save Settings

Click *Save Settings* to save the current **Grid Data** dialog settings from all dialog pages (**Select Data**, **Variogram** (if applicable), **Options**, **Cross Validation**, and **Output**) to a Grid Data Settings file. Grid Data Settings files can be loaded on the Select Data page when creating grids.

What's Next?

Click *Finish* to create the grid file, or click *Back* to return to previous dialog pages.

Reports

Surfer can generate several types of detailed grid reports. This table provides a snapshot of the types of information produced for each report type. Descriptions of the report sections are below the table.

If you make changes to the data selection (i.e. changing a data column or changing the data filtering method) generate a new report by repeating one of the processes listed with the report type.

Information Produced	**Report Types and How to Access Them**					
	Cross Validation Report	Data Statistics Report	Grid Contour Vol/Area Report	Gridding Report	Grid Information Report	Variogram Data Report
	Click *Report* in the Cross Validation dialog	Click the *Statistics* button in the Grid Data dialog	Click the **Grids | Calculate | Contour Vol/Area** command	Check the *Grid Report* Option on the **Grid Data** dialog	Click the **Grids | Info | Grid Info** command; or click Options | Grid Info in the **Grid Editor**	In the Variogram Properties click the *Display Statistics* button on the **Statistics** page and then click **Report**
Data Counts, includes Filtered and at Validation Points	X	X		X		X
Filtering, includes Breakline, Duplicate, Exclusion		X		X		
Grid Information, includes Grid Geometry				X	X	
Gridding Rules	X			X		
Inter-Variable Correlation, Rank Correlation and Covariance	X	X		X		X
Output Grid				X		
Principal Component Analysis		X		X		

Polygon Used for Statistics					X	
Regression, includes Planar: Z=AX+BY+C and Residual: R=AX+BY +C	X	X				X
Statistics, such as Univariate, Nearest Neighbor, and Cross-Validation	X	X		X	X	X
Variogram Grid						X
Volume and Area			X			
Z Transform, includes Grid and Data				X		

Information Contained in Each Report Section

Each section of the report contains information about the grid, data, or variogram.

Time Stamp

Time of report	Date and time the report was created in Mon Oct 14 10:43:13 2013 format
Elapsed time to create grid	Seconds required when gridding. Only included in the Gridding Report.

Data Source

Source Data File Name	path and file name of the data used in gridding
X Column	X data column
Y Column	Y data column
Z Column	Z data column
Detrending	variogram data detrending method selected on the General page in the **New Variogram** dialog. Only included in the Variogram Grid Report.

Data Counts and Filtered Data Counts

Active Data	number of data after applying filters

Original Data	number of original data points (excludes breakline data)
Excluded Data	number of data excluded by the *Data Exclusion Filter* - the excluded data are listed in the *Exclusion Filtering* section
Deleted Duplicates	number of duplicates deleted by the *Duplicate Data* filter - the deleted duplicates are listed in the *Duplicate Filtering* section
Retained Duplicates	number of duplicates retained by the *Duplicate Data* filter (this statistic is not computed if the duplicate rule is ALL) - the retained duplicates are listed in the *Duplicate Filtering* section including any artificial data
Artificial Data	number of artificial data created by the *Sum*, *Average*, and *Midrange Duplicate Data* filters
Superseded Data	Superseded data are number of data eliminated by breaklines in the Data Statistics Report and the Gridding Report. Breakline data always supersede point data. If point data are on, or in the immediate vicinity of, breakline data the point data are eliminated.

Data Counts at Validation Points

The *Data Counts at Validation Points* section is only included in the Cross Validation Report.

Active Results	locations at which the cross validation interpolation was successfully carried out
NoData Results	The NoData results are the locations at which cross validation interpolation was attempted, but was not successful. For example, the natural neighbor gridding algorithm can only interpolate at locations within the convex hull of the active data. As such, an observation that lies on the convex hull of the original, complete, data set will lie outside of the convex hull of the active data when that observation is the cross validation point. Cross validation is not possible using the natural neighbor algorithm at such a point, so it is assigned the NoData value.
Attempted Results	reports the number of locations at which cross validation interpolation was attempted
Requested Results	contains the original number of random data

Z Data Transform

Includes the transformation method (if any) applied to the Z values. Lists the data that was unable to be transformed in a table.

The rest of the report information is calculated using the active data, including any artificial data generated by duplicate filtering. Excluded, deleted, or superseded data are not included in the following calculations.

Exclusion Filtering

Exclusion Filter String	shows the Data Exclusion Filterstring
Excluded Data	number of data excluded by the *Data Exclusion Filter* -
Excluded Data Table	the excluded data are listed in a table. The *ID* is equal to the line number in the original data file. This list is 100 data rows long by default.

Duplicate Filtering

Duplicate Points to Keep	*To Keep* filter used
X Duplicate Tolerance	maximum X spacing of points to be considered a duplicate
Y Duplicate Tolerance	maximum Y spacing of points to be considered a duplicate
Deleted Duplicates	number of duplicates deleted by the *Duplicate Data* filter - the deleted duplicates are listed in the **Duplicate Filtering** section
Retained Duplicates	number of duplicates retained by the *Duplicate Data* filter (this statistic is not computed if the duplicate rule is ALL) - the retained duplicates are listed in the **Duplicate Filtering** section including any artificial data
Artificial Data	number of artificial data created by the *Sum*, *Average*, and *Midrange Duplicate Data* filters
Duplicate Data Table	the duplicate data table lists all of the duplicate points with X, Y, Z, ID, and Status. The *ID* is equal to the line number in the original data file. When the status is artificial, no *ID* is given since this data does not come from the original data file. The *Status* (*Retained, Deleted,* or *Artificial*) reports how the duplicate was handled. This list is 100 data rows long by default.

Breakline Filtering

When breaklines are used, data that is within the *X Tolerance* and *Y Tolerance,* as set in the Filter dialog, of the breakline are deleted due to breakline data superseding original data.

Anisotropy Angle	the anisotropy angle reported for the default variogram

Anisotropy Ratio	the anisotropy ratio reported for the default variogram
X Tolerance	maximum X spacing of points to be considered a duplicate
Y Tolerance	maximum Y spacing of points to be considered a duplicate
Superseded Data	Superseded data are number of data eliminated by breaklines in the Data Statistics Report and the Gridding Report. Breakline data always supersede point data. If point data are on, or in the immediate vicinity of, breakline data the point data are eliminated.
Breakline Data Table	the breakline data table lists all of the superseded data points with X, Y, Z, ID, and Status. The *ID* is equal to the line number in the original data file. The *Status* (*Retained, Deleted,* or *Artificial*) reports how the duplicate was handled. This list is 100 data rows long by default.

Inter-Variable Correlation

The Inter-Variable Correlation table shows the correlation between the X, Y, and Z variables. The Cross Validation Report also contains *Estimated Z* and *Residual Z* columns and rows. The correlations are computed with

$$\text{Correlation}(X,Y) = \frac{\text{covariance}(X,Y)}{(\text{standard deviation}(X))(\text{standard deviation}(Y))}$$

The correlation is positive when both variables increase or decrease together. The correlation is negative when one variable increases while the other variable decreases. A correlation of zero shows that there is no linear relationship between the variables.

Inter-Variable Covariance

The Inter-Variance Covariance table shows the covariance between the X, Y, and Z variables. The covariances are computed with

$$\text{Covariance}(X,Y) = \frac{1}{N}\sum_{i=1}^{N}(X_i - \text{mean}(X))(Y_i - \text{mean}(Y))$$

The covariance is positive if, on average, the variables are both above the mean. The covariance is negative if one variable is above the mean and the other variable is below the mean.

Inter-Variable Rank Correlation

The Inter-Variance Rank Correlation table shows the rank correlation between the X, Y, and Z variables. The data is ordered and then assigned a rank value from 1 to the count of values. Rank values range from -1 to +1. The correlation is positive when both variables increase or decrease together. The correlation is negative when one variable increases while the other variable decreases. A correlation of zero shows that there is no linear relationship between the variables.

Univariate Statistics

This group of statistics shows information for X, Y, and Z data. These statistics do not include breakline data.

Count	total number of points
1%-tile	1 percent of the values are smaller than this number and 99 percent of the values are larger
5%-tile	5 percent of the values are smaller than this number and 95 percent of the values are larger
10%-tile	10 percent of the values are smaller than this number and 90 percent of the values are larger
25%-tile	lower quartile; 25 percent of the values are smaller than this number and 75 percent of the values are larger
50%-tile	middle data value, 50 percent of the data values are larger than this number and 50 percent of the data are smaller than this number
75%-tile	upper quartile; 75 percent of the values are smaller than this number and 25 percent of the values are larger than this number
90%-tile	90 percent of the values are smaller than this number and 10 percent of the values are larger
95%-tile	95 percent of the values are smaller than this number and 5 percent of the values are larger
99%-tile	99 percent of the values are smaller than this number and 1 percent of the values are larger
Minimum	minimum value
Maximum	maximum value
Mean	arithmetic average of the data $Mean = \frac{1}{N}\sum_{i=1}^{N} X_i$
Median	middle data value, 50 percent of the data values are larger than this number and 50 percent of the data are smaller than this number
Geometric Mean	geometric mean of the data
Harmonic Mean	harmonic mean of the data
Root Mean Square	square root of the mean square

Trim Mean (10%)	Trim Mean is the mean without the upper five percent and lower five percent of the data, therefore, extreme value influence is removed. If there are fewer than 20 data points, the minimum and maximum data points are removed instead of the upper and lower five percent.
Interquartile Mean	interquartile mean, or midmean, is a truncated mean using only the data in the second and third quantiles (all data beween the 25%%-tile and 75%%-tile)
Midrange	the value halfway between the minimum and maximum values Midrange = (Minimum + Maximum) / 2
Winsorized Mean	Winsorized mean is a truncated mean. This method replaces the extreme highs and lows values with a more central value. This mean is less sensitive to outliers.
TriMean	the trimean, or Tukey's trimean, is a measure of probability distribution location. This is equivalent to the the sum of (quartile 1, 2 times the quartile 2, and quartile 3) divided by four.
Variance	$Variance = \frac{1}{N}\sum_{i=1}^{N}(X_I - Mean)^2$
Standard Deviation	square root of the variance
Interquartile Range	separation distance between the 25%-tile and 75%-tile this shows the spread of the middle 50 percent of the data, similar to standard deviation, though this statistic is unaffected by the tails of the distribution
Range	separation between the minimum and maximum value Range = Maximum - Minimum
Mean Difference	the mean or average of the absolute difference of two random variables X and Y.

Median Abs. Deviation	Median Absolute Deviation is the median value of the sorted absolute deviations. It is calculated by 1. computing the data's median value 2. subtracting the median value from each data value 3. taking the absolute value of the difference 4. sorting the values 5. calculating the median of the values
Average Abs Deviation	Average Absolute Deviation is the average value of the sorted absolute deviations. It is calculated by 1. computing the data's average mean value 2. subtracting the mean value from each data value 3. taking the absolute value of the difference 4. calculating the average value
Quartile Dispersion	Measures dispersion of the data using: (Quartile 3 - Quartile 1)/(Quartile 3 + Quartile 1)
Relative Mean Difference	The mean difference of the entire data set divided by the sample mean of the data set
Standard Error	The standard error of the mean is the standard deviation of those sample means over all possible samples drawn from the population. This is calculated by dividing the standard deviation by the square root of the number of samples.
Coef. of Variation	The Coefficient of Variation is calculated by dividing the standard deviation by the mean. If a "-1" is reported, the coefficient of variation could not be computed. The coefficient of variation is computed only for the Z values.
Skewness	The Coefficient of Skewness is calculated by $\gamma_1 = \frac{1}{N\,\sigma^3}\sum_{i=1}^{N}(x_i - \mu)^3$ If a "-1" is reported, the coefficient of skewness could not be computed. The coefficient of skewness is computed only for the Z values.

Kurtosis	The Coefficient of Kurtosis is calculated by $$\gamma_2 = \left(\frac{1}{N\,\sigma^4} \sum_{i=1}^{N} (x_i - \mu)^4 \right)$$
Sum	the sum of all X, Y, or Z values
Sum Absolute	the absolute value of the sum of all X, Y, or Z values
Sum Squares	the sum of all squared X, Y, or Z values
Mean Square	$$\mathrm{MeanSquare} = \frac{1}{N} \sum_{i=1}^{N} X_i^2$$

Planar Regression

Planar regression is an ordinary least-squares fit where Z=AX+BY+C.

- The *Parameter Values* are the A, B, and C values.
- The *Standard Error* is the estimated standard deviation of the parameters.
- The *Inter-Parameter Correlations* are the correlation between A, B, and C coefficients.
- The *ANOVA Table* shows regression statistics on the planar fit where *df* are the degrees of freedom and *F* is the ratio of the mean squares.
- The *Coefficient of Multiple Determination (R*2) is calculated with

$$R^2 = 1 - \frac{\Sigma(Z_i - AX_i - BY_i - C)^2}{\Sigma(Z_i - \bar{Z})^2} = 1 - \frac{SSE}{SST}$$

For the Cross Validation Report, the planar regression is the residual regression at the validation points.

Nearest Neighbor Statistics

The nearest neighbor statistics represent aspects of the data values and of the data locations. The nearest neighbor to a data point uses a simple separation distance without taking anisotropy into account. If two or more points tie as the nearest neighbor, the tied data points are sorted on X, then Y, then Z, and then ID. The smallest value is selected as the nearest neighbor.

The *Separation* column shows the separation distances between the observation and its nearest neighbor. The |*Delta Z*| column shows the absolute values of the differences between the observation Z value and the nearest neighbor Z value.

The statistics are the same as the Univariate Statistics (see above).

The *Nearest Neighbor Statistics* also includes the *Complete Spatial Randomness* section. The *Complete Spatial Randomness* statistics measure how random locations are in space. **Surfer** does not correct for edge effects so the statistics may be biased.

Lamda	is the average spatial density $\lambda = \frac{N}{(Range(X))(Range(Y))}$
Clark and Evans	$= \frac{2\sqrt{\lambda}}{N}\sum S_i$ where λ = average spatial density Si= separation distance between the observation and the nearest neighbor The distribution of this statistic is normal, with a mean equal to one and a variance of $\frac{(4-\pi)}{(N\pi)}$ See Clark and Evans (1954) and Cressie (1991) for more information.
Skellam	$= 2\pi\lambda\sum S_i^2$ where λ = average spatial density Si= separation distance between the observation and the nearest neighbor The distribution is Chi-Squared with 2N degrees of freedom. See Skellam (1952) and Cressie (1991) for more information.

Principal Component Analysis

Principal component analysis (PCA) is a mathematical procedure that uses orthogonal transformation to convert a set of observations of possibly correlated variables into a set of values of linearly uncorrelated variables called principal components. The number of principal components is less than or equal to the number of original variables. The principal components are calculated for the X, Y, and Z values. A λ value is also reported for each principal component.

Variogram Grid

Max Lag Distance	*Max Lag Distance* set on the General page in the **New Variogram** dialog
Angular Divisions	number of *Angular Divisions* set on the General page in the **New Variogram** dialog
Radial Divisions	number of *Radial Divisions* set on the General page in the **New Variogram** dialog

Output Grid

Grid File Name	name of the output grid file
Grid Size	number of rows and columns in the grid
Total Nodes	number of columns times the number of rows
Filled Nodes	number of grid nodes containing interpolated values
NoData Nodes	number of grid nodes containing the NoData value
NoData Value	reports the Z value associated with NoData nodes
X Minimum	minimum X grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
X Maximum	maximum X grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
X Spacing	X spacing set in the Grid Data dialog
Y Minimum	Minimum Y grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
Y Maximum	Maximum Y grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
Y Spacing	Y spacing set in the Grid Data dialog

Grid Information

Grid File Name	name of the output grid file
Grid Size	number of rows and columns in the grid
Total Nodes	number of columns times the number of rows
Filled Nodes	number of grid nodes containing interpolated values
NoData Nodes	number of grid nodes containing the NoData value
NoData Value	reports the Z value associated with NoData nodes

Grid Geometry

X Minimum	minimum X grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
X Maximum	maximum X grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
X Spacing	X spacing set in the Grid Data dialog

Y Minimum	Minimum Y grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
Y Maximum	Maximum Y grid line value specified in the *Output Grid Geometry* group in the Grid Data dialog
Y Spacing	Y spacing set in the Grid Data dialog

Polygon used for statistics

Side	Whether statistics were calculated for values Inside or Outside the polygon(s)
Polygons	Number of polygons used for calculating statistics
Number of points	Total number of vertices among the polygon(s)

Gridding Rules

This section displays the gridding method used, as well as the option settings for each gridding method.

Univariate Grid Statistics

The Univariate Grid Statistics are the same as those reported in the Univariate Statistics and Nearest Neighbor Statistics sections.

Univariate Cross-Validation Statistics

The Univariate Cross Validation Statistics section are the same as those reported in the Univariate Statistics. It also contains an additional column of data, called *Data Used*. This column shows the number of data points used in the calculation.

Volumes and Areas

Volume and area calculations are based on the Simple level method for contour line properties. Minimum and maximum contours along with the contour interval must be specified. Surface area and planar area calculations are made above, below, and between contours. Contour volume calculations are made above, below, and between contours.

Search

Search options control which data points are considered by the gridding operation when interpolating grid nodes. To access the search options, if available with the gridding method, click the *Advanced Options* button in the Grid Data dialog. If searching is available with the selected gridding method, a *Search Neighborhood* section appears in the Grid Data Advanced Options dialog. If search options are not available for a gridding method this means that all the data points from the data file must be used when calculating the grid.

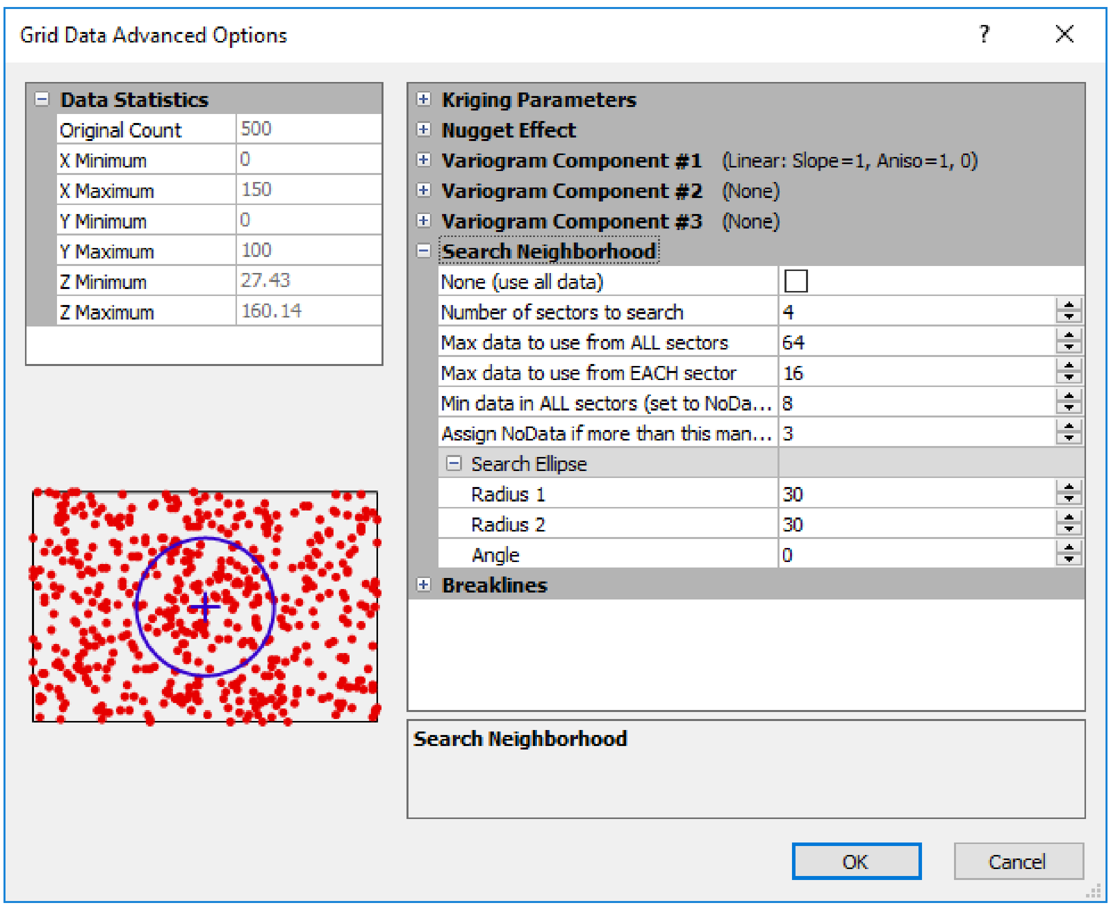
Search Rules

Search rules define the number of points included in interpolating a grid node value. Clear the *None (Use all data)* option to enable the search rules edit controls. Search rules limit the number of data points to include in the interpolation at each grid node. Search rules work in concert with the *Search Ellipse*. The *Search Ellipse* specifies the size of the local neighborhood in which to look for data, and the search rules specify the number of points to actually consider within the neighborhood.

If the number of data points defined by the *Min data in ALL sectors* is not found within the *Search Ellipse* distance, the NoData value is assigned at the grid node. NoData values indicate that insufficient data existed to satisfy the search criteria at that particular location. Grid nodes assigned the NoData value truncate contour lines on contour maps, and produce low flat regions on wireframes.

Search Neighborhood

To access the search options, if available with the gridding method, click the *Advanced Options* button in the Grid Data dialog and expand the *Search Neighborhood* section of the **Grid Data Advanced Options** dialog.



Specify the search options for the selected gridding method in the ***Search Neighborhood*** *section.*

No Search

The None (use all data) option tells **Surfer** to use all data when interpolating each grid node. Clear the *None (use all data)* box to activate the search during the gridding process. If the search is enabled, the following search options can be specified.

Number of Sectors to Search

The *Number of sectors to search* option divides the search area into smaller sections to which you can apply the following three search rules. You can specify up to 32 search sectors.

Maximum Number of Data to Use from ALL Sectors

The *Max data to use from ALL sectors* value limits the total number of points used when interpolating a grid node.

Maximum Number of Data to Use from EACH Sector

The *Max data to use from EACH sector* value specifies the maximum number of points to be used from each sector. Data points beyond the nearest points in a sector are ignored even if the data points in another sector are farther from the grid node being calculated.

Minimum Number of Data in All Sectors

The *Min data in ALL sectors (set to NoData if fewer)* value assures that the specified number of points are encountered when interpolating a grid node. If the minimum number of points is not found, the NoData value is assigned at the grid node.

Assign NoData to Node if More Than This Many Sectors are Empty

Assign NoData if more than this many sectors are empty assures that if more empty sectors than the specified value are encountered, the NoData value is assigned at the grid node.

Search Ellipse

Search ellipses are specified by defining the ellipse radii and the angle for the ellipse.

Radius 1 and 2

Radius 1 and *Radius 2* are positive values indicating the distance in data units. The *Radius 1* and *Radius 2* values are the length that is searched in the direction indicated by the *Angle*. For all gridding methods except *Nearest Neighbor* and *Modified Shepard's Method*, the default values are calculated by taking half the distance of the diagonal.

Angle

Angle is the inclination between the positive X axis and the ellipse axis associated with *Radius 1*. This can be any value between -360 and +360 degrees.

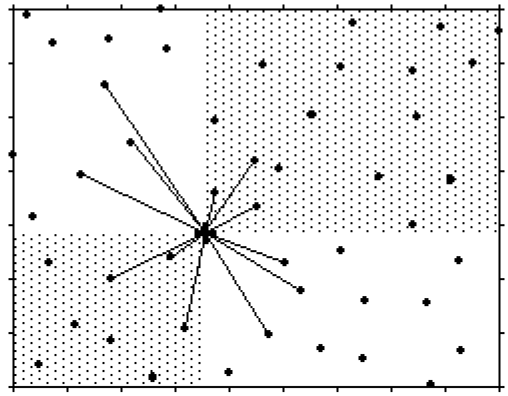
No Search

The search options tell **Surfer** how to find data points during the calculation of grid node values. The *No Search (use all of the data)* option tells **Surfer** to use all data when interpolating each grid node. Uncheck the *No Search* box to activate the search during the gridding process.

For small data sets (up to 250 points) the *No Search* option is usually the most appropriate. This type of search increases gridding speed. *No Search* uses all data points in the calculation of every grid node. The distance weighting factors are still applied. Therefore, although a point far removed from the grid node is still used when calculating the grid node value, it carries relatively little weight compared to data points close to the grid node.

When data points are evenly distributed over the map area, the *No Search* option is adequate. When observations are heavily clustered within the map area, a four-sector or eight-sector search is recommended. These types of searches are also appropriate when you have data collected on widely spaced traverses. A one-sector search might attempt to estimate grid nodes using data points from a single direction. This might generate unrealistic slopes between traverses, and unrealistic polygonal shaped plateaus across the map area. Four- or eight- sector searches should eliminate or reduce this effect. Up to 32 sectors can be specified.

Some gridding methods construct an internal matrix based on the number of search points (such as *Kriging* and *Radial Basis Function*). This matrix can consume a large amount of memory if too many search points are used. For these methods, the default cut off for using all data (*No Search*) versus searching is 250 data points. If there are more than 250 points, **Surfer** defaults to performing a search. If there are 250 or fewer data points, **Surfer** selects the *No Search (use all of the data)* option by default. The absolute maximum number of data points that can be used with the *No Search (use all of the data)* option with these methods is 10,000. If you have more than 10,000 points the *No Search (use all of the data)* option is disabled.



This is an example of a four-sector search. The three nearest points in each quadrant are included during gridding in this example.

Search Ellipse

The *Search Ellipse* defines the local neighborhood of points to consider when interpolating each grid node. This defines the distance in data units from the grid node that **Surfer** looks to find data points when calculating grid nodes. Data points outside the search ellipse are not considered during grid node interpolation.

Elliptical searches do not impart extra weight to data points in the various directions, but do search farther along one ellipse axis. The default *Search Ellipse* is circular; meaning that **Surfer** looks the same distance in all directions.

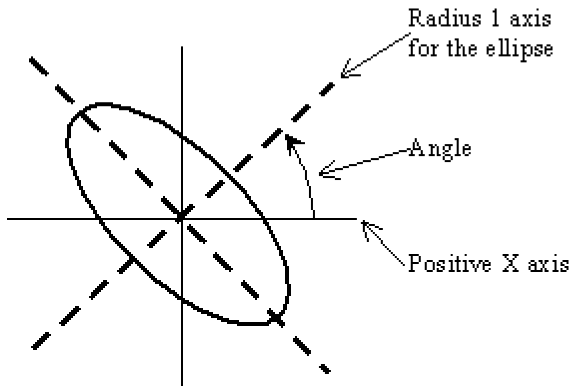
Search ellipses are specified by defining the ellipse radii and the angle for the ellipse.

Radius 1 and 2

Radius 1 and *Radius 2* are positive values indicating the distance in data units. The *Radius 1* and *Radius 2* values are the length that is searched in the direction indicated by the *Angle*. For all gridding methods except *Nearest Neighbor* and *Modified Shepard's Method*, the default values are calculated by taking half the distance of the diagonal.

Angle

Angle is the inclination between the positive X axis and the ellipse axis associated with *Radius 1*. This can be any value between -360 and +360 degrees.

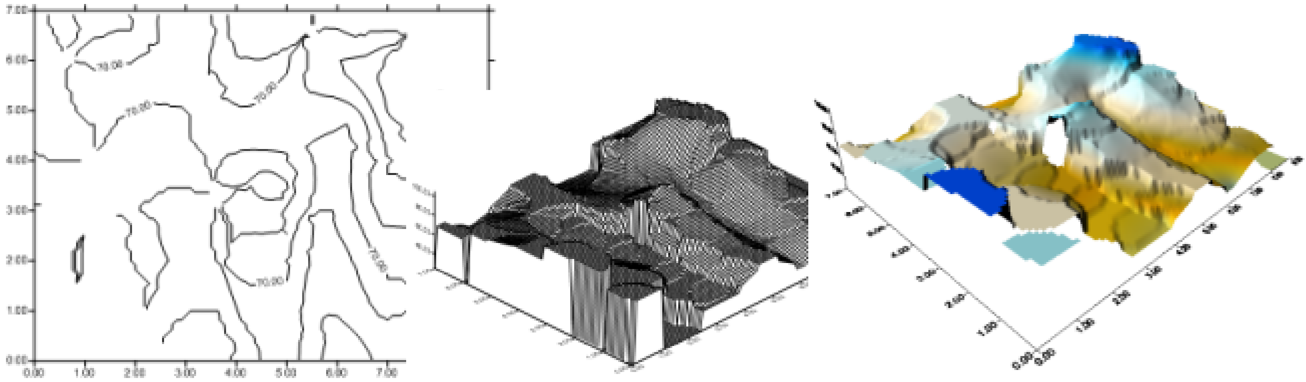


The search ellipse angle is the angle between the positive X axis and the ellipse axis associated with Radius 1.

Radius 1 = 2
Radius 2 = 1
Angle = 0
is the same ellipse as
Radius 1 = 1
Radius 2 = 2
Angle = 90

NoData Values

NoData values indicate that insufficient data existed to generate a grid node value at that particular location based on the specified search rules. For example, if you inadvertently set your search ellipse size to be smaller than half the distance between your data points, a significant number of grid nodes may be assigned the NoData value in the grid file. Grid nodes assigned the NoData value cause truncation of contour lines within contour maps, and flat regions on wireframes. Also, NoData values are assigned outside of the convex hull of the data for non-extrapolating gridding methods, like *Triangulation with Linear Interpolation* and *Natural Neighbor* and when the *Assign NoData outside convex hull of data* box is checked in the **Grid Data** dialog.



When a search ellipse is set too small, the NoData value is assigned to the grid nodes where not enough data were found within the search ellipse. This can result in an incomplete contour map, with truncation of contour lines, or a wireframe or surface map with pronounced discontinuities.

Anisotropy (Optional)

Although not required, in most cases it works well to set the search ellipse ratio and direction to coincide with the anisotropy ratio and direction. For more information on anisotropy, see Anisotropy.

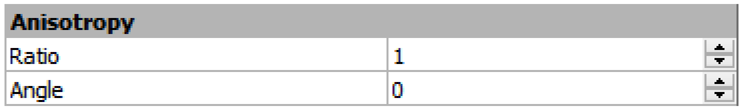
Anisotropy

Natural phenomena are created by physical processes. Often these physical processes have preferred orientations. For example, at the mouth of a river the coarse material settles out fastest, while the finer material takes longer to settle. Thus, the closer one is to the shoreline the coarser the sediments while the further from the shoreline the finer the sediments. When interpolating at a point, an observation 100 meters away but in a direction parallel to the shoreline is more likely to be similar to the value at the interpolation point than is an equidistant observation in a direction perpendicular to the shoreline. Anisotropy takes these trends in the data into account during the gridding process.

Usually, points closer to the grid node are given more weight than points farther from the grid node. If, as in the example above, the points in one direction have more similarity than points in another direction, it is advantageous to give points in a specific direction more weight in determining the value of a grid node. The relative weighting is defined by the anisotropy ratio. The underlying physical process producing the data as well as the sample spacing of the data are important in the decision of whether or not to reset the default anisotropy settings. Anisotropy is also useful when data sets use fundamentally different units in the X and Y dimensions. See below for examples.

Detailed discussions of anisotropy and *Kriging* as well as anisotropy equations are given in the variogram help topics.

The *Anisotropy* options are displayed in the Options page of the **Grid Data** dialog when a gridding method that supports anisotropy is selected..



*Set the anisotropy options in the **Grid Data Options** dialog.*

Ratio

The *Ratio* is the maximum range divided by the minimum range. An anisotropy ratio less than two is considered mild, while an anisotropy ratio greater than four is considered severe. Typically, when the anisotropy ratio is greater than three the effect is clearly visible on grid-based maps.

Unless there is a good reason to use an anisotropy ratio, you should accept the default value of 1.0.

Angle

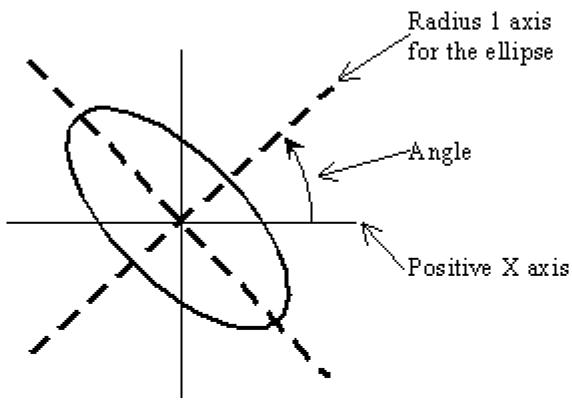
The *Angle* is the preferred orientation (direction) of the major axis in degrees. 0 degrees is defined as east-west orientation. 90 degrees is defined as north-south orientation. Angles rotate counterclockwise.

Ellipse

In the most general case, anisotropy can be visualized as an ellipse. The ellipse is specified by the lengths of its two orthogonal axes and by an orientation angle. In **Surfer**, the lengths of the axes are called Radius 1 and Radius 2. The orientation angle is defined as the counterclockwise angle between the positive X axis and Radius 1. Since the ellipse is defined in this manner, an ellipse can be defined with more than one set of parameters.

For example:

Radius 1 = 2
Radius 2 = 1
Angle = 0
is the same ellipse as
Radius 1 = 1
Radius 2 = 2
Angle = 90



The anisotropy angle is the angle between the positive X axis and the ellipse axis associated with Radius 1.

For most of the gridding methods in **Surfer**, the relative lengths of the axes are more important than the actual length of the axes. The relative lengths are expressed as a *Ratio* in the *Anisotropy* section. The ratio is defined as Radius 1 divided by Radius 2. Using the examples above, the ratios are 2 and 0.5. The ratio of 2 indicates that Radius 1 is twice as long as Radius 2. The *Angle* is the counterclockwise angle between the positive X axis and Radius 1. This means that:

Ratio = 2
Angle = 0
is the same ellipse as
Ratio = 0.5
Angle = 90

The small picture in the *Anisotropy* group displays a graphic of the ellipse to help describe the ellipse.

Example 1: Plotting a Flood Profile Along a River

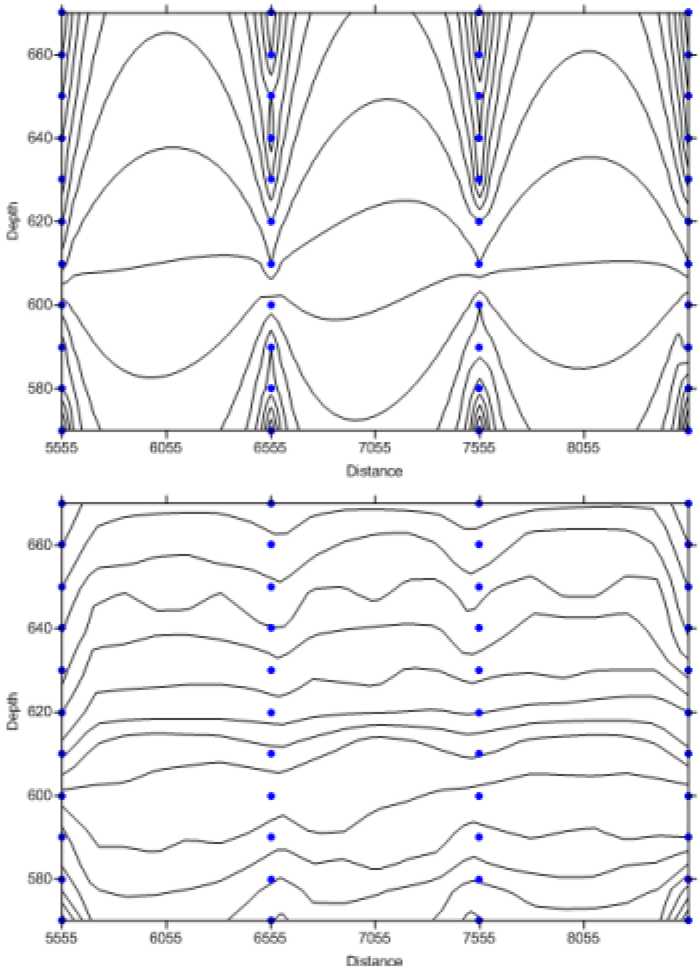
For an example when data sets use fundamentally different units in the X and Y directions, consider plotting a flood profile along a river. The X coordinates are locations, measured in miles along the river channel. The Y coordinates are time, measured in days. The Z values are river depth as a function of location and time. Clearly in this case, the X and Y coordinates would not be plotted on a common scale, because one is distance and the other is time. One unit of X does not equal one unit of Y. While the resulting map can be displayed with changes in scaling, it may be necessary to apply anisotropy as well.

Example 2: Isotherm Map of Average Daily Temperature

Another example of when anisotropy might be employed is an isotherm map (contour map) of average daily temperature over the upper Midwest. Although the X and Y coordinates (easting and northing) are measured using the same units, along the east-west lines (X lines) the temperature tends to be very similar. Along north-south lines (Y lines) the temperature tends to change more quickly (getting colder as you head north). When gridding the data, it would be advantageous to give more weight to data along the east-west axis than along the north-south axis. When interpolating a grid node, observations that lie in an east-west direction are given greater weight than observations lying an equivalent distance in the north-south direction.

Example 3: Oceanographic Survey to Determine Water Temperature at Varying Depths

A final example where an anisotropy ratio is appropriate is an oceanographic survey to determine water temperature at varying depths. Assume the data are collected every 1000 meters along a survey line, and temperatures are taken every ten meters in depth at each sample location. With this type of data set in mind, consider the problem of creating a grid file. When computing the weights to assign to the data points, closer data points get greater weights than points farther away. A temperature at 10 meters in depth at one location is similar to a sample at 10 meters in depth at another location, although the sample locations are thousands of meters apart. Temperatures might vary greatly with depth, but not as much between sample locations.



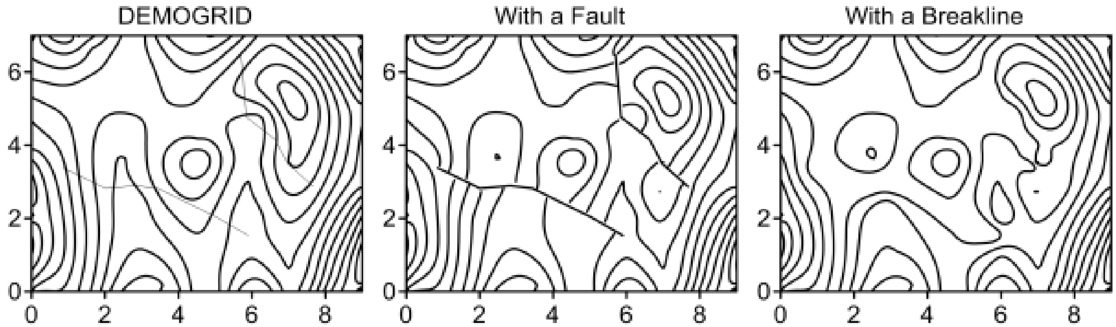
In the oceanographic survey described here, the contour lines cluster around the data points when an anisotropy ratio is not employed. In the lower contour map, an anisotropy ratio results in contour lines that are a more accurate representation of the data.

Breaklines and Faults

Breaklines and faults are a means to show discontinuities in the surface. Select gridding methods support breaklines and/or faults. Breaklines and faults are defined with Golden Software Blanking .BLN files.

If your grid is not dense enough, the breakline or fault will not show very well in the map. If you cannot see any indication of the breakline or fault (i.e. contours do not bend properly), regrid the data with a denser grid.

To include the breakline or fault as a line on your map, select the map and use the **Home | Add to Map | Layer | Base**. If the grid file used to create a contour map contains a fault, the contour properties General page has a *Fault Line section*, so you do not need to use a base map to show faults on contour maps.



The map on the left is created from demogrid.dat *using default gridding settings. The center map is created with two fault lines. The right map is created with breaklines.*

Using Breaklines and Faults when Gridding

To reach the **Breaklines and Faults** page, click the *Advanced Options* button in the Grid Data dialog, then expand the *Breaklines* and/or *Faults* section in the **Grid Data Advanced Options** dialog. The **Breaklines and Faults** page contains two fields for entering the file names of the faults and breaklines to use when gridding. If faults are not available with the selected gridding method, only breaklines are available. The page is then named **Breaklines**.

- Click the button next to *File Containing Breaklines* to select the blanking file .BLN containing the breaklines. In the **Open** dialog, specify the blanking file and click *Open*. The blanking file will be displayed in the *File Containing Breaklines.* The number of traces and the number of total vertices are displayed under the file name. Click the *Clear* button to delete the file name to exclude the breaklines from the gridding process. Breaklines must contain 3 columns: X, Y, and Z. If the Z column is missing, the .BLN file cannot be used as a breakline.
- Click the button next to *File Containing Fault Traces* to select the blanking file .BLN containing the fault traces. In the **Open** dialog, specify the blanking file and click *Open*. The blanking file will be displayed in the *File Containing Fault Traces.* The number of traces and the number of total vertices are displayed under the file name. Click the *Clear* button to delete the file name to exclude the fault traces from the gridding process.

Faults

Faults are used to show discontinuity when gridding, similar to *breaklines*. A fault is a two-dimensional boundary file defining a line acting as a barrier to information flow when gridding. When gridding a data set, data on one side of a fault is not directly used when calculating grid node values on the other side of the fault.

If the fault line is a closed polygon, the gridding algorithm will grid the data on the side of the polygon where the data are located. If the fault line is not a closed polygon, the gridding algorithm can search around the end of the fault to see a point on the other side of the fault, but this longer distance reduces the weight of the point in interpolating the grid node value. If a point lies directly on the fault line, random round-off error determines which side of the fault captures the point.

The following gridding methods support faults:

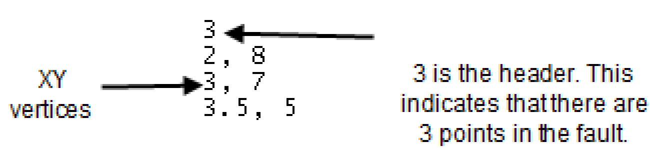
- Inverse Distance to a Power
- Minimum Curvature
- Nearest Neighbor
- Data Metrics

You can create a blanking file to define a fault in the **Surfer** worksheet or any text editor. Enter a header containing the number of vertices in the breakline, followed by the X, Y coordinates of each vertex, one per line.

Faults consume memory and increase gridding time in proportion to the square of the number of fault segments. In addition, they may cause some gridding methods to fail. With *Minimum Curvature*, the failure is due to lack of convergence. The only grid file format that retains fault information is the Surfer GRD grid file. If you use another grid file format, the faulting information is lost. If the grid is modified, the fault is removed.

Fault Example

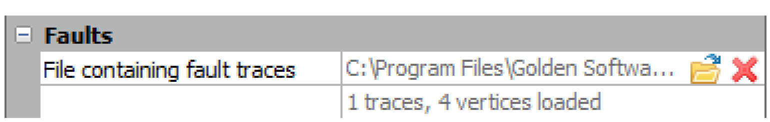
An example of a fault .BLN format is:



A fault is defined by X and Y values in a .BLN file.

Using Faults when Gridding

The faults options are displayed in the *Faults* section of the **Grid Data** dialog **Options** page when the selected interpolation method supports faults.

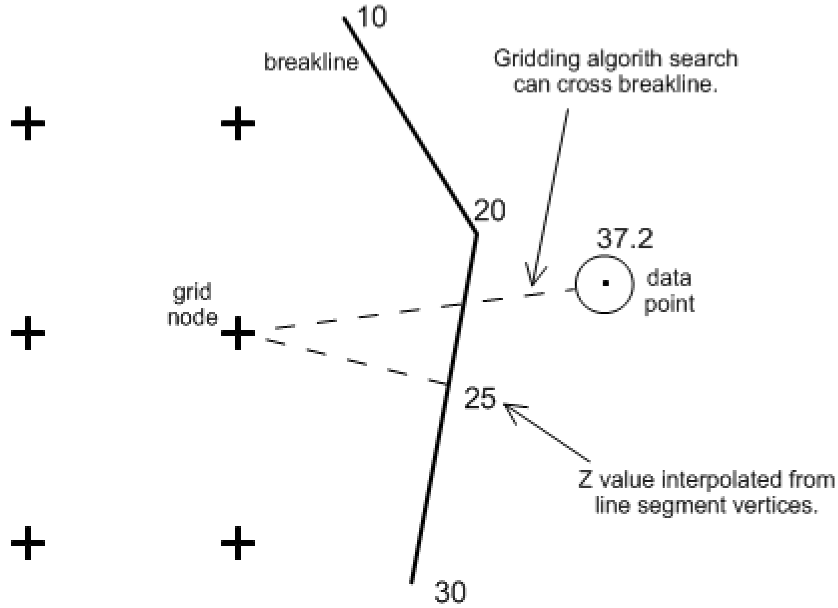


The Faults section is displayed when an interpolation method supports faults.

Click the button next to *File containing fault traces* to select the blanking file BLN containing the fault traces. In the **Open** dialog, specify the blanking file and click *Open*. The blanking file will be displayed in the *File containing fault traces* field. The number of traces and the number of total vertices are displayed under the file name. Delete the file name to exclude the fault traces from the interpolation process. Note that .TXT files can be used to define faults, but the data must be formatted similarly to the .BLN format.

Breaklines

Breaklines are used when gridding to show discontinuity in the grid. A breakline is a three-dimensional boundary file that defines a line with X, Y, and Z values at each vertex. When the gridding algorithm sees a breakline, it calculates the Z value of the nearest point along the breakline, and uses that value in combination with nearby data points to calculate the grid node value. **Surfer** uses linear interpolation to determine the values between breakline vertices when gridding. Unlike faults, breaklines are not barriers to information flow, and the gridding algorithm can cross the breakline to use a point on the other side of the breakline. If a point lies on the breakline, the value of the breakline takes precedence over the point. Breakline applications include defining streamlines, ridges, and other breaks in the slope.



The gridding algorithm search uses a single Z value at the closest point along the line. The search can cross the breakline to search for data.

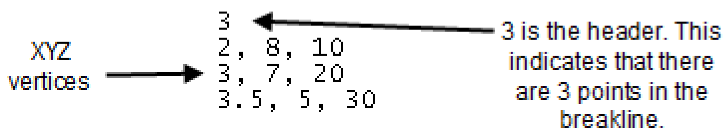
Gridding algorithm searches use the specified anisotropy ratio when determining the distance to the breakline. The use of complex breaklines or a large number of breaklines slows the gridding process significantly. Breaklines cannot cross other breaklines or faults. Breaklines are not allowed with the *No Search (use all of the data)* search method for any gridding method.

The following gridding methods support breaklines:

- Inverse Distance to a Power
- Kriging
- Minimum Curvature
- Nearest Neighbor
- Radial Basis Function
- Moving Average
- Data Metrics
- Local Polynomial

Breakline Example

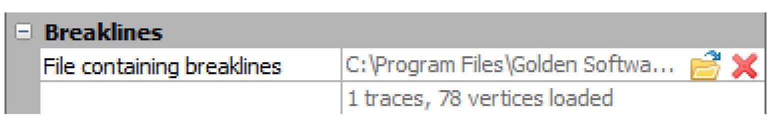
An example of a breakline .BLN format is:



A breakline file contains X, Y, and Z values in the .BLN file.

Using Breaklines when Gridding

The breaklines options are displayed in the *Breaklines* section of the **Grid Data** dialog **Options** page when the selected interpolation method supports breaklines.

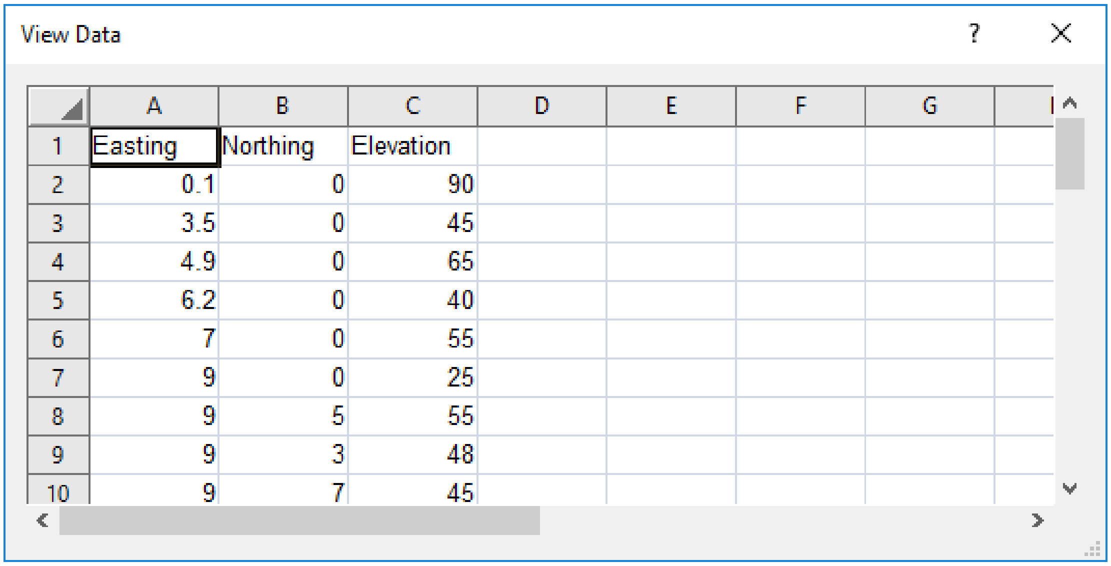


The Breaklines section is displayed when an interpolation method supports breaklines.

Click the button in the *File containing breaklines* field to select the blanking file BLN containing the breaklines. In the **Open** dialog, specify the blanking file and click **Open**. The blanking file will be displayed in the *File containing breaklines* field. The number of traces and the number of total vertices are displayed under the file name. Delete the file name to exclude the breaklines from the interpolation process. Breaklines must contain 3 columns: X, Y, and Z. If the Z column is missing, the .BLN file cannot be used as a breakline. Note that .TXT files can be used to define breaklines, but the data must be formatted similarly to the .BLN format.

View Data

The *View Data* button in the Grid Data and Base from XY Data dialogs displays a preview of the data file in the **View Data** dialog. Use the **View Data**dialogto be sure you are selecting the correct X, Y, and Z data columns in the **Grid Data** or **Base from XY Data** dialog.



	A	B	C	D	E	F	G
1	Easting	Northing	Elevation				
2	0.1	0	90				
3	3.5	0	45				
4	4.9	0	65				
5	6.2	0	40				
6	7	0	55				
7	9	0	25				
8	9	5	55				
9	9	3	48				
10	9	7	45				

*The **View Data** dialog displays a preview of the data file.*

Introduction to Gridding Methods

Grid method parameters control the interpolation procedures. When you create a grid file, you can usually accept the default gridding method and produce an acceptable map. The differences between gridding methods are in the mathematical algorithms used to compute the weights during grid node interpolation. Each method results in a different representation of your data. It is advantageous to test each method with a typical data set to determine the gridding method that provides you with the most satisfying interpretation of your data.

Because **Surfer** maps are created from gridded data, the original data are not necessarily honored in the grid file. When you post the original data points on a contour map, some of the contour lines might be positioned "wrong" relative to the original data. This happens because the locations of the contour lines are determined solely by the interpolated grid node values and not directly by the original data. Some methods are better than others in preserving your data, and sometimes some experimentation (i.e. increasing grid density) is necessary before you can determine the best method for your data.

The gridding method is selected in the **Grid Data** dialog. The **Grid Data** dialog is accessed through the Grids | New Grid | Grid Data command.

Gridding methods include:

- Kriging
- Cokriging
- Inverse Distance to a Power
- Minimum Curvature
- Modified Shepard's Method
- Natural Neighbor
- Nearest Neighbor
- Polynomial Regression
- Radial Basis Function
- Triangulation with Linear Interpolation
- Moving Average
- Data Metrics
- Local Polynomial

The General Gridding Recommendations give a quick overview of each gridding method with some advantages and disadvantages of each.

General Gridding Recommendations

The following list gives you a quick overview of each gridding method and some advantages and disadvantages in selecting one method over another.

Some gridding methods, such as *Natural Neighbor,* do not generate grid node values outside the data by default. To make any gridding method not generate grid node values outside the data, check the *Assign NoData outside convex hull of data* option. Any grid nodes outside the data boundary are automatically assigned the NoData value.

- Inverse Distance to a Power is fast but has the tendency to generate "bull's-eye patterns of concentric contours around the data points.Inverse Distance to a Powerdoes not extrapolate Z values beyond the range of data.
- Kriging is one of the more flexible methods and is useful for gridding almost any type of data set. With most data sets, *Kriging* with the default linear variogram is quite effective. In general, we would most often recommend this method. *Kriging* is the default gridding method because it generates a good map for most data sets. For larger data sets, *Kriging* can be rather slow. *Kriging* can extrapolate grid values beyond your data's Z range.
- Cokriging is a geostatistical gridding method that uses a densely sampled second correlated variable to improve the estimation of a primary variable with the kriging algorithm.
- Minimum Curvature generates smooth surfaces and is fast for most data sets but it can create high magnitude artifacts in areas of no data. The *internal tension* and *boundary tension* allow you control over the amount of smoothing. *Minimum Curvature* can extrapolate values beyond your data's Z range.
- Natural Neighbor generates good contours from data sets containing dense data in some areas and sparse data in other areas. It does not generate data

in areas without data. *Natural Neighbor* does not extrapolate Z grid values beyond the range of data.
- Nearest Neighbor is useful for converting regularly spaced (or almost regularly spaced) XYZ data files to grid files. When your observations lie on a nearly complete grid with few missing holes, this method is useful for filling in the holes, or creating a grid file with the NoData value assigned to those locations where no data are present. *Nearest Neighbor* does not extrapolate Z grid values beyond the range of data.
- Polynomial Regression processes the data so that underlying large-scale trends and patterns are shown. This is used for trend surface analysis. *Polynomial Regression* is very fast for any amount of data, but local details in the data are lost in the generated grid. This method can extrapolate grid values beyond your data's Z range.
- Radial Basis Function is quite flexible. It compares to *Kriging* since it generates the best overall interpretations of most data sets. This method produces a result quite similar to *Kriging*.
- Modified Shepard's Method is similar to *Inverse Distance to a Power* but does not tend to generate "bull's eye" patterns, especially when a smoothing factor is used. *Modified Shepard's Method* can extrapolate values beyond your data's Z range.
- Triangulation with Linear Interpolation is fast. When you use small data sets, *Triangulation with Linear Interpolation* generates distinct triangular faces between data points. *Triangulation with Linear Interpolation* does not extrapolate Z values beyond the range of data.
- Moving Average is most applicable to large and very large data sets (e.g. > 1,000 observations). Moving Average extracts intermediate-scale trends and variations from large, noisy data sets, and it is fast even for very large data sets. This gridding method is a reasonable alternative to Nearest Neighbor for generating grids from large, regularly spaced data sets.
- Data Metrics is used to create grids of information about the data.
- Local Polynomial is most applicable to data sets that are locally smooth (i.e. relatively smooth surfaces within the search neighborhoods). The computational speed of the method is not significantly affected by the size of the data set.

Choosing Methods Based on the Number of XYZ Data Points

The size of your input data set should be considered when selecting a gridding method. For example, some gridding methods interpret small data sets more effectively than others do. **Surfer** requires a minimum of three X, Y, and Z points to perform the gridding process.

- Ten or fewer points are not enough to define more than a general trend in your data. Triangulation with Linear Interpolation and Moving Average are not effective with few points. As with most data sets, Kriging and Radial Basis Function methods will produce the best representation of your data in this situation. If you want only to define the trend of the data, you can use Polynomial Regression. With 10 or fewer points, gridding is extremely fast, so you might want to try the different methods to determine the most effective method for your data.

- With small data sets (<250 observations), *Kriging* with the default linear variogram, or Radial Basis Function with the *multiquadric* function produce good representations of most data sets.

- With moderate-sized data sets (from 250 to 1000 observations), Triangulation with Linear Interpolation is fast and creates a good representation of your data. Although Kriging or Radial Basis Function generate the grids more slowly, they also produce good data representations.

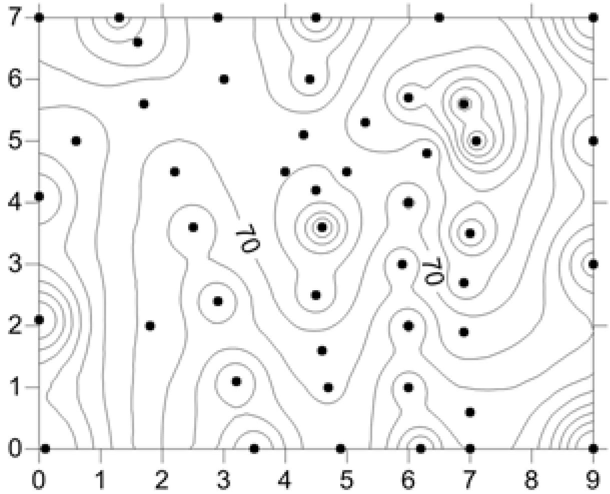
- For large data sets (>1000 observations), both Minimum Curvature and Triangulation with Linear Interpolation are quite fast, and both produce good representations. As with most other data sets, Kriging or Radial Basis Function probably produce the best maps but are quite a bit slower.

- Using Kriging or Radial Basis Function with large data sets does not result in significantly different gridding times. For example, if your data file contains 3,000 or 30,000 data points, the gridding time with *Kriging* and *Radial Basis Function* is not significantly different. Either data set (i.e. 3,000 or 30,000 data points) might take a considerable amount of time to grid, but the two gridding methods (i.e. *Kriging* and *Radial Basis Function*) will take approximately the same amount of time.

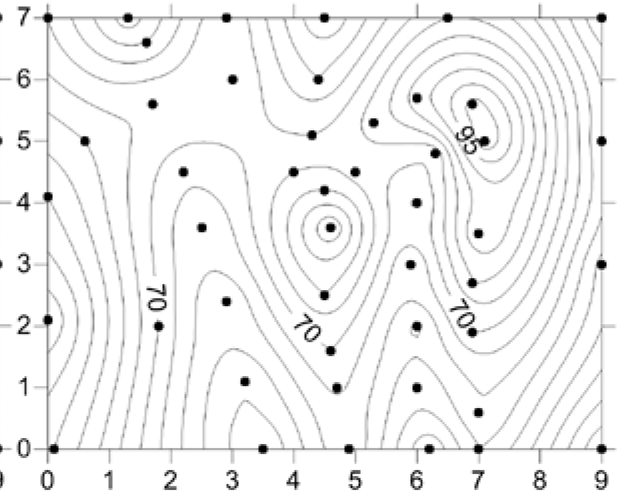
Gridding Method Comparison

It is recommended that you try each of the different gridding methods, accepting the defaults, in much the same fashion as you have seen here. This gives you a way to determine the best gridding method to use with the same data set.

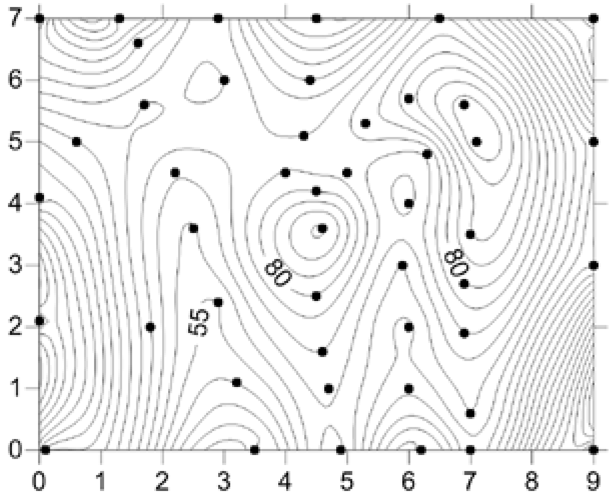
In this example, we will use the *Demogrid.dat* file located in the Samples folder. Each image shows the contour and post map combination created from the default settings for each of the various gridding methods.



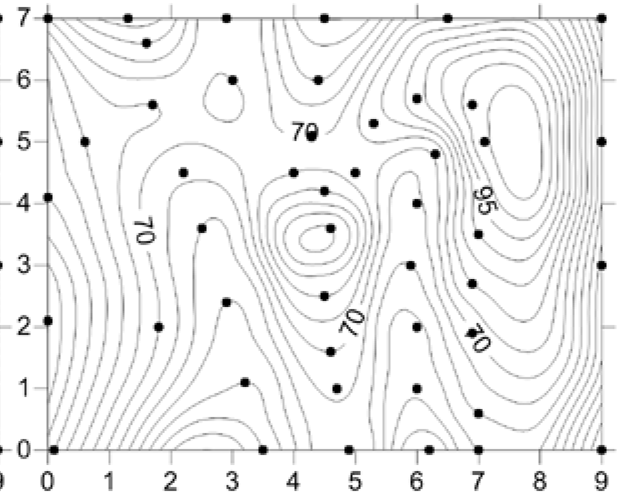
Inverse Distance to a Power



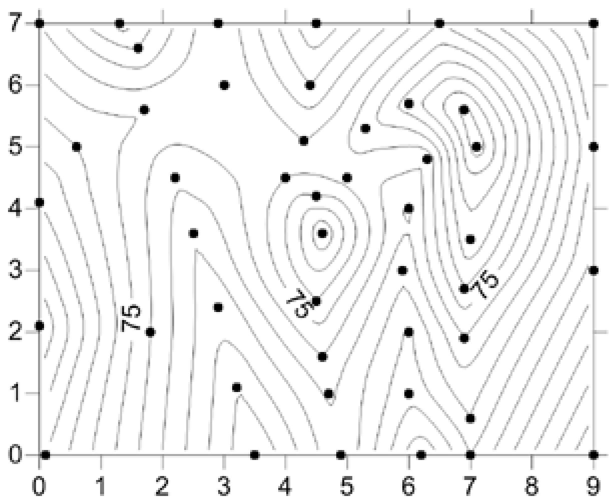
Kriging



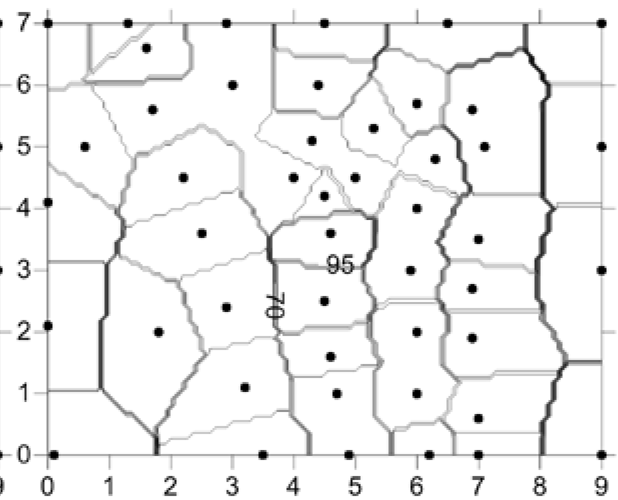
Minimum Curvature



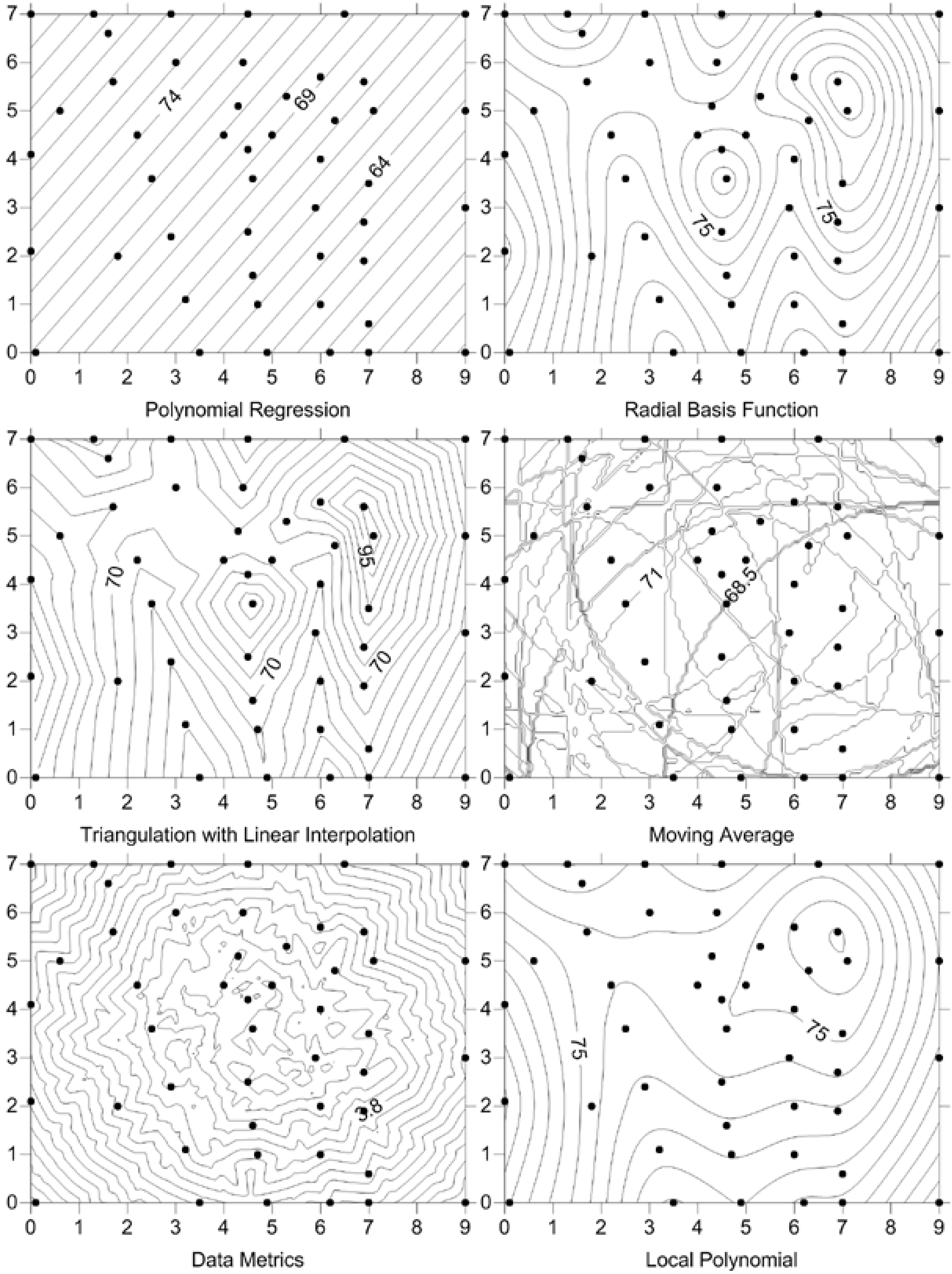
Modified Shepard's Method



Natural Neighbor



Nearest Neighbor



This is a comparison of different gridding methods. For these examples, the *sample file,* Demogrid.dat*, was used. All the defaults for the various methods were accepted. This data set contains 47 data points, irregularly spaced over the extent of the map. The data point locations are displayed as a post map layer.*

Achieving Different Results with Different Gridding Methods:

Using the same data set from the *Demogrid.dat* file and applying different gridding methods produces maps that emphasize different patterns.

Smooth Appearance

Kriging, Minimum Curvature, Natural Neighbor, and Radial Basis Function all produced acceptable contour maps with smooth appearance.

Bulls Eye Pattern

Inverse Distance to a Power and Modified Shepard's Method both tended to generate "bull's eye" patterns.

Triangular Facets

With Triangulation with Linear Interpolation, there are too few data points to generate an acceptable map, and this explains the triangular facets apparent on the contour map.

Blocky

Nearest Neighbor shows as a "blocky" map because the data set is not regularly spaced and therefore a poor candidate for this method.

Tilted Plane

Polynomial Regression shows the trend of the surface, represented as a tilted plane.

Discontinuities

Due to the small number of data in *Demogrid.dat*, the Moving Average method is not applicable. The results of using this method with an inadequate data set are shown as discontinuities are created as data are captured and discarded by the moving search neighborhoods.

Median Distance

Data Metrics can show many different types of information about the data and about the gridding process, depending on which metric is selected. This example shows the median distance between each grid node and the original 47 data points.

Smooth Local Variation

Local Polynomial models smooth local variation in the data set.

Exact and Smoothing Interpolators

Gridding methods included with **Surfer** are divided into two general categories: exact interpolators and smoothing interpolators. Some exact interpolators can incorporate a smoothing factor that causes them to become smoothing interpolators.

Exact Interpolators

Exact interpolators honor data points exactly when the point coincides with the grid node being interpolated. In other words, a coincident point carries a weight of essentially 1.0 and all other data points carry a weight of essentially zero. Even when using exact interpolators, it is possible that the grid file does not honor specific data points if the data points do not exactly coincide with the grid nodes. Refer to Weighted Averaging for more information on weights assigned during interpolation.

To increase the likelihood that your data are honored, you can increase the number of grid nodes in the X and Y direction. This increases the chance that grid nodes coincide with data points, thereby increasing the chance that the data values are applied directly to the grid file.

The following methods are exact interpolators:

- Inverse Distance to a Power when you do not specify a smoothing factor
- Kriging when you do not specify a nugget effect
- Nearest Neighbor under all circumstances
- Radial Basis Function when you do not specify an R^2 value
- Modified Shepard's Method when you do not specify a smoothing factor
- Triangulation with Linear Interpolation
- Natural Neighbor

Smoothing Interpolators

Smoothing interpolators or smoothing factors can be employed during gridding when you do not have strict confidence in the repeatability of your data measurements. This type of interpolation reduces the effects of small-scale variability between neighboring data points. Smoothing interpolators do not assign weights of 1.0 to any single point, even when a point is exactly coincident with the grid node. When smoothing is used, weighting factors are assigned so the map is smoother. In the extreme case, all data points are given equal weight and the surface becomes a horizontal plane at the average for all data in the data file.

The following methods are smoothing interpolators:

- Inverse Distance to a Power when you specify a smoothing factor
- Kriging when you specify an error nugget effect
- Polynomial Regression
- Radial Basis Function when you specify an R^2 value
- Modified Shepard's Method when you specify a smoothing factor
- Local Polynomial
- Moving Average

Weighted Averaging

The gridding methods in **Surfer** use weighted average interpolation algorithms. This means that, with all other factors being equal, the closer a point is to a grid node, the more weight it carries in determining the Z value at that grid node. The difference between gridding methods is how the weighting factors are computed and applied to data points during grid node interpolation.

To understand how weighted averages are applied, consider the equation shown here.

Given N data values:

$$\{Z_1, Z_2, \ldots, Z_N\}$$

the interpolated value at any grid node (for example, G_j) can be computed as the weighted average of the data values:

$$G_j = \sum_{i=1}^{N} w_{ij} Z_i$$

where

Gj	= interpolated grid node value at node *j*;
N	= number of points used to interpolate at each node;
Zi	Z value at the ithpoint

wij	weight associated with the ithdata value when computing

Inverse Distance to a Power

The *Inverse Distance to a Power* gridding method is a weighted average interpolator, and can be either an exact or a smoothing interpolator.

With *Inverse Distance to a Power*, data are weighted during interpolation such that the influence of one point relative to another declines with distance from the grid node. Weighting is assigned to data through the use of a weighting power that controls how the weighting factors drop off as distance from a grid node increases. The greater the weighting power, the less effect points far from the grid node have during interpolation. As the power increases, the grid node value approaches the value of the nearest point. For a smaller power, the weights are more evenly distributed among the neighboring data points.

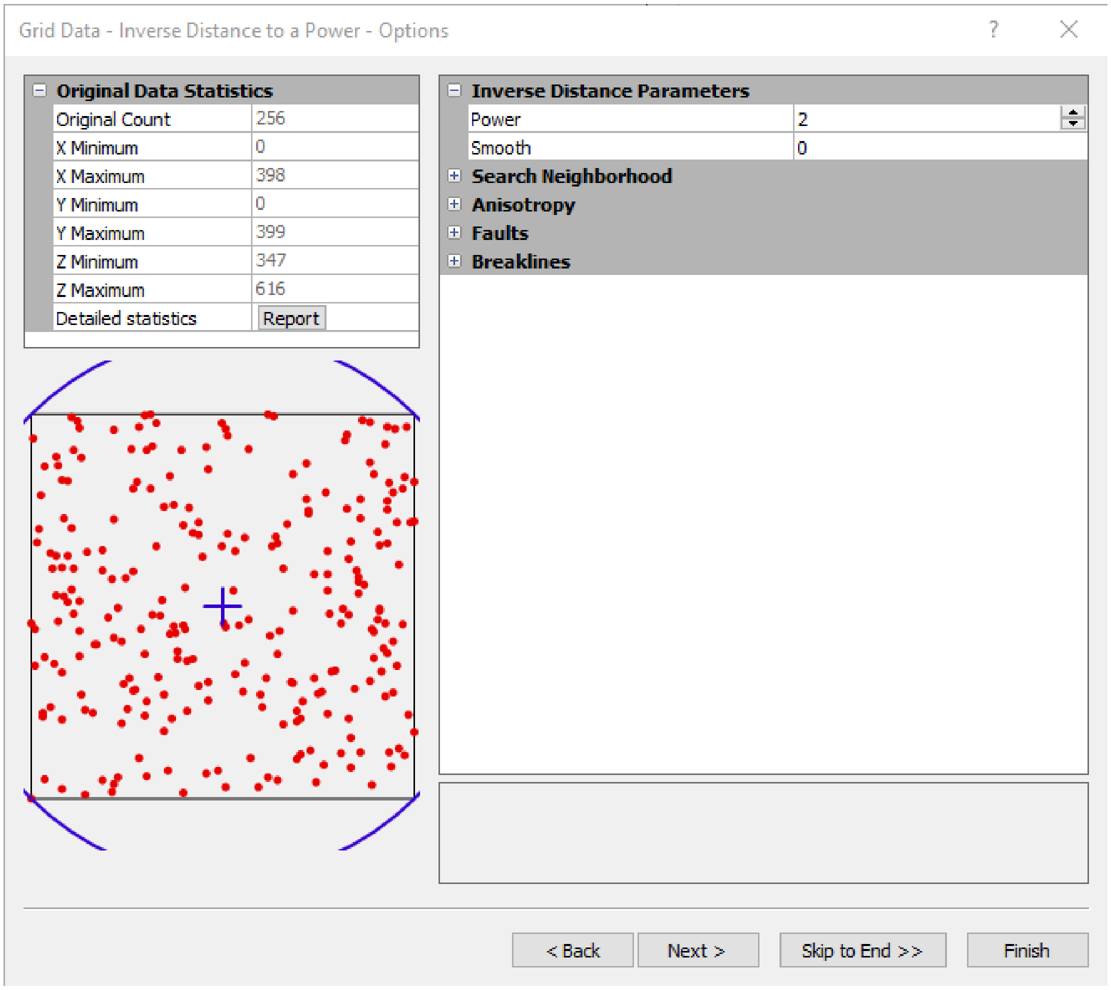
Normally, *Inverse Distance to a Power* behaves as an exact interpolator. When calculating a grid node, the weights assigned to the data points are fractions, and the sum of all the weights are equal to 1.0. When a particular observation is coincident with a grid node, the distance between that observation and the grid node is 0.0, and that observation is given a weight of 1.0, while all other observations are given weights of 0.0. Thus, the grid node is assigned the value of the coincident observation. The *Smoothing* parameter is a mechanism for buffering this behavior. When you assign a non-zero *Smoothing* parameter, no point is given an overwhelming weight so that no point is given a weighting factor equal to 1.0.

One of the characteristics of *Inverse Distance to a Power* is the generation of "bull's-eyes surrounding the position of observations within the gridded area. You can assign a smoothing parameter duringInverse Distance to a Powerto reduce the bull's-eye effect by smoothing the interpolated grid.

Inverse Distance to a Power is a very fast method for gridding. With less than 500 points, you can use the *All Data* search type and gridding proceeds rapidly.

Inverse Distance to a Power Options Dialog

In the Grid Data dialog, specify *Inverse Distance to a Power* as the *Gridding Method* and click the *Next* button to open the **Grid Data Inverse Distance to a Power Options** dialog.



Specify the parameters to use when gridding with the inverse distance to a power gridding method.

Power

The weighting *Power* parameter determines how quickly weights fall off with distance from the grid node. As the power parameter approaches zero, the generated surface approaches a horizontal planar surface through the average of all observations from the data file. As the power parameter increases, the generated surface is a "nearest neighbor" interpolator and the resultant surface becomes polygonal. The polygons represent the nearest observation to the interpolated grid node. Power values between 1.2e-038 and 1.0e+038 are accepted, although powers should usually fall between one and three.

Smooth

The *Smooth* factor parameter allows you to incorporate an "uncertainty" factor associated with your input data. The larger the smoothing factor parameter, the less overwhelming influence any particular observation has in computing a neighboring grid node.

Search Neighborhood

Specify search rules. For more information about search rules, see Search.

Anisotropy

Set anisotropy settings if needed. For more information about anisotropy options, see Anisotropy.

Breaklines and Faults

Specify a breakline and/or fault. For more information, see Breaklines and Faults.

References

Davis, John C. (1986), *Statistics and Data Analysis in Geology*, John Wiley and Sons, New York.

Franke, R. (1982), Scattered Data Interpolation: Test of Some Methods,Mathematics of Computations, v. 33, n. 157, p. 181-200.

Inverse Distance to a Power Math

The equation used for *Inverse Distance to a Power* is:

$$\hat{Z}_j = \frac{\sum_{i=1}^{n} \frac{Z_i}{h_{ij}^{\beta}}}{\sum_{i=1}^{n} \frac{1}{h_{ij}^{\beta}}}$$

$$h_{ij} = \sqrt{d_{ij}^2 + \delta^2}$$

where:

hij	is the effective separation distance between grid node "j" and the neighboring point "i."
$\hat{Z}_j$	is the interpolated value for grid node "j";
Zi	are the neighboring points;
dij	is the distance between the grid node "j" and the neighboring point "i";
β	is the weighting power (the *Power* parameter); and
δ	is the *Smoothing* parameter.

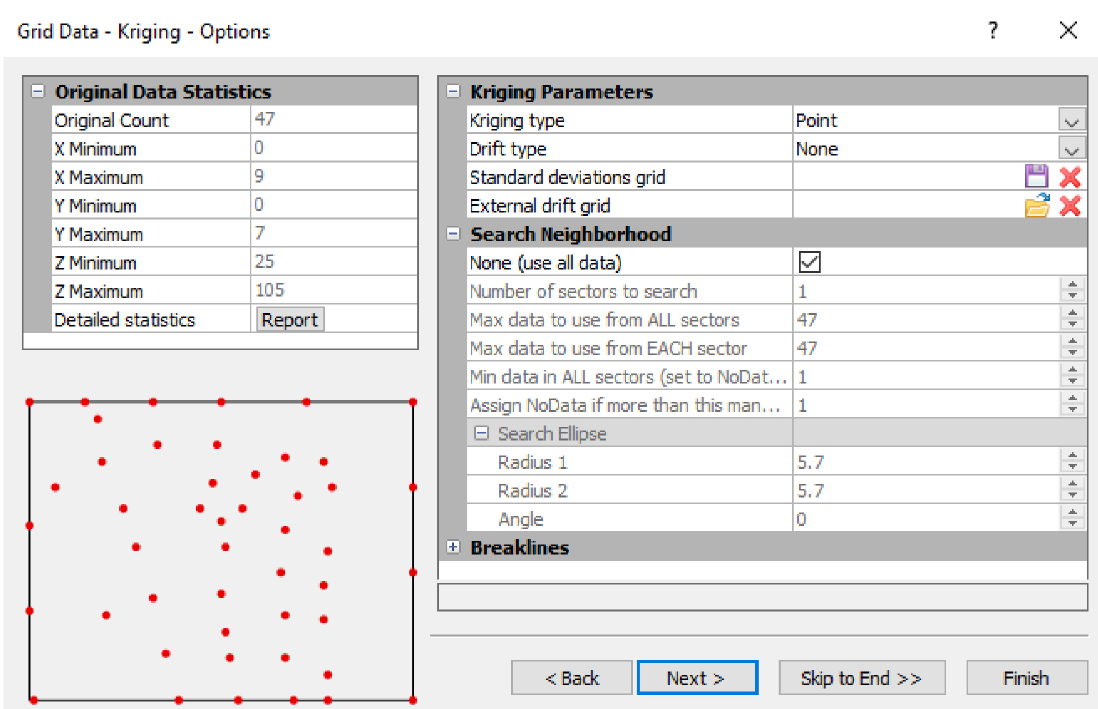
Kriging

Kriging is a geostatistical gridding method that has proven useful and popular in many fields. This method produces visually appealing maps from irregularly spaced data. *Kriging* attempts to express trends suggested in your data, so that, for example, high points might be connected along a ridge rather than isolated by bull's-eye type contours.

Kriging is a very flexible gridding method. You can accept the *Kriging* defaults to produce an accurate grid of your data, or *Kriging* can be custom-fit to a data set by specifying the appropriate variogram model. Within **Surfer**, *Kriging* can be either an exact or a smoothing interpolator depending on the user-specified parameters. It incorporates anisotropy and underlying trends in an efficient and natural manner.

Kriging Options Dialog

In the Grid Data dialog, specify *Kriging* as the *Gridding Method* and click the *Next* button to open the **Grid Data Kriging Options** dialog.



*Specify the variogram parameters, Kriging standard deviations file, Kriging type, and drift type in the **Grid Data Kriging Options** dialog.*

Kriging Type

Choose *Point* or *Block* Kriging from the *Kriging Type* list box.

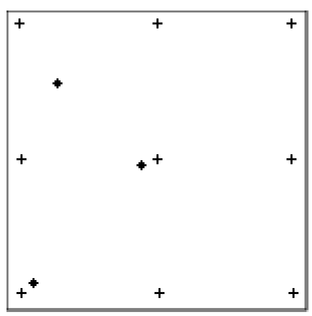
Surfer includes two *Kriging* types: *Point* Kriging and *Block* Kriging. A detailed discussion of the two methods can be found in Isaaks and Srivastava (1989, Chapters 12 and 13).[1] *Ordinary* (no drift) and *Universal Kriging* (linear or quadratic drift) algorithms can be applied to both *Kriging* types.

Both *Point* Kriging and *Block* Kriging generate an interpolated grid. *Point* Kriging estimates the values of the points at the grid nodes. *Block* Kriging estimates the average value of the rectangular blocks centered on the grid nodes. The blocks are the size and shape of a grid cell. Since *Block* Kriging is estimating the average value of a block, it generates smoother contours (block averaging smooths). Furthermore, since *Block* Kriging is not estimating the value at a point, *Block* Kriging is not a perfect interpolator. That is even if an observation falls exactly on a grid node, the *Block* Kriging estimate for that node does not exactly reproduce the observed value.

When a *Kriging* standard deviation grid is generated with *Block* Kriging, the generated grid contains the *Block* Kriging standard deviations and not the *Point* Kriging standard deviations.

The numerical integration required for point-to-block variogram calculations necessary for *Block* Kriging are carried out using a 3x3, two-dimensional Gaussian-Quadrature. In particular, Surfer uses Section 25.4.62 of Aramowitz and Stegun (1972).[2]

Note that *Point* Kriging is the default method.



The crosses indicate a block of grid nodes and the diamonds indicate data points.

In the diagram above, if we were to interpolate the center grid node with *Point* Kriging, the data point closest to the center grid node would have the greatest weight in determining the value of the grid node. If we were to interpolate the center grid node with *Block* Kriging, all three data points within the block of grid nodes are averaged to interpolate the grid node value.

Drift Type

Select a *Linear* or *Quadratic* drift type. Drift type *None* is *Ordinary Kriging*, while *Linear* or *Quadratic* drift type is *Universal Kriging*.

If you use the *Get Variogram* command to apply a variogram model, be sure to set the *Drift Type* option to the same value as you used in the General page for *Detrend* when creating the variogram.

Output Grid of Kriging Standard Deviations

Click the button to enter a file name into the *Standard deviations grid* field to produce an estimation standard deviation grid. If this box is empty, then the estimation standard deviation grid is not created. Click the button to remove the destination path from the *Standard deviations grid* field.

The Kriging standard deviation grid output option greatly slows the Kriging process. This is contrary to what you may expect since the Kriging variances are usually a by-product of the Kriging calculations. However, **Surfer** uses a highly optimized algorithm for calculating the node values. When the variances are requested, a more traditional method must be used, which takes much longer.

There are several cases where a standard deviation grid is incorrect or meaningless. If the variogram model is not truly representative of the data, the standard deviation grid is not helpful to your data analysis. Also, the Kriging standard deviation grid generated when using a variogram model estimated with the *Standardized Variogram* estimator or the *Autocorrelation* estimator is not correct. These two variogram estimators generate dimensionless variograms, so the Kriging standard deviation grids are incorrectly scaled. Similarly, while the default linear variogram model will generate useful contour plots of the data, the associated Kriging standard deviation grid is incorrectly scaled and should not be used. The default linear model slope is one, and since the Kriging standard deviation grid is a function of slope, the resulting grid is meaningless.

The Kriging standard deviation grid cannot be created when the *None (use all data)* option in the Search section is selected.

Kriging with External Drift

Load a grid in the *External drift grid* field to perform Kriging with external drift interpolation. Click the button and select the external drift grid file in the **Open Grid** dialog. Click the button to remove the file from the *External drift grid* field.

Kriging with external drift uses a grid of a correlated variable to refine the interpolation of grid values. The *External drift grid* is sampled at the grid nodes during interpolation. The *External drift grid* should be strongly correlated with and predictive of the variable of interest. Kriging with external drift extends the covariance matrix with the values of auxiliary predictors. Otherwise, Kriging with external drift is similar to ordinary Kriging.

The *External drift grid* must have larger areal extents than the input data set and the *Output grid geometry* in the Grid Data dialog. Any interpolated grid node that does not include a sample from the *External drift grid* will be assigned the NoData value.

Search Neighborhood

The options in the *Search Neighborhood* section specify search rules. Search options control which data points are considered by the gridding operation when interpolating grid nodes.

Breaklines

The *Breaklines* section is used to add breaklines to the gridding process. Faults are not supported with *Kriging*.

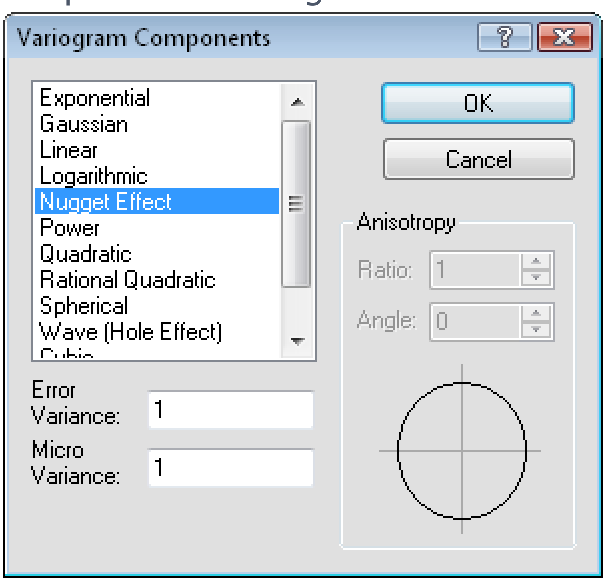
[1]Isaaks, E. H., and Srivastava, R. M. (1989), *An Introduction to Applied Geo-statistics*, Oxford University Press, New York, 561 pp.

[2]Abramowitz, M., and Stegun, I. (1972), *Handbook of Mathematical Functions*, Dover Publications, New York.

Variogram Components

Click the *Add* button in the Grid Data Kriging Options dialog (accessed through the *Advanced Options* button in Grids | New Grid | Grid Data when *Kriging* is the selected gridding method) to open the **Variogram Components** dialog. If you have not modeled the variogram or if you do not understand variograms, leave the variogram set at the original defaults (linear variogram; slope=1, aniso-tropy=1,0; no nugget effect).

The Variogram Components Dialog



Customize the Variogram Components *in the* ***Variogram Components*** *dialog, if necessary.*

Variogram Model List

Choose from the *Exponential*, *Gaussian*, *Linear*, *Logarithmic*, *Nugget Effect*, *Power*, *Quadratic*, *Rational Quadratic*, *Spherical*, *Wave (Hole Effect)*, *Cubic*, or *Pentaspherical* models.

Model Options

Depending on which model is selected, the options below the model list changes. Set the *Scale* (vertical scale) and *Length* (horizontal range) for *Exponential*, *Gaussian*, *Logarithmic*, *Quadratic*, *Rational Quadratic*, *Spherical*, *Wave (Hole Effect)*, *Cubic*, or *Pentaspherical* models. In addition to the *Scale* and *Length*, set the *Power* for the *Power* model. Set the *Slope* for the *Linear* model. When the Nugget Effect is selected set the *Error Variance* and *Micro Variance*.

Anisotropy Ratio and Angle

The Anisotropy *Ratio* and *Angle* can be set for all models except the *Nugget Effect*.

Kriging References

For a detailed derivation and discussion of *Kriging* see Cressie (1991) or Journel and Huijbregts (1978). Journel (1989) is, in particular, a concise presentation of geostatistics (and Kriging). Isaaks and Srivastava (1989) offer a clear introduction to the topic, though it does not cover some of the more advanced details.

For those who need to see computer code to really understand an algorithm, Deutsch and Journel (1992) includes a complete, well-written, and well-documented source code library of geostatistics computer programs (in FORTRAN). Finally, a well-researched account of the history and origins of *Kriging* can be found in Cressie (1990).

Abramowitz, M., and Stegun, I. (1972), *Handbook of Mathematical Functions*, Dover Publications, New York.

Cressie, N. A. C. (1990), "The Origins of Kriging", *Mathematical Geology*, v. 22, p. 239-252.

Cressie, N. A. C. (1991), *Statistics for Spatial Data*, John Wiley and Sons, Inc., New York, 900 pp.

Deutsch, C. V., and Journel, A. G. (1992), *GSLIB - Geostatistical Software Library and User's Guide*, Oxford University Press, New York, 338 pp.

Isaaks, E. H., and Srivastava, R. M. (1989), *An Introduction to Applied Geostatistics*, Oxford University Press, New York, 561 pp.

Journel, A. G., and Huijbregts, C. (1978), *Mining Geostatistics*, Academic Press, 600 pp.

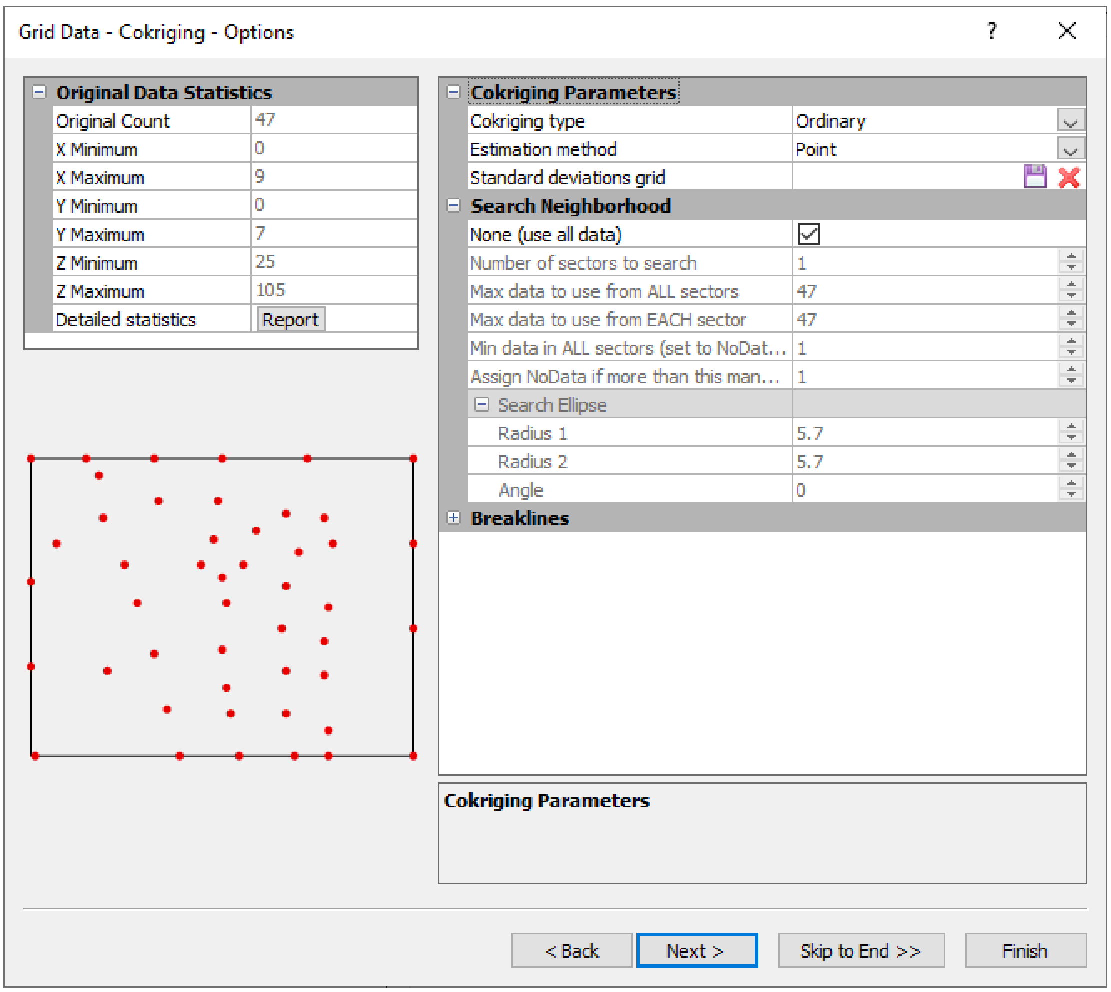
Journel, A. G. (1989), *Fundamentals of Geostatistics in Five Lessons*, American Geophysical Union, Washington D.C.

Cokriging

Cokriging is a geostatistical gridding method that uses a densely sampled second correlated variable to improve the estimation of a primary variable. For cokriging to be effective, the two variables must be correlated and the primary variable must be undersampled with respect to the secondary variable. See Kriging for more information about the interpolation method.

Cokriging Options Dialog

In the Grid Data dialog, specify *Cokriging* as the *Gridding Method* and click the *Next* button twice to open the **Grid Data Cokriging Options** dialog.



*Specify the cokriging standard deviations file, cokriging type, and estimation method in the **Grid Data Cokriging Options** dialog.*

Cokriging Type

Select the desired variation of the cokriging algorithm from the *Cokriging type* list.

- *Simple* cokriging imposes no constraints on the sum of the weights of the variables. This version requires working on data residuals or equivalently on variables whose means have all been standardized to zero.
- *Ordinary* cokriging sets the sum of the weights applied to all variables to one. The *Ordinary* type is the default selection.
- *Traditional Ordinary* cokriging sets the sum of the weights applied to the primary variable to one and the sum of the weights applied to the secondary variable to zero. This formulation tends to limit severely the influence of the secondary variable.

Estimation Method

Choose *Point* or *Block* Kriging from the *Estimation method* list.

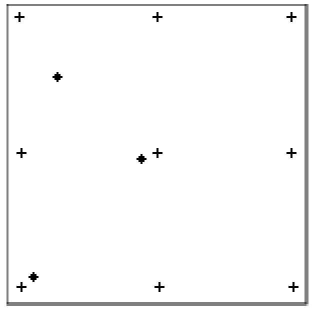
Surfer includes two *Estimation methods*: *Point* Kriging and *Block* Kriging. A detailed discussion of the two methods can be found in Isaaks and Srivastava (1989, Chapters 12 and 13)[1]. *Ordinary* (no drift) and *Universal Kriging* (linear or quadratic drift) algorithms can be applied to both *Kriging* types.

Both *Point* Kriging and *Block* Kriging generate an interpolated grid. *Point* Kriging estimates the values of the points at the grid nodes. *Block* Kriging estimates the average value of the rectangular blocks centered on the grid nodes. The blocks are the size and shape of a grid cell. Since *Block* Kriging is estimating the average value of a block, it generates smoother contours (block averaging smooths). Furthermore, since *Block* Kriging is not estimating the value at a point, *Block* Kriging is not a perfect interpolator. That is even if an observation falls exactly on a grid node, the *Block* Kriging estimate for that node does not exactly reproduce the observed value.

When a *Kriging* standard deviation grid is generated with *Block* Kriging, the generated grid contains the *Block* Kriging standard deviations and not the *Point* Kriging standard deviations.

The numerical integration required for point-to-block variogram calculations necessary for *Block* Kriging are carried out using a 3x3, two-dimensional Gaussian-Quadrature. In particular, Surfer uses Section 25.4.62 of Aramowitz and Stegun (1972).[2]I

Point Kriging is the default method.



In the diagram above, the crosses indicate a block of grid nodes and the filled circles indicate data points. If we were to interpolate the center grid node with *Point* Kriging, the data point closest to the center grid node would have the greatest weight in determining the value of the grid node. If we were to interpolate the center grid node with *Block* Kriging, all three data points within the block of grid nodes are averaged to interpolate the grid node value.

Output Grid of Kriging Standard Deviations

Click the button to enter a file name into the *Standard deviations grid* field to produce an estimation standard deviation grid. If this box is empty, then the estimation standard deviation grid is not created. Click the button to remove the destination path from the *Standard deviations grid* field.

The Kriging standard deviation grid output option greatly slows the Kriging process. This is contrary to what you may expect since the Kriging variances are usually a by-product of the Kriging calculations. However, **Surfer** uses a highly optimized algorithm for calculating the node values. When the variances are requested, a more traditional method must be used, which takes much longer.

There are several cases where a standard deviation grid is incorrect or meaningless. If the variogram model is not truly representative of the data, the standard deviation grid is not helpful to your data analysis. Also, the Kriging standard deviation grid generated when using a variogram model estimated with the *Standardized Variogram* estimator or the *Autocorrelation* estimator is not correct. These two variogram estimators generate dimensionless variograms, so the Kriging standard deviation grids are incorrectly scaled. Similarly, while the default linear variogram model will generate useful contour plots of the data, the associated Kriging standard deviation grid is incorrectly scaled and should not be used. The default linear model slope is one, and since the Kriging standard deviation grid is a function of slope, the resulting grid is meaningless.

The Kriging standard deviation grid cannot be created when the *None (use all data)* option in the Search section is selected.

Search Neighborhood

The options in the *Search Neighborhood* section specify search rules. Search options control which data points are considered by the gridding operation when interpolating grid nodes. The same search parameters are used for both the primary and secondary variables.

Breaklines

The *Breaklines* section is used to add breaklines to the gridding process. Faults are not supported with *Kriging*.

[1]Isaaks, E. H., and Srivastava, R. M. (1989), *An Introduction to Applied Geostatistics*, Oxford University Press, New York, 561 pp.

[2]Abramowitz, M., and Stegun, I. (1972), *Handbook of Mathematical Functions*, Dover Publications, New York.

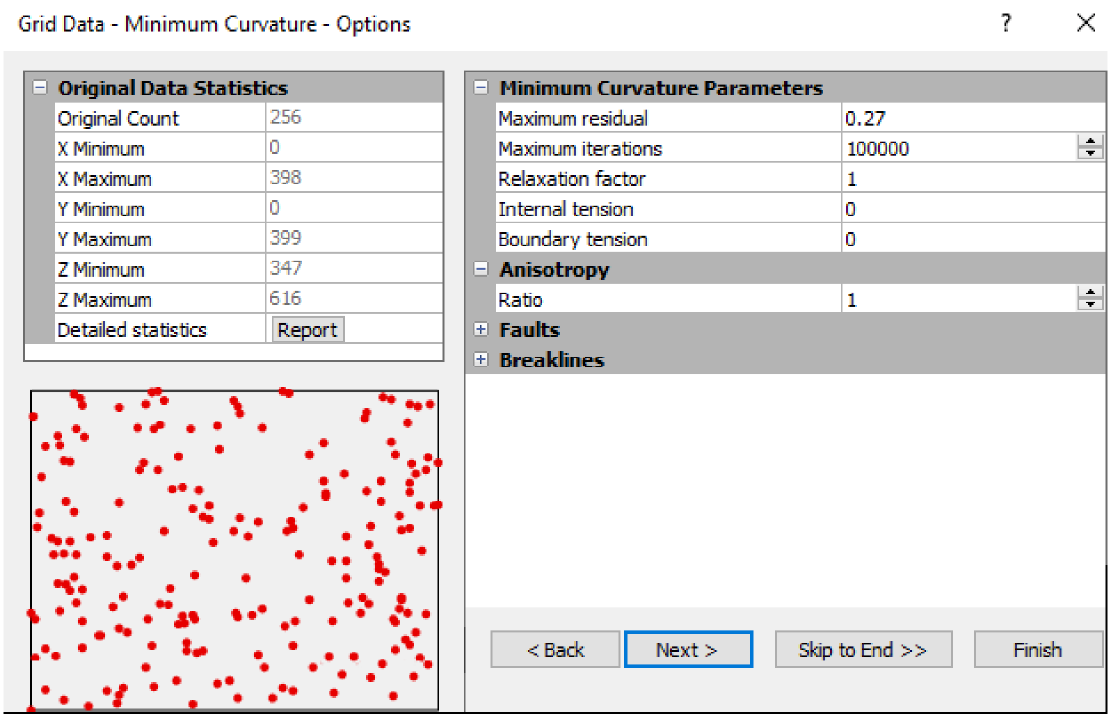
Minimum Curvature

Minimum Curvature is widely used in the earth sciences. The interpolated surface generated by *Minimum Curvature* is analogous to a thin, linearly elastic plate passing through each of the data values with a minimum amount of bending. *Minimum Curvature* generates the smoothest possible surface while attempting to honor your data as closely as possible. *Minimum Curvature* is not an exact interpolator, however. This means that your data are not always honored exactly.

Minimum Curvature produces a grid by repeatedly applying an equation over the grid in an attempt to smooth the grid. Each pass over the grid is counted as one iteration. The grid node values are recalculated until successive changes in the values are less than the *Maximum Residuals* value, or the maximum number of iterations is reached (*Maximum Iteration* field).

Minimum Curvature Options Dialog

In the Grid Data dialog, specify *Minimum Curvature* as the *Gridding Method* and click the *Next* button to open the **Grid Data Minimum Curvature Options** dialog.



Click the Advanced Options *button in the* ***Grid Data Minimum Curvature Options*** *dialog to set the minimum curvature options.*

Maximum Residual

The *Maximum Residual* parameter has the same units as the data, and an appropriate value is approximately 10% of the data precision. If data values are measured to the nearest 1.0 units, the *Maximum Residual* value should be set at 0.1. The iterations continue until the maximum grid node correction for the entire iteration is less than the *Maximum Residual* value. The default *Maximum Residual* value is given by:

Default Max Residual = 0.001 (Z*max* - Z *min*)

Maximum Iteration

The *Maximum Iteration* parameter should be set at one to two times the number of grid nodes generated in the grid file. For example, when generating a 50 by 50 grid using *Minimum Curvature*, the *Maximum Iteration* value should be set between 2,500 and 5,000.

Relaxation Factor

The *Minimum Curvature* gridding algorithm solves the specified partial differential equation using a successive over-relaxation algorithm. The interior is updated using a "chessboard" strategy, as discussed in Press, et al. (1988, p. 868). The only difference is that the biharmonic equation must have nine different "colors," rather than just black and white.

The *Relaxation Factor* is as described in Press et al. (1988). In general, the *Relaxation Factor* should not be altered. The default value (1.0) is a good generic value. Roughly, the higher the *Relaxation Factor* (closer to two) the faster the *Minimum Curvature* algorithm converges, but the more likely it will not converge at all. The lower the *Relaxation Factor* (closer to zero) the more likely the *Minimum Curvature* algorithm will converge, but the algorithm is slower. The optimal *Relaxation Factor* is derived through trial and error.

Internal and Boundary Tension

Qualitatively, the *Minimum Curvature* gridding algorithm is attempting to fit a piece of sheet metal through all of the observations without putting any creases or kinks in the surface. Between the fixed observation points, the sheet bows a bit.

The *Internal Tension* is used to control the amount of this bowing on the interior: the higher the tension, the less the bowing. For example, a high tension makes areas between observations look like facets of a gemstone.

The *Boundary Tension* controls the amount of bowing on the edges. The range of values for *Internal Tension* and *Boundary Tension* are 0 to 1.

By default, the *Internal Tension* and the *Boundary Tension* are set to 0.

Anisotropy

Set anisotropy settings if needed. For more information about anisotropy options, see Anisotropy.

Breaklines and Faults

Specify a breakline and/or fault. For more information, see Breaklines and Faults.

Minimum Curvature Math

The **Surfer** code fully implements the concepts of tension as described and detailed in Smith and Wessel (1990). Also, as recommended by Smith and Wessel, this routine first fits a simple planar model using least squares regression:

$$AX + BY + C = Z(X,Y)$$

Thus, there are four steps to generate the final grid using the minimum curvature method.

1. The least squares regression model is fit to the data.
2. The values of the planar regression model at the data locations are subtracted from the data values; this yields a set of residual data values.

3. The minimum curvature algorithm is used to interpolate the residuals at the grid nodes.

4. The values of the planar regression model at the grid nodes are added to the interpolated residuals, yielding a final interpolated surface.

Unlike Smith and Wessel (1990), the fixed nodes are defined as the average of the neighboring observed values. That is, consider a rectangle the size and shape of a grid cell. The neighborhood of a grid node is defined by this rectangle centered on the grid node. If there are any observed data within the neighborhood of a grid node, the value of that grid node is fixed equal to the arithmetic average of contained data.

The *Minimum Curvature* algorithm generates the surface that interpolates the available data and solves the modified biharmonic differential equation with tension:

$$(1 - T_i)\nabla^2(\nabla^2 Z) - (T_i)\nabla^2 Z = 0$$

There are three sets of associated boundary conditions:

On the edges: $(1 - T_b)\dfrac{\partial^2 Z}{\partial n^2} + (T_b)\dfrac{\partial Z}{\partial n} = 0$

On the edges: $\dfrac{\partial(\nabla^2 Z)}{\partial n} = 0$

At the corners: $\dfrac{\partial^2 Z}{\partial x \partial y} = 0$

where:

∇^2 is the Laplacian operator

n is the boundary normal

T_i is the internal tension

T_b is the boundary tension

Convergence

As mentioned in Briggs (1974), and strongly recommended in Smith and Wessel (1990), **Surfer** uses a "multiple lattice strategy." It starts with a coarse grid and then incrementally refines the grid until the final density is achieved.

The relaxation approach is a local smoothing process and, consequently, short-wavelength components of Z are found quickly. On the other hand, the relaxation process does not propagate the effects of the data constraints to longer wavelengths efficiently.

As recommended by Briggs (1974) and Smith and Wessel (1990), this routine determines convergence by comparing the largest magnitude nodal change in one iteration to the specified tolerance (*Maximum Residual*).

The status of the algorithm is reflected on the status line. For example: *pass 2 of 4, iteration 360 (0.1234 > 0.08)*

This says that there are four levels of grids considered (the fourth is the final grid), and the algorithm is currently working on the second. The algorithm is currently on iteration 360. If the iteration number exceeds the *Maximum Iterations* parameter, the algorithm terminates without generating the grid and provides a failure-to-converge warning. The numbers in the parentheses are the current largest residual and the largest allowed residual. The largest allowed residual equals the *Maximum Residual* parameter on the final pass.

The *Minimum Curvature* method requires at least four data points.

Minimum Curvature References

Briggs, I. C. (1974), Machine Contouring Using Minimum Curvature, *Geophysics*, v. 39, n. 1, p. 39-48.

Press, W.H., , *Numerical Recipes in C* , Cambridge University Press.

Smith, W. H.

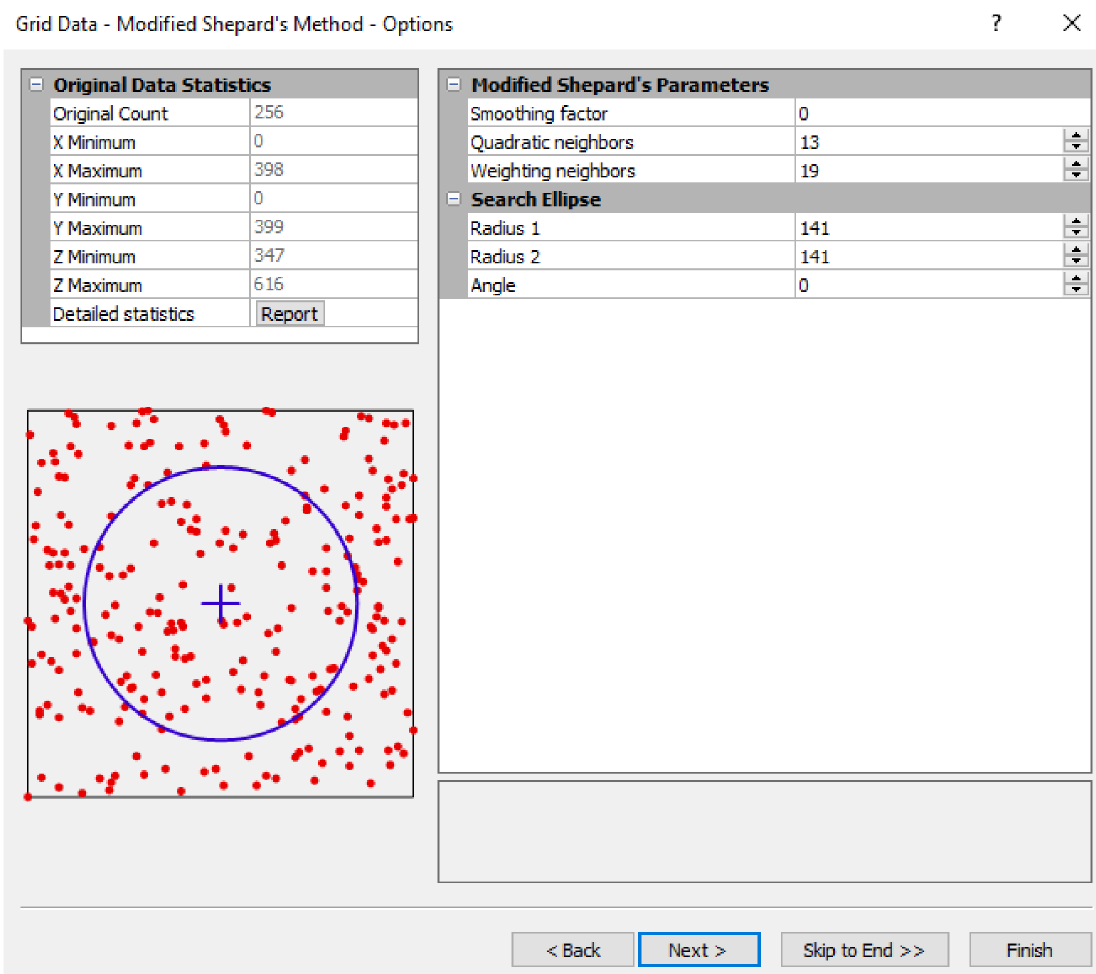
Modified Shepard's Method

Modified Shepard's Method uses an inverse distance weighted least squares method. As such, *Modified Shepard's Method* is similar to the Inverse Distance to a Power interpolator, but the use of local least squares eliminates or reduces the "bull's-eye" appearance of the generated contours. *Modified Shepard's Method* can be either an exact or a smoothing interpolator.

The **Surfer** algorithm implements Franke and Nielson's (1980) Modified Quadratic Shepard's Method with a full sector search as described in Renka (1988). (Surfer 6 was based upon Franke and Nielson, not Renka.)

Modified Shepard's Method Options Dialog

In the Grid Data dialog, specify *Modified Shepard's Method* as the *Gridding Method* and click the *Advanced Options* button to open the **Grid Data Modified Shepard's Method Options** dialog.



Set the options, including search, and smoothing factor, in the ***Grid Data Modified Shepard's Method Options*** *dialog.*

Smoothing Factor

You can assign a smoothing parameter to the gridding operation. The *Smoothing Factor* parameter allows *Modified Shepard's Method* to operate as a smoothing interpolator. Greater smoothing occurs as you increase the value of the smoothing parameter. In general, values between zero and one are most reasonable.

Quadratic Neighbors

The *Modified Shepard's Method* starts by computing a local least squares fit of a quadratic surface around each observation. The *Quadratic Neighbors* parameter specifies the size of the local neighborhood by specifying the number of local neighbors. The local neighborhood is a circle of sufficient radius to include exactly this many neighbors. The default value follows the recommendation of Renka (1988).

Weighting Neighbors

The interpolated values are generated using a distance-weighted average of the previously computed quadratic fits associated with neighboring observations. The *Weighting Neighbors* parameter specifies the size of the local neighborhood by specifying the number of local neighbors. The local neighborhood is a circle of sufficient radius to include exactly this many neighbors. The default value follows the recommendation of Renka (1988).

Search

You can set *Search* parameters in this dialog. For more information about search options, see Search.

References

Franke, R., and Nielson, G. (1980), 'Smooth Interpolation of Large Sets of Scattered Data', *International Journal for Numerical Methods in Engineering*, v. 15, p. 1691-1704.

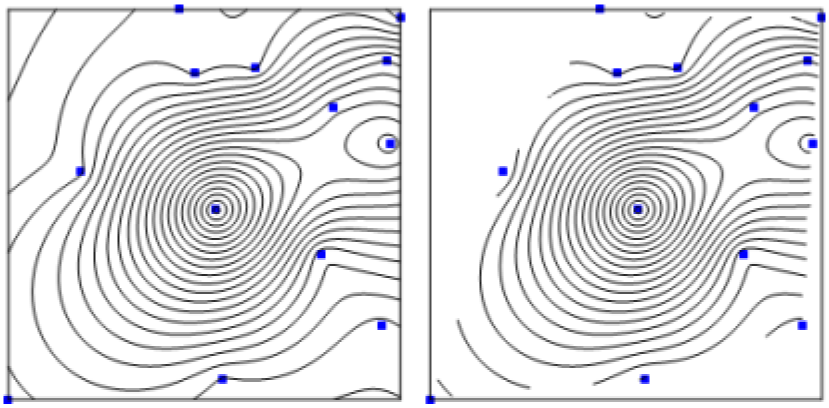
Renka, R. J. (1988), 'Algorithm 660: QSHEP2D: Quadratic Shepard Method for Bivariate Interpolation of Scattered Data', *ACM Transactions on Mathematical Software*, v. 14, No. 2, p. 139-148.

Shepard, D. (1968), A two dimensional interpolation function for irregularly spaced data, *Proc. 23rd Nat.*, p. 517-523.

Natural Neighbor

The *Natural Neighbor* gridding method interpolates grid values by weighting neighboring data points based on proportionate areas. Consider a set of Thiessen polygons (the dual of a Delaunay triangulation). If a new point (target) were added to the data set, these Thiessen polygons would be modified. In fact, some of the polygons would shrink in size, while none would increase in size. The area associated with the target's Thiessen polygon that was taken from an existing polygon is called the "borrowed area." The *Natural Neighbor* interpolation algorithm uses a weighted average of the neighboring observations, where the weights are proportional to the "borrowed area."

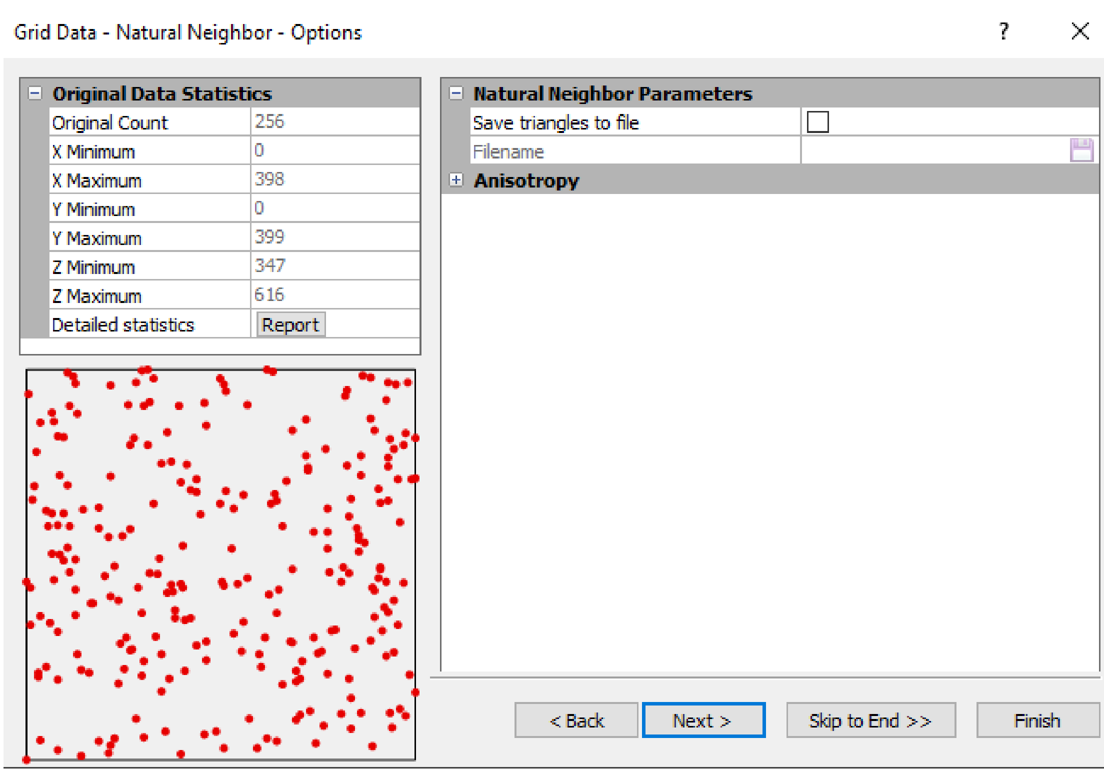
The *Natural Neighbor* method assigns the NoData value at and beyond the convex hull of the data locations (i.e. the outline of the Thiessen polygons).



The map on the left shows contours generated by the inverse distance to a power method. The map on the right shows contours generated by the natural neighbor method. Contours are not extrapolated beyond the boundary of the Thiessen polygons with the natural neighbor gridding method.

Natural Neighbor Advanced Options Dialog

In the Grid Data dialog, specify *Natural Neighbor* as the *Gridding Method* and click the *Advanced Options* button to open the **Natural Neighbor Advanced Options** dialog.



Set the anisotropy options and save Delaunay triangles in the ***Grid Data Natural Neighbor Options*** *dialog.*

Save Triangles

Check the *Save Triangles To* box to export the Delaunay triangulation. The resulting file can be loaded as a base map and combined with other maps (i.e. contour maps, 3D surface maps, etc.).

Click the button to open the **Export Triangles** dialog. Specify a *Save In* location, *File Name*, and *Save as type*. The Delaunay triangle export formats include: .BLN, .BNA, .EMF, .GSB, .MIF, .PDF (Vector), and .WMF. Click the *Save* button to return to the **Grid Data Advanced Options** dialog.

Anisotropy

For more information about anisotropy options see Anisotropy.

References

The main reference for this method is Sibson (1981), however, also refer to Sibson (1980) and Watson (1994). Watson (1994) discusses the *Natural Neighbor* gridding technique in some detail, although Surfer does not use any of Watson's source code.

Sibson, R. (1980), 'A Vector Identity for the Dirichlet Tesselation', *Math. Proc. Cambridge Phil. Soc.*, v. 87, p. 151-155.

Sibson, R. (1981), 'A Brief Description of Natural Neighbor Interpolation', *Interpreting Multivariate Data*, V. Barnett editor, John Wiley and Sons, New York, p. 21-36.

Watson, Dave (1994), *Nngridr - An Implementation of Natural Neighbor Interpolation*, David Watson, P.O. Box 734, Clarement, WA 6010, Australia.

Nearest Neighbor

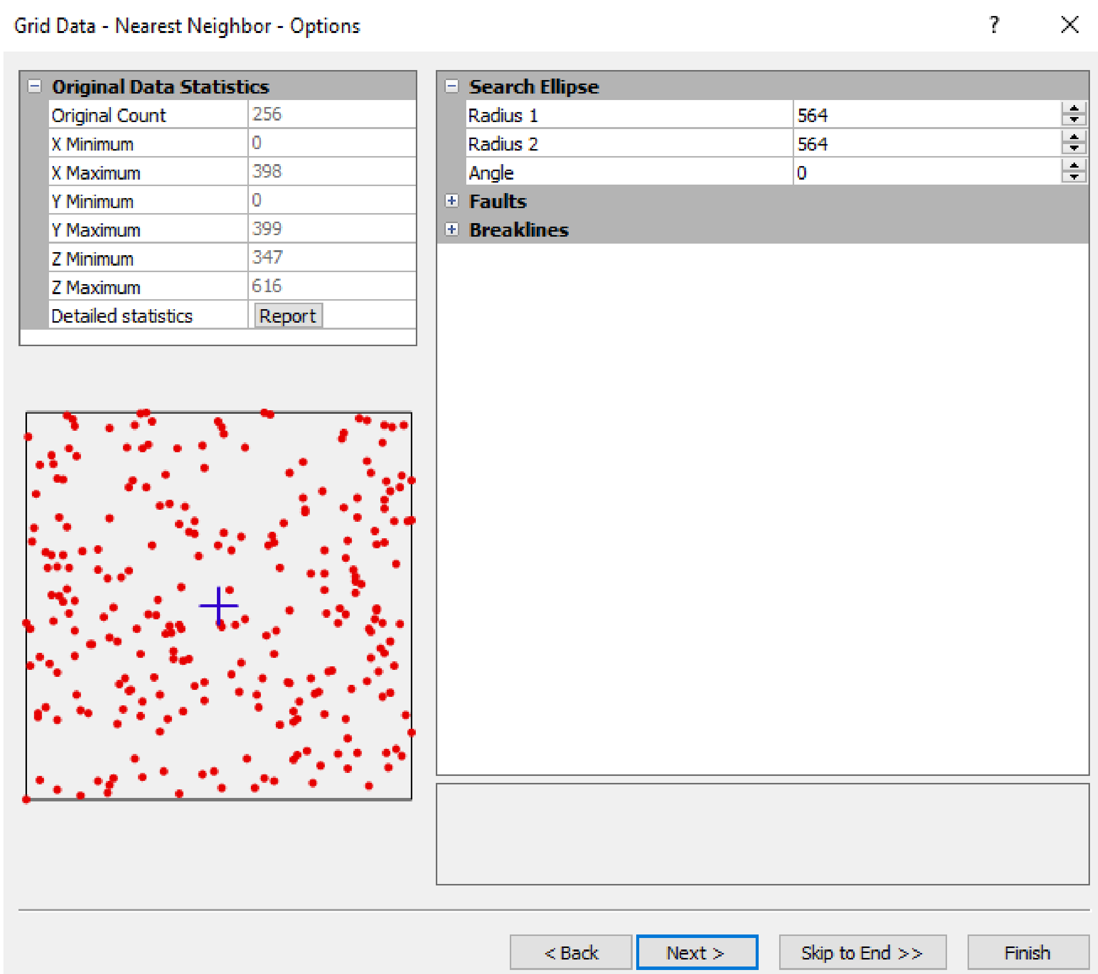
The *Nearest Neighbor* gridding method assigns the value of the nearest point to each grid node. This method is useful when data are already evenly spaced, but need to be converted to a **Surfer** grid file. Alternatively, in cases where the data are nearly on a grid with only a few missing values, this method is effective for filling in the holes in the data.

Sometimes with nearly complete grids of data, there are areas of missing data that you want to exclude from the grid file. In this case, you can set the Search Ellipse to a value so the areas of no data are assigned the NoData value in the grid file. By setting the search ellipse radii to values less than the distance between data values in your file, the NoData value is assigned at all grid nodes where data values do not exist.

When you use the *Nearest Neighbor* method to convert regularly spaced XYZ data to a grid file, you can set the grid spacing equal to the spacing between data points in the file. Refer to*Producing a Grid File from a Regular Array of XYZ Data* for the procedure of converting regularly spaced XYZ data into a **Surfer** grid file.

Nearest Neighbor Options Dialog

In the Grid Data dialog, specify *Nearest Neighbor* as the *Gridding Method* and click the *Next* button to open the **Grid Data Nearest Neighbor Options** dialog.



Set the search parameters page in the ***Grid Data Nearest Neighbor Options*** *dialog.*

Search Neighborhood

Specify search rules. For more information about search rules, see Search.

Breaklines and Faults

Specify a breakline and/or fault. For more information, see Breaklines and Faults.

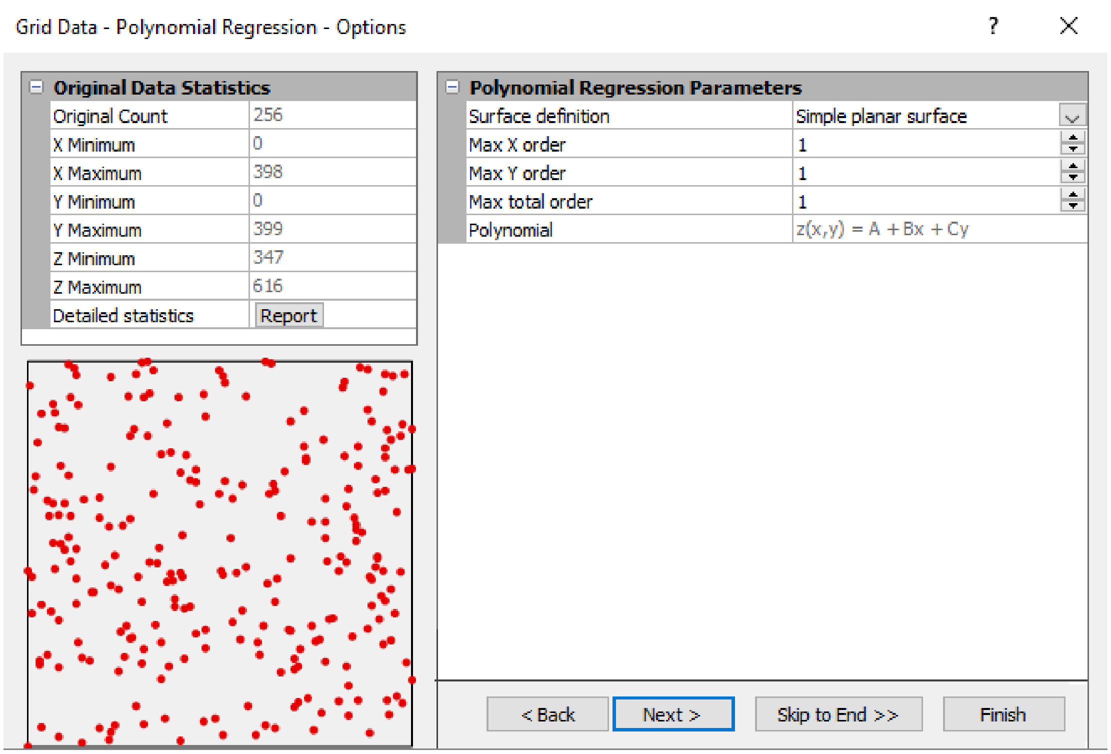
Polynomial Regression

Polynomial Regression is used to define large-scale trends and patterns in your data. *Polynomial Regression* is not really an interpolator because it does not attempt to predict unknown Z values. There are several options you can use to define the type of trend surface.

If you would like a report of the coefficients used in the calculation of the surface, make sure to select the *Grid Report* check box in the Grid Data dialog.

Polynomial Regression Advanced Options Dialog

In the Grid Data dialog, specify *Polynomial Regression* as the *Gridding Method* and click the *Next* button to open the **Grid Data Polynomial Regression Options** dialog.



Specify the Surface Definition *and* Parameters *in the* ***Grid Data Polynomial Regression Options*** *dialog.*

Surface Definition

You can select the type of polynomial regression to apply to your data from the *Surface Definition* group. As you select the different types of polynomials, a generic polynomial form of the equation is presented in the dialog, and the values in the *Parameters* group change to reflect the selection. The available choices are:

- Simple planar surface
- Bi-linear saddle
- Quadratic surface
- Cubic surface
- User defined polynomial

Parameters

The *Parameters* group allows you to specify the maximum powers for the X and Y component in the polynomial equation. As you change the *Parameters* values, the options are changed in the *Surface Definition* group to reflect the defined parameters.

- The *Max X Order* specifies the maximum power for the X component in the polynomial equation.
- The *Max Y Order* specifies the maximum power for the Y component in the polynomial equation.
- The *Max Total Order* specifies the maximum sum of the *Max X Order* and *Max Y Order* powers. All of the combinations of the X and Y components are included in the polynomial equation as long as the sum of the two powers does not exceed the *Max Total Order* value.

Reference

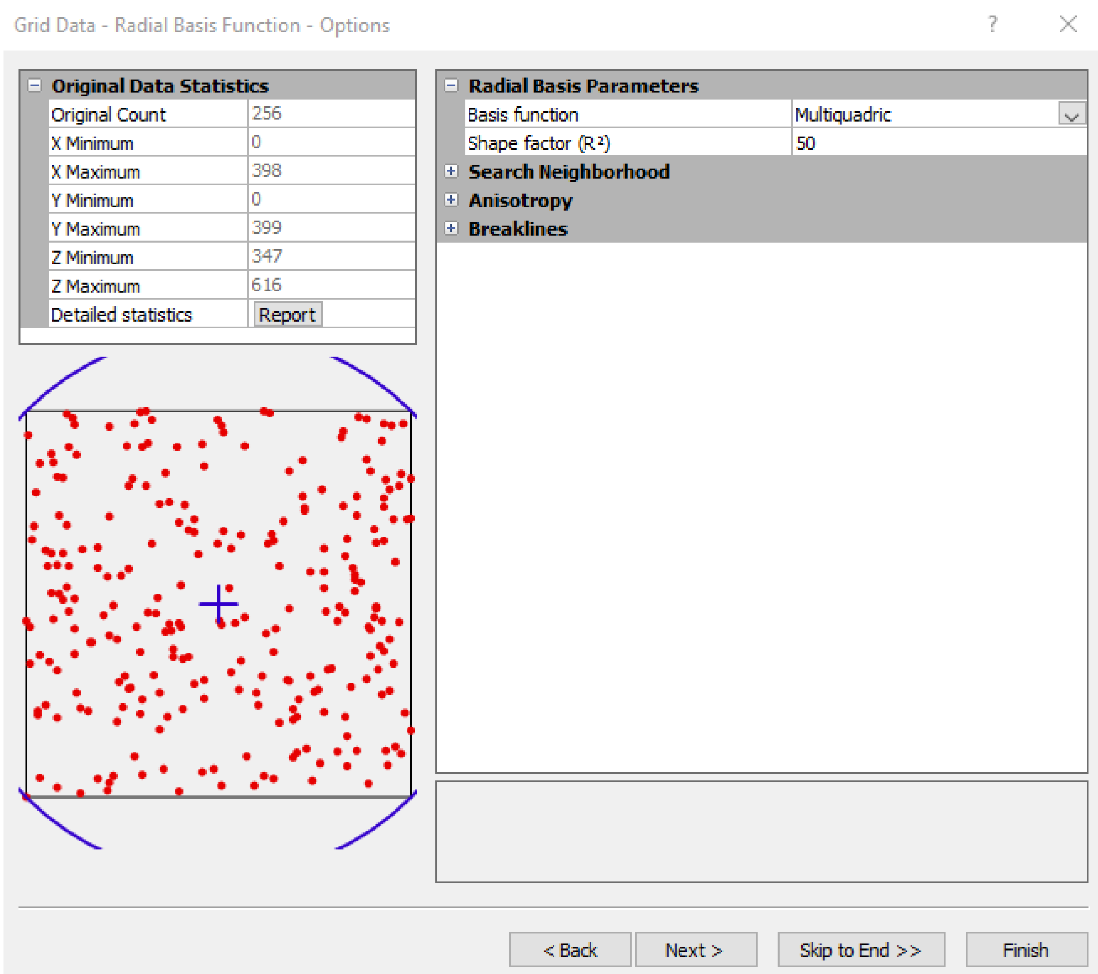
Draper, N., and Smith, H. (1981), *Applied Regression Analysis*, second edition, Wiley-Interscience, 709 pp.

Radial Basis Function

Radial Basis Function interpolation is a diverse group of data interpolation methods. In terms of the ability to fit your data and to produce a smooth surface, the *Multiquadric* method is considered by many to be the best. All of the *Radial Basis Function* methods are exact interpolators, so they attempt to honor your data. You can introduce a shaping factor to all the methods in an attempt to produce a smoother surface. Regardless of the R^2 value, the *Radial Basis Function* is an exact interpolator.

Radial Basis Function Options Dialog

In the Grid Data dialog, specify *Radial Basis Function* as the *Gridding Method* and click the *Next* button to open the **Grid Data Radial Basis Function Options** dialog.



Set the type of Basis Function *and* Anisotropy *in the* ***Grid Data Radial Basis Function Options*** *dialog.*

Basis Function

Specify the function type from the *Basis Function* list. The basis kernel functions are analogous to variograms in Kriging. The basis kernel functions define the optimal set of weights to apply to the data points when interpolating a grid node. The available basis kernel functions are listed in the *Basis Function* list in the **Grid Data Radial Basis Function Options** dialog.

Type	Equation

Inverse Multiquadric	$B(h) = \frac{1}{\sqrt{h^2 + R^2}}$
Multilog	$B(h) = \log(h^2 + R^2)$
Multiquadric	$B(h) = \sqrt{h^2 + R^2}$
Natural Cubic Spline	$B(h) = \left(h^2 + R^2\right)^{3/2}$
Thin Plate Spline	$B(h) = \left(h^2 + R^2\right)\log\left(h^2 + R^2\right)$

where:

h	= the anisotropically rescaled, relative distance from the point to the node
R 2	= the shaping factor specified by the user

R^2 Parameter

The default value for R^2 in the Radial Basis Function gridding algorithm is calculated as follows:

(length of diagonal of the data extent)^2 / (25 * number of data points)

Search Neighborhood

Specify search rules. For more information about search rules, see Search.

Anisotropy

Set anisotropy settings if needed. For more information about anisotropy options, see Anisotropy.

Breaklines

The Breaklines section is used to add breaklines to the gridding process. Faults are not supported with *Radial Basis Function.*

References

A concise and readable introduction to *Radial Basis Function* interpolation can be found in Carlson and Foley (1991a). Given the clarity of presentation and the numerous examples, Hardy (1990) provides an excellent overview of the method, although this paper focuses exclusively on the special case of multiquadrics.

Carlson, R.E., and Foley, T.A. (1991a), *Radial Basis Interpolation Methods on Track Data*, Lawrence Livermore National Laboratory, UCRL-JC-1074238.

Carlson, R. E., and Foley, T. A. (1991b), *The Parameter R2 in Multiquadric Interpolation*, Computers Math. Applic, v. 21, n. 9, p. 29-42.

Franke, R. (1982), *Scattered Data Interpolation: Test of Some Methods*, Mathematics of Computations, v. 33, n. 157, p. 181-200.

Hardy, R. L. (1990), *Theory and Applications of the Multiquadric-BiHarmonic Method*, Computers Math. Applic, v. 19, n. 8/9, p. 163-208.

Powell, M.J.D. (1990), *The Theory of Radial Basis Function Approximation in 1990*, University of Cambridge Numerical Analysis Reports, DAMTP 1990/NA11.

Triangulation with Linear Interpolation

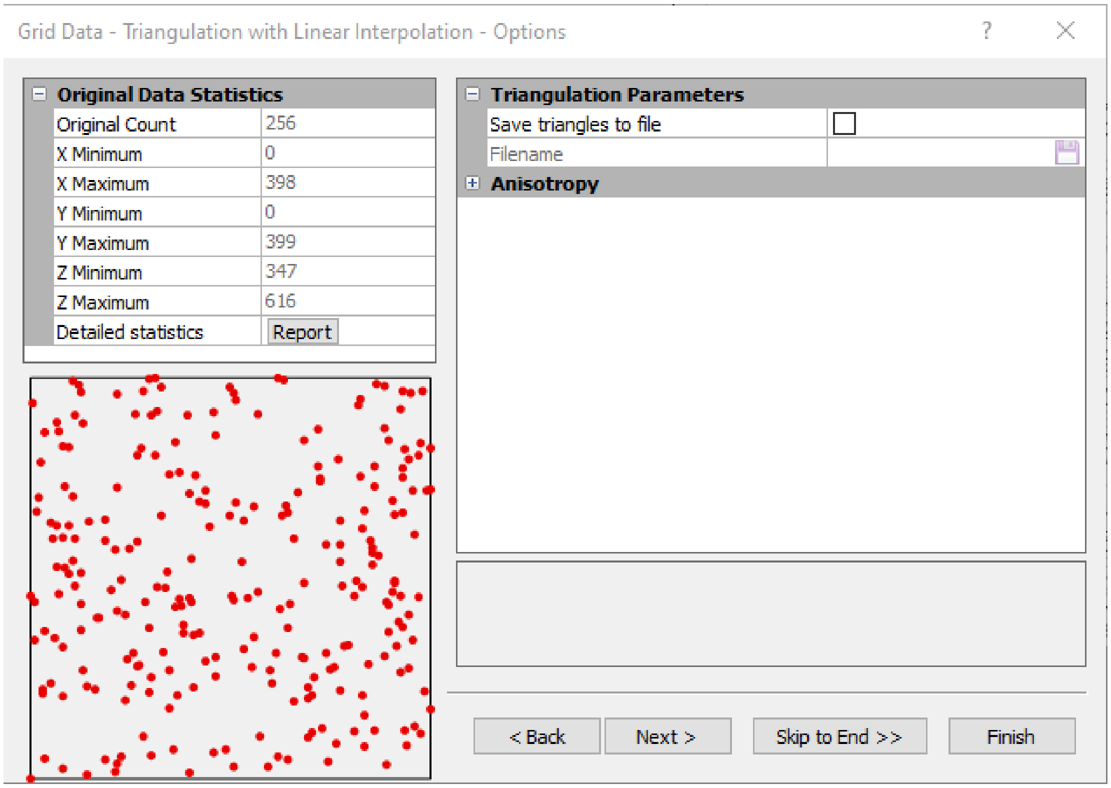
The *Triangulation with Linear Interpolation* method in **Surfer** uses the optimal Delaunay triangulation. The algorithm creates triangles by drawing lines between data points. The original points are connected in such a way that no triangle edges are intersected by other triangles. The result is a patchwork of triangular faces over the extent of the grid. This method is an exact interpolator.

Each triangle defines a plane over the grid nodes lying within the triangle, with the tilt and elevation of the triangle determined by the three original data points defining the triangle. All grid nodes within a given triangle are defined by the triangular surface. Because the original data are used to define the triangles, the data are honored very closely.

Triangulation with Linear Interpolation works best when your data are evenly distributed over the grid area. Data sets that contain sparse areas result in distinct triangular facets on the map.

Triangulation with Linear Interpolation Options Dialog

In the Grid Data dialog, specify *Triangulation with Linear Interpolation* as the *Gridding Method* and click the *Next* button to open the **Grid Data Triangulation with Linear Interpolation Options** dialog.



Set the anisotropy options and save Delaunay triangles in the ***Grid Data Triangulation with Linear Interpolation Options*** *dialog.*

Save Triangles

Check the *Save Triangles To* box to export the Delaunay triangles to a file. The resulting file can be loaded as a base map and combined with other maps (i.e. contour maps, 3D surface maps, etc.).

Click the button to open the **Export Triangles** dialog. Specify a *Save In* location, *File name*, and *Save as type*. The Delaunay triangle export formats include: .BLN, .BNA, .EMF, .GSB, .MIF, .PDF (Vector), and .WMF. Click the *Save* button to return to the **Grid Data Triangulation with Linear Interpolation Options** dialog.

Anisotropy

For more information about anisotropy options see Anisotropy.

References

The **Surfer** implementation of *Triangulation with Linear Interpolation* is based upon three papers. Lee and Schachter (1980) present a complete discussion of (Delaunay) triangulation, including the details of two algorithms and the underlying mathematical proofs. Lawson (1977) is equally informative. The algorithm presented in Guibas and Stolfi (1985) form the basis for the **Surfer** implementation.

Guibas, L., and J. Stolfi (1985), 'Primitives for the Manipulation of General Subdivisions and the Computation of Voronoi Diagrams', *ACM Transactions on Graphics*, v. 4, n. 2, p. 74-123.

Lawson, C. L. (1977), 'Software for C1 surface interpolation', *Mathematical Software III*, J. Rice (ed.), Academic Press, New York, p. 161-193.

Lee, D. T., and Schachter, B. J. (1980), 'Two Algorithms for Constructing a Delaunay Triangulation', *International Journal of Computer and Information Sciences*, v. 9, n. 3, p. 219-242.

Moving Average

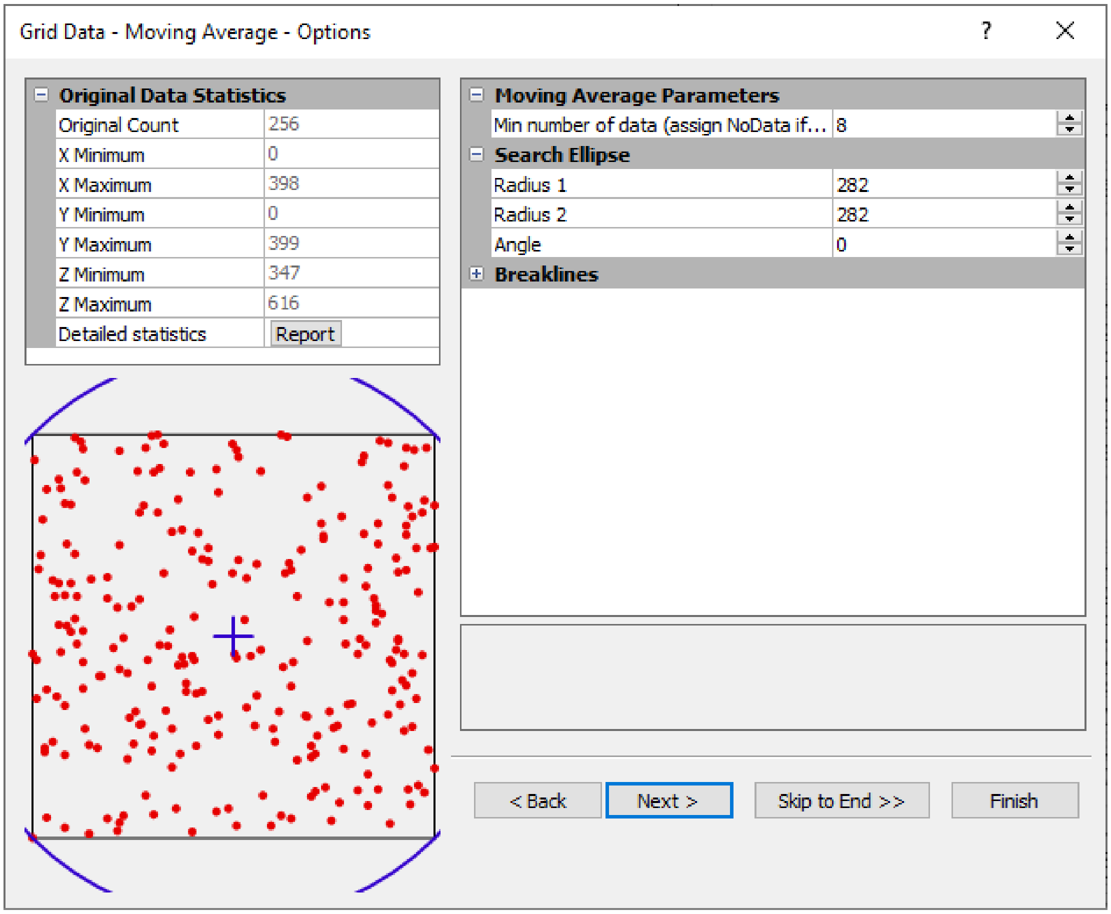
The *Moving Average* gridding method assigns values to grid nodes by averaging the data within the grid node's search ellipse.

Moving Average Tips

- The moving average gridding method can incorporate breakline data.
- The moving average gridding method is not recommended for generating maps from small and moderate-sized data sets. It is, however, a useful tool for characterizing and investigating large and very large spatial data sets.

Moving Average Options Dialog

In the Grid Data dialog, specify *Moving Average* as the *Gridding Method* and click the *Next* button to open the **Grid Data Moving Average Options** dialog.



Set the NoData options and the search ellipse in the ***Grid Data Moving Average Options*** *dialog.*

Minimum Number of Data

The **Minimum number of data (node is set to NoData if fewer)** value sets the specified number of points when interpolating a grid node. If the minimum number of points is not found, the NoData value is assigned to the grid node.

Search Neighborhood

Specify search rules. For more information about search rules, see Search.

Breaklines

The Breaklines section is used to add breaklines to the gridding process. Faults are not supported with *Moving Average.*

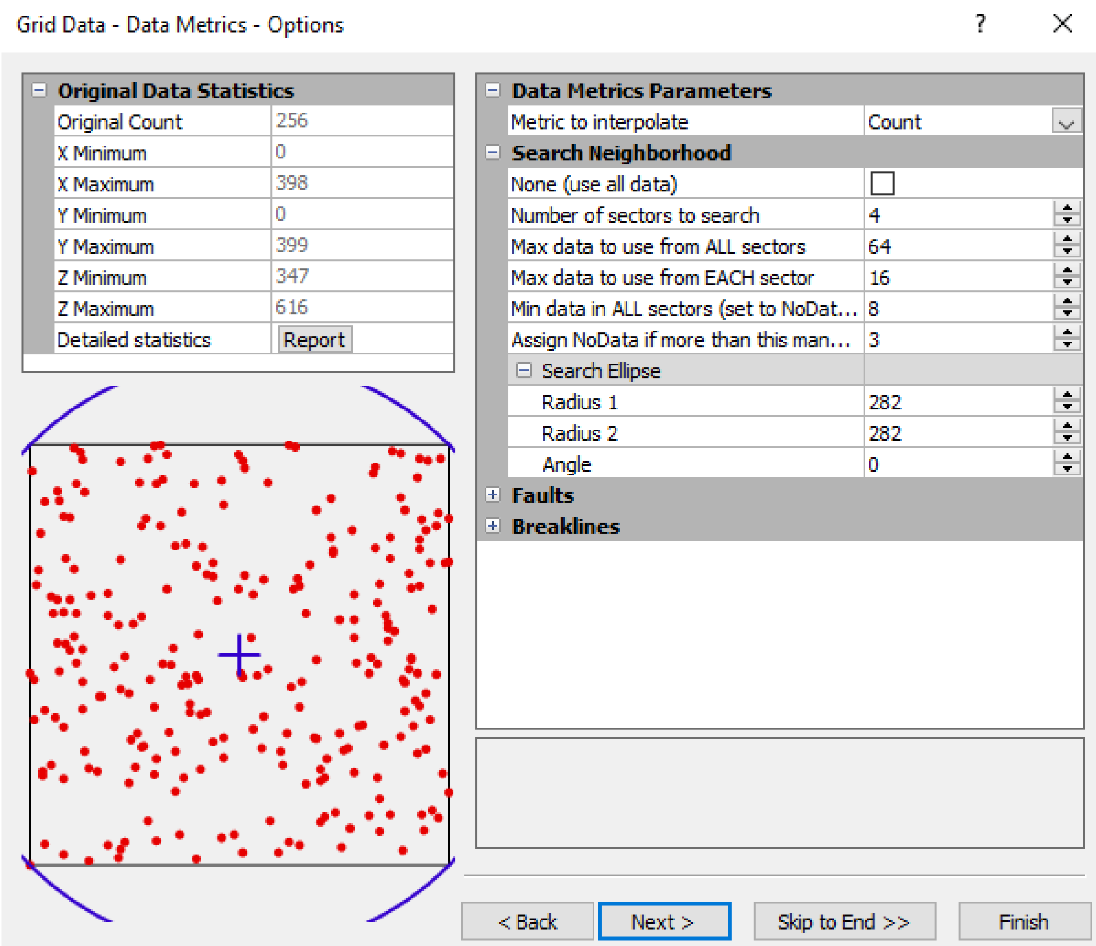
Data Metrics

The collection of data metrics gridding methods creates grids of information about the data on a node-by-node basis. The data metrics gridding methods are not, in general, weighted average interpolators of the Z values. For example, you can obtain information such as:

- The number of data points used to interpolate each grid node. If the number of data points used are fairly equal at each grid node, then the quality of the grid at each grid node can be interpreted.
- The standard deviation, variance, coefficient of variation, and median absolute deviation of the data at each grid node. These are measures of the variability in space of the grid, and are important information for statistical analysis.
- The distance to the nearest data point. For example, if the XY values of a data set are sampling locations, use the *Distance to Nearest* data metric to determine locations for new sampling locations. A contour map of the distance to the nearest data point, quantifies where higher sampling density may be desired.

Data Metrics Options Dialog

In the Grid Data dialog, specify *Data Metrics* as the *Gridding Method* and click the *Next* button to open the **Grid Data - Data Metrics - Options** dialog.



*Select the metric to interpolate in the **Grid Data - Data Metrics - Options** dialog.*

Data metrics use the local data set including breaklines, for a specific grid node for the selected data metric. The local data set is defined by the search parameters. These search parameters are applied to each grid node to determine the local data set. In the following descriptions, when computing the value of a grid node (r, c), the local data set S(r, c) consists of data within the specified search parameters centered at the specific grid node only. The set of selected data at the current grid node (r,c), can be represented by S(r,c), where

$$S(r,c) = \{(x_1, y_1, z_1), (x_2, y_2, z_2), \ldots, (x_n, y_n, z_n)\}$$

where *n* = number of data points in the local data set.

The Z(r,c) location refers to a specific node within the grid.

Data Metrics Descriptions

There are five groups of data metrics:

- Z Order Statistics
- Z Moment Statistics
- Other Z Statistics
- Data Location Statistics
- Terrain Statistics

Planar Grids

Data metrics is used to provide information about your data. After information is obtained from data metrics, it is likely you will grid the data again using one of the other gridding methods. When using data metrics, you will usually want to use the same output grid geometry, search, breakline, and fault parameters as when you grid the data using another gridding method.

When using some data metrics, a horizontal planar or sloping planar grid is generated. This is usually a result of the selected search method. For example, consider using *Demogrid.dat* and the *Count* data metric. The *Count* data metric determines the number of data points used in determining the grid node value. Since *Demogrid.dat* contains 47 data points, *No search (use all of the data)* is the default search method. Using *No search (use all of the data)* means for each calculated grid node, all 47 points are used in determining the grid node value. The resulting data metric grid is horizontal planar because all grid nodes have a Z value of 47. The grid report shows both the Z minimum and the Z maximum as 47.

Other data metrics can yield similar results. For example, if the search radius is large enough to include all of the data using *Terrain Statistics*, the moving average is computed with such a large search radius that the resulting grid will be a planar surface at the data average. When interpreting data metrics results, keep the gridding parameters and the data metrics calculation approach in mind.

Search Neighborhood

Specify search rules. For more information about search rules, see Search.

Breaklines and Faults

Specify a breakline and/or fault. For more information, see Breaklines and Faults.

Data Metrics Z Order Statistics

There are five groups of data metrics: *Z Order Statistics*, *Z Moment Statistics*, *Other Z Statistics*, *Data Location Statistics*, and *Terrain Statistics*.

The data at a grid node within the search parameters are sorted for the Z order statistics.

$$\{z[1], z[2], \ldots, z[n]\}$$

where $z_{[1]} \le z_{[2]} \le \ldots \le z_{[n]}$. Square brackets indicate ordered Z values.

Data Metric	**Definition**	**Equation**
Minimum	each nodal value is the minimum Z value of data selected by the specified sector search centered at that node	$Z(r,c) = \min(z_1, z_2, \ldots, z_n) = z_{[1]}$
Lower Quartile	each nodal value is the 25th percentile Z value of data selected by the specified sector search centered at that node	$Z(r,c) = z_{\left[\frac{n}{4}\right]}$
Median	each nodal value is the median Z value of data selected by the specified sector search centered at that node	$Z(r,c) = z_{\left[\frac{n}{2}\right]}$
Upper Quartile	each nodal value is the 75th percentile Z value of data selected by the specified sector search centered at that node	$Z(r,c) = z_{\left[\frac{3n}{4}\right]}$
Maximum	each nodal value is the maximum Z value of data selected by the specified sector search centered at that node	$Z(r,c) = \max(z_1, z_2, \ldots, z_n) = z_{[n]}$
Range	each nodal value is the difference between the maximum Z value and the minimum Z value for the data selected by the specified sector search centered at that node	$Z(r,c) = z_{[n]} - z_{[1]}$
Midrange	each nodal value is the average of the maximum Z value and the minimum Z value, for the data selected by the specified sector search centered at that node	$Z(r,c) = \frac{z_{[n]} + z_{[1]}}{2}$

Interquartile Range	The *Interquartile Range* data metric generates a grid for which each nodal value is the difference between the 75th percentile Z value and the 25th percentile Z value, for the data selected by the specified sector search centered at that node. This data metric shows the spatial variation of variability of the data, but it focuses on the middle fifty percent of the data only. Thus, it is insensitive to variations in the tails of the local distributions	$Z(r,c) = z_{\left[\frac{3n}{4}\right]} - z_{\left[\frac{n}{4}\right]}$

Data Metrics Z Moment Statistics

There are five groups of data metrics: *Z Order Statistics*, *Z Moment Statistics*, *Other Z Statistics*, *Data Location Statistics*, and *Terrain Statistics*. The *Z Moment Statistics* calculate information about the values at the nodes.

Data Metric	**Definition**	**Equation**
Mean	each nodal value is the arithmetic average of the data selected by the specified sector search centered at that node	$Z(r,c) = \frac{1}{n}\sum_{i=1}^{n} z_i$
Standard Deviation	each nodal value is the standard deviation of the data selected by the specified sector search centered at that node	$Z(r,c) = \sqrt{\frac{1}{n}\sum_{i=1}^{n}(z_i - \bar{z})^2}$ where $\bar{z}$ is the mean of the selected data
Variance	each nodal value is the variance of the data selected by the specified sector search centered at that node	$Z(r,c) = \frac{1}{n}\sum_{i=1}^{n}(z_i - \bar{z})^2$ where $\bar{z}$ is the mean of the selected data

Coef. of Variation	The *Coef. of Variation* data metric generates an output grid for which each nodal value is the local standard deviation divided by the local mean of the data selected by the specified sector search centered at that node. Note that this measure is useless for data whose local mean values are close to zero; this includes data that changes sign within the domain of interest.	$Z(r,c) = \frac{\text{standard deviation}(z)}{\text{mean}(z)}$

Data Metrics Other Z Statistics

There are five groups of data metrics: *Z Order Statistics*, *Z Moment Statistics*, *Other Z Statistics*, *Data Location Statistics*, and *Terrain Statistics*. The *Other Z Statistics* calculate additional information about the values at the nodes.

Data Metric	**Definition**	**Equation**
Sum	The *Sum* data metric generates an output grid for which each nodal value is the sum of the Z values of the data selected by the specified sector search centered at that node.	$Z(r,c) = \sum_{i=1}^{n} z_i$
M.A.D.	The Median Absolute Deviation (*M.A.D.*) data metric generates an output grid for which each nodal value is the median absolute deviation of the data selected by the specified sector search centered at that node.	$Z(r,c) = d_{\left[\frac{n}{2}\right]}$ where $d_i = \left\| z_i - z_{\left[\frac{n}{2}\right]} \right\|$ These d_i are sorted into ascending order and indexed as $\{d_{[1]}, d_{[2]}, \ldots, d_{[n]}\}$
R.M.S.	The Root Mean Square (*R.M.S.*) data metric generates an output grid for which each nodal value is the root mean square of the data selected by the specified sector search centered at that node.	$Z(r,c) = \sqrt{\frac{1}{n}\sum_{i=1}^{n} z_i^2}$

Data Metrics Data Location Statistics

There are five groups of data metrics: *Z Order Statistics*, *Z Moment Statistics*, *Other Z Statistics*, *Data Location Statistics*, and *Terrain Statistics*. The *Data Location Statistics* calculate information about the points around each of the nodes.

The separation distances between the current grid node and each of the selected data are used in the computation of the data location statistics. In the following discussion, let the location of the current grid node be represented as (x_0, y_0). The list of separation distances are defined as

$$R_i = \sqrt{(x_i - x_0)^2 + (y_i - y_0)^2}$$

The R_i are sorted in ascending order and indexed as

$$\{R_{[1]}, R_{[2]}, \ldots, R_{[n]}\}$$

Data Metric	Definition	Equation
Count	The *Count* data metric generates an output grid for which each nodal value is the number of data selected by the specified sector search centered at that node. Under most circumstances, the best interpolation results occur when the *Count* is approximately homogeneous throughout the area of interest.	$Z(r,c) = n$
Approximate Density	The *Approximate Density* data metric generates an output grid for which each nodal value is the number of data selected by the specified sector search centered at that node, divided by πR_n^2, where $R_{[n]}$ is the distance from the node to the farthest selected datum. The area over which the density is computed is bounded by the distance to the farthest selected datum. As such, when the data are relatively sparse, the computed density is slightly overstated.	$Z(r,c) = \frac{n}{\pi R_{[n]}^2}$
Distance to Nearest	each nodal value is the distance to the nearest datum selected by the specified sector search centered at that node	$Z(r,c) = R_{[1]}$

Distance to Farthest	each nodal value is the distance to the farthest datum selected by the specified sector search centered at that node	$Z(r,c) = R_{[n]}$
Median Distance	each nodal value is the median separation distance between the node and all of the data selected by the specified sector search centered at that node	$Z(r,c) = R_{\left[\frac{n}{2}\right]}$
Average Distance	each nodal value is the average separation distance between the node and all of the data selected by the specified sector search centered at that node	$Z(r,c) = \frac{1}{n}\sum_{i=1}^{n} R_i$
Offset Distance	each nodal value is the distance between the node and the centroid of all of the data selected by the specified sector search centered at that node	$Z(r,c) = \sqrt{(x_0 - x_c)^2 + (y_0 - y_c)^2}$ where the centroid coordinates (x_c, y_c) are $x_c = \frac{1}{n}\sum_{i=1}^{n} x_i$ and $y_c = \frac{1}{n}\sum_{i=1}^{n} y_i$

Terrain Statistics

There are five groups of data metrics: *Z Order Statistics*, *Z Moment Statistics*, *Other Z Statistics*, *Data Location Statistics*, and *Terrain Statistics*. The *Terrain Statistics* calculate information about the slope and aspect of the grid around each node.

Terrain statistics are all based upon a locally fitted planar surface. For each grid node, the specified sector search is performed. Then, using ordinary least squares, the following equation is fitted to the selected data:

$$z_i = Ax_i + By_i + C + \varepsilon_i$$

Least squares fitting is carried out using the data coordinates and it ignores faults. A sector search and the subsequent least squares fit are carried out for each grid node.

These data metrics are similar to Grids | Calculate | Calculus terrain modeling, however, data metrics work on a local subset of the data rather than the grid.

Data Metric	**Definition**	**Equation**

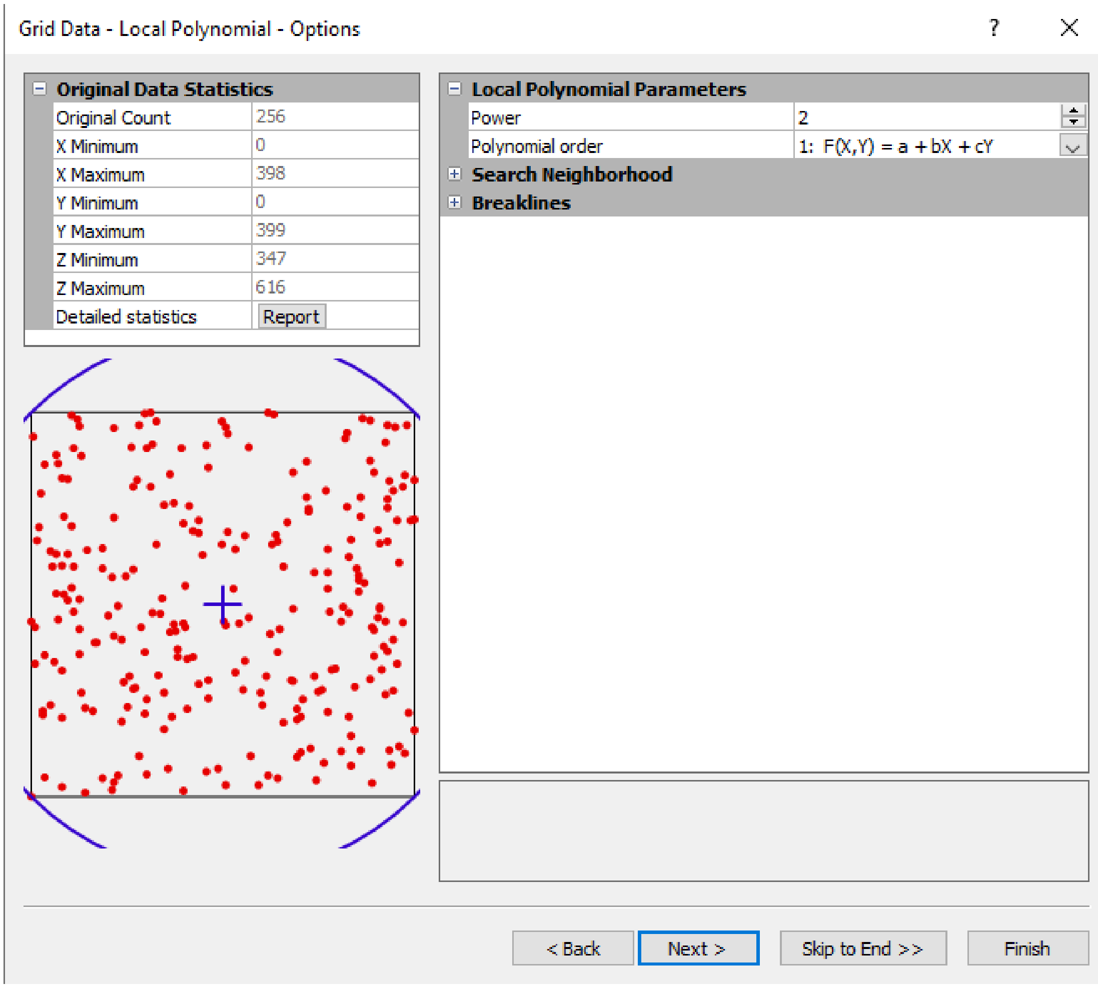
Terrain Slope	each nodal value is the terrain slope of the least-squares-fit plane of the data selected by the specified sector search centered at that node	$Z(r,c) = RadiansToDegrees(atan(hypot(A,B)))$
Terrain Aspect	each nodal value is the terrain aspect, as an angle from zero to 360 degrees, of the least-squares-fit plane of the data selected by the specified sector search centered at that node	$Z(r,c) = 270.0 - RadiansToDegrees(atan2(A,B))$

Local Polynomial

The *Local Polynomial* gridding method assigns values to grid nodes by using a weighted least squares fit with data within the grid node's search ellipse.

Local Polynomial Options Dialog

In the Grid Data dialog, specify *Local Polynomial* as the *Gridding Method* and click the *Next* button to open the **Grid Data Local Polynomial Options** dialog.



Choose a Power *and* Polynomial Order *in the* ***Grid Data Local Polynomial Options*** *dialog.*

Power

Set the *Power* to a number between 0 and 20. The *Power* parameter is used to assign the power for the weighted least square function, see Local Polynomial Math. This allows data closer to the grid node to have more impact on the value of the grid node than data farther away. This parameter is similar to the Inverse Distance to a Power gridding method.

Polynomial Order

Select a *Polynomial Order*, 1, 2, or 3.

1: $F(X,Y) = a + bX + cY$

2: $F(X,Y) = a + bX + cY + dXY + eX^2 + fX^2$

3: $F(X,Y) = a + bX + cY + dXY + eX^2 + fX^2 + gX^2Y + hXY^2 + iX^3 + iY^3$

Search Neighborhood

Specify search rules. For more information about search rules, see Search.

Breaklines

The Breaklines section is used to add *breaklines* to the gridding process. Faults are not supported with *Local Polynomial*.

Local Polynomial Math

For each grid node, the neighboring data are identified by the user-specified sector search. Using only these identified data, a local polynomial is fit using weighted least squares, and the grid node value is set equal to this value. Local polynomials can be order 1, 2, or 3.

The form of these polynomials are:

Order 1	$F(X,Y) = a + bX + cY$
Order 2	$F(X,Y) = a + bX + cY + dXY + eX^2 + fY^2$
Order 3	$F(X,Y) = a + bX + cY + dXY + eX^2 + fY^2 + gX^2Y + hXY^2 + iX^3 + jY^3$

The weighted least squares function weights data closer to the grid node higher and data further away lower. The weighting function depends on the search ellipse, the power, and the specific data geometry. The actual calculations for the weights are somewhat involved. Define TXX,

$$T_{XX} = \frac{\cos(\phi)}{R_1}$$
$$T_{XY} = \frac{\sin(\phi)}{R_1}$$
$$T_{YX} = \frac{-\sin(\phi)}{R_2}$$
$$T_{YY} = \frac{\cos(\phi)}{R_2}$$

where

ϕ	is the angle of the search ellipse
R_1	is search radius 1
R_2	is search radius 2

Define AXX,

$$A_{XX} = T_{XX}^2 + T_{YX}^2$$
$$A_{XY} = 2\left(T_{XX}T_{XY} + T_{YX}T_{YY}\right)$$
$$A_{YY} = T_{YY}^2 + T_{XY}^2$$

Note that these values (AXX,

Next, consider a datum at location (Xi,

$$dX = X_i - X_0$$
$$dY = Y_i - Y_0$$

then

$$R_i = \sqrt{A_{XX}dX^2 + A_{XY}dXdY + A_{YY}dY^2}$$

and finally,

$$W_i = \left(1 - R_i\right)^p$$

where Wiis the weight for datum i andpis the specified power.

Let the collection of neighboring data be enumerated as

$$\left\{\left(x_i, y_i, z_i\right) \text{for } i = 1,2,\ldots, N\right\}$$

The local least squares parameters are computed by minimizing the weighted sum of the squared residuals:

$$Minimize \sum_{i=1}^{N} W_i \left[F\left(x_i, y_i\right) - z_i\right]^2$$

Grid from Server

Click the **Grids | New Grid | Grid from Server** command or the  button to download data from an online web mapping server and create a new grid file. Clicking the **Grid from Server** command opens the Download Online Grids dialog.

The **Download Online Grids** dialog can also be accessed from the Open Grid dialog by clicking *Download*. The downloaded grid will be used to create a grid-based map layer or in the following grid command dialog, depending on which command was used to open the **Open Grid** dialog.

When using the **Download Online Grids** dialog, a server with appropriate data must be selected. Surfer currently supports four types of servers, web mapping server (WMS), Open Street Map server (OSM), web coverage server (WCS), and web feature server (WFS). WMS, OSM, WCS, and WFS servers provide different types data.

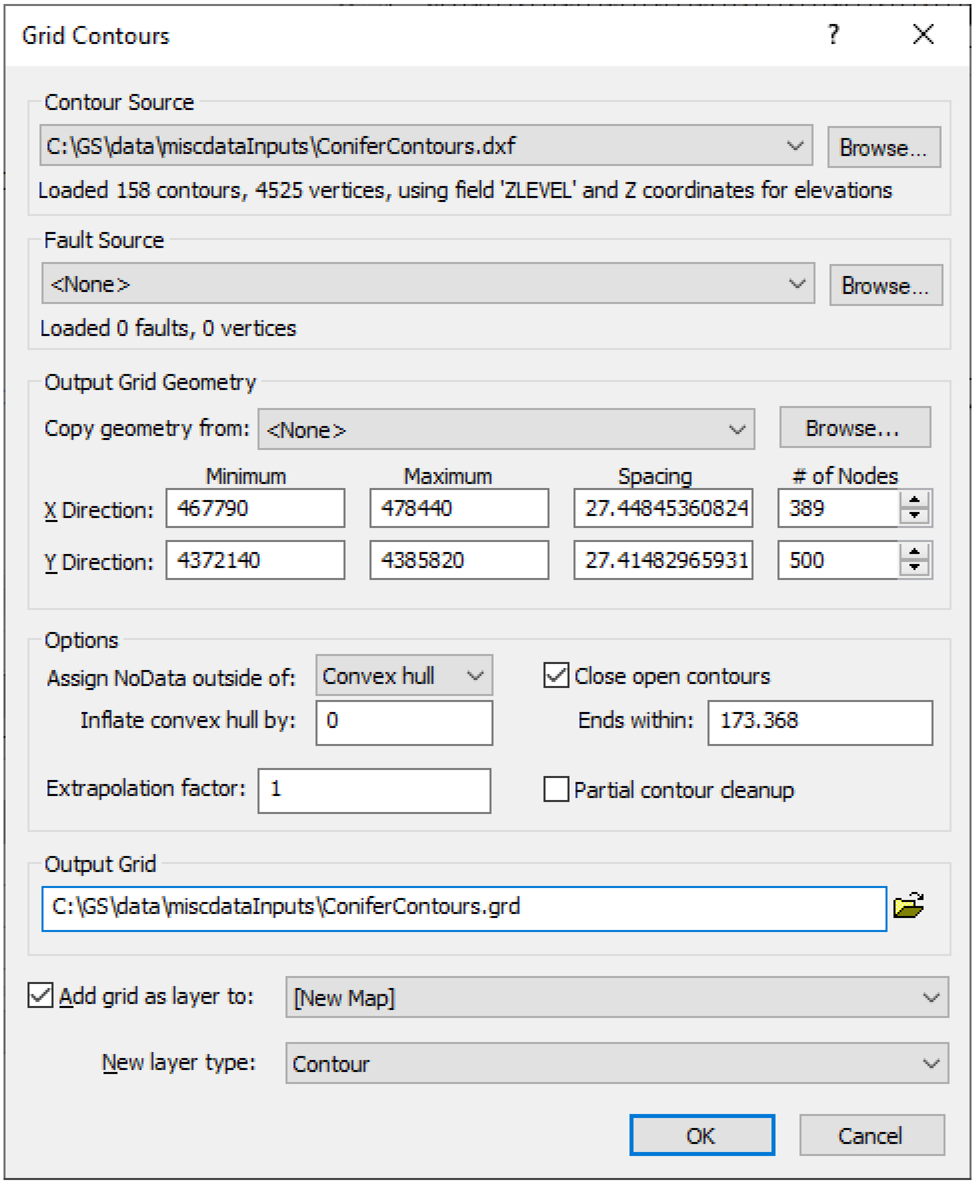
- When selecting a WMS or OSM server, an image is downloaded to be used as a grid file. There are no Z values associated with the image. When the image is used as a grid file, a value between 0 and 1 is assigned to each pixel in the image based on the color of that pixel. WMS and OSM servers are generally used to create base maps with the **Home | New Map | Base | Base from Server** command.
- When selecting a WCS server, data values are downloaded to be used as a grid file.
- When selecting a WFS server, vector data is downloaded with points, poly-lines, and/or polygons representing features. WFS servers cannot be used with the **Grid from Server** command.

Grid from Contours

Click the **Grids | New Grid | Grid from Contours** command or the button to create a grid file from contour lines. The contour lines can be loaded as a base layer or the vector file containing the contour lines can be used to generate a grid file. The **Grid from Contours** command uses the contour line polyline, polygon, 3D polyline and 3D polygon vertices and height attribute (commonly ZLEVEL) to create a grid. The **Grid from Contours** command algorithm interpolates grid node values as a linear combination of the two nearest contour values. The **Grid Contours** dialog is opened when the **Grids | New Grid | Grid from Contours** command is clicked.

Grid Contours Dialog

The **Grid Contours** dialog contains the input vector file or layer, input fault file or layer, gridding options, and output grid file settings.



*Create grid files from contour lines with the **Grid Contours** dialog.*

Contour Source

The *Contour Source* is the vector data file, base layer, or contour layer containing the contour lines from which to generate a grid. There are two methods for specifying the *Contour Source*:

- Select a map layer from the *Contour Source* list by clicking the current selection and then clicking the desired layer. Base layers are only included in the *Contour Source* list if the objects in the layer contain one or more numeric attribute fields.
- Specify a vector data file as the *Contour Source* by clicking Browse. The Open dialog is displayed. Navigate to the desired vector file and click *Open*. An

Import Options dialog may be displayed depending on the vector file format.

Once a *Contour Source* is selected, the Select Field dialog is displayed. Specify which attribute field contains elevation information in the **Select Field** dialog. For 3D polylines and 3D polygons, select *[None]* and check *Use Z coordinate for 3D geometry* to use the Z coordinates rather than the attributes. If an attribute field and *Use Z coordinate for 3D geometry* are both selected, the Z coordinate will take precedence. To exclude all 2D objects, select *[None]*.

The layer name or file path and name is displayed in the *Contour Source* field when a *Contour Source* has been loaded. The number of contours, number of vertices, and elevation field selection are displayed below the *Contour Source*.

Fault Source

The *Fault Source* is the vector data file or base layer containing any faults to include in the grid generation. Select a base layer from the *Fault Source* list by clicking the current selection and then clicking the desired layer. Specify a vector data file as the *Fault Source* by clicking *Browse*. The Open dialog is displayed. Navigate to the desired vector file and click *Open*. An **Import Options** dialog may be displayed depending on the vector file format.

The layer name or file path and name is displayed in the *Fault Source* field when a *Fault Source* has been loaded. The number of faults and number of vertices is displayed below the *Fault Source*.

Supported Formats

The following file formats are supported in the *Contour Source* and *Fault Source* fields:

BLN Golden Software Blanking
BNA Atlas Boundary
DDF SDTS TVP
DXF AutoCAD Drawing
E00 Esri ArcInfo Export Format
GSB Golden Software Boundary
GSI Golden Software Interchange
MIF MapInfo Interchange Format
SHP Esri Shapefile

Output Grid Geometry

The *Output Grid Geometry* section defines the grid limits and grid density.

Copy Geometry

The *Copy geometry from* option copies the grid geometry from an existing map layer or grid file. This option is useful when creating grids that will become overlaid map layers, processed with the Grid Math command, or used to calculate a volume between two surfaces. The **Math** and **Volume** commands require the input grids to have the same geometry.

To copy the geometry from an existing layer, select the layer in the *Copy geometry from* list. To copy the geometry from a grid file, click *Browse* and select the file in the Open Grid dialog. Select <*None*> to return the *Output Grid Geometry* options to their default values and to manually edit the grid geometry.

Minimum and Maximum X and Y Coordinate (Grid Limits)

Grid limits are the minimum and maximum X and Y coordinates for the grid. **Surfer** computes the minimum and maximum X and Y values from the *Contour Source* layer or data file. These values are used as the default minimum and maximum coordinates for the grid.

Grid limits define the X and Y extent of contour maps, color relief maps, vector maps, 3D wireframes, and 3D surfaces created from grid files. When creating a grid file, you can set the grid limits to the X and Y extents you want to use for your map. Once a grid file is created, you cannot produce a grid-based map larger than the extent of the grid file. If you find you need larger grid limits, you must regrid the data. You can, however, read in a subset of the grid file to produce a map smaller than the extent of the grid file.

When either the X, Y, or Z value is in a date/time format, the date/time values are converted and stored in the grid as numbers.

Spacing and # of Nodes (Grid Density)

Grid density is usually defined by the number of columns and rows in the grid, and is a measure of the number of grid nodes in the grid. The *# of Nodes* in the *X Direction* is the number of grid columns, and the *# of Nodes* in the *Y Direction* is the number of grid rows. The direction (*X Direction* or *Y Direction*) that covers the greater extent (the greater number of data units) is assigned 500 grid nodes by default. The number of grid nodes in the other direction is computed so that the grid nodes *Spacing* in the two directions are as close to one another as possible.

By defining the grid limits and the number of rows and columns, the *Spacing* values are automatically determined as the distance in data units between adjacent rows and adjacent columns.

You can also increase or decrease the grid density by using the Grids | Edit | Spline Smooth, Grids | Resize | Extract, or Grids | Resize | Mosaic commands after a grid is created. See the Grid Data topic for information on memory requirements for large and/or dense grids and a grid geometry example.

Options

The *Options* section of the **Grid Contours** dialog contains options for assigning the NoData value outside the convex hull of the data or outside of the alpha value, control over extrapolation in peaks and basins, contour closing, and contour cleanup.

Assign NoData

Convex Hull

Select *Convex hull* from the *Assign NoData outside of* list to assign the NoData value to the grid nodes outside the convex hull of the data. The convex hull of a data set is the smallest convex polygon containing all the data. The convex hull can be thought of as a rubber band that encompasses all data points. The rubber band only touches the outside points. Areas inside the convex hull without data are still gridded. See the Grid Data topic for an example of assigning NoData outside the convex hull of the data.

If *Convex hull* is selected from the *Assign NoData outside of* list, the **Grid Contours** dialog displays the *Inflate convex hull by* box. Enter a value to expand or contract the convex hull. When set to zero, the boundary connects the outside data points exactly. When set to a positive value, the area assigned the NoData value is moved outside the convex hull boundary by the number of map units specified. When set to a negative value, the area assigned the NoData value is moved inside the convex hull boundary by the number of map units specified. Values are in horizontal (X) map units. If the value is set to a large positive value, the grid values may extend all the way to the minimum and maximum X and Y limits of the grid, essentially overriding the *Assign NoData outside convex hull of data* option. If the value is set to a large negative value, the entire grid may be assigned the NoData value, resulting in no grid file being created.

Alpha Shape

Select *Alpha shape* from the *Assign NoData outside of* list to assign the NoData value to the grid nodes outside the alpha value of the data. Select *Alpha shape* instead of *Convex hull* to have a tighter boundary, especially if a boundary could form concave areas. If *Alpha shape* is selected from the *Assign NoData outside of* list, the **Grid Contours** dialog displays the *Alpha value* box. Enter a value to define the radius of the circles that are created by the edges of the triangulation of the points. Larger alpha values create a more convex boundary and smaller alpha numbers create a tighter, concave boundary. If too low or too high of an alpha value was entered, a message will appear stating that the grid could not be assigned NoData because of the alpha value. For more information on alpha shapes or to determine an appropriate alpha value, see the Alpha Shape help topic.

Extrapolation Factor

The *Extrapolation factor* controls the rate of change in the Z for grid node values with only one bounding contour. These grid nodes are usually located near peaks and basins. The slope between the nearest two contour lines is multiplied by the *Extrapolation factor* to perform linear extrapolation. Type the desired factor in the *Extrapolation factor* field.

The *Extrapolation factor* must be greater than or equal to zero, and the default *Extrapolation factor* is one. An *Extrapolation factor* of zero generates a grid with flat peaks and basins. An *Extrapolation factor* less than one will lessen the rate of change in Z with distance from the nearest contour and generate lower peaks and shallower basins. An *Extrapolation factor* greater than one will increase the rate of change in Z with distance from the nearest contour and generate higher peaks and deeper basins.

Close Open Contours

Check the *Close open contours* check box to automatically close any contour lines with ends within the specified *Ends within* tolerance. Contours will not be closed if the added section intersects any existing contours. Contours will also not be closed if the ends are further apart than the *Ends within distance.*

The *Close open contours* option is enabled by default. This improves the algorithm performance near the edges of the map and for any open contours in the map interior. The default *Ends within* distance is 1% of the diagonal distance of the grid. Specify an *Ends within* distance in horizontal (X) map units by typing the value in the *Ends within* field.

Clear the *Close open contours* check box to generate the grid from the contour lines in the layer or vector file without closing open contours. The *Ends within* field is not available when *Close open contours* is not checked.

Partial Contour Cleanup

Check the *Partial contour cleanup* check box to improve performance around open contours that terminate in the interior of the map, especially near the edges of the map or when the contour lines indicate faulting. However for some input layers or files, the *Partial contour cleanup* option will decrease performance in the resulting grid. By default the *Partial contour cleanup* check box is not checked.

The *Partial contour cleanup* option changes the algorithm to ignore "invisible" contour lines when calculating grid node values. First, the algorithm determines the nearest contour levels. Next, the algorithm determines if any of the levels are blocked from view by nearer levels. If this is the case, the occluded contour levels are ignored and the algorithm treats the grid node value as a bounded point (see the *Extrapolation factor* section).

Output Grid

Choose a path and file name for the grid file in the *Output Grid* group. You can type a path and file name, or click the  button to browse to a new path and enter a file name in the **Save Grid As** dialog.

Add New Map or Layer

Check *Add grid as layer to* to add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

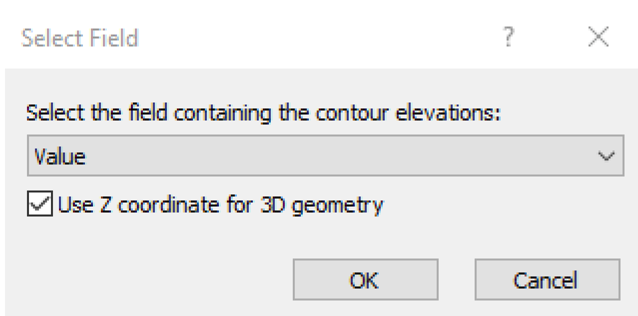
Notes: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command. When exporting to a BLN or DXF file, the Z coordinates for 3D polylines are not exported.

Reference for Grid from Contours

See Koshel, S. M., *Algorithm for Topologically Correct Gridding of Contour Data*, Department of Cartography and Geoinformatics, Faculty of Geography, Lomonosov Moscow State University, for more information regarding the gridding algorithm.

Select Field Dialog

The **Select Field** dialog specifies which attribute value contains elevation information for the contour lines in the map layer or vector data file.



*Specify which field contains elevation information in the **Select Field** dialog.*

For 2D polylines and polygons, click the current attribute in the *Select the field containing the contour elevations* field and select the desired attribute from the list.

With 3D polylines and 3D polygons, check the *Use Z coordinate for 3D geometry* check box to use the Z coordinate, or unclick the *Use Z coordinate for 3D geometry* and select the attribute to use from *Select the field containing the contour elevations*.

To ignore 2D objects when both 2D objects and 3D objects are present, set *Select the field containing the contour elevations* to *[None]*.

If both *Use Z coordinate for 3D geometry* is checked and an attribute is selected, 3D objects will use the Z coordinate and the 2D objects will use the attribute. For 3D objects, the Z coordinate always takes precedent.

Click *OK* when you have selected the desired field.

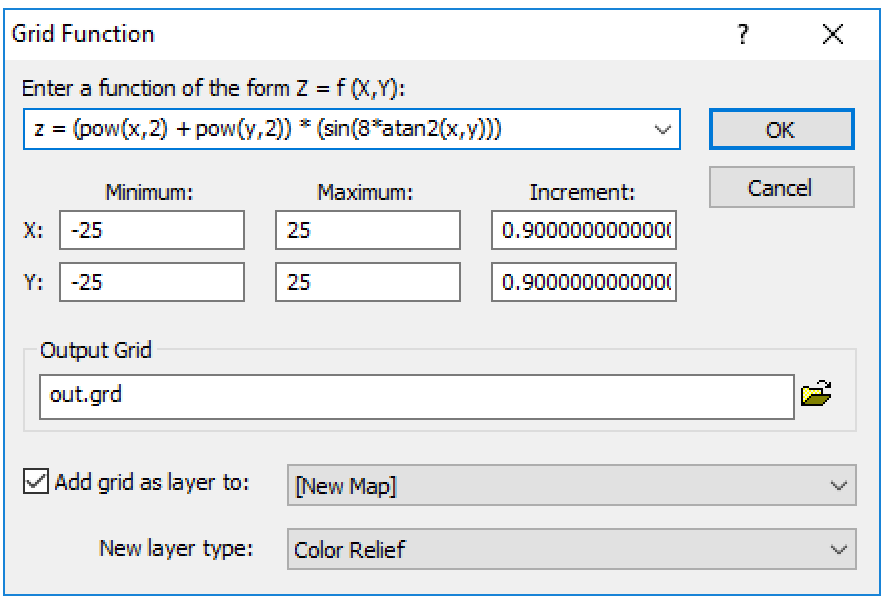
Click *Cancel* to return to the Grid Contours dialog without selecting the elevation field. If you click *Cancel* the map layer or vector file will not be loaded.

Grid Function

The **Grids | New Grid | Function** command creates a grid file from a user-defined two variable equation of the form Z = f(X,Y). The density of the generated grid is a function of the *Minimum*, *Maximum*, and *Increment* values for both X and Y. The **Grids | New Grid | Function** command can use any of the mathematical functions. Grids created as functions can be plotted in the same way as grids created using the **Grid Data** command.

Grid Function Dialog

Click the **Grids | New Grid | Function** command or the button to open the **Grid Function** dialog.



Set the mathematical function, the minimum and maximum X and Y values, and the function increment values in the ***Grid Function*** *dialog.*

Enter a Function

Enter the function of interest into the *Enter a function of the form Z = f (X,Y)* box. The function calculation is repeated for each Z value to be written to the grid file. The number of grid nodes in the output grid and, therefore, the number of calculations to be performed, is based on the *Minimum*, *Maximum* and *Increment* values specified.

To use a stored function, click the next to the current function. This will display the ten most recent functions used. The functions are stored in the registry, so the equations are stored between **Surfer** sessions. You can also start typing the function in the function box. If the function is in the ten function history, the entire function will auto-complete.

Minimum X and Y

In the *Minimum* X and Y boxes, specify the beginning values used in the specified function. The first grid node, defined by these boxes, is calculated based on these values. These values also specify the lower X and Y limits in the grid file.

Maximum X and Y

In the *Maximum* X and Y boxes, specify upper values to be applied to the function and the upper X and Y limits for the grid.

Increment X and Y

In the *Increment* boxes, specify the amount to step in the X and Y directions for each grid line. This is similar to the grid *Spacing* setting in the Grid Data dialog.

Output Grid

Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the button and specify the path and file name for the grid file in the Save Grid As dialog.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

Producing a Grid File from a Regular Array of XYZ Data

When your XYZ data are collected or generated on regular intervals it is possible to produce a grid file that uses your values directly and does not interpolate the values for the grid nodes. You can use the Grid Data command to create a grid file in the correct format.

When you have a complete array of XYZ data (or a nearly complete set of data with only a few "holes"), you can use the Nearest Neighbor gridding method to convert your data directly to a grid file. The *Nearest Neighbor* method does not interpolate data but merely picks the closest point and assigns that value to the grid node.

To use the **Home | Grid Data | Grid Data** command to produce a .GRD grid file from regularly spaced data:

1. Create an XYZ data .DAT file from your data.
2. Select the Home | Grid Data | Grid Data command.
3. Specify the name of the XYZ data file in the **Open** dialog and click *Open*.
4. In the **Grid Data** dialog, select the *Nearest Neighbor* gridding method from the *Gridding Method* list. During gridding, the nearest Z value from the data file is assigned to the grid node.
5. Set the *Spacing* values in the *Output Grid Geometry* group to match the spacing of your data in the X and Y directions. This assures that the grid nodes coincide with your data. For example, if your data are separated by 100 meters over the extent of your map, enter 100 for the *X Direction* and *Y Direction* boxes.
6. Click *OK* in the **Grid Data** dialog and the grid file is created.

You can also produce grid files directly from an evenly spaced array of Z values. When your Z values are organized correctly in an ASCII file, you can use the file directly as a TXT Formatted Text Grid file. Alternatively, add some header information identifying the data as a grid file, specify the limits of the data, and then save the file. The ASCII grid file format is given in Surfer 6 Text Grid Format.

Chapter 5 - Introduction to Variograms

Variogram Overview

Surfer includes an extensive variogram modeling subsystem. This capability was added to **Surfer** as an integrated data analysis tool. The primary purpose of the variogram modeling subsystem is to assist you in selecting an appropriate variogram model when gridding with the kriging algorithm. Variogram modeling may also be used to quantitatively assess the spatial continuity of data even when the kriging algorithm is not applied.

Surfer's variogram modeling feature is intended for experienced variogram users who need to learn **Surfer**'s variogram modeling features. The novice variogram user may find the following four authors helpful: Cressie (1991), Isaaks and Srivastava (1989), Kitanidis (1997), and Pannatier (1996). Please refer to Suggested Reading for full references for each of the previous books. If you do not understand variograms or if you are unsure about which model to apply, use **Surfer**'s default linear variogram with the kriging algorithm.

Variogram modeling is not an easy or straightforward task. The development of an appropriate variogram model for a data set requires the understanding and application of advanced statistical concepts and tools: this is the *science* of variogram modeling. In addition, the development of an appropriate variogram model for a data set requires knowledge of the tricks, traps, pitfalls, and approximations inherent in fitting a theoretical model to real world data: this is the *art* of variogram modeling. Skill with the science and the art are both necessary for success.

The development of an appropriate variogram model requires numerous correct decisions. These decisions can only be properly addressed with an intimate knowledge of the data at hand, and a competent understanding of the data genesis (i.e. the underlying processes from which the data are drawn). The cardinal rule when modeling variograms is *know your data*.

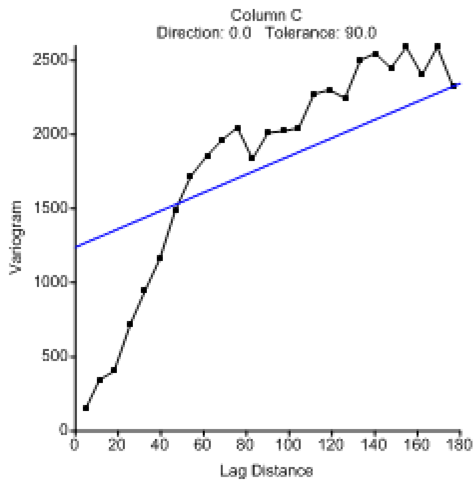
Variogram

The variogram is a measure of how quickly things change on the average. The underlying principle is that, on the average, two observations closer together are more similar than two observations farther apart. Because the underlying processes of the data often have preferred orientations, values may change more quickly in one direction than another. As such, the variogram is a function of direction.

The variogram is a three dimensional function. There are two independent variables (the direction θ, the separation distance h) and one dependent variable (the variogram value g(q,h)). When the variogram is specified for Kriging we give the sill, range, and nugget, but we also specify the anisotropy information. The variogram grid is the way this information is organized inside the program.

The variogram (XY plot) is a radial slice (like a piece of pie) from the variogram grid, which can be thought of as a "funnel shaped" surface. This is necessary because it is difficult to draw the three-dimensional surface, let alone try to fit a three dimensional function (model) to it. By taking slices, it is possible to draw and work with the directional experimental variogram in a familiar form - an XY plot.

Remember that a particular directional experimental variogram is associated with a direction. The ultimate variogram model must be applicable to all directions. When fitting the model, the user starts with numerous slices, but must ultimately mentally integrate the slices into a final 3D model.



This is a variogram using the default parameters and the sample data file VARIO1.DAT.

Variogram Components

Kriging and Variograms

Variogram Grid

Variogram Properties

Experimental Page

Model Page

Statistics Page

Plot Page

Kriging and Variograms

The kriging algorithm incorporates four essential details:

1. When computing the interpolation weights, the algorithm considers the spacing between the point to be interpolated and the data locations. The algorithm considers the inter-data spacings as well. This allows for declustering.

2. When computing the interpolation weights, the algorithm considers the inherent *length scale* of the data. For example, the topography in Kansas varies much more slowly in space than does the topography in central Colorado. Consider two observed elevations separated by five miles. In Kansas it would be reasonable to assume a linear variation between these two observations, while in the Colorado Rockies such an assumed linear variation would be unrealistic. The algorithm adjusts the interpolation weights accordingly.

3. When computing the interpolation weights, the algorithm considers the inherent trustworthiness of the data. If the data measurements are exceedingly precise and accurate, the interpolated surface goes through each and every observed value. If the data measurements are suspect, the interpolated surface may not go through an observed value, especially if a particular value is in stark disagreement with neighboring observed values. This is an issue of *data repeatability.*

4. Natural phenomena are created by physical processes. Often these physical processes have preferred orientations. For example, at the mouth of a river the coarse material settles out fastest, while the finer material takes longer to settle. Thus, the closer one is to the shoreline the coarser the sediments, while the further from the shoreline the finer the sediments. When computing the interpolation weights, the algorithm incorporates this natural *anisotropy.* When interpolating at a point, an observation 100 meters away but in a direction parallel to the shoreline is more likely to be similar to the value at the interpolation point than is an equidistant observation in a direction perpendicular to the shoreline.

Items two, three, and four all incorporate something about the underlying process from which the observations were taken. The *length scale*, *data repeatability*, and *anisotropy* are not a function of the data locations. These enter into the Kriging algorithm via the variogram. The *length scale* is given by the variogram *range* (or *slope*), the *data repeatability* is specified by the nugget effect, and the *anisotropy* is given by the anisotropy.

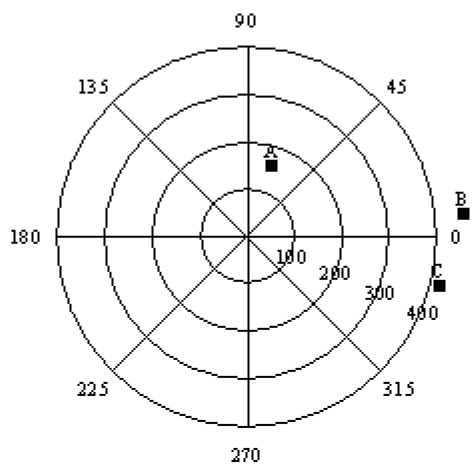
Variogram Grid

Surfer uses a variogram grid as a fundamental internal data representation, in lieu of a pair comparison file. The pair comparison file can be extremely large for moderately sized data sets. For example, 5000 observations create N(N-1)/2 pairs (12,497,500). Each pair requires 16 bytes of information for a pair comparison file, so a 5000-observation pair comparison file would take approximately 191 megabytes of memory to merely hold the pair comparison information. The time to read and search through this large file makes this approach impractical for many Surfer users.

Computational speed and storage are gained by using the variogram grid approach. Once the variogram grid is built, any experimental variogram can be computed instantaneously. This is independent of the number of observations. However, the ability to carry out on-the-fly editing of variograms on a pair-by-pair basis is lost by using the variogram grid approach in **Surfer**.

Unlike the grids used elsewhere in **Surfer**, which are rectangular grids, variogram grids are polar grids. Polar grids cannot be viewed in **Surfer**, and are only used within the context of variogram computation. The first coordinate in a variogram grid is associated with the polar angle, and the second coordinate is associated with the radial distance out from the origin.

Consider the following variogram grid:



Polar variogram grid for three pairs (A, B, and C) of observation points, showing separation angles and separation distances.

There are eight angular divisions: {0°, 45°, 90°, 135°, 180°, 225°, 270°, 315°} and four radial divisions: {100, 200, 300, 400}. Thus, there are 32 individual cells in this variogram grid. Users familiar with VarioWin® will notice similarities between **Surfer**'s variogram grid and the "variogram surface" in VarioWin® 2.2. In **Surfer**, only the upper half of the grid is used. See the General Page for a more detailed explanation.

Consider the following three observation locations: {(50,50), (100, 200), and (500,100)}. There are three observations, so there are 3*(3-1)/2 = 3 pairs. The pairs are:

A (50,50), (100,200)

B (50,50), (500,100)

C (100,200), (500,100)

Each pair is placed in a particular cell of the variogram grid based upon the separation distance and separation angle between the two observation locations.

The separation distance is:

$$h = \sqrt{(X_2 - X_1)^2 + (Y_2 - Y_1)^2}$$

The separation angle is:

$$\theta = \arctan\left[\frac{Y_2 - Y_1}{X_2 - X_1}\right]$$

Using the above equations, the separation angle for the first pair of observations {(50,50), (100,200)} is 71.57 degrees and the separation distance is 158.11. This pair is placed in the cell bounded by the 100 circle on the inside, the 200 circle on the outside, the 45° line in the clockwise direction, and the 90° line in the counterclockwise direction. The location of this pair in the variogram grid is shown on the previous page as point A.

Pair	*Separation Angle*	*Separation Distance*
A	*71.57*	*158.11*
B	*6.34*	*452.77*
C	*-14.04*	*412.31*

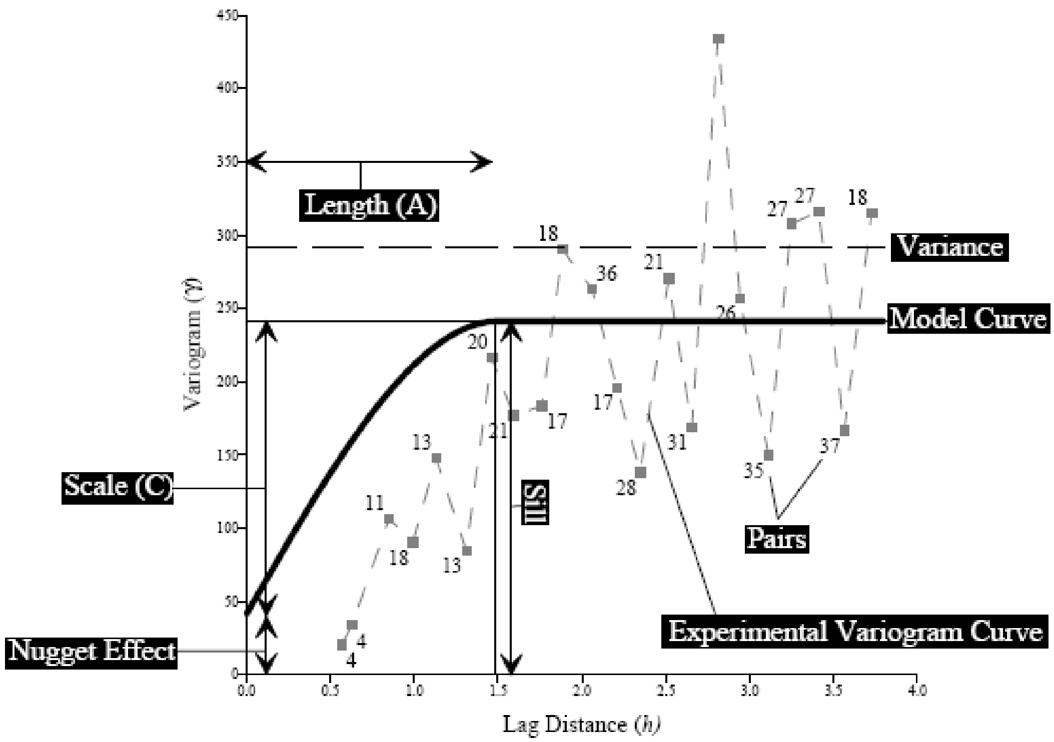
The separation angle and separation distance for each pair are calculated using the previous equations.

Since the separation distance of pairs B and C are greater than the radius of the largest circle (400), these pairs fall outside of the variogram grid. Pairs B and C are not included in the variogram grid and therefore, not included in the variogram. Using the above equations, every pair is placed into one of the variogram grid cells or it is discarded if the separation distance is too large.

For a large data set there could be millions of pairs (or more) and the associated pair comparison file would be very large. On the other hand, with the variogram grid in the example above there are only 32 grid cells regardless of the number of pairs contained in a particular grid cell. Herein lies the computational saving of the variogram grid approach. It is not necessary that every pair is stored in a variogram grid cell; each variogram grid cell stores only a small set of summary statistics which represent all of the pairs contained within that cell.

Variogram Model

The variogram model mathematically specifies the spatial variability of the data set and the resulting grid file. The interpolation weights, which are applied to data points during the grid node calculations, are direct functions of the variogram model.



A variogram model demonstrating the variogram parameters.

Nugget Effect

The *Nugget Effect* quantifies the sampling and assaying errors and the short scale variability (i.e. spatial variation occurring at distance closer than the sample spacing).

Partial Sill

The *Partial sill (C)* is the vertical scale for the structured component of the variogram. Each component of a variogram model has its own scale.

Sill

The *Sill* is the total vertical scale of the variogram (Nugget Effect + *Sum of all component Scales*). *Linear*, *Logarithmic*, and *Power* variogram models do not have a sill.

Range

The *Range (Length)* is the horizontal range of the variogram. (Some variogram models do not have a range parameter; e.g., the linear model has a slope instead.)

Variance

The *Variance* is the mean squared deviation of each value from the mean value. Variance is indicated by the dashed horizontal line in the diagram shown above.

$$\mathrm{var} = \frac{\sum_{i=1}^{N} z_i^2}{N} - \left(\frac{\sum_{i=1}^{N} z_i}{N} \right)^2$$

where: N is the number of data

Pairs

Pairs represents the average variogram value for the group of pairs separated by a specified distance (lag width). The number adjacent to the square symbols indicates the number of pairs within each lag distance.

Model Curve

Model Curve shows the shape of the variogram model.

Experimental Curve

Experimental Curve displays the groups of variogram pairs on a plot of separation distance versus the estimated variogram.

For more information on the nugget effect, scale, and length settings, see the Model Page.

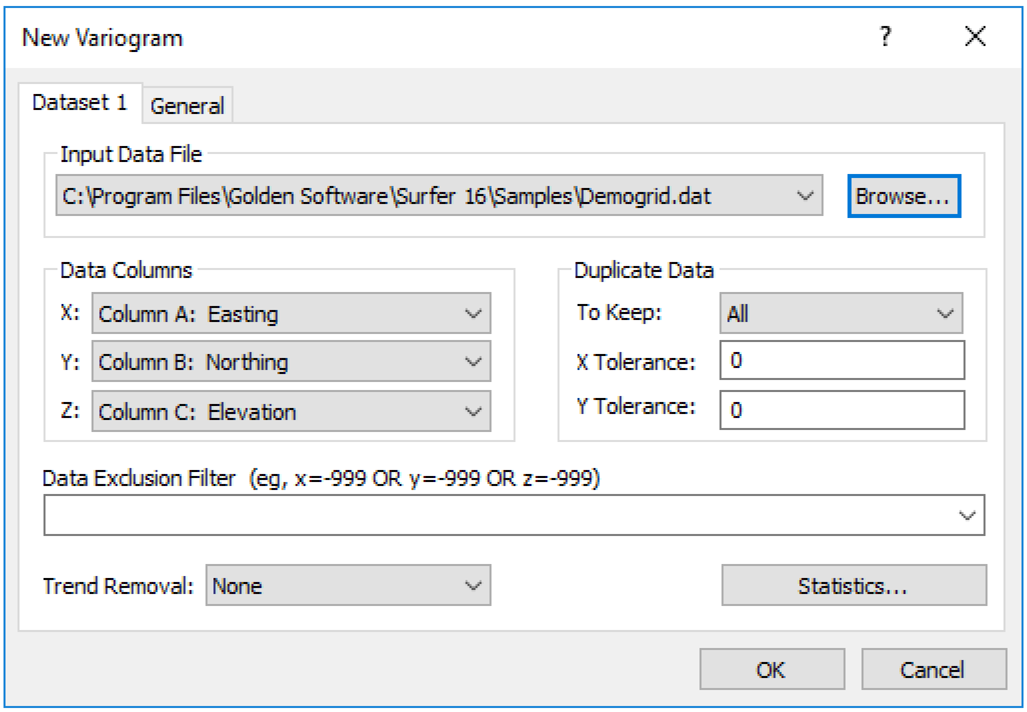
New Variogram

Click the **Grids | New Grid | Variogram | New Variogram** command to create a new variogram from a data file in the plot window. When you create a new variogram, the **New Variogram** dialog is displayed after the data file is selected.

The **New Variogram** dialog contains options for setting up the variogram grid used for the variogram. Once the variogram is created, the variogram properties, such as the variogram model used, are set through the Variogram Properties located in the Properties window. Once the variogram grid is created, it is used for the duration of the variogram modeling effort. See Variogram Grid for more information on variogram grids.

New Variogram Dialog

Click the **Grids | New Grid | Variogram | New Variogram** command or the  button from the plot window to open the Open Data dialog. Select a data file and click *Open* to open the **New Variogram** dialog.



*Specify options for a new variogram in the **New Variogram** dialog.*

Dataset Page

Select data for a new variogram on the Dataset page.

General Page

Define the variogram grid on the General page.

Creating a Variogram

The following directions contain the general steps to create a variogram. Decisions about the various options should be made based on knowledge of the data and based on knowledge of variograms. As such, specific modeling recommendations can not be made.

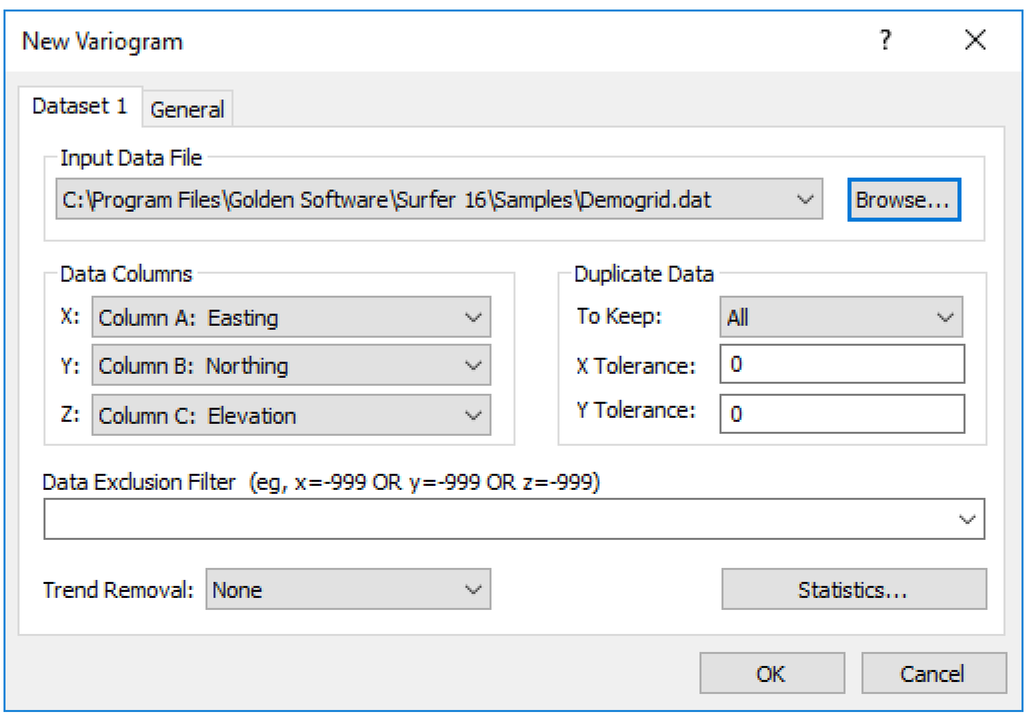
1. Click **Grids | New Grid | Variogram | New Variogram** .
2. Select a data file in the **Open Data** dialog and click *Open*.
3. Set the data columns to use on the Data page in the **New Variogram** dialog.
4. Select the variogram grid settings on the General page.
5. Click *OK* to create the variogram.
6. Once the variogram has been created, click once on the variogram to select it. The Properties window allows you to select a variogram options, including a variogram model other than the default linear model.

New Variogram Dataset Properties

The **New Variogram** dialog is used to create a new variogram. Click Grids | New Grid | Variogram | New Variogram and select a data file to open the **New Variogram** dialog.

Dataset Page

The **Dataset** page defines the input data in the data file.



*The **Data** page in the **New Variogram** dialog is used to set the data columns, filter duplicate data, and to view data statistics.*

Input Data File

The *Input Data File* is the data file from which the variogram is created. Click *Browse* to select a new data file in the Open Data dialog.

Data Columns

You can specify the columns for the X data, the Y data, and the Z data in the *Data Columns* group. **Surfer** defaults to *X:* column A, *Y:* column B, and *Z:* column C. Your data can be in any three columns, however. Click the down arrow on each box and select the appropriate column for each variable.

Duplicate Data

You can choose the type of duplicate data to use in the variogram gridding process from the *To Keep* list. The *X Tolerance* and *Y Tolerance* boxes set the distance, in data units, that define whether or not the data are duplicates. If the data do not share identical coordinates, but you would like close-by points to be considered duplicates, use this setting.

Data Exclusion Filter

To set up rules for excluding data when gridding, use the Data Exclusion Filter. The *Data Exclusion Filter* can exclude data based on X, Y, or Z values or by a number in another column in the worksheet.

To use a stored function, click the next to the current function. This will display the ten most recent functions used. The functions are stored in the registry, so the equations are stored between **Surfer** sessions. You can also start typing the function in the function box. If the function is in the ten function history, the entire function will auto-complete.

Trend Removal

The *Trend Removal* list contains three options for detrending the data before creating a variogram. The *Trend Removal* options carry out a simple polynomial least squares regression of the data and computes the variogram grid for the resulting residuals only. See Section 6.12 in Kitanidis (1997) for a more detailed discussion of detrending data.

- *None* should be used most of the time.
- *Linear* detrending should be used when the Kriging algorithm is applied with a linear drift.
- *Quadratic* detrending should be used when the Kriging algorithm is applied with a quadratic drift.

Creating a variogram with *Linear or Quadratic* detrending does not automatically change the Kriging Variogram options in the Grid Data dialog. You must manually import the variogram model by clicking *Get Variogram* to create a grid with the Kriging algorithm.

Statistics

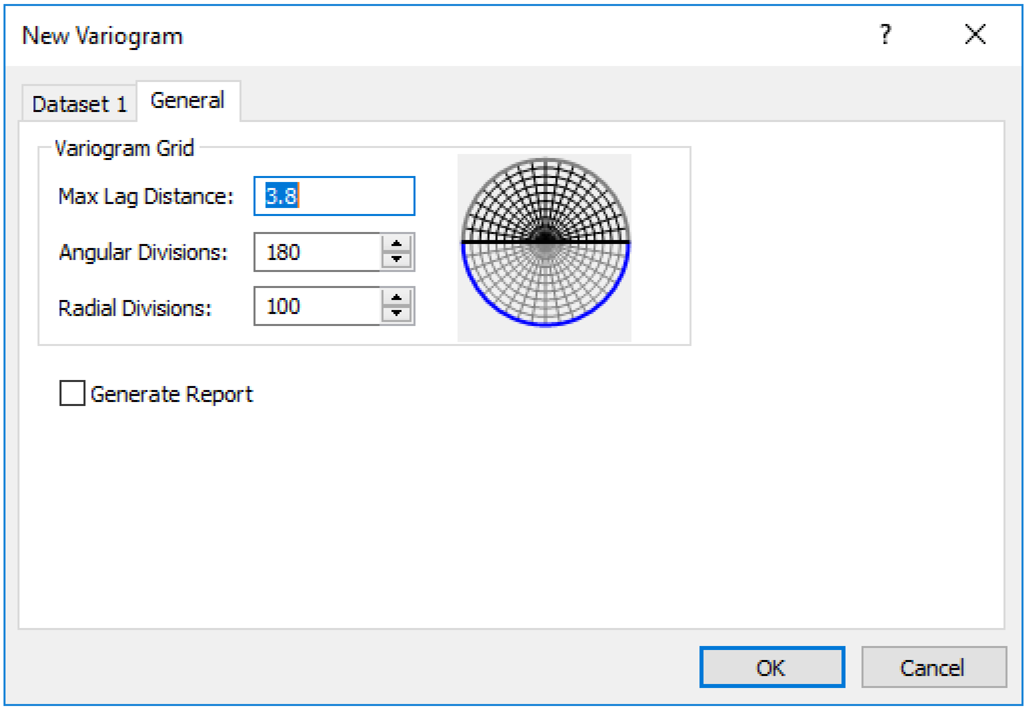
Click *Statistics* to display statistics about the data in the Statistics window.

New Variogram General Properties

The **New Variogram** dialog is used to create a new variogram. Click Grids | New Grid | Variogram |New Variogram and select a data file to open the **New Variogram** dialog.

General Page

Use the New Variogram command to open the **New Variogram** dialog. The **General** page is used to define the variogram grid.



*You can set the variogram grid properties on the **General** page in the **New Variogram** dialog.*

Variogram Grid

The *Variogram Grid* group contains the maximum lag distance, angular divisions, and the radial divisions settings for the variogram grid, as pictured on the right side of the group.

The picture to the right of the edit controls depicts the specified variogram grid. However, to make the picture legible, the number of concentric circles drawn is equal to the number of radial divisions divided by ten. The number of radial spokes drawn is equal to the number of angular divisions divided by ten.

Max Lag Distance

Max Lag Distance specifies the largest separation distance contained in the variogram grid. Any pairs separated by more than the maximum lag distance are not included in the variogram grid. The default value of the *Max Lag Distance* is one third the diagonal extent of the data rounded down at the second significant figure (i.e. 1.2345 becomes 1.2).

Angular Divisions

The *Angular Divisions* specify the number of angular divisions: i.e. the number of spaces between "spokes" in the variogram grid. 0° is located on the positive X axis, not the positive Y axis (azimuth). The angular divisions only go from 0° to 180°, and not all the way back around to 360°. If a vector were drawn from the first point in the pair to the second point in the pair, the arrow symbol would point to one direction in the polar grid. In **Surfer**, the first point is the point with the smaller Y value and the second point is the point with the larger Y value. As such, the lower half of the variogram grid is empty and unnecessary. The experimental variogram for any direction between 180° and 360° can be computed from the upper half of the grid using the inherent symmetry of the variogram. The default value of 180 makes each variogram grid cell span 1° of arc.

Radial Divisions

The *Radial Divisions* specify the number of concentric circles in the variogram grid. The default value is 100. Keep in mind that increasing the number of *Angular Divisions* and *Radial Divisions* increases the amount of memory required to store the grid. See The Variogram Grid for more information on memory requirements.

Generate Report

Click in the *Generate Report* check box to create a report of the data, including statistics. For more information about the contents of the report, see Reports.

Variogram Properties

Once the variogram grid has been created through the with Grids | New Grid | Variogram | New Variogram, the properties of the variogram can be changed in the Properties window. To change the features of an existing variogram click once on the variogram in either the Contents window or in the plot window to select it. The properties are displayed in the Properties window.

Variogram Properties

The variogram properties contains five pages: **Experimental, Model, Statistics, Plot,** and **Info**.

Experimental Page

The Experimental page contains parameters for the experimental variogram such as lag direction, estimator type, maximum lag distance, number of lags, lag width, and vertical scale.

Model Page

The Model page allows you to set a specific variogram model and its parameters such as variogram components, error variance, micro variance, automatic variogram fitting, and anisotropy.

Statistics Page

The Statistics page shows histograms and scatter plots of the data, as well as providing statistics on the data.

Plot Page

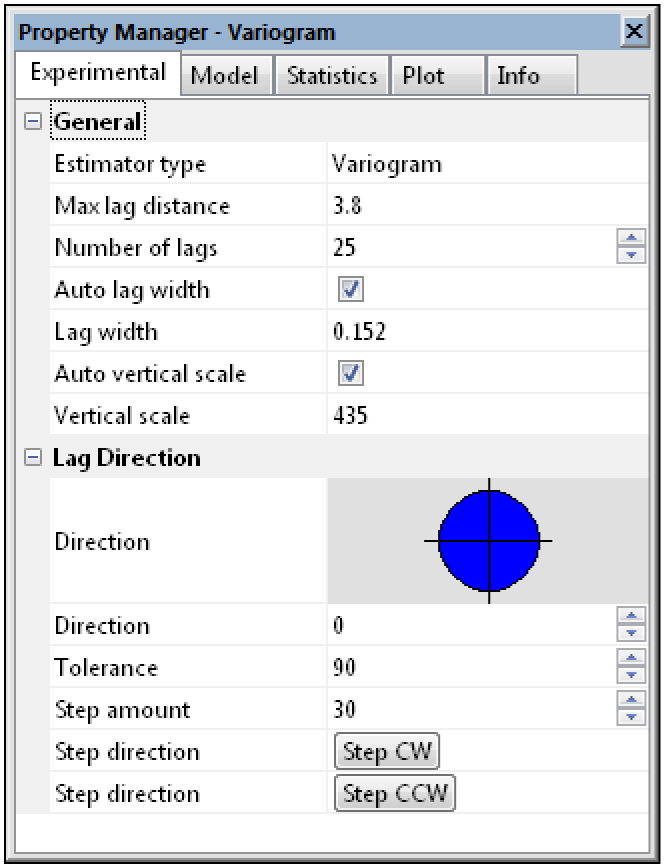
Most of the visual characteristics of the variogram display are controlled through the Plot page.

Info Page

Information on attributes and the ability to copy attribute data are available on the Info page.

Variogram Experimental Properties

The **Experimental** page in the **Properties** window contains parameters for the experimental variogram such as the lag direction, estimator type, maximum lag distance, number of lags, lag width, and vertical scale.



The ***Experimental*** *page in the* ***Properties*** *window contains the parameters for the experimental variogram. The diagram in the Lag Direction section shows the direction and size of the angular window of the current variogram.*

Estimator Type

Surfer contains four estimating options when create a variogram: the *variogram*, *standardized variogram*, *autocovariance*, and *autocorrelation*. When in doubt use the default classical variogram. When cokriging, the cross variogram, *Var1-Var2*, estimator is *Covariance*.

Maximum Lag Distance

The *Max lag distance* specifies the extent of the separation distance axis (abscissa or X axis). This value should never be larger than the *Max Lag Distance* specified in the New Variogram dialog. Pairs with separation values beyond the **New Variogram** *Max lag distance* have been eliminated from the variogram already. The *Max lag distance* specified when creating the variogram grid (**New Variogram**) limits the maximum separation distance that you ***can*** look (without recreating a new variogram grid). The *Max lag distance* specified in the variogram properties dialog limits the maximum separation distance you ***will*** look. The *Max lag distance* specified in the **Experimental** page in the variogram properties dialog is often less than the *Max lag distance* specified when creating the variogram grid. Select a value >0.

Number of Lags

The *Number of lags* specifies how many experimental variogram points are computed and plotted. The default *Number of Lags* is 25. Select a value >=1.

Lag Width/Lag Size and Auto Lag Width

When the *Auto lag width* box is checked, the *Lag width* is automatically set equal to the *Max lag distance* divided by the *Number of lags*. For example, if the *Max lag distance* is 100, and the *Number of lags* is five, the *Lag width* would be set to 100 / 5 = 20. In this case, the first plotted point would summarize all pairs with separation distances greater than or equal to zero and less than 20. The second plotted point would summarize all pairs with separation distances greater than or equal to 20 and less than 40, etc. Lag intervals are 0 to 20, 20 to 40, ..., 80 to 100; they do not overlap. You can use *Lag Width* to smooth a variogram.

Vertical Scale and Auto Vertical Scale

The *Vertical scale* specifies the extent of the variogram axis (ordinate or Y axis). If the *Auto vertical scale* box is checked, the vertical scale is determined on the fly so that the entire variogram fits on the plot. When the *Auto vertical scale* box is checked the vertical scale is recomputed every time the variogram is redrawn. As you step around with the *Step CCW* or *Step CW* buttons, the vertical scale changes with each step.

When the *Auto vertical scale* box is not checked, the *Vertical scale* is held constant at the specified value. When using the *Step CCW* and *Step CW* buttons to rotate the lag direction, it is advisable to uncheck the *Auto vertical scale* box to avoid having the vertical scale change for every plot. This allows for easier visual comparison between plots.

Lag Direction

The *Lag Direction* section controls the direction from which the variogram is viewed.

Direction

The *Direction* specifies the focal direction when computing the experimental variogram. In keeping with the standard notation used in geostatistics, this direction is given using a mathematical convention: 0° is along the positive X axis, and 90° is along the positive Y axis. The direction is not given as an azimuth.

Tolerance

The direction *Tolerance* specifies the size of the angular window for the experimental variogram. The angular window is:

Direction - Tolerance < Angle < Direction + Tolerance

thus, the width of the entire angular window is twice the *Tolerance*. Select a *Tolerance* value between 0 and 90.

For example, the default *Tolerance* of 90° captures all directions (omni-directional variogram). The diagrammatic representation of the current angular window appears in theLag Directiongroup.

Step Amount and Step Direction

When building a complete model of a variogram, it is necessary to review the variogram and the model in many directions. **Surfer** allows you to step through as many directions as desired in an animated fashion. Since **Surfer** uses a variogram grid approach it can compute and replot directional variograms almost instantaneously. The Step amount specifies the increment in the lag Direction each time the Step or Step buttons are clicked. When the Step CW button is clicked, the lag *Direction* is decremented by the *Step amount* and the variogram plot is automatically updated. When the *Step CCW* button is clicked, the lag *Direction* is incremented by the *Step amount* and the variogram plot is automatically updated.

Estimator Type

There are a number of different formulae for estimating the variogram. Each method has different strengths, weaknesses, and proponents. **Surfer** contains four estimating options: the *variogram, standardized variogram, autocovariance*, and *autocorrelation*. When in doubt use the default classical *variogram.*

An observation is an (X, Y, Z) triple. A pair of observations contains two such triples. Consider the set of N pairs of observations that go into the calculation of a single plotted point on an experimental variogram. These N pairs are ordered pairs; the two observations are not randomly ordered within a pair. The first observation in the pair is the point with the smaller Y value and the second observation is the point with the larger Y value. The first observation is called the "head" (denoted by an "h"), and the second observation is called the "tail" (denoted by a "t).

Let

Zh_i be the observed value of the head observation of the ithpair, and

Zt_i be the observed value of the tail observation of the ithpair.

The required sums over the set of all N pairs are denoted:

$$Zh = \sum Zh_i$$
$$Zhh = \sum Zh_i^2$$
$$Zt = \sum Zt_i$$
$$Ztt = \sum Zt_i^2$$
$$Zht = \sum Zh_i \times Zt_i$$

The average and variance for all of the observed head values are:

$$AVEh = \frac{Zh}{N}$$
$$VARh = \frac{Zhh}{N} - (AVEh)^2$$

Similarly, the average and variance for all of the observed tail values are:

$$AVEt = \frac{Zt}{N}$$
$$VARt = \frac{Ztt}{N} - (AVEt)^2$$

Then define the following two intermediate quantities:

$$\text{Lag Variance} = \sqrt{VARh \times VARt}$$
$$\text{Lag Covariance} = \left(\frac{Zht}{N}\right) - (AVEh \times AVEt)$$

Variogram

The "classical variogram" estimator is:

$$\gamma = \frac{(Zhh - 2Zht + Ztt)}{2N}$$

(seePannatier, 1996, p. 38).

Standardized Variogram

The standardized variogram estimator is:

$$\gamma = \frac{\frac{(Zhh - 2Zht + Ztt)}{2N}}{\text{Lag Variance}}$$

(see Pannatier, 1996, p. 39).

Autocovariance

The autocovariance estimator is:

$$\gamma = \text{Lag Variance} - \text{Lag Covariance}$$

(see Pannatier, 1996, p. 41).

Autocorrelation

The autocorrelation estimator is:

$$\gamma = 1 - \frac{\text{AutoCovariance}}{\text{Lag Variance}}$$

(see Pannatier, 1996, p. 42).

Smoothing a Variogram with Lag Width

On the variogram properties **Experimental** page, when the associated *Auto* check box is not checked, the Lag Width may be independently specified. In this case, the *Lag Width* is not a function of the *Max lag distance* and the *Number of Lags*. This flexibility allows for overlapping lag intervals and, therefore, a smoothed experimental variogram.

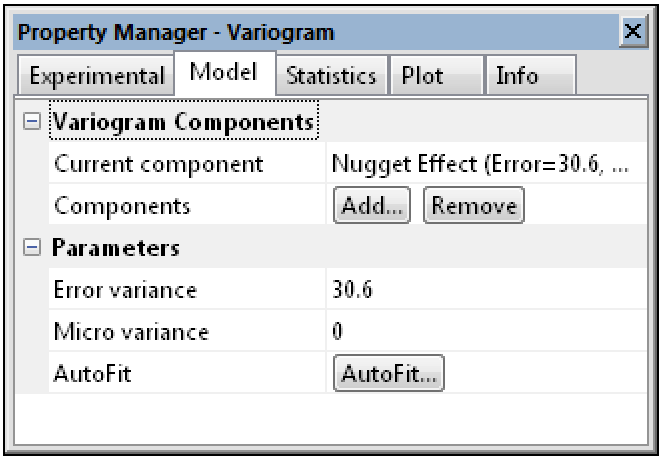
For example, consider the case where the *Max lag distance* is 100, the *Number of Lags* is five, and the *Lag Width* is 40. The resulting lag intervals would be 0 to 40, 10 to 50, 30 to 70, 50 to 90, and 60 to 100. Notice these intervals overlap. Overlapping lag intervals causes a moving average type smoothing of the experimental variogram. Often, it is easier to select an appropriate model for such smoothed experimental variograms.

In either case, the formulae for determining the intervals are given by the following pseudo-code fragment.

```
for(lag = 0; lag Number of Lags; ++lag )
     {
     center = Max Lag Distance / Number of Lags * (lag+0.5);
     from = max( center - ( Lag Width / 2.0 ), 0.0 );
     to = min( center + ( Lag Width / 2.0 ), Max Lag Distance );
     }
```

Variogram Model Properties

The **Model** page in the **Properties** window allows you to set a specific variogram model and its parameters. Computing an experimental variogram from your data is the only certain way to determine which variogram model you should use. There are lengthy chapters in many geostatistics textbooks discussing the tools and techniques necessary to generate a variogram (e.g. Isaaks and Srivastava, 1989).



*Options such as the variogram model are set on the **Model** page.*

Surfer allows for a general nested variogram model. There are no limits on the number of models that can be nested in **Surfer**. Because of this, there are hundreds of possible combinations of variogram models.

Current Component

Click the item next to *Current component* to select a different variogram component to edit. A list appears containing all of the components defined for the current variogram. When you select a component, the properties of that component are shown in the *Parameters* section.

Add Components

Click the *Add* button to open the Add Component dialog and add a variogram component. There are twelve common variogram functions: *Nugget Effect*, *Spherical*, *Exponential*, *Linear*, *Gaussian*, *Wave* (*Hole-Effect*), *Quadratic*, *Rational Quadratic*, *Logarithmic*, *Power*, *Cubic*, and *Pentaspherical*. Each of the components allow for independent specification of the anisotropy, except for the nugget effect.

Remove Components

Select a *Variogram Component* and click the *Remove* button to remove a variogram component.

Partial Sill (C)

With the exception of the *Linear, Logarithmic,* and *Power* variogram models (which do not have a sill), the *Partial sill* parameters (denoted by *C* in the variogram equations) define the sill for the variogram components you select. Thus, the sill of the variogram model equals the *Nugget Effect* plus the sum of the component's *Partial sill (C)* parameters. In many situations, the variogram model sill is approximately equal to the variance of the observed data (see Barnes, 1991).

Range (A)

The Range *(A)* parameters define how rapidly the variogram components change with increasing separation distance. The *Range (A)* parameter for a variogram component is used to scale the physical separation distance. For the *Spherical* and *Quadratic* variogram functions, the *Range (A)* parameter is also known as the variogram range. In an isotropic setting, the relative separation distance, *h*, is computed by the following equation:

$$h = \frac{\sqrt{\Delta x^2 + \Delta y^2}}{A}$$

where

$\begin{bmatrix} \Delta x & \Delta y \end{bmatrix}$ is the separation vector (in map coordinates), and

A is the component's *Range* parameter.

The anisotropically rescaled relative separation distance for the variogram equations is computed by the following matrix equation

$$h = \sqrt{\begin{bmatrix} \Delta x & \Delta y \end{bmatrix} \begin{bmatrix} \cos\theta & -\sin\theta \\ \sin\theta & \cos\theta \end{bmatrix} \begin{bmatrix} \left(1/A\right)^2 & 0 \\ 0 & \left(\rho/A\right)^2 \end{bmatrix} \begin{bmatrix} \cos\theta & \sin\theta \\ -\sin\theta & \cos\theta \end{bmatrix} \begin{bmatrix} \Delta x \\ \Delta y \end{bmatrix}}$$

where

$\begin{bmatrix} \Delta x & \Delta y \end{bmatrix}$ is the separation vector (in map coordinates),

A is the component's *Range* parameter.

θ is the anisotropy *Angle*, and

ρ is the anisotropy *Ratio*.

With a *Linear* variogram model, the slope is given by the *Partial sill* divided by the *Radius*. By allowing an anisotropic radius, it is possible to specify an anisotropic linear variogram slope. The slope of the linear variogram makes no difference in the kriging algorithm if the nugget effect is zero.

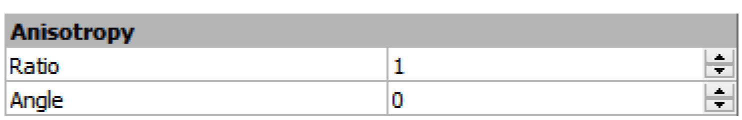
Anisotropy

Natural phenomena are created by physical processes. Often these physical processes have preferred orientations. For example, at the mouth of a river the coarse material settles out fastest, while the finer material takes longer to settle. Thus, the closer one is to the shoreline the coarser the sediments while the further from the shoreline the finer the sediments. When interpolating at a point, an observation 100 meters away but in a direction parallel to the shoreline is more likely to be similar to the value at the interpolation point than is an equidistant observation in a direction perpendicular to the shoreline. Anisotropy takes these trends in the data into account during the gridding process.

Usually, points closer to the grid node are given more weight than points farther from the grid node. If, as in the example above, the points in one direction have more similarity than points in another direction, it is advantageous to give points in a specific direction more weight in determining the value of a grid node. The relative weighting is defined by the anisotropy ratio. The underlying physical process producing the data as well as the sample spacing of the data are important in the decision of whether or not to reset the default anisotropy settings. Anisotropy is also useful when data sets use fundamentally different units in the X and Y dimensions. See below for examples.

Detailed discussions of anisotropy and *Kriging* as well as anisotropy equations are given in the variogram help topics.

The *Anisotropy* options are displayed in the Options page of the **Grid Data** dialog when a gridding method that supports anisotropy is selected..



*Set the anisotropy options in the **Grid Data Options** dialog.*

Ratio

The *Ratio* is the maximum range divided by the minimum range. An anisotropy ratio less than two is considered mild, while an anisotropy ratio greater than four is considered severe. Typically, when the anisotropy ratio is greater than three the effect is clearly visible on grid-based maps.

Unless there is a good reason to use an anisotropy ratio, you should accept the default value of 1.0.

Angle

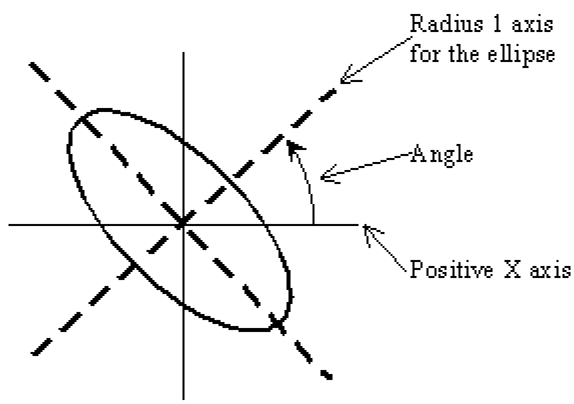
The *Angle* is the preferred orientation (direction) of the major axis in degrees. 0 degrees is defined as east-west orientation. 90 degrees is defined as north-south orientation. Angles rotate counterclockwise.

Ellipse

In the most general case, anisotropy can be visualized as an ellipse. The ellipse is specified by the lengths of its two orthogonal axes and by an orientation angle. In **Surfer**, the lengths of the axes are called Radius 1 and Radius 2. The orientation angle is defined as the counterclockwise angle between the positive X axis and Radius 1. Since the ellipse is defined in this manner, an ellipse can be defined with more than one set of parameters.

For example:

Radius 1 = 2
Radius 2 = 1
Angle = 0
is the same ellipse as
Radius 1 = 1
Radius 2 = 2
Angle = 90



The anisotropy angle is the angle between the positive X axis and the ellipse axis associated with Radius 1.

For most of the gridding methods in **Surfer**, the relative lengths of the axes are more important than the actual length of the axes. The relative lengths are expressed as a *Ratio* in the *Anisotropy* section. The ratio is defined as Radius 1 divided by Radius 2. Using the examples above, the ratios are 2 and 0.5. The ratio of 2 indicates that Radius 1 is twice as long as Radius 2. The *Angle* is the counterclockwise angle between the positive X axis and Radius 1. This means that:

Ratio = 2
Angle = 0
is the same ellipse as
Ratio = 0.5
Angle = 90

The small picture in the *Anisotropy* group displays a graphic of the ellipse to help describe the ellipse.

Example 1: Plotting a Flood Profile Along a River

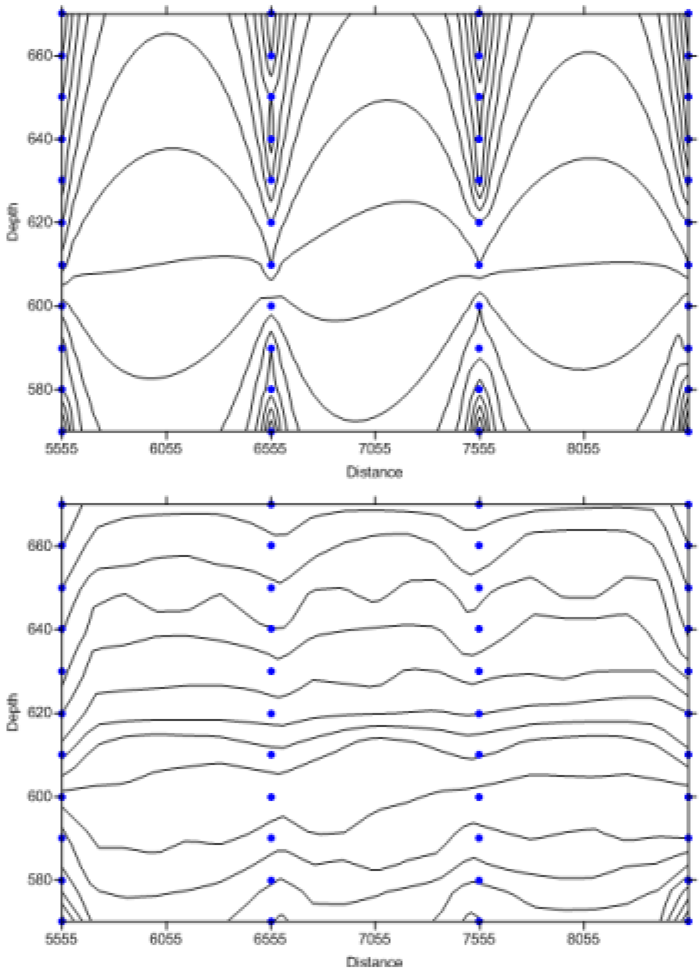
For an example when data sets use fundamentally different units in the X and Y directions, consider plotting a flood profile along a river. The X coordinates are locations, measured in miles along the river channel. The Y coordinates are time, measured in days. The Z values are river depth as a function of location and time. Clearly in this case, the X and Y coordinates would not be plotted on a common scale, because one is distance and the other is time. One unit of X does not equal one unit of Y. While the resulting map can be displayed with changes in scaling, it may be necessary to apply anisotropy as well.

Example 2: Isotherm Map of Average Daily Temperature

Another example of when anisotropy might be employed is an isotherm map (contour map) of average daily temperature over the upper Midwest. Although the X and Y coordinates (easting and northing) are measured using the same units, along the east-west lines (X lines) the temperature tends to be very similar. Along north-south lines (Y lines) the temperature tends to change more quickly (getting colder as you head north). When gridding the data, it would be advantageous to give more weight to data along the east-west axis than along the north-south axis. When interpolating a grid node, observations that lie in an east-west direction are given greater weight than observations lying an equivalent distance in the north-south direction.

Example 3: Oceanographic Survey to Determine Water Temperature at Varying Depths

A final example where an anisotropy ratio is appropriate is an oceanographic survey to determine water temperature at varying depths. Assume the data are collected every 1000 meters along a survey line, and temperatures are taken every ten meters in depth at each sample location. With this type of data set in mind, consider the problem of creating a grid file. When computing the weights to assign to the data points, closer data points get greater weights than points farther away. A temperature at 10 meters in depth at one location is similar to a sample at 10 meters in depth at another location, although the sample locations are thousands of meters apart. Temperatures might vary greatly with depth, but not as much between sample locations.



In the oceanographic survey described here, the contour lines cluster around the data points when an anisotropy ratio is not employed. In the lower contour map, an anisotropy ratio results in contour lines that are a more accurate representation of the data.

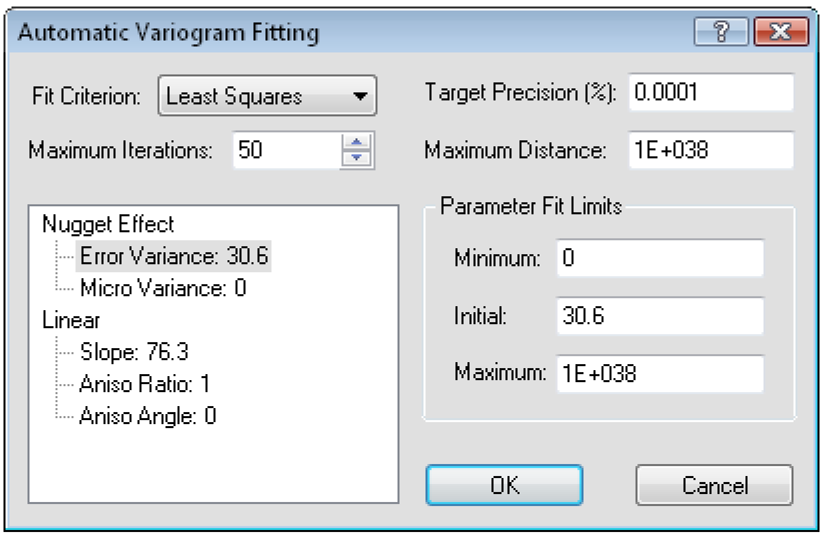
AutoFit

AutoFit takes a user-specified variogram model and an initial set of parameters, and attempts to find a better set of parameter values. For example, if you have selected a spherical variogram model with a nugget effect, the AutoFit tool can be used to determine the "best" fitting anisotropy parameters.

The *AutoFit* button on the Model page in the variogram properties and the Variogram page in the Grid Data dialog opens the **Automatic Variogram Fitting** dialog. Please note that the AutoFit tool is meant to hone and polish a set of user specified parameters. ***AutoFit does not select the appropriate model form.*** It should not be used as an exploratory tool in the search for an appropriate model. It merely fits the parameters ***once you select a reasonable model***.

The underlying fitting algorithm is a simple implementation of Powell's Search Method. This search does not involve the use of derivatives. This means there are no gradient calculations. One iteration of this algorithm includes a line search along each of the individual parameter directions, and then a line search along the accumulated direction moved during this iteration. There is some useful discussion of this method in Press et al.(1988, Section 10.1). However, this routine does not attempt to build a set of conjugate directions.

While the fitting algorithm is working, the current iteration number and the current value of the objective function are printed on the status bar. The fit algorithm can take some time to work.



*The **Automatic Variogram Fitting** dialog is opened by clicking the AutoFit button on the **Model** page of the **Variogram Properties** dialog. The AutoFit feature does not select an appropriate model. Before using AutoFit, select reasonable model parameters and then use AutoFit to fine tune the model.*

Fit Criterion

There are two fit criteria available in AutoFit, *Least Squares* and *Least Absolute Value*. An important consideration is that both of these criteria work by fitting the model to the variogram grid in all directions at once and not to the current directional variogram plot shown on the screen.

The *Least Squares* criterion is an implementation of the simple weighted least squares method discussed in Cressie (1991, Section 2.6.2). See Zimmerman and Zimmerman (1991) for a rather detailed comparison among various methods. The *Least Squares* method minimizes the sum of the square of the error, and the *Least Absolute Value* method minimizes the sum of the absolute value of the errors.

The *Default* method uses the default linear variogram. Default calculates a slope and nugget effect similar to the default linear variogram as first displayed with New Variogram.

Maximum Iterations

The maximum number of iterations specifies a termination criterion for AutoFit. After this number of iterations the search stops, though the iterations also terminate if a target precision is met. One iteration includes a line search for each of the free parameters, plus a line search in the direction of the accumulated change.

One feature of AutoFit is that the search can be stopped and restarted multiple times. Therefore, you could run the search for a few iterations, see what is happening, and then try a few more iterations. You can terminate the search by clicking on the ESC button on your keyboard and selecting the *Abort* button in the dialog that appears. If you stop AutoFit, the best set of parameters so far is saved.

Target Precision (%)

The target precision is also a termination criterion for the search with AutoFit. The target precision specifies a maximum relative change in the objective function during one iteration. If the change in the objective function is less than the specified amount, the search is terminated. If the change in the objective function is greater than the specified amount, the search is continued for another iteration (assuming that the maximum number of iterations has not been reached). The actual formula is

Maximum Distance

AutoFit attempts to fit the model to the data contained in the variogram grid. The extent of the variogram grid is given by the *Max Lag Distance* parameter specified when the variogram grid is created with the New Variogram dialog. Often, the *Max Lag Distance* specified when the variogram grid is created is greater than the maximum lag distance that a user is interested in working with while fitting. (Remember that the variogram grid is created before you start modeling).

The *Maximum Distance* parameter limits the extent of the search. The fitting algorithm ignores any pairs separated by more than the *Maximum Distance*, even if they are included in the variogram grid.

Parameter Fit Limits

The*Parameter Fit Limits*define the range of values over which the fitting algorithm searches with AutoFit. The*Parameter Fit Limits*allow the user a great deal of control in applying the fitting algorithm. For example, if you specify the*Maximum*equal to the*Minimum,*the fitting algorithm does not adjust the specified parameter.

Although the fit algorithm attempts to fit parameters with a large range between the *Maximum* and the *Minimum*, you will have greater success with the fitting algorithm if the limits are as tight as you can make them. Use a "reasonable value" approach in specifying the limits.

Minimum

The *Minimum* is the lower bound on the fitted parameter value. No value lower than this is considered by the fitting algorithm.

Initial

The *Initial* option is the guess for the parameter value. Since the objective function is non-linear, the final fitted values can be sensitive to the starting point. The objective function (fit criterion) may have numerous local minima which are distinct from the global minima.

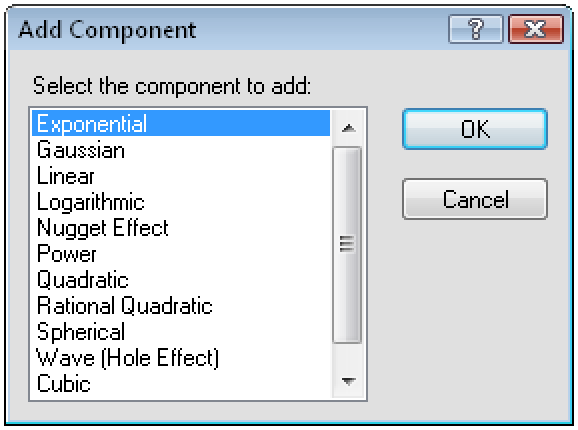
Maximum

The *Maximum* is the upper bound on the fitted parameter value. No value greater than this is considered by the fitting algorithm.

Variogram Model Graphics

When in doubt, you should use the Linear variogram model with the default *Scale (C)* and *Length (A)* parameters.

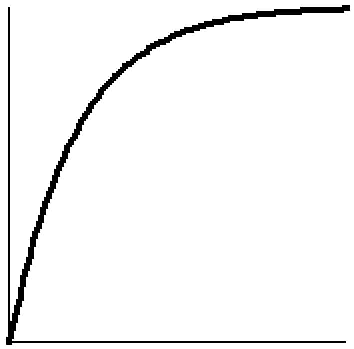
In the **Variogram Properties** in the Properties window, click on the Model tab. Click the *Add* button to open the **Add Component** dialog. Select one of the variogram model components and click the *OK* button.



*Select a variogram model component in the **Add Component** dialog.*

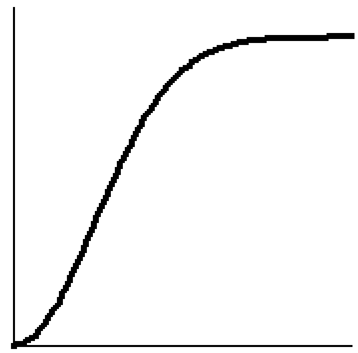
Variogram Model Components

The **variogram model** components available in **Surfer** are:



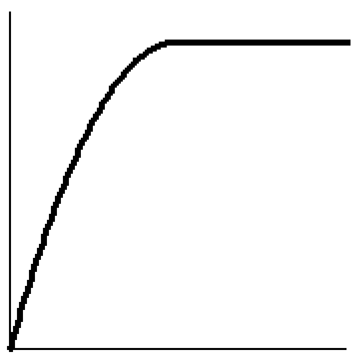
Exponential Model
Cressie (1991, p. 61)

$$\gamma(h) = C\left[1 - e^{-h}\right]$$



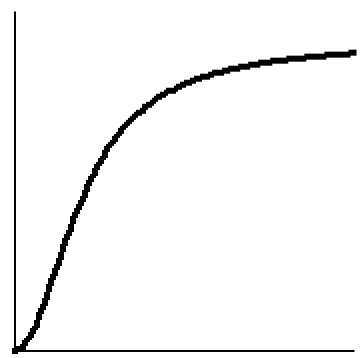
Gaussian Model
Pannatier (1996, p. 50)

$$\gamma(h) = C\left[1 - e^{-h^2}\right]$$



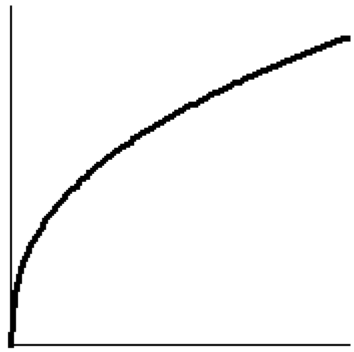
Quadratic Model
Alfaro (1980, p. 31)

$$\gamma(h) = \begin{cases} C\left[2h - h^2\right] & h < 1 \\ C & h \geq 1 \end{cases}$$

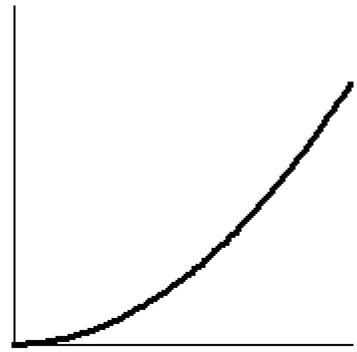


Rational Quadratic Model
Cressie (1991, p. 61)

$$\gamma(h) = C\left[\frac{h^2}{1 + h^2}\right]$$



$0 < h < 1$



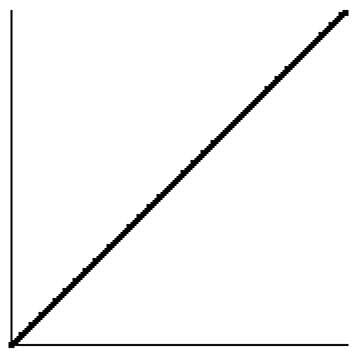
$1 < h < 2$

Power Models
Pannatier (1996, p. 51)

$$\gamma(h) = C\left|h^{n}\right|$$

where $0 < n < 2$

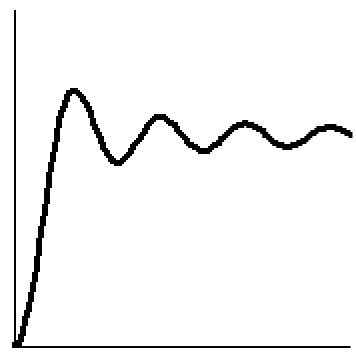
when n=1, it is a Linear Model



Linear Model
Kitanidis (1997, p. 61)

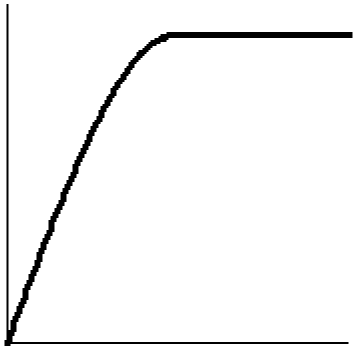
$$\gamma(h) = C(h)$$

The Linear Model is a special case of the Power Model where the power is equal to one



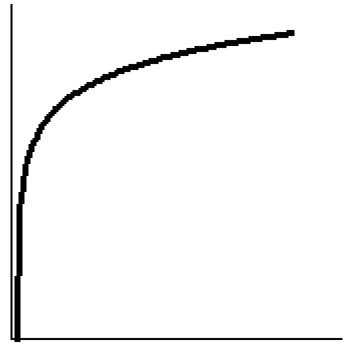
Wave (Hole Effect) Model
Cressie (1991, p. 62)

$$\gamma(h) = C\left[1 - \frac{\sin h}{h}\right]$$



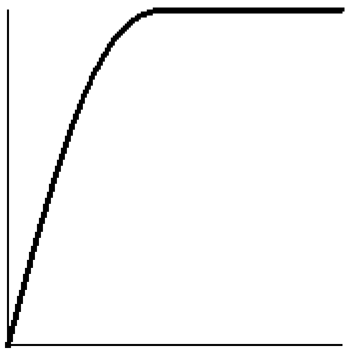
Spherical Model
Pannatier (1996, p. 48)

$$\gamma(h) = \begin{cases} C\left[1.5h - 0.5h^3\right] & h < 1 \\ C & h \geq 1 \end{cases}$$



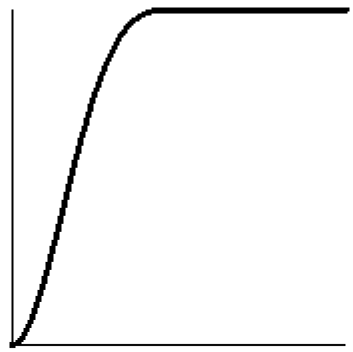
Logarithmic Model
Kitanidis (1997, p. 61)

$$\gamma(h) = C\left[\log_e(h)\right] \quad h > 0$$



Pentaspherical Model
Olea (1999, p. 79)

$$\gamma(h) = C\left(1.875h - 1.25h^3 + 0.375h^5\right)$$

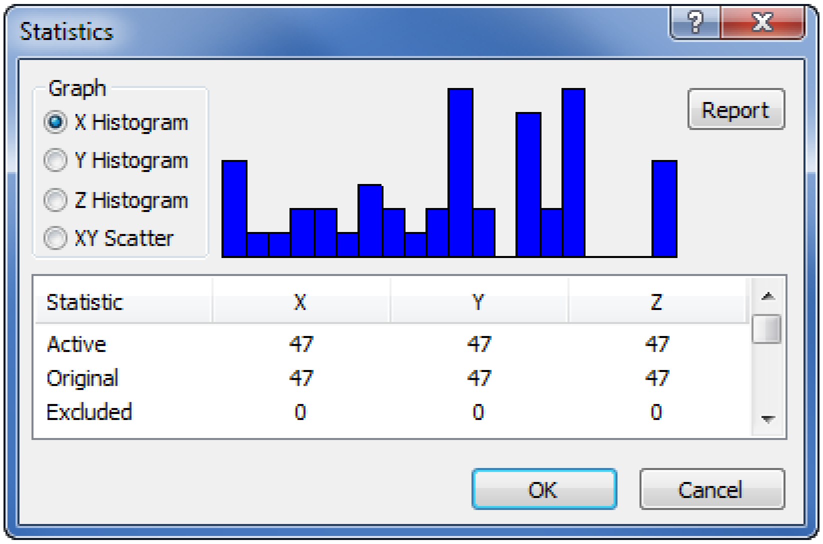


Cubic Model
Olea (1999, p. 79)

$$\gamma(h) = C\left(7h^2 - 8.75h^3 + 3.5h^5 - 0.75h^7\right)$$

Variogram Statistics Properties

Clicking the *Display Statistics* button on the **Statistics** page in the Properties window, *Statistics* button on the Grid Data - Variogram page, or the *Statistics* button on the New Variogram - Dataset page the shows histograms and scatter plots of the data, as well as providing statistics on the data.



*In the **Statistics** dialog, you can show an X, Y, or Z histogram, view statistics, or generate a variogram report.*

Graph X, Y, Z Histogram or XY Scatter

The *X, Y, Z Histogram* options show the distribution of the X, Y, and Z data respectively. The histogram is shown to the right of the options. The *XY Scatter* option shows the distribution of the data as a simple scatter diagram.

Report

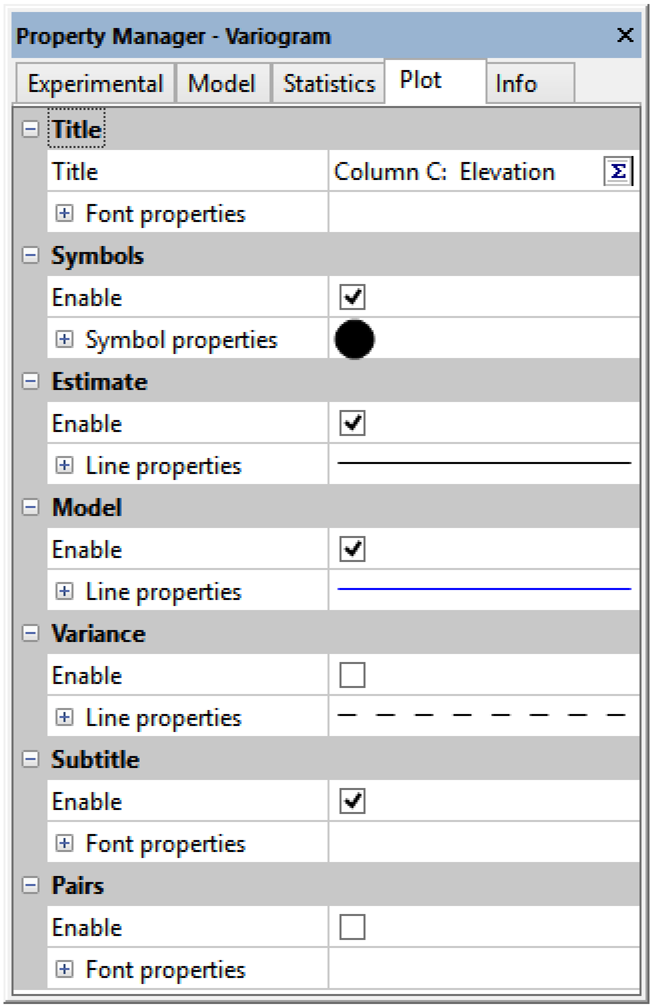
Click the *Report* button to open the statistics and variogram information in a report window. The statistics can be printed if they are opened in the report window.

Statistics

You can use the scroll bar on the right of the statistics box to view the statistics.

Variogram Plot Properties

Most of the visual characteristics of the variogram display are controlled though the **Plot** page.



Control the display of the variogram graph on the ***Plot*** *page of the variogram properties in the* ***Properties*** *window.*

Title

You can change the title and title font in the *Title* section. Click next to *Title* and type in the desired title text.

Titles can contain multiple lines. To create a multiple line title, click the Σ button. The Text Editor appears. Type the text in the dialog. To add a new line of text, press the ENTER key on the keyboard. To change the properties of the text, highlight the text and change the desired property. When finished, click *OK*. You are returned to the **Properties** window.

To edit all of the text properties for the entire title at once, click the ⊞ next to *Font Properties*. You can then set any desired font settings, including font, size, and text color.

Symbols

To display symbols, check the *Enable* box in the *Symbols* section.

To change the symbol properties, click the ⊞ next to *Symbol Properties*. You can change any desired symbol properties, including the symbol type, size, and color.

Estimate

To display the curve showing the actual data, check the *Enable* box in the *Estimate* section.

To change the line properties, click the ⊞ next to *Line Properties*. You can change any desired line properties, including the line style, width, and color.

Model

To display the model line, check the *Enable* box in the *Model* section.

To change the line properties, click the ⊞ next to *Line Properties*. You can change any desired line properties, including the line style, width, and color.

Variance

To display the variance line, check the *Enable* box in the *Variance* section.

To change the line properties, click the ⊞ next to *Line Properties*. You can change any desired line properties, including the line style, width, and color.

Subtitle

To display the subtitle, check the box next to *Enable* in the *Subtitle* section. The subtitle lists the direction and tolerance of the variogram being displayed.

To edit the text properties, click the ⊞ next to *Font Properties*. You can then set any desired font settings, including font, size, and text color.

Pairs

To display the number of pairs for each point on the variogram, check the box next to *Enable* in the *Pairs* section.

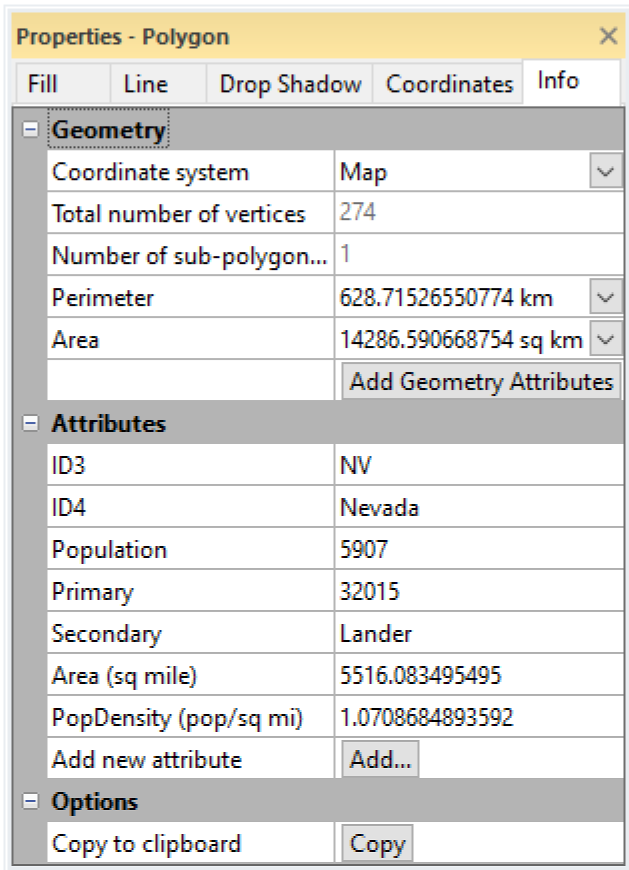
To edit the text properties, click the ⊞ next to *Font Properties*. You can then set any desired font settings, including font, size, and text color.

Info Properties - Objects

The **Info** properties page contains information about the selected object. **Info** properties are available with drawn objects, such as polylines, polygons, points, grouped objects, images, and metafiles, with map objects, axes, and map layers, such as contours or base layers. The information displayed is dependent on the type of object, and the image below is just one example of an **Info** properties page.

The **Info** page is located in the Properties window for the selected object. A single object must be selected to display the **Info** page.

Note: Grid layers have their own Info properties page.



*The **Info** page displays information for the selected object.*

Geometry Section

The **Geometry** section includes properties that specify the *Coordinate system* for calculating the geometry values and display the perimeter, length, position, or area of the selected object. Click on the ⊞ next to *Geometry* to open the section. This section is only available for polylines, polygons, symbols, rectangles, rounded rectangles, ellipses, or spline polylines.

Select the units for the units for the *Perimeter*, *Area*, *Length*, or *Position* values by clicking the value and selecting the desired units from the list. The selected units are moved to the top of the list, otherwise the units are displayed alphabetically.

Coordinate System

Specify a coordinate system from the *Coordinate system* list to be used for the information displayed for the object.

- *Page* uses the page coordinate system (inches or centimeters depending on the setting in the **Options** dialog General page).
- *Local* uses the units from the layer in which the geometry was originally specified. For example, if the geometry was loaded from a file, this is the original file units. If the geometry was created interactively, this is an internal coordinate system with 0.0 in the center of the page (in inches).
- *Map* displays information in map units if the geometry is part of a map. If the object is not part of a map, this option is not available.

The default *Coordinate system* property is *Map* for map objects and *Page* for objects outside a map. Changes to the *Coordinate system* selection are applied to all objects. For example, if the property is changed to *Local* for one object, the *Local* information will be displayed for the next selected object. If the selected *Coordinate system* is a geographic coordinate system, i.e. uses spherical coordinates, some geometry information cannot be displayed.

Position

This displays the XY position of a point, for example in a base (vector) map.

Total Number of Vertices

The *Total number of vertices* displays how many vertices are used in the selected object. The Reshape command can be used to see the vertex locations for polygons and polylines.

Number of Sub-Polygons

If the object contains a complex polygon, the *Number of sub-polygons (rings)* displays how many polygons are included in the complex polygon. For example, the sample file ca2010.gsb has polygons of California counties. Some of the counties are complex polygons (i.e. Santa Barbara) that include sub-polygon islands.

Number of Curves

The *Number of curves* is displayed for a spline polyline. The *Number of curves* is the number of inflection points along the spline polyline. This is one less than the total number of vertices.

Perimeter/Length

Depending on the object selected, either *Perimeter* or *Length* will be displayed.

- The *Perimeter* displays the calculated perimeter of the selected closed object (i.e. polygon).
- The *Length* displays the calculated length of the selected open object (i.e. polyline). The units displayed are dependent on what is selected for the *Coordinate system*.

The *Perimeter* or *Length* is not displayed in spherical coordinates (i.e. degrees for *Unprojected Lat/Long* coordinate systems). If your map is in a geographic coordinate system, change the Target Coordinate System to a projected coordinate system to view the *Perimeter* or *Length*. Then specify *Map* in the *Coordinate system* field on the **Info** page.

Area

The *Area* displays the calculated area of the selected object. The units displayed are dependent on what is selected for the *Coordinate system*.

The *Area* is not displayed in spherical coordinates (i.e. degrees squared for *Unprojected Lat/Long* coordinate systems). If your map is in a geographic coordinate system, change the Target Coordinate System to a projected coordinate system to view the *Area*. Then specify *Map* in the *Coordinate system* list on the **Info** page.

Add Geometry Attributes

Click the *Add Geometry Attributes* command to add the geometry values displayed above to the object's attributes. If you wish to add the geometry attributes to all objects in the base layer, use the Attribute Table.

Info Section

The **Info** section allows you to view the *Type, Description, Number of records, size in bytes, Objects in group* and image information for the selected object. Click on the ⊞ next to *Info* to open the section. This section is available for text, images, metafiles, grouped objects, and point cloud layers.

Type

The *Type* displays the type of metafile being displayed.

Description

The *Description* displays any information about the metafile.

Number of Records

The *Number of records* displays the number of objects in the metafile.

Pixel Format

The *Pixel format* option displays the type of image imported, including the number of bits per pixel included in the image.

Size in Bytes

The *Size in bytes* or the *Size (bytes)* option displays the file size of the image or metafile in bytes.

Size in Pixels

The *Size (pixels)* option displays the number of pixels in the image.

Image Source

The *Image source* displays the name of the file imported for an image.

Number of Points

The *Number of points* displays the number of points in the point cloud layer.

Extents

The *xMin*, *xMax* , *yMin* , *yMax*, *Zmin*, and *Zmax* values display the extents of the point cloud layer.

Attributes Section

The *Attributes* section contains any information that is available about the selected object. To open the *Attributes* section, click the ⊞ next to *Attributes*.

Attributes can be image properties, such as *TIFF_Compression*, or can be information about a specific object that was imported from a file, such as a .DEM, .DXF, or .SHP file. Each object (such as polylines in a base map) can have its own attribute information. The *Attributes* can also contain user generated information.

Adding Attributes

To add an attribute to the Attribute Table,

1. Click on the object to select it.
2. In the **Properties** window, click on the **Info** page.
3. Open the *Attributes* section by clicking the ⊞ next to *Attributes*.
4. Click on the *Add* button next to *Add new attribute*.
5. Type the new attribute name, such as *Surveyor's Name*, in the *Attribute name* field of the New Attribute Name dialog.
6. Click *OK* in the **New Attribute Name** dialog. The new attribute is added to the object displayed on the **Info** page.
7. Click in the value column next to the new attribute name. Type the desired

description, such as *Thomas Denver*.
8. Now the new attribute and its value has been added to the object.

To rename or remove an attribute, use the **Attribute Table**.

Edit Attributes

Edit attribute values by typing the new value into the field next to the attribute name in the **Properties** window. Edit values for multiple attributes in a base layer with the Attribute Table.

Options Section

The *Options* section contains the option to copy the attribute data for a selected object. To open the *Options* section, click the ⊞ next to *Options*.

Copy Attributes

Click the *Copy* button next to *Copy to clipboard* to copy all of the information on the **Info** page for the selected object to the clipboard. The text can be pasted into **Surfer** or any other program using the Paste command. All information on the **Info** page is copied, including the information in the *Geometry* and *Info* sections. If the *Copy* button is not available, no attribute information is available for the selected item.

Exporting Attributes

Attribute information is exported for polygons, polylines, and points to file formats that support metadata. The file type will determine how many attributes are exported. Refer to the specific file type pages for specific information.

Displaying Attributes

Attributes can be added to base maps as labels. Click on the *Base* map layer object in the **Contents** window to select the base layer. In the **Properties** window, click on the Labels tab to turn label display on. Any attribute can be used as a label for polylines, polygons, or symbols on a base map layer.

Information Displayed for Objects

All objects display an *Attributes* and *Options* section. The information in the *Geometry* section changes depending on the object selected.

Polyline

The **Info** properties for Polylines displays the *Coordinate system, Number of vertices,* and *Length.*

Polygon

The **Info** properties for Polygons displays the *Coordinate system, Total number of vertices, Number of sub-polygons (rings), Perimeter,* and *Area.*

Point

The **Info** properties for Symbols displays the *Coordinate system* and *Position* in X and Y units.

Rectangle

The **Info** properties for Rectangles displays the *Coordinate system*, *Perimeter* and *Area.*

Rounded Rectangle

The **Info** properties for Rounded Rectangles displays the *Coordinate system, Perimeter* and *Area.*

Ellipse

The **Info** properties for Ellipses displays the *Coordinate system*, *Perimeter* and *Area.*

Spline Polyline

The **Info** properties for Spline Polylines displays the *Coordinate system*, *Number of curves* and *Length*.

Range Ring

The **Info** properties for Range Rings displays the *Coordinate system*, *Perimeter*, and *Area*. Note that when *Number of rings* is greater than one, the *Perimeter* and *Area* values are for the largest ring.

Group Objects

The **Info** page for Group Properties displays the number of *Objects in group*. Each group object can be selected in the **Contents** window to display the object info.

Metafile

The **Info** properties for Metafiles displays the *Type of metafile*, *Description*, *Number of records*, and *Size in bytes*.

Image

The **Info** properties for Images displays the *Pixel format*, *Size (pixels)*, *Size (bytes)*, and *Image source*.

Default Linear Variogram

The default linear variogram takes the form

$$\gamma(h) = C_0 + S \cdot h$$

where C_0 is the unknown nugget effect, and S is the unknown slope. We need two defining equations to solve for these two unknown parameters.

According to theory, the expected value of the sample variance is the average value of the variogram between all possible pairs of sample locations (Barnes, 1991); this yields one equation. The second equation is generated by equating the experimental sample variogram for nearest neighbors to the modeled variogram. Thus, we can write

$$Var = C_0 + S \cdot D_{avg}$$
$$G_{nn} = C_0 + S \cdot D_{nn}$$

where

D_{nn} = average distance to the nearest neighbor

D_{avg} = average inter-sample separation distance

G_{nn} = one half the averaged squared difference between nearest neighbors

Var = sample variance

By solving the two equations for the two unknown parameters, and checking for unreasonable values, we get the final formulae used in **Surfer**:

$$S = \max\left[\frac{Var - G_{nn}}{D_{avg} - D_{nn}}, 0\right]$$
$$C_o = \max\left[\frac{G_{nn} \cdot D_{avg} - Var \cdot D_{nn}}{D_{avg} - D_{nn}}, 0\right]$$

Nugget Effect

The *Nugget Effect* is used when there are potential errors in the collection of your data. The nugget effect is implied from the variogram you generate of your data. Specifying a nugget effect causes Kriging to become more of a smoothing interpolator, implying less confidence in individual data points versus the overall trend of the data. The higher the *Nugget Effect*, the smoother the resulting grid. The units of the nugget effect are the units of the observations squared.

The nugget effect is made up of two components:

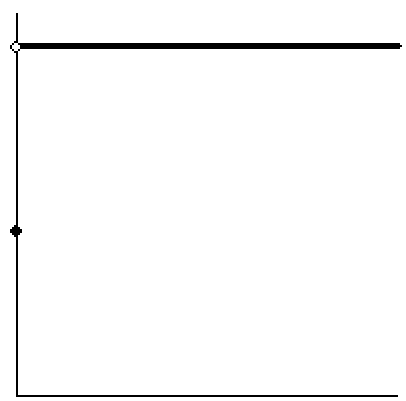
Nugget Effect = *Error Variance* + *Micro Variance*.

Error Variance

The *Error Variance* is a measure of the direct repeatability of the data measurements. If you took an observation at one point, would the second measurement at that exact point be exactly the same as the first measurement? The *Error Variance* values take these variances in measurement into account. A non-zero *Error Variance* means that a particular observed value is not necessarily the exact value of the location. Consequently, *Kriging* tends to smooth the surface: it is not a perfect interpolator.

Micro Variance

The *Micro Variance* is a measure of variation that occurs at separation distances of less than the typical nearest neighbor sample spacing. For example, consider a nested variogram where both of the models are spherical. The range of one of the structures is 100 meters while the range of the second structure is five meters. If our closest sample spacing were 10 meters, we would not be able to see the second structure (five meter structure). The *Micro Variance* box allows you to specify the variance of the small-scale structure. If you do not know the micro variance, leave the setting at 0.0.



Nugget Effect Model

$$\gamma(h) = \begin{Bmatrix} \text{Error Variance} & h = 0 \\ \text{Error Variace} + \text{Micro Variance} & h > 0 \end{Bmatrix}$$

This is the nugget effect model.

Reference

Surfer's implementation of the *Nugget Effect* follows the recommendation of Cressie (1991, Section 3.2.1), and this reference should be consulted for a more rigorous presentation of the nugget effect and its partitioning.

Export Variogram

You can use **Grids | New Grid | Variogram | Export Variogram** to export the XY coordinates of the symbols representing the pairs for each lag distance.

To Export a Variogram:

1. Create a variogram using the New Variogram command.
2. Click once on the variogram to select it.
3. Click the **Grids | New Grid | Variogram | Export Variogram** command or the  button. In the **Save As** dialog, set the *File name* and location. Click *Save* and the file is created.

The resulting file is an ASCII text file [.DAT] with the X coordinates (separation distance) in column A, the Y coordinates (variogram - g(

Using Variogram Results in Kriging

Once you have completed the variogram, you can use the results of the variogram model when gridding the data with the Kriging gridding option.

To use model results:

1. Make sure the variogram is displayed in the plot window. If you switch to a new plot window, the variogram results cannot be used.

2. Select Grids | New Grid | Grid Data.

3. In the **Open Data** dialog, choose the data file to be modeled and click *Open*.

4. In the **Grid Data** dialog, select *Kriging* for the gridding method.

5. Click the *Next* button twice to open the Grid Data Kriging Options dialog.

6. Click the *Get Variogram* button to use the variogram model open in the plot window. If more than one variogram exists in the document, a dialog is displayed prompting you for the variogram to use.

Suggested Reading - Variograms

Alfaro, M. (1980), *The Random Coin Method: Solution of the Problem of the Simulation of a Random Function in the Plane,Mathematical Geology*, v. 12, n. 1, p. 25-32.

Barnes, R. (1991), The Variogram Sill and the Sample Variance, *Mathematical Geology*. v. 23, n. 4, p. 673-678.

Barnes, R. Variogram Tutorial. http://www.goldensoftware.com/variogramTutorial.pdf

Cressie, Noel (1991), *Statistics for Spatial Data* , John Wiley and Sons, New York, 900 pp., ISBN 0-471-84336-9.

Isaaks, Edward and Mohan Srivastava (1989), *An Introduction to Applied Geostatistics* , Oxford University Press, New York, 561 pp., ISBN 0-19-505013-4.

Kitanidis, Peter (1997), *Introduction to Geostatistics - Applications in Hydrogeology* , Cambridge University Press, New York, 249 pp., ISBN 0-521-58747-6.

Olea, R.A., 1999, *Geostatistics for Engineers and Earth Scientists*, Kluwer Academic Publishers, Boston, 303 pp. ISBN 0-7923-8523-3.

Pannatier, Yvan (1996), *VarioWin - Software for Spatial Data Analysis in 2D* , Springer-Verlag, New York, 91 pp., ISBN 0-387-94579-9.

Press, W.H., , Numerical Recipes in C , Cambridge University Press.

Zimmerman, D. L., and *Technometrics* , v. 33, n. 1, p. 77-91.

Variogram Tutorial

Surfer's variogram modeling feature is intended for experienced variogram users who need to learn **Surfer**'s variogram modeling features.

The novice variogram user may find the following four authors helpful: Cressie (1991), Isaaks and Srivastava (1989), Kitanidis (1997), and Pannatier (1996). A tutorial for use with **Surfer**'s variogram modeling is also available. Please refer to Suggested Reading for full references for each of the previous books.

If you do not understand variograms or if you are unsure about which model to apply, use **Surfer**'s default linear variogram with the kriging algorithm. The following tutorial is intended for those who have a basic knowledge of variograms and need assistance with using **Surfer**'s variogram feature.

Variogram Introduction

The variogram characterizes the spatial continuity or roughness of a data set. Ordinary one-dimensional statistics for two data sets may be nearly identical, but the spatial continuity may be quite different. Refer to What Does a Variogram Represent? for a partial justification of the variogram.

Variogram analysis consists of the experimental variogram calculated from the data and the variogram model fitted to the data. The experimental variogram is calculated by averaging one-half the difference squared of the z-values over all pairs of observations with the specified separation distance and direction. It is plotted as a two-dimensional graph. Refer to What is a Variogram? for details about the mathematical formulas used to calculate the experimental variogram.

The variogram model is chosen from a set of mathematical functions that describe spatial relationships. The appropriate model is chosen by matching the shape of the curve of the experimental variogram to the shape of the curve of the mathematical function.

Refer to the Variogram Model Graphics for graphs illustrating the curve shapes for each function. To account for geometric anisotropy (variable spatial continuity in different directions), separate experimental and model variograms can be calculated for different directions in the data set.

What Does a Variogram Represent?

Consider two synthetic data sets; we will call them A and B. Some common descriptive statistics for these two data sets are given in Table 1.1.

	A	B
Count	15251	15251
Average	100.00	100.00
Standard Deviation	20.00	20.00
Median	100.35	100.92
10 Percentile	73.89	73.95
90 Percentile	125.61	124.72

Table 1.1 Some common descriptive statistics for the two example data sets.

The histograms for these two data sets are given in Figures 1.1 and 1.2. According to this evidence the two data sets are almost identical.

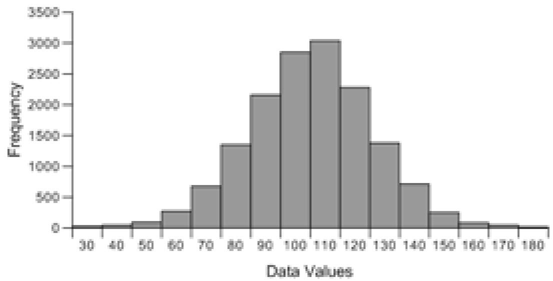


*Figure 1.1 Data Set **A** Histogram*

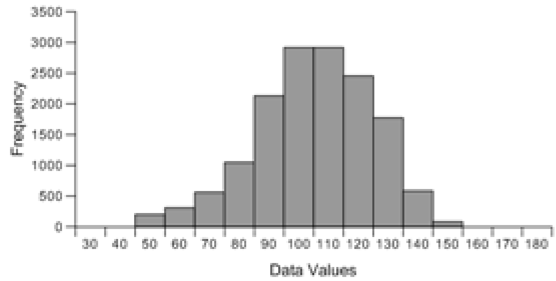


*Figure 1.2 Data Set **B** Histogram*

However, these two data sets are significantly different in ways that are not captured by the common descriptive statistics and histograms. As can be seen by comparing the associated contour plots (see Figures 1.3 and 1.4), data set A is rougher than data set B. Note that we can not say that data set A is "more variable" than data set B, since the standard deviations for the two data sets are the same, as are the magnitudes of highs and lows. The visually apparent difference between these two data sets is one of texture and not variability.

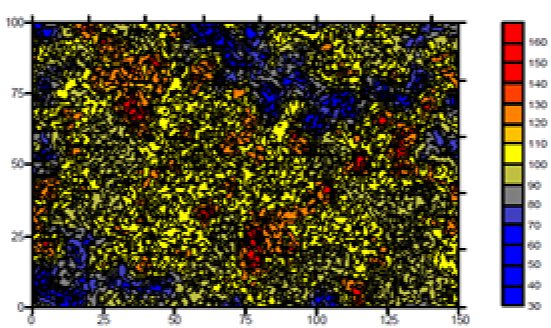


*Figure 1.3 Data Set **A** Contour Plot*

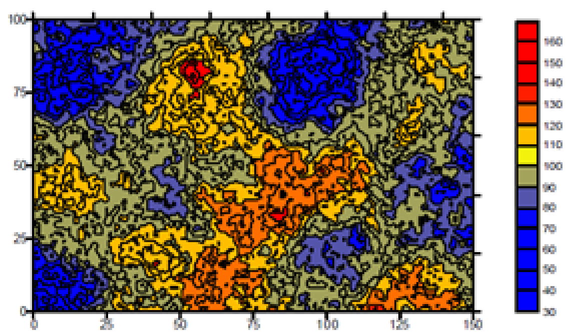


*Figure 1.4 Data Set **B** Contour Plot*

In particular, data set A changes more rapidly in space than does data set B. The continuous high zones (red patches) and continuous low zones (blue patches) are, on the average, smaller for data set A than for data set B. Such differences can have a significant impact on sample design, site characterization, and spatial prediction in general.

It is not surprising that the common descriptive statistics and the histograms fail to identify, let alone quantify, the textural difference between these two example data sets. Common descriptive statistics and histograms do not incorporate the spatial locations of data into their defining computations.

The variogram is a quantitative descriptive statistic that can be graphically represented in a manner which characterizes the spatial continuity (i.e. roughness) of a data set. The variograms for these two data sets are shown in Figures 1.5 and 1.6. The difference in the initial slope of the curves is apparent.

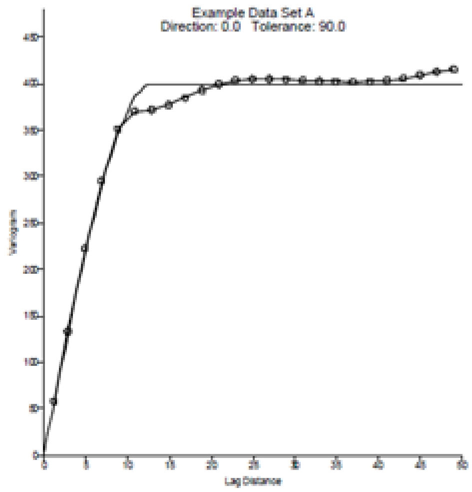


*Figure 1.5 Data Set **A** Variogram and Model*

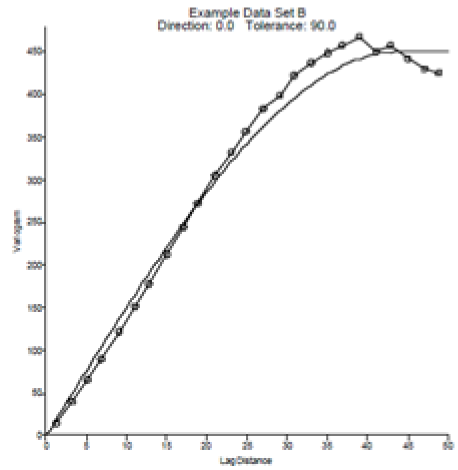


*Figure 1.6 Data Set **B** Variogram and Model*

What is a Variogram?

The mathematical definition of the variogram is

$$\gamma(\Delta x,\Delta y) = \frac{1}{2}\,\varepsilon\left[\{Z(x+\Delta x, y+\Delta y) - Z(x,y)\}^2\right]$$

(3.1)

where **Z(x,y)** is the value of the variable of interest at location **(x, y)**, and **e[ ]** is the statistical expectation operator. Note that the variogram, **g( )**, is a function of the separation between points **(Dx,Dy)**, and not a function of the specific location **(x, y)**. This mathematical definition is a useful abstraction, but not easy to apply to observed values.

Consider a set of n observed data: $\{(x_1, y_1, z_1),(x_2, y_2, z_2), \ldots (x_n, y_n, z_n)\}$, where (x_i,y_i) is the location of observation **i**, and z_i is the associated observed value. There are **n(n - 1)/2** unique pairs of observations. For each of these pairs we can calculate the associated separation vector:

$$(\Delta x_{i,j}, \Delta y_{i,j}) = (x_i - x_j, y_i - y_j)$$

(3.2)

When we want to infer the variogram for a particular separation vector, **(Δx,Δy)**, we will use all of the data pairs whose separation vector is approximately equal to this separation of interest:

$$(\Delta x_{i,j}, \Delta y_{i,j}) \approx (\Delta x, \Delta y)$$

(3.3)

Let **S(Δx,Δy)** be the set of all such pairs:

$$S(\Delta x, \Delta y) = \{ (i,j) \mid (\Delta x_{i,j}, \Delta y_{i,j}) \approx (\Delta x, \Delta y) \}$$

(3.4)

Furthermore, let **N(Δx,Δy)** equal the number of pairs in **S(Δx,Δy)**. To infer the variogram from observed data we will then use the formula for the *experimental variogram*.

$$\hat{\gamma}(\Delta x,\Delta y) = \frac{1}{2N(\Delta x, \Delta y)} \sum_{(i,j)\in S(\Delta x, \Delta y)} (Zi-Zj)^2$$

(3.5)

That is, the experimental variogram for a particular separation vector of interest is calculated by averaging one-half the difference squared of the z-values over all pairs of observations separated by approximately that vector.

Variogram Tutorial - Variogram Grid

If there are **n** observed data, there are **n(n - 1)/2** unique pairs of observations. Thus, even a data set of moderate size generates a large number of pairs. For example, if **n** = 500, **n(n - 1)/2** = 124,745 pairs. The manipulation of such a large number of pairs can be time consuming, even for a fast computer. **Surfer** pre-computes all of the pairs and stores the necessary sums and differences in the *variogram grid*. Note: a variogram grid is not the same format as a grid used in creating a map.

To create a new variogram, choose the **Grid | Variogram | New Variogram** menu command, specify the data file name in the **Open** dialog box, and click *Open*. Specify the *X*, *Y*, and *Z* columns, *Duplicate Data* settings, *Data Exclusion Filter* (if any), and review the data *Statistics*.

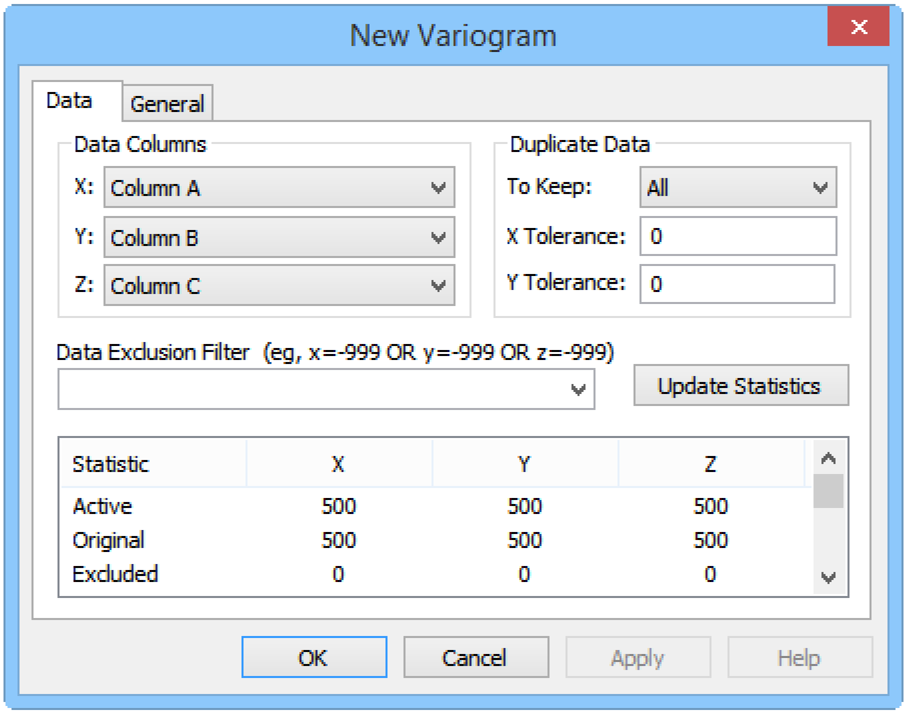


*Figure 4.1 Choose the **Grid | Variogram | New Variogram** menu to display the **Data** tab of the **New Variogram** dialog box.*

Click the **General** tab to view the *Variogram Grid* and *Detrending* options. The *Max Lag Distance* is the maximum separation distance to be considered during variogram modeling. By default, this is approximately one-third the diagonal extent of the observed data. The *Angular Divisions* of 180 and the *Radial Divisions* of 100 are adequate for almost any setting.

The *Detrend* options offer advanced data handling options for universal kriging. Typically, the appropriate option is *Do not detrend the data*. However, if you know that a strong trend exists in the data, you may want to consider *Linear* detrending. Click the *Generate Report* option to create a list of the *Data Filter Settings* and *Data Statistics*.

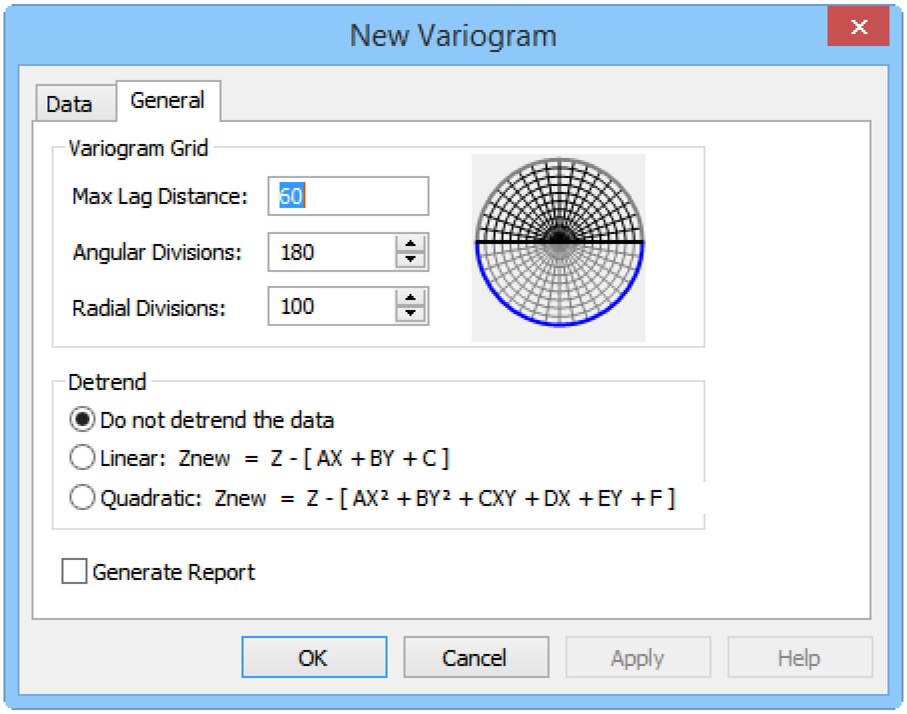


*Figure 4.2 Click the **General** tab in the **New Variogram** dialog box to display the* Variogram Grid *,* Detrend *, and* Generate Report *options.*

Without changing any of the settings, click *OK*. Figure 4.3 is displayed.

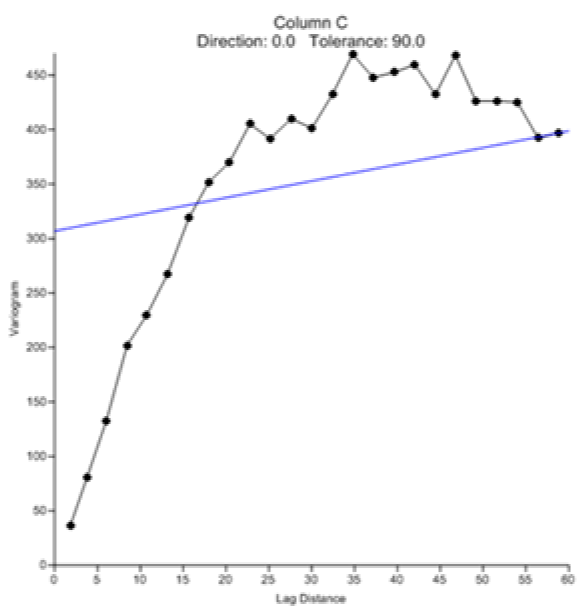


Figure 4.3 Resulting variogram with default variogram settings using ExampleDataSetC.xls.

The black line with the dots is the omni-directional experimental variogram, while the blue line is a first pass (albeit a poor one) at a fitted variogram model.

Modeling the Omni-Directional Variogram

By default, this first plot is the omni-directional variogram (the directional tolerance is 90 degrees). Choose the model type, the sill, and the nugget effect based upon the omni-directional variogram.

5.1 Selecting the variogram model type

There are infinitely many possible variogram models. **Surfer** allows for the construction of thousands of different variogram models by selecting combinations of the ten available component types. When combined with a nugget effect, one of three models is adequate for most data sets: the linear, the exponential, and the spherical models. Examples of these three models are shown in Figure 5.1.

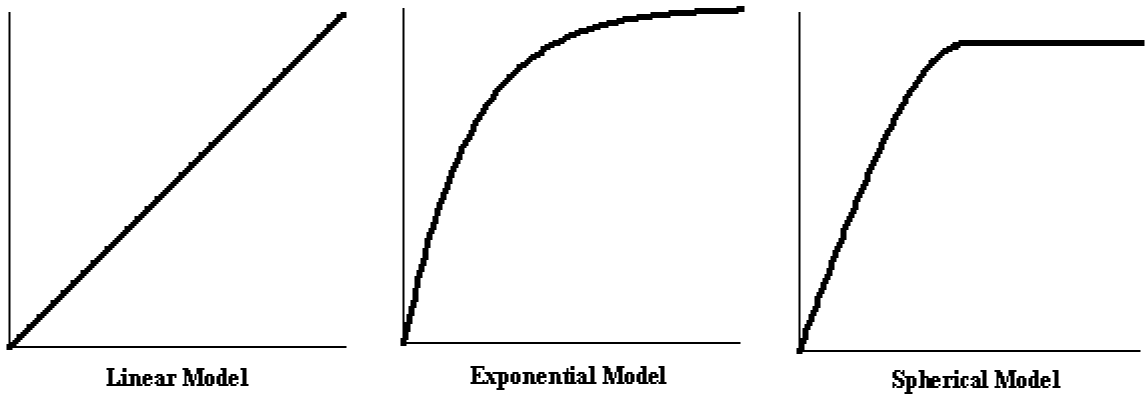


Figure 5.1 Variogram Models

If the experimental variogram never levels out, then the linear model is usually appropriate. If the experimental variogram levels out, but is "curvy" all the way up, then the exponential model should be considered. If the experimental variogram starts out straight, then bends over sharply and levels out, the spherical model is a good first choice.

For the data in ExampleDataSetC.xls, a spherical model appears appropriate, though one could also try an exponential model. Click on the variogram plot and select the **Model** tab in the **Properties** window.

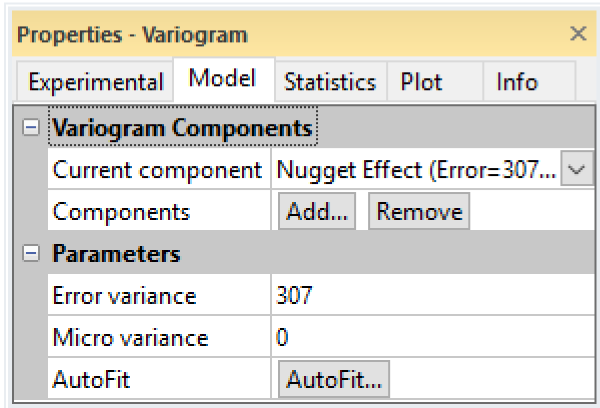


*Figure 5.2 The **Model** tab of the Variogram properties in the **Properties** window.*

To see the current model, click on the *Nugget Effect* next to *Current component.* You can see that there is also a *Linear* component to the default model. Next to *Components*, click the *Remove* button twice to remove the inappropriate default model. Then click the *Add* button. In the **Add Component** dialog, select the *Spherical* model and click *OK*.

5.2 Selecting the variogram model scale and length parameters

We must now set the *Scale* and the *Length (A)* parameters using an iterative approach (i.e. guess and check). The *Scale* is the height on the y-axis at which the variogram levels off. By simply looking at the plot, a value between 400 and 450 seems reasonable: enter 425. The*Length (A)* for a spherical model is the lag distance at which the variogram levels off. Again, from the plot a value between 30 and 40 seems reasonable: enter 35. The new candidate variogram model is automatically drawn.

This is not a bad first guess, but upon examination of the redrawn curve, it appears that the *Length (A)* is a little bit too long since the model (blue line) lies to the right of the experimental variogram plot (black line and dots). Reset the *Length (A)* to 30. This is still a little bit too long. Try 29 for the*Length (A)* This is a good fit for a variogram.

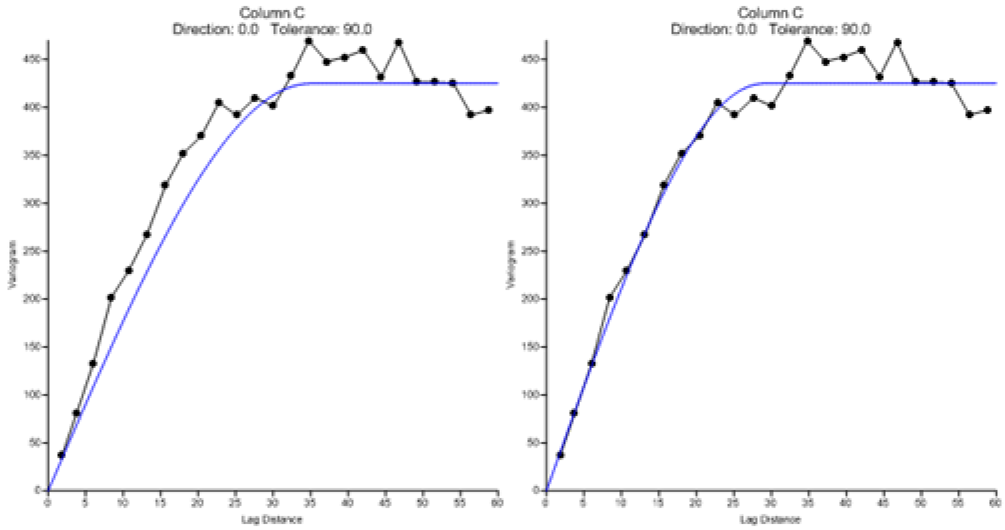


Figure 5.3 Variogram model with initial assumptions Left: Scale = 425, Length (A) = 35. Right: Scale =425. Length (A) = 29.

5.3 Selecting the variogram nugget effect

If the experimental variogram appears to have a non-zero intercept on the vertical axis, then the model may need a nugget effect component. The variance of *Delta Z* in the *Nearest Neighbor Statistics* section of the Variogram Grid Report offers a quantitative upper bound for the nugget effect in most circumstances.

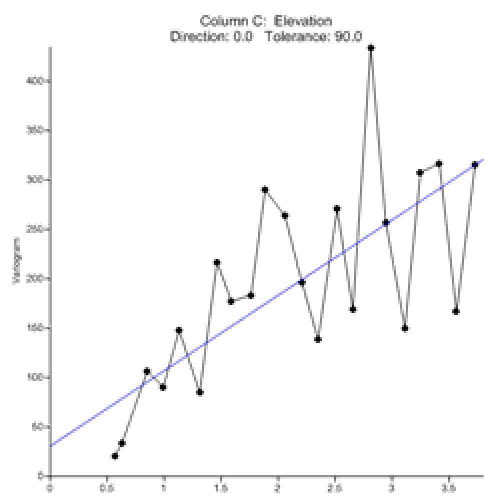


Figure 5.4 Linear Variogram model with nugget effect for data set Demogrid.dat.

In **Surfer** the nugget effect is partitioned into two sub-components: the *error variance* and the *micro variance*. Both of these sub-components are non-negative,and the sum of these two sub-components should equal the apparent non-zero intercept.

The *error variance* measures the reproducibility of observations. This includes both sampling and assaying (analytical) errors. The error variance is best selected by computing the variance of differences between duplicate samples.

The *micro variance* is a substitute for the unknown variogram at separation distances of less than the typical sample spacing. This is best selected by taking the difference between the apparent non-zero intercept of the experimental variogram and the error variance.

The model for our example appears to intersect the vertical axis at 0, so we will not apply a nugget effect.

Modeling the Variogram Anisotropy

Often, the experimental variogram shows different length scales in different directions. This is called *geometric anisotropy*. For a linear variogram model this would appear as a different slope in different directions, while a spherical model manifests geometric anisotropy as different apparent *Length (A)* parameters in different directions.

To investigate the variogram anisotropy click on the variogram. In the **Properties** window, click on the **Experimental** tab. The omni-directional experimental variogram averages the behavior over all directions. To look in particular directions open the *Lag Direction* section and change the *Direction* and the *Tolerance* parameters. Start by setting the *Tolerance* to 30 degrees. Notice how the small circular diagram shows the blue sector centered at 0 degrees (east in this model), with an angular tolerance (plus or minus) of 30 degrees.

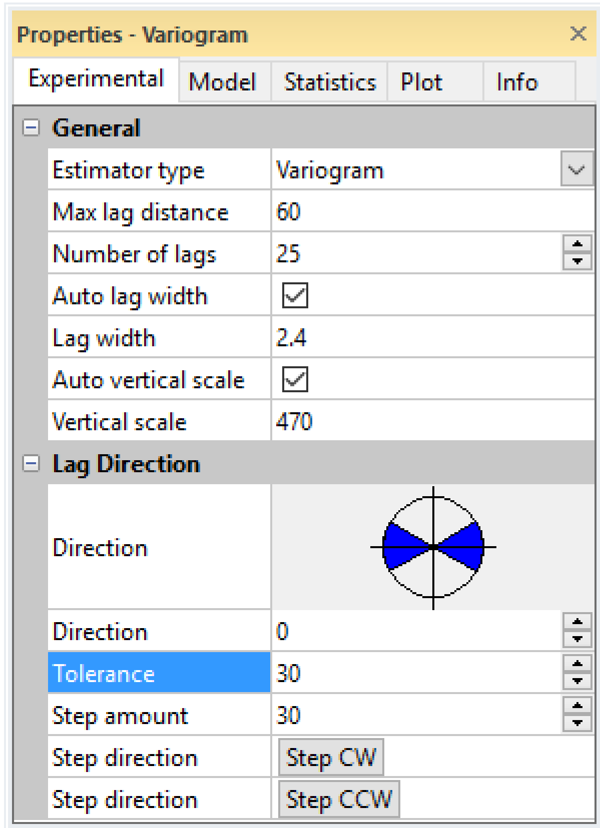


Figure 6.1 Set the Tolerance to 30 on the ***Experimental*** *tab of the Variogram properties in the* ***Properties*** *window.*

To quickly sweep through all directions press the *Step CCW* button. Press it once and the plot is now set for a direction of 30 degrees. Press it again and you have selected 60 degrees. Press it again and again and again and notice how the small circular diagram helps you keep track of the current direction.

Sweep through all directions a few times. Notice how the scaling of the vertical axis changes. To fix the vertical scale of the plot, in the *General* section, set the *Vertical Scale* to 515.

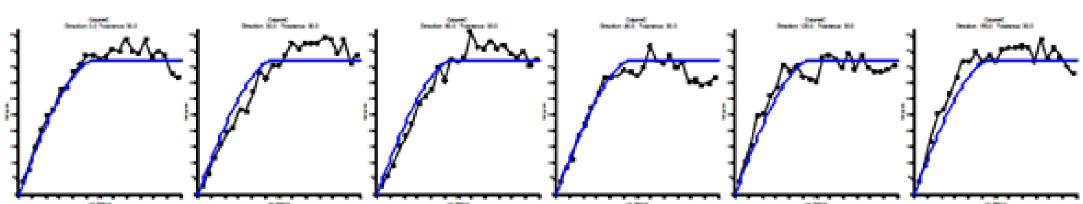


Figure 6.2 Directional variogram plots with Tolerance set to 30°. From left to right: Direction = 0°, 30°, 60°, 90°, 120°, 150°.

Change the *Step Amount* to 15 degrees and sweep through all directions a couple of times. It appears that for directions between 90 and 180 degrees the omni-directional model is consistently to the right of the ascending portion of the experimental variogram. Conversely, the omni-directional model is consistently to the left of the ascending portion of the experimental variogram for directions between 0 and 90 degrees. This is a clear indication of geometric anisotropy.

By pressing the *Step CCW* button move to the direction at which the apparent length and scale of the experimental variogram is maximized: say, 30 degrees. Click on the **Model** tab and change the *Length (A)* parameter to better fit this directional variogram. 40 is too long, 30 is too short, and 37 appears to fit quite well. Change the *Anisotropy Angle* to 30 degrees. .

Click on the **Experimental** tab and change the *Direction* to 120 degrees (this is orthogonal to the direction of maximum length, 120 = 30 + 90). Click on the **Model** tab. Change only the *Anisotropy Ratio* to fit the variogram model to the current directional experimental variogram. As you change the setting, the model will automatically update the plot. A *Anisotropy Ratio* value of 1.6 appears to be a good fit.

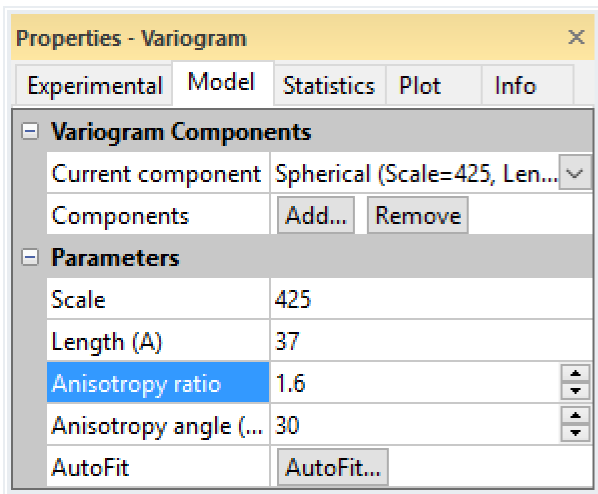


Figure 6.3 Change the variogram model properties to add anisotropy.

To get a final visual verification of your model, take it for a "spin". Click on the **Experimental** tab and repeatedly click the *Step CCW* button. Notice how the variogram model mimics the changing shape of the experimental variogram. You can fine tune the model parameters to better match the model to the experimental variogram, but do not over-model: one or two significant digits is all that you can hope for.

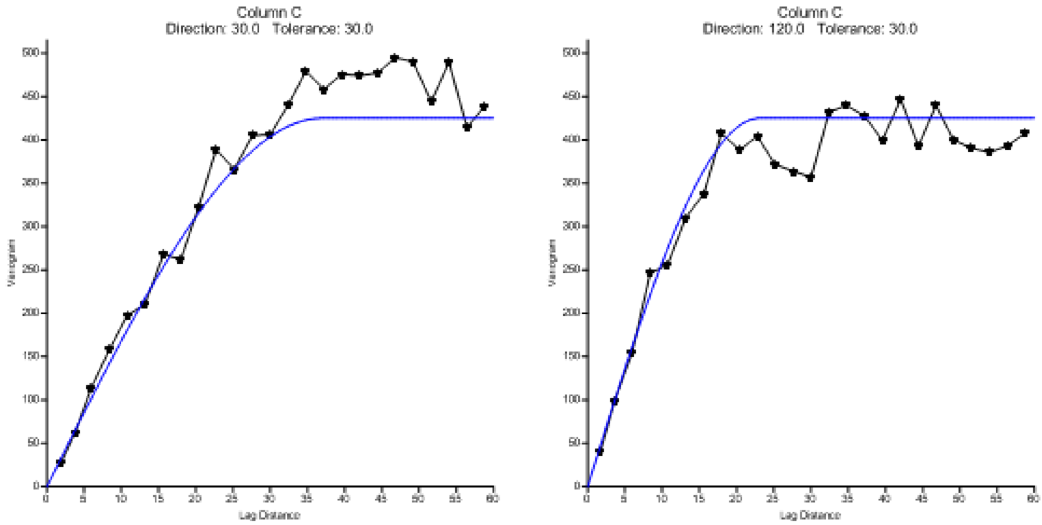


Figure 6.4 Directional variograms with anisotropy settings. Left: Direction = 30°, Right: Direction = 120° Anisotropy Ratio = 1.6, Angle = 30°.

Click the *AutoFit* button on the **Model** tab to fine-tune the model after choosing appropriate initial parameters. Click *OK* to accept the defaults, and AutoFit returns an *Anisotropy Ratio* of 1.741 and *Anisotropy Angle* of 38.57°.

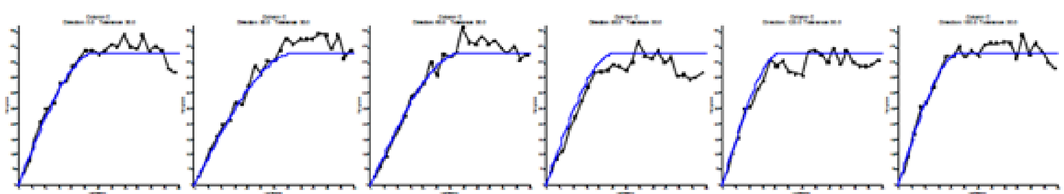


Figure 6.5 Directional Variogram plots with Anisotropy Ratio *= 1.75 and* Anisotropy Angle *= 38.49°. From left to right: Direction = 0°, 30°, 60°, 90°, 120°, 150°.*

Variogram Modeling Rules of Thumb

The following rules of thumb are general tips for variogram modeling:

- Know your data! Before calculating the experimental variogram, calculate regular non-spatial statistics. Use the Variogram Report to display the data minimum, maximum, median, mean, standard deviation, variance, and skewness. Create a post map or classed post map in **Surfer** to display scatter plots. Use **Grapher** to create histograms and cumulative frequency plots.

- Do not over model. The simplest model that reproduces the important features of the experimental variogram is the best model.

- When in doubt, use the default variogram model for gridding. A simple linear variogram model usually generates an acceptable grid; this is especially true for initial data analysis. Remember, however, that the kriging standard deviation grid generated using the default variogram is meaningless.

- Unless there is a clear, unambiguous, physical justification, do not use an anisotropy ratio of greater than 3 to 1. If the experimental variogram appears to support an anisotropy of greater than 3 to 1, and there is no unambiguous, physical justification for such a severe anisotropy, there may be a trend in the data. Consider detrending the data before carrying out your variogram analysis.

- Try the model and see how the resulting grid looks in a contour map. If you have competing candidate variogram models, generate a grid and contour map from each. If there are no significant differences, choose the simplest variogram model.

- The range of the variogram is often close to the average size of physical anomalies in the spatial fluctuation of the Z values. In the absence of a reliable experiment variogram, this rule of thumb may be used to postulate a variogram range.

- An experimental variogram that fluctuates around a constant value is not an ill-behaved variogram. It is an indication that the Z values are uncorrelated at the scale of the typical sample spacing. In such a situation a contour map, regardless of the gridding method used, is an unreliable representation of the data, and more data at closer sample spacing are needed for detailed local characterization.

- If the following three conditions are met, then the sample variance is a reasonable approximation for the variogram sill:

    1. The data are evenly distributed across the area of interest, as displayed in a post map.

    2. There is no significant trend in the data across the area of interest.

    3. The dimension of the area of interest is more than three times the effective variogram range.

Variogram Frequently Asked Questions

Q. What is a variogram?

A. A variogram is a statistically-based, quantitative, description of a surface's roughness. A variogram is a function of a separation vector: this includes both distance and direction, or a Δx and a Δy. The variogram function yields the average dissimilarity between points separated by the specified vector (dissimilarity is measured by the squared difference in the Z-values).

Q. What is the difference between a variogram and a semi-variogram?

A. The difference is simply a factor of 2. The variogram was originally defined as

$$2\gamma(\Delta x,\Delta y) = \varepsilon\left[\{Z(x + \Delta x, y + \Delta y) - Z(x,y)\}^2\right]$$

where 2γ( ) was given the name variogram. The function of practical interest was γ( ) which was given the name semi-variogram since it is one-half the variogram function. Since only the function γ( ) is used in kriging, the prefix semi- is regularly dropped, and the function γ( ) is interchangeably called the variogram and the semi-variogram in the geostatistical literature.

Q. What are the units of the variogram plot?

A. Let (X, Y) represent the location of an observation, and let Z represent the associated observed value. The abscissa (lag distance axis) has the same units as X and Y. The ordinate (variogram axis) has the units equal to the units of Z squared (like the variance of Z). For

example, let

- X represent the distance east of a benchmark measured in [m],
- Y represent the distance north of a benchmark also measured in [m], and
- Z represent rainfall intensity measured in [cm/hr].

The abscissa subsequently has units of [m], and the ordinate has units of [(cm/hr)2].

Q. Why can't I specify the sill in the variogram parameters? How is the scale related to the sill?

A. The sill is the value of the ordinate at which the variogram levels off. A variogram model may be comprised of multiple components (e.g. a spherical component plus a nugget effect). While the individual components may be said to have a sill, such usage often leads to confusion between the sill of the overall model and the various sills of the individual components. A further source of confusion is that some components, and therefore models including such components, do not have a sill (e.g. the linear model).

To reduce confusion, the word sill was used to describe the leveling-off of the entire model, and not of the individual components. When discussing the leveling-off of an individual component, we use the term scale.

Q. What does the variogram direction do? Why does the variogram change when I change the direction?

A. This behavior is called anisotropy. The variogram measures dissimilarity as a function of separation distance and direction. In many physical settings it is possible for the variable of interest to change more rapidly in one direction than in another. For example, the distribution of grain size changes more rapidly in a direction perpendicular to the shoreline than it does parallel to the shoreline. Similarly, in an arid climate the prevalence of certain species of plants changes more rapidly as one moves in a direction perpendicular to a river than it does as one moves along the river.; i.e. different behavior in different directions. The direction parameter allows you to investigate the variogram in different directions. Thus, you can identify and quantify the anisotropy.

Q. Are the various variogram directions used when gridding with Surfer?

A. Yes. The variogram anisotropy is explicitly entered when specifying the variogram for gridding with the kriging algorithm.

Q. Does Surfer have a multi-directional gridding algorithm so that more than just the anisotropy can be changed (different sills in different directions)?

A. The variogram model contains the necessary information to describe the behavior of the variogram in all directions. The spherical model, for example, is a funnel-like surface. The variogram plot on which a single line is drawn to represent the variogram model is a vertical slice through the variogram model.

The anisotropy ratio and angle describe the ratio of the Lengths (A) in different directions (geometric anisotropy). Sometimes the apparent sill of the experimental variogram differs in different directions (zonal anisotropy). For example, an experimental variogram appears to be well described by a spherical model with a Length (A) of 100 meters. It levels off (sill) at 2.0 in the East/West direction (Angle = 0 degrees), while it levels off at 3.0 in the North/South direction (Angle = 90 degrees). Surfer can reproduce such behavior using a nested model. Make the first component an isotropic spherical model with a length of 100 and a scale of 2. Make the second component be an anisotropic model with length 100, ratio1000, direction 90, and scale of 1. Adding the two components results in a compound model with a sill of 2 in the East/West direction and a sill of 3 in the North/South direction.

Q. Why are some of the cells blank when exporting variogram data?

A. A cell is blank when there are no data pairs separated by the specified lag distance interval and angular tolerance represented by the cell.

Q. How do I perform kriging cross-validation, jack-knifing (leave-out-one modeling), and such.

A. The **Grids | New Grid | Grid Data** command in **Surfer** offers an extensive set of cross-validation tools. The *Cross Validation Report* (using the Kriging gridding method) offers numerous quantitative measures that can be used as a goodness-of-fit statistic for the variogram. We recommend three particular statistics as the most consistently useful: the median absolute deviation of the cross validation residuals, the standard deviation of the cross validation residuals, and the rank correlation between the measurements and the estimates.

Q. How do I plot the 3D variogram surface?

A. This option is not available in **Surfer**.

Q. Does Surfer calculate S2 (variance of the data) and how do we find this value in the program outputs?

A. **Surfer** uses the formula for calculating the sample variance in the experimental variogram. This value is reported in the **Statistics** tab of the Variogram Properties and in the Variogram Report.

Q. Why doesn't the grid change when using a linear variogram with different slope values?

A. This occurs if there is no nugget effect. Changing the slope of a linear variogram is equivalent to changing only the scale of the observation locations; thus, the ratio of the variogram values between any two pairs of points is unchanged by changing the slope. It is important to note, however, that the kriging variance is sensitive to the slope even if the interpolation weights and interpolated value are not.

Q. How can I get information on the distance between data points?

A. Plot your data as a post map (see the **Home | New Map | Post** command). **Surfer** reports some useful quantitative statistics in the Variogram Report and at the bottom of the table on the **Statistics** tab of the variogram properties. For example, the Nearest Neighbor (NN) values for the ExampleDataSetC.xls are:

- Avg. distance to nearest neighbor: 2.90403
- Min. distance to nearest neighbor: 1

- Max. distance to nearest neighbor: 9.48683
- Gamma for nearest neighbors: 64.735

In addition, the **Grids | New Grid | Grid Data** menu with the Data Metrics gridding method offers an extensive suite of tools for characterizing the data geometry. This includes a neighborhood count, data density, distance to the nearest neighbor, etc.

Q. When using an exponential variogram model for kriging, is the input parameter Length (A) the correlation length (i.e. the length at which the variogram value reaches 63 percent of the sill) or the practical range (i.e. the length at which the variogram value reaches 95 percent of the sill)?

A. Different geostatistical software packages use various definitions for the length parameter of an exponential model. **Surfer** defines the *Length (A)* parameter, as the correlation length. Graphically, the correlation length corresponds to the lag distance at which the tangent line to the curve at lag distance zero intersects the sill of the exponential model. The exponential model achieves about 63 percent of the sill at Lag Distance (A). At a lag distance of 3A the exponential model reaches about 95 percent of the sill.

Q. When applying the Gaussian model, I seem to be getting a mismatch between what I see on the plot of the variogram and the length reported by the software after AutoFit has finished. The length at which the sill is reached seems to be much larger than the value reported under the Model tab when AutoFit has finished.

A. As with the exponential model, different geostatistical software packages use various definitions for the Length (A) parameter of a Gaussian model. **Surfer** defines the *Length (A)* parameter as the correlation length. Graphically, the correlation length corresponds to the lag distance of the inflection point of the curve. The Gaussian model achieves about 63 percent of the sill at Lag Distance (A). At a lag distance of 3A the Gaussian model reaches about 99.99 percent of the sill.

Q. Is there a function in the variogram procedure in the new Surfer that can actually tell me the exact nugget, length, and scale?

A. To view the nugget, length, and scale, choose the **Grid | Variogram | New Variogram** command. Specify the data file, click the *Open* button, and click *OK* to use the defaults. Click on the variogram graph and the variogram properties are listed in the **Properties** window on the **Model** tab. The *Nugget Error* and *Micro Variance* are listed for the nugget and the default linear model *Slope* and *Anisotropy* parameters are listed for the linear component. Other components have other options, such as *Scale* or *Length (A)* listed.

Q. I tried to fit a spherical variogram using AutoFit and all the standard settings. The fit is poor. Could you explain how Surfer does the fitting? How can I make an appropriate fit using the spherical function?

A. The *AutoFit* function requires reasonable initial parameters to arrive at a good fit. Examine the plot of the experimental variogram and estimate the *Length (A)* and *Scale* visually. If it is difficult to estimate the scale from your data set, use the Z variance listed in the Variogram Report and on the **Statistics** tab of the variogram properties.

Q. I set the Lag Width to a particular value. Why aren't the points on the variogram plotted at multiples of this value?

A. The *Lag Width* defines the midpoint of a range plus or minus half the lag width. The point is plotted at the average separation distance for all points within that range.

Q. How can I get a list of the average separation distances?

A. Export the variogram to a data file. This file includes the average lags, half the average variance, and the number of pairs in the range.

Q. Is it possible to create a grid file from two juxtaposed areas on which different variogram models are applied? The areas have spatially contrasted morphologies.

A. If the areas are defined by rectangles, you may be able to grid each area with a separate model, convert the grid to XYZ data files, import the data files in the worksheet and regrid using the Nearest Neighbor method to prevent further interpolation.

Q. How is the drift used within the kriging algorithm? Should the modeling be done on transformed data (data minus drift) or on non-transformed data? Kriging with prior removal of the trend is different than Universal Kriging.

A. We recommend calculating the variogram model of the detrended data prior to applying drift during kriging, after Kitanidis, 1997, p. 143, section 6.12.

Q. I haven't found any procedure in Surfer that can help me to tell HOW well the theoretical variogram models are fitted to the experimental data in my variogram. I understand that I can use AutoFit but I would like a quantitative number that measures the goodness-of-fit.

A. The best measure of "how well" the theoretical model fits the data is to try it out and see. Grid the data with the selected variogram model. Create a contour map and overlay a post map of the data file. Judge the quality of the model by examining the contour map. The Cross Validation Report (using the Kriging gridding method) offers numerous quantitative measures that can be used as a goodness-of-fit statistic for the variogram. We recommend three particular statistics as the most consistently useful: the median absolute deviation of the cross validation residuals, the standard deviation of the cross validation residuals, and the rank correlation between the measurements and the estimates.

Q. Can Surfer import a variogram model from Variowin, GeoEAS, GSLIB, or other software?

A. **Surfer** does not have this feature. Submit a request to surfer-support@goldensoftware.com with the name of the software you would like to support.

Q. I've read somewhere that the scale should not exceed the variance. Is this true? For the data set ExampleDatSetC.xls, the variance of Z is 405.371, but you recommend a scale of 425.

A. Under a reasonable and commonly applicable set of assumptions, the sill of the variogram should be equal to the variance of the underlying population; however, the sample variance is often an inappropriate measure of the population variance (the standard formula for estimating the variance assumes independent data, which is invalid in most geological settings). Thus, a rule like "the scale should not exceed the variance" is not a useful generalization. The critical result is

$$\varepsilon\left[S^2\right] = \frac{1}{n^2}\sum_{i-1}^{n}\sum_{j=1}^{n}\gamma(x_i - x_j,\, y_i - y_j)$$

This result states that the expected value of the sample variance, **S 2**, is equal to the average value of the variogram between all **n 2** pairs of sample values. If the **n** sample values are evenly distributed over an areal extent many times larger than the range of the variogram, then there will be significantly more pairs at long separation distances and fewer pairs at short separation distances. Thus, the average value of the variogram between all pairs will be the average of many values equal to the sill and a few values less than the sill. In this case, the sample variance is a reasonable first estimate for the variogram sill.

Geostatistical References

Armstrong, M. [ed.], 1989, *Geostatistics*, volumes 1 and 2, Kluwer Academic Publishers, 1027 pp.

Armstrong, M., 1997, *Basic Linear Geostatistics*, Springer-Verlag, New York, ISBN: 3540618457, 220 pp.

Baafi, E. Y., 1997, *Geostatistics Wollongong '96 – Quantitative Geology and Geostatistics*, Kluwer Academic Publishers, ISBN: 0792302044, 1333 pp.

Bras, R., and Rodriguez-Iturbe, R., 1985, *Random Functions and Hydrology*, Addison-Wesley, 559 pp.

Chiles, J., and Delfiner, P., 1999, *Geostatistics: Modeling Spatial Uncertainty*, John Wiley and Sons, New York, ISBN: 0471083151, 695 pp.

Christakos, G., 2001, *Modern Spatiotemporal Geostatistics*, Oxford University Press; ISBN: 0195138953, 312 pp.

Clark, I., 1979, *Practical Geostatistics*, Applied Science Publishers, 129 pp.

Clark, I., and W. V. Harper, 2000, *Practical Geostatistics 2000*, Ecosse North America, LLC; ISBN: 0970331703, 442 pp.

Cressie, N., 1991, *Statistics for Spatial Data*, John Wiley and Sons, New York, 900 pp.

David, M., 1977, *Geostatistical Ore Reserve Estimation*, 2nd edition, Elsevier, 364 pp.

David, M., 1988, *Handbook of Applied Advanced Geostatistical Ore Reserve Estimation*, Elsevier, 216 pp.

Davis, J., 1973, *Statistics and Data Analysis in Geology*, Wiley, 550 pp.

deMarsily, G., 1986, *Quantitative Hydrogeology*, Academic Press, 440 pp.

Deutsch, Clayton, V., Journel, Andre G., 1992, *GSLIB - Geostatistical Software Library and User's Guide*, Oxford University Press, New York, 338 pp.

Draper, N., and Smith, H., 1981, *Applied Regression Analysis*, second edition, Wiley-Interscience, 709 pp.

Dubrule, O. 1998, *Geostatistics in Petroleum Geology*, American Association of Petroleum Geologists; ISBN: 0891811877.

Gelhar, L. W., 1993, *Stochastic Subsurface Hydrology*, Prentice Hall, Englewood Cliffs, New Jersey, 390 pp.

Geostatistics, 1980, McGraw-Hill, 168 pp.

Gilbert, R.O., 1987, *Statistical Methods for Environmental Pollution Monitoring,* Van Nostrand Reihold Company, 320 pp.

Gomez-Hernandez, J., A. Soares, and R. Froidevaux [eds.], *Geoenv II -- Geostatistics for Environmental Applications,* Kluwer Academic Publishers, 562 pp.

Goovaerts, P. 1997, *Geostatistics for Natural Resources Evaluation,* Oxford University Press, New York, 483 pp.

Guarascio, M., David, M., and Huijbregts, C. [eds.], 1976, *Advanced Geostatistics in the Mineral Industry,* Reidel, 491 pp.

Gy, P., 1979, *Sampling of Particulate Material*s, Elsevier, 431 pp.

Houlding, S. W., 2000, *Practical Geostatistics - Modeling and Spatial Analysis,* Springer-Verlag,New York, 160 pp.

Hohn, M. E., 1998, *Geostatistics and Petroleum Geology,* 2nd edition, Kluwer Academic Publishers; ISBN: 041275780X, 238 pp.

Isaaks, E. H., and Srivastava, R. M., 1989, *An Introduction to Applied Geostatistics,* Oxford University Press, New York, 561 pp.

Johnson, M. E., 1987, *Multivariate Statistical Simulation*, John Wiley and Sons, New York, 230 pp.

Journel, A.G., and Huijbregts, C., 1978, *Mining Geostatistics,* Academic Press, 600 pp.

Journel, A.G., 1988, *Geostatistics for the Environmetal Sciences,* EPA Project Report, Project No. CR 811893, EPA/EMSL, Las Vegas, Nevada.

Journel, A.G., 1989, *Fundamentals of Geostatistics in Five Lessons,* American Geophysical Union, Washington D.C.

Kitanidis, P. K., 1997, *Introduction to Geostatistics: Applications in Hydrogeology,* ISBN 0-521-58747-6.

Koch, G., and Link, R., 1971, *Statistical Analysis of Geological Data,* volumes I and II, Wiley, 375 pp. now available through Dover reprints.

Krige, D., 1978, *Lognormal DeWijsian Geostatistics for Ore Evaluation,* South African I.M.M., 55 pp.

Knudsen, H.P, 1988, *A Short Course on Geostatistical Ore Reserve Estimation,* Division of Mining and Minerals Engineering, Montana Tech.

Manoukian, E.B., 1986, *Modern Concepts and Theorems of Mathematical Statistics,* Springer-Verlag, 156 pp.

Matheron, G., 1965, *Les Variables Regionalisees et leur Estimation,* Editions Masson, Paris.

Matheron, G., and Armstrong, M. [eds.], 1987, *Geostatistical Case Studies,* Reidel, 248 pp.

Monestiez, P., D. Allard, R. Froidevaux [eds.], 2001, *Geoenv III -- Geostatistics for Environmental Applications,* Kluwer Academic Publishers, ISBN: 0792371070, 555 pp.

Olea, R., 1999, *Geostatistics for Engineers and Earth Scientists,* Kluwer Academic Publishers, Boston, MA, 303 pp.

Pannatier, Y., 1996, *VARIOWIN: Software for Spatial Data Analysis in 2D,* Springer-Verlag, New York, 91 pp.

Rendu, J., 1978, *An Introduction to Geostatistical Methods of Mineral Evaluation,* South African I.M.M., 100pp.

Ripley, B., 1981, *Spatial Statistics,* John Wiley and Sons, New York, 252 pp.

Rouhani, S., and M. Srivastava, 1997, *Geostatistics for Environmental and Geotechnical Applcations,* ASTM Special Technical Publication 1283, American Society for Testing and Materials, ISBN: 0803124147.

Rivoirard, J., J. Simmonds, K. G. Foote, P. Fernandes, and N. Bez [eds.], *Geostatistics for Estimating Fish Abundance,* Blackwell Science Inc; ISBN: 0632054441, 206 pp.

Soares, A. [ed.], 1992, *Geostatistics Troia '92,* volumes 1 and 2, Reidel, 1088 pp.

Soares, A., Gomez-Hernandez, J., and R. Froidevaux [eds.], *Geoenv I -- Geostatistics for Environmental Applications,* Kluwer Academic Publishers, ISBN: 0792345908, 516 pp.

Thompson, S. K., 1992, *Sampling,* John Wiley and Sons, New York, 343 pp.

Verly, G., David, M., Journel, A.G., and Marachel, A. [eds.], 1984, *Geostatistics for Natural Resources Characterization,* volumes 1 and 2, Reidel, 1092 pp.

Webster, R. and M. A. Oliver, 2001, *Geostatistics for Environmental Scientists,* John Wiley and Sons, Ltd., Chichester, England, 271 pp.

Chapter 6 - Grid Editor

Grid Editor

The **File | Open**, **Grids | Editor | Grid Editor** command, the  button, and the **Map Tools | Edit Layer | Grid** commands open the grid editor as a new document.

- The **Grids | Editor | Grid Editor** command and the  button open a grid file with the **Open Grid** dialog.
- The **Map Tools | Edit Layer | Grid** command opens the grid file from the selected map layer in the plot document. You can also edit the grid for a map layer by right-clicking on the map layer and clicking **Edit Grid**. This command enables the **Update Layer** command in the grid editor. The **Map Tools | Edit Layer | Grid** command is not available for 1-grid vector and 2-grid vector layers. When accessed this way, the Grid Editor will display any Post, Classed Post, and vector or raster Base layers in the Map as context layers.

The grid editor contains various methods for editing the grid Z values. Editing the grid Z values will change the appearance of any grid-based maps. For example, the grid editor can be used to edit contours on a contour map or change the surface in a 3D surface map.

Each grid node is indicated with a black "+" in the grid editor window by default. Each NoData grid node is indicated with a blue "x" by default. The active node is highlighted with a red diamond. To move between grid nodes, press the arrow keys, or click a node with the **Select** tool active to make it the active node. The grid editor also includes contours, node labels, and a color fill. The grid appearance is controlled by the items in the **Contents** window and the properties displayed in the **Properties** window. Note the **Undo** command does not undo changes in the **Properties** window in the grid editor.

Context Layers

When the grid editor is accessed either using the **Map Tools | Edit Layer | Grid** command or by right-clicking on the map layer and clicking **Edit Grid**, any Post, Classed Post, and vector or raster Base layers will be displayed as context layers to assist with editing the grid. Any layers being used as context layers will be displayed in the **Contents** window. Visibility for these context layers can be controlled by toggling the layers on and off in the **Contents** window, or, by toggling visibility for those layers in the 2D plot window.

The context layer will have labels in the grid editor if the layer has labels applied in the 2D plot window. These labels can be toggled on and off by selecting the desired context layer in the **Contents** window and toggling the *Show labels* option in the **Properties** window. If a layer does not have labels applied in the 2D plot window, then there are no properties for the context layers in the grid editor – all the properties are changed in the 2D plot window.

Note, no context layers will be displayed if the grid file being edited has been geo-transformed. To remove a geotransform from a grid layer, set the map target coordinate system to the same coordinate system as the grid layer source coordinate system.

Images in the Grid Editor

The grid editor also allows you to open an image file and save as a grid file.

A grid requires a single floating point value at each grid node. Images contain colors which are three separate values (Red, Green, Blue) at each pixel.

Color Image

Color image formats are converted to a single floating point value by calculating the intensity of each color value using the intensity equation:

I = A(.30R + .59G + .11B)

where I = intensity, R,G,B,A are the red, green, blue, and normalized alpha values.

For example, a pixel from a color image with Red=255, Green=0, and Blue=0 would be mapped to a grid node with the value of:

I = .30*255 + .59*0 + .11*0 = 77

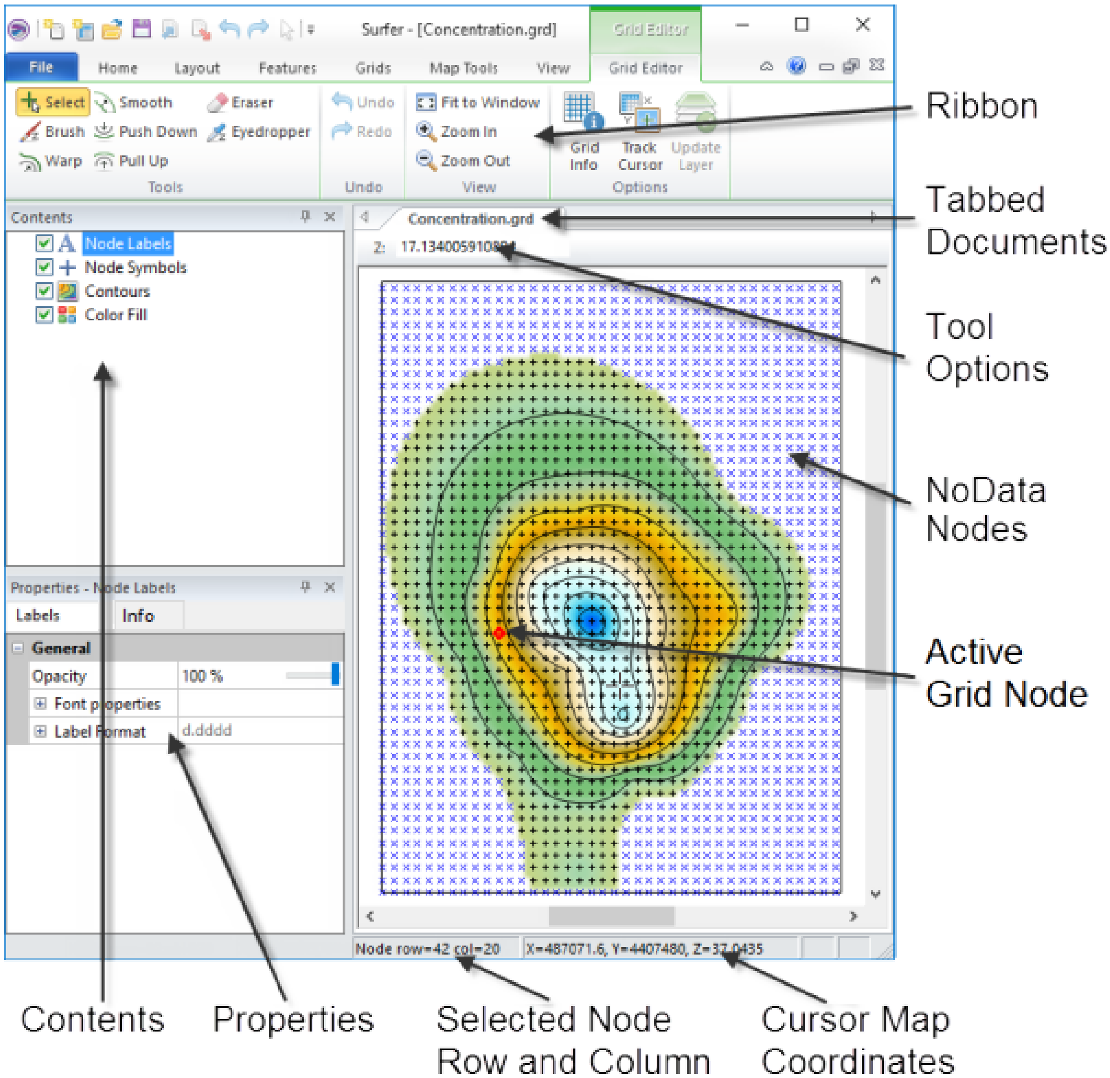
Grayscale Image

Grayscale images are imported directly. Grayscale images have a single color value and do not need to use the intensity equation. Surfer does not normalize the grayscale value. The value is used exactly as specified in the image.

For example, consider a grayscale image with a pixel that contains a value of 55. The grid node value would be set to 55.

Grid Editor Window

The following image and table explain the purpose of the grid editor window components.



*This is the **Surfer** grid editor with the **Contents** and **Properties** windows on the left and grid editor window on the right.*

Component Name	Component Definition
Ribbon	The ribbon contains the **Grid Editor** commands.
Contents	Toggle the display of the *Node Labels*, *Node Symbols*, *Contours*, *Context Layers*, and *Color Fill* with the **Contents** window.
Properties	Edit *Node Labels*, *Node Symbols*, *Contours*, *Context Layers*, and *Color Fill* **Labels** properties and view **Info** properties in the **Properties** window.
Tabbed Documents	Plot windows, worksheet windows, and grid editor windows are displayed as tabbed documents.
Tool Options	The tool options bar contains the *Z value box*, *Brush size*, *Density*, and/or *Pressure* depending on the selected tool mode.

Active Node	The node that is currently selected. The active node is highlighted with a red diamond.
Grid Node	Each grid node is indicated with a black "+" in the grid editor window by default. NoData nodes are indicated with a blue "x".
Status Bar	The status bar includes information about the selected property, active node grid coordinates, and cursor map coordinates.

Grid Editor Commands

The **Grid Editor** ribbon tab includes the following commands:

Select	Select a grid node to edit the grid Z values one node at a time
Brush	Apply a specific Z value to one or more nodes
Warp	Drag grid values from one region into another
Smooth	Apply weighted averaging to grid nodes
Push Down	Decrease grid node values
Pull Up	Increase grid node values
Eraser	Assign the NoData value to grid nodes
Eyedropper	Acquire a grid node value by clicking on the grid
Undo	Undo the last operation
Redo	Redo the last undone operation
Fit to Window	Fits the entire grid in the grid editor window
Zoom In	Increase the grid editor window magnification
Zoom Out	Decrease the grid editor window magnification
Zoom Rectangle	Zoom in to an area of interest
Grid Info	Display information about the grid in a report window
Track Cursor	Track cursor location across plot, worksheet, and grid editor windows for maps, data files, and grids.
Update Layer	Updates the associated map layer with the edited grid

Using the Grid Editor

The grid editor can be used on existing map layers or on grid files without first creating a map.

To edit a map layer's grid:

1. Select the map layer created from a grid file to edit in the plot document **Contents** window. Only the grid for this map layer will be edited even when multiple layers, such as contour or color relief, use the same grid file.

2. Click **Map Tools | Edit Layer | Grid** in the plot window. The grid file is opened and is represented by a filled contour map. The location of each grid node in the file is marked with a black "+". NoData nodes are marked with a blue "x".
3. Use the **Grid Editor | Tools** commands to make the desired adjustments to the grid.
4. When you are done editing the grid, click the **Grid Editor | Options | Update Layer** command to update the map layer in the plot document with your grid.
5. Click the plot document tab to view the changes to the map layer. If you wish to revert the changes to the map layer, click the **Undo** command while viewing the plot window. If you are satisfied with the changes to the map layer, you may wish to save the edited grid to a file.
6. If you wish to save your edits to a file, click **File | Save As** to create a new grid file. Click **File | Save** to overwrite the existing grid file. It is necessary to save your edits to a file with **Save** or **Save As** if you wish to update all layers in your map to use the edited grid.
7. To close the grid editor window, click the **File | Close** command or click the X in the grid editor document tab. To view an existing window and keep the grid editor window open, click on another document tab.

To edit a grid file:

1. Click the **Grids | Editor | Grid Editor** command and select the grid file in the **Open Grid** dialog. The grid file is opened and is represented by a filled contour map. The location of each grid node in the file is marked with a black "+". NoData nodes are marked with a blue "x".
2. Use the **Grid Editor | Tools** commands to make the desired adjustments to the grid.
3. When you are done editing the grid, click **File | Save As** to create a new grid file. Click **File | Save** to overwrite the existing grid file. It is necessary to save your edited grid to a file with **Save** or **Save As** if you wish to create map layers with the grid.
4. To close the grid editor window, click the **File | Close** command or click the X in the grid editor document tab. To view an existing window and keep the grid editor window open, click on another document tab.

Node Labels - Grid Editor

The *Node Labels* object in the grid editor **Contents** window contains the grid editor node label properties. Click the *Node Labels* object to change the grid node label properties in the **Properties** window. Click the **Info** property page to view information about the grid file.

The visibility check box next to the *Node Labels* object is used to turn on or off the node label display in the grid editor window. When the check box is checked, a number appears below each grid node when zoomed in a sufficient amount.

When the zoom or density is such that the labels would be too close together, no labels appear. NoData node labels appear as "n/a".

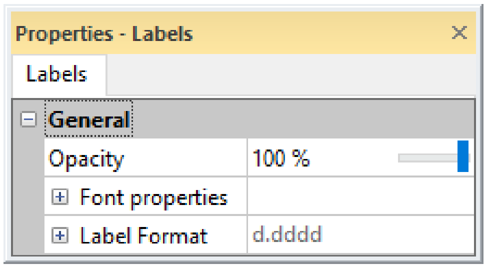
This image displays grid nodes, grid labels, and contour lines for a grid file.

Info Properties

An **Info** page is displayed in the Properties window when the *Node Labels* object is selected. See the Grid Info Properties help topic for more information on these common, information only, properties.

Grid Node Labels Properties

A **Labels** page is displayed in the Properties window when the *Node Labels* object is selected. Edit the label opacity, font properties, and format in the **Properties** window.



Edit label properties in the ***Properties*** *window.*

Opacity

Change the *Opacity* of the labels by entering a value from 0% (completely transparent) to 100% (completely opaque). To change the opacity, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Or, click and drag the to change the opacity percentage.

Font Properties

Click the ⊞ next to *Font Properties* to display the Font Properties section. The font properties are applied to all grid node labels.

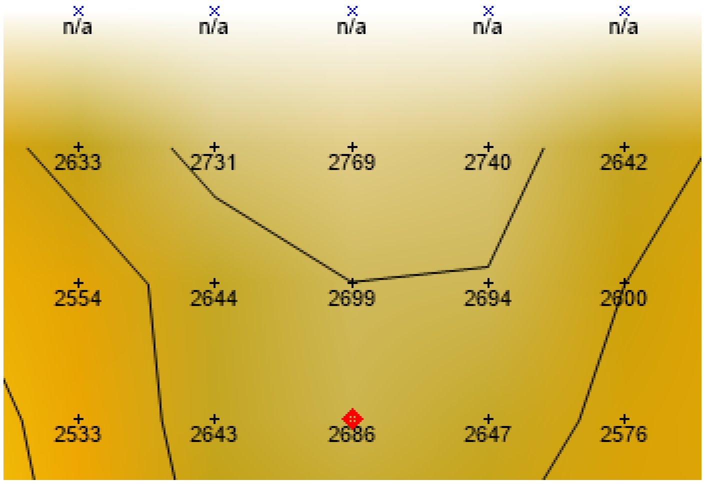
Label Format

Click the ⊞ next to *Label Format* to display the Label Format section. The numeric format is applied to all grid node labels.

Node Symbols - Grid Editor

The *Node Symbols* object in the grid editor **Contents** window contains the grid editor symbol properties. Click the *Node Symbols* object to change the grid node symbol properties in the **Properties** window. Click the **Info** property page to view information about the grid file.

The visibility check box next to the *Node Symbols* object is used to turn on or off the symbol display in the grid editor window. When the check box is checked, a black "+" appears at the location of each grid node. NoData nodes appear with a blue "x" at each grid node location. When the command is not checked, the symbols are not displayed, but you can still edit the grid. Some node markers are not displayed if the zoom or density is such that they are too close together. Although not all of the markers are shown, the active node indicator still "snaps" to each grid node as the arrow keys and mouse are used.



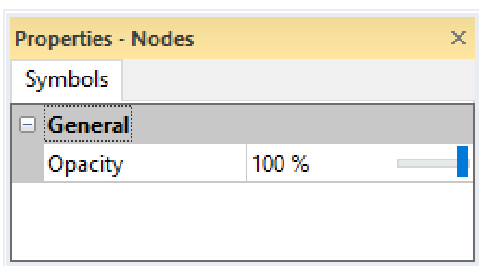
This image displays grid node symbols, labels, contour lines, and a color fill for a grid file.

Info Properties

An **Info** page is displayed in the Properties window when the *Node Symbols* object is selected. See the Grid Info Properties help topic for more information on these common, information only, properties.

Grid Node Symbol Properties

The **Symbols** page is displayed in the Properties window when the *Node Symbols* object is selected. Edit the symbol opacity in the **Properties** window.



*Edit symbol opacity in the **Properties** window.*

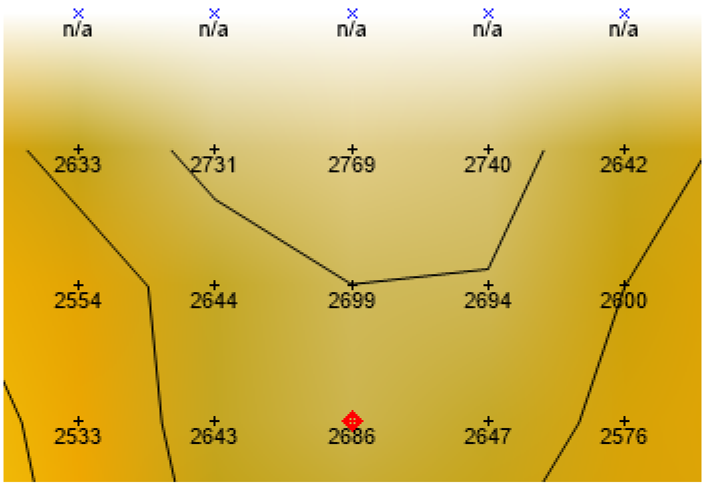
Opacity

Change the *Opacity* of the grid node symbols by entering a value from 0% (completely transparent) to 100% (completely opaque). To change the opacity, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Or, click and drag the  to change the opacity percentage.

Contours - Grid Editor

The *Contours* object in the grid editor **Contents** window contains the grid editor contour level and line properties. Click the *Contours* object to change the contour properties in the **Properties** window. Click the **Info** property page to view information about the grid file.

The visibility check box next to the *Contours* object is used to turn on or off the contour lines in the grid editor window. When the check box is checked, contour lines are displayed on the grid in the grid editor. When the command is not checked, the contour lines are not displayed.



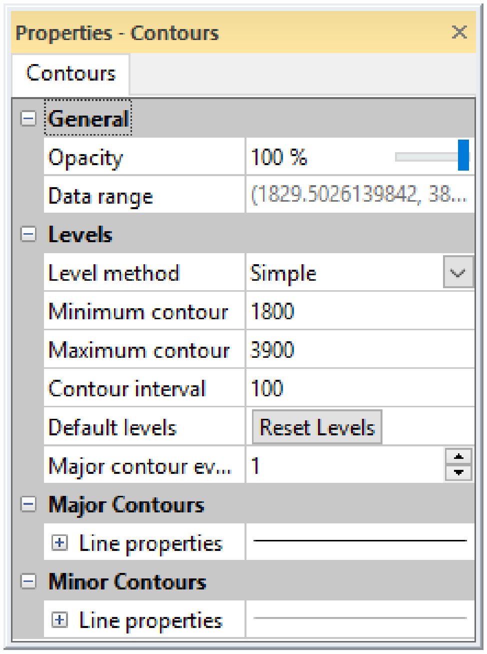
This image displays grid node symbols, labels, contour lines, and a color fill for a grid file.

Info Properties

An **Info** page is displayed in the Properties window when the *Contours* object is selected. See the Grid Info Properties help topic for more information on these common, information only, properties.

Grid Contour Properties

The **Contours** page is displayed in the Properties window when the *Contours* object is selected. Edit the contour opacity, level method and properties, and line properties in the **Properties** window.



Edit contour properties in the ***Properties*** *window.*

Opacity

Change the *Opacity* of the labels by entering a value from 0% (completely transparent) to 100% (completely opaque). To change the opacity, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Or, click and drag the  to change the opacity percentage.

Data Range

The *Data range* displays the Z minimum and maximum values read from the grid file.

Level Method

The *Level method* determines which options are available for setting the grid editor contour level properties. Available options are *Simple, Logarithmic,* and *Advanced. Simple* allows basic options to be set. These are discussed below. *Logarithmic* uses a log(10) interval to display the contours. These are also discussed below. *Advanced* allows individual lines to have z-level and line properties set individually. These options are explained on the Contour Levels - Grid Editor topic. To set the *Level method*, click on the existing method and select the desired method from the list.

Minimum Contour

Set the Minimum contour to the value of the first contour line you want displayed in the grid editor. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard or click another command in the **Properties** window for the value to be applied. Values are in map Z units. When the *Level method* is set to *Logarithmic* , the *Minimum contour* value must be greater than 0.

Maximum Contour

Set the *Maximum contour* to the value of the last contour line you want displayed in the grid editor. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard or click another command in the **Properties** window for the value to be applied. Values are in map Z units.

Contour Interval

The *Contour interval* option is available when the *Level method* is set to *Simple* . The *Contour interval* defines the spacing units between adjacent contour lines. This will affect how many contours appear in the grid editor and how close those contours are to one another. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard or click another command in the **Properties** window for the value to be applied. Values are in map Z units.

Surfer uses the *Contour interval* , the *Minimum contour* value, and the *Maximum contour* value to determine how many contour lines are created on the map. **Surfer** will issue a warning message if the number of contour lines increases above 5000. Click *OK* in the warning message and alter the *Contour interval* , the *Minimum contour* value, or the *Maximum contour* value so that the total number of lines is less than 5000.

Default Levels

The *Default levels* option is available when the *Level method* is set to *Simple*. If the *Minimum contour*, *Maximum contour*, or *Contour interval* has changed, you can return to the **Surfer** default values by clicking the Reset Levels button next to *Default levels*. **Surfer** returns the *Minimum contour*, *Maximum contour*, and *Contour interval* to reasonable values.

Major Contour Every

The *Major contour every* option is available when the *Level method* is set to *Simple*. The *Major contour every* option sets the frequency for how often a major contour line appears on the map. Major contour lines and minor contour lines can have different line properties. By default, the first line is the *Minimum contour*. The *Major contour every* value tells the program how many contour lines to skip before the next major contour line. If the *Major contour every* value is set to 1, every contour line is a major contour line. If the value is 2, every other line is a major contour line. If the value is 3, every third line is a major contour line, and so on.

Minor Levels Per Decade

The *Minor levels per decade* option is available when the *Level method* is set to *Logarithmic.* The *Minor levels per decade* controls how many minor contour lines appear between major contour lines. When the *Level method* is set to *Logarithmic*, the major contour lines occur at multiples of 10 (0.001, 0.01, 0.1, 1, 10, 100, 1000, 10000, etc.). The number of contour lines between each major contour line is set by the *Minor levels per decade*. To change the *Minor levels per decade*, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the to increase or decrease the value. The default value is 8.

If the value is 8, minor contour lines appear at 2, 3, 4, 5, 6, 7, 8, and 9 between 1 and 10 or 20, 30, 40, 50, 60, 70, 80, and 90 between 10 and 100. 8 is the most common number of minor contour lines per major. Any value between 0 and 500 can be specified, though. Some common levels are defined below, with the values of the contour lines that would appear.

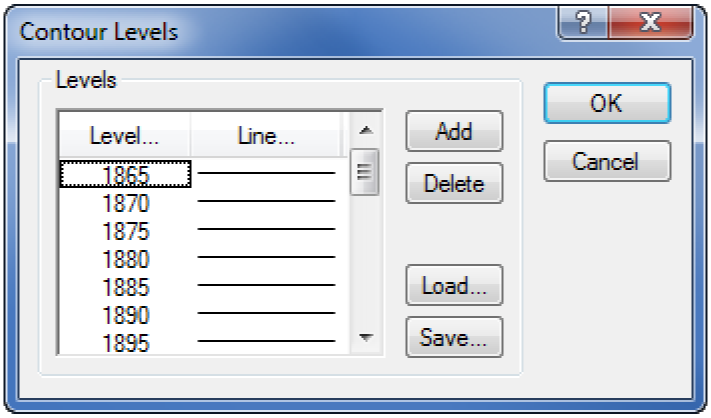
Minor Levels per Decade	Values of Contour Lines
8	1, 2, 3, 4, 5, 6, 7, 8, 9
7	2.125, 3.250, 4.375, 5.500, 6.625, 7.750, 8.875
5	2.5, 4.0, 5.5, 7.0, 8.5
4	2.8, 4.6, 6.4, 8.2
3	3.25, 5.5, 7.75
2	4, 7
1	5.5
0	no minor lines are displayed

Contour Levels

When the *Level method* is set to *Advanced*, the *Contour levels* option is available. Click the *Edit Levels* button to open the Contour Levels dialog, where the advanced level options are set.

Contour Levels - Grid Editor

The **Contour Levels** dialog controls the *Advanced* contours in the Grid Editor. Click the *Edit Levels* button in the Contours page to open the **Contour Levels** dialog.



*The **Contour Levels** dialog controls the display of the contour map in the grid editor window.*

Levels Group

The *Levels* group lists the contour levels and colors of contour lines used in the grid editor window.

Levels

Click the *Level* button to display the Contour Levels dialog. The **Contour Levels** dialog controls the minimum and maximum contour levels as well as the contour interval to use between levels.

Line

The *Line* button displays the Line Spectrum dialog that controls the colors and widths of the minimum and maximum contour lines. **Surfer** automatically produces a gradational spectrum of colors between the minimum and maximum colors.

Add

The *Add* button adds a contour line halfway between selected contour level and the next lower contour. If the minimum contour level is selected, the added contour level is the same as the minimum level.

Delete

The *Delete* button removes the selected contour from the list of contours.

Load

The *Load* button allows you to select an existing level file [.LVL] containing contour level information. All grid files using the level file must have comparable Z data ranges, otherwise contour lines will not appear on the map.

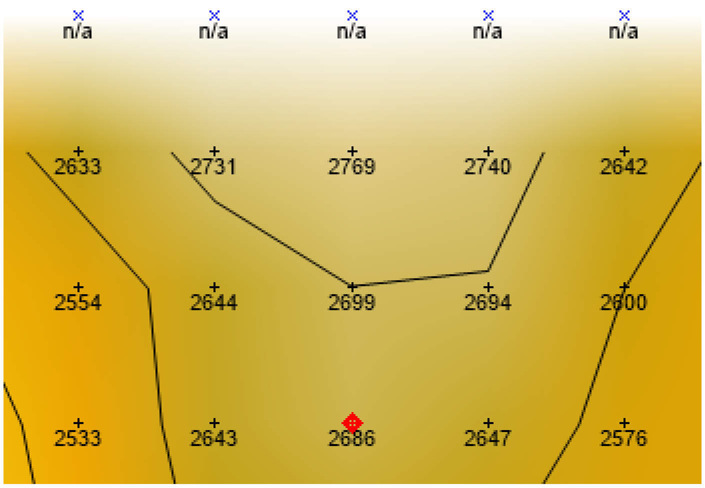
Save

The *Save* button allows you to save the current contour level and line property information to a level file .LVL.

Color Fill - Grid Editor

The *Color Fill* object in the grid editor **Contents** window contains the grid editor color fill properties. Click the *Color Fill* object to change the grid node colors in the **Properties** window.

The visibility check box next to the *Color Fill* object is used to turn on or off the color fill display in the grid editor window. When the check box is checked, the grid editor displays a color relief map. When the command is not checked, the labels, nodes, and contours are displayed on a white background.



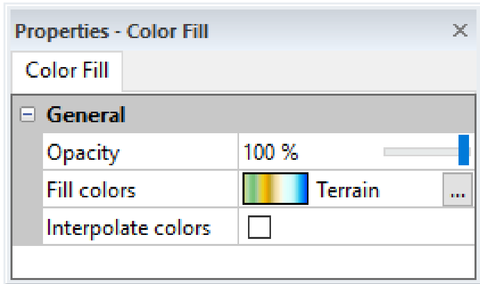
This image displays grid node symbols, labels, contour lines, and a color fill for a grid file.

Info Properties

An **Info** page is displayed in the Properties window when the *Color Fill* object is selected. See the Grid Info Properties help topic for more information on these common, information only, properties.

Grid Node Color Fill Properties

The **Color Fill** page is displayed in the Properties window when the *Color Fill* object is selected. Edit the fill opacity, color map, and interpolation option in the **Properties** window.



*Edit color fill properties in the **Properties** window.*

Opacity

Change the *Opacity* of the color fill by entering a value from 0% (completely transparent) to 100% (completely opaque). To change the opacity, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Or, click and drag the slider to change the opacity percentage.

Fill Colors

The *Fill colors* property controls which color map is used to fill the grid editor window. To use a predefined colormap, click the current selection and select a colormap from the list. The default *Fill colors* is *Terrain*. To edit a colormap or create a custom colormap, click the ... button to open the Colormap Editor.

Interpolate Colors

The *Interpolate colors* check box activates color smoothing on the map. When checked, *Interpolate colors* uses bilinear interpolation to calculate colors on the map. Bilinear interpolation makes the color gradations smoother, but it can slightly slow the on-screen drawing of the grid editor.

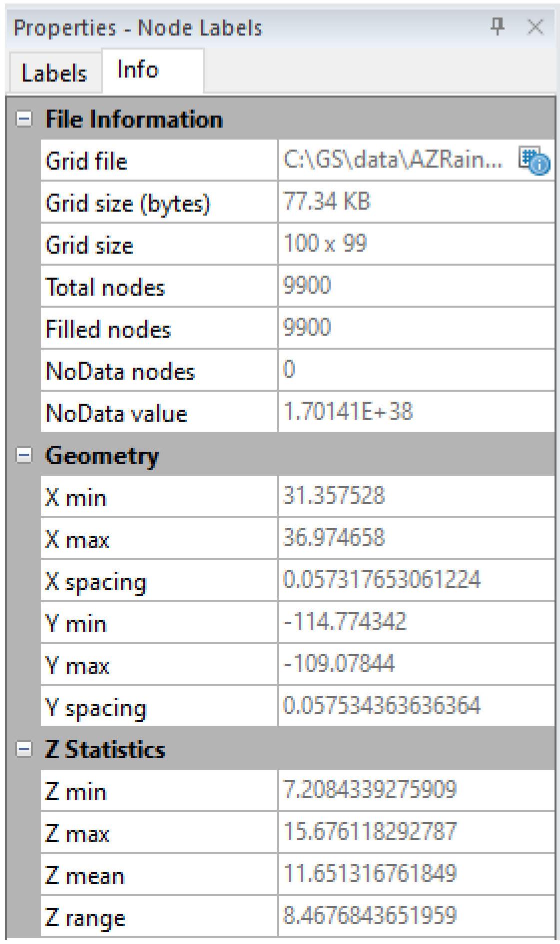
When a dense grid (a grid with relatively large numbers of rows and columns) is used, little difference is seen between the final display whether the *Interpolate colors* option is checked or not. For dense grids, on-screen drawing time can be reduced when the *Interpolate colors* option is unchecked.

When a coarse grid (a grid with relatively few rows and columns) is used and the *Interpolate colors* option is not checked, all pixels within a single grid square are assigned the same color. This creates a grid-square map, and can result in an color fill with a very blocky appearance. For coarse grids, therefore, a smoother appearance results when the *Interpolate colors* check box is checked.

Note there is a difference between a grid node and a grid cell. The non-interpolated color fill maps each grid node to a pixel, which is expanded to a block of pixels to make the map the correct size. The interpolated color fill positions a grid node at the four corners and then interpolates all the interior pixels.

Grid Info Properties

Each of the four content types in the **Grid Editor Contents** window (*Node Labels*, *Node Symbols*, *Contours*, *Color Fill*) have an **Info** properties page. This information contains information about the grid being edited, such as grid resolution and descriptive statistics.



Grid Info Properties

Select - Grid Editor

Click the **Grid Editor | Tools | Select** command or the button to select individual grid nodes in the grid editor. The cursor changes to a crosshair when **Select** mode is active. The active grid nodes is indicated by a red diamond. The active grid node's grid coordinates are displayed in the status bar. Press ESC when in a different tool mode, such as Eraser, to quickly return to select mode.

Select a different grid node by clicking the node with the **Select** tool or pressing the ARROW keys.

Z Value Box

The *Z value box* in the tool options bar at the top of the grid editor window displays the Z value for the active grid node. Type a new value in the *Z value box* and press ENTER to change the grid node value. The Label, Node, Contours, and Color FIll display updates in the grid editor once you press ENTER or select a different grid node. Press ESC while editing a value in the *Z value box* to cancel the change.

Brush

The **Grid Editor | Tools | Brush** command or the button in the grid editor assigns the specified Z value in the *Z value box* to the grid nodes within the brush. Click the **Grid Editor | Tools | Brush** command to start brush mode. The cursor changes to a crosshair with a circle to visually indicate the *Brush size* . The **Brush** command button is highlighted in yellow to indicate brush mode as well.

Click to apply the value in the *Z value box* to grid nodes within the circle. Click and drag to continuously apply the Z value as the cursor moves across the grid. The grid Nodes, Labels, Color Fill, and Contours update automatically as you use the **Brush** command.

Press ESC or click another Grid Editor | Tools command to end **Brush** mode.

Tool Options

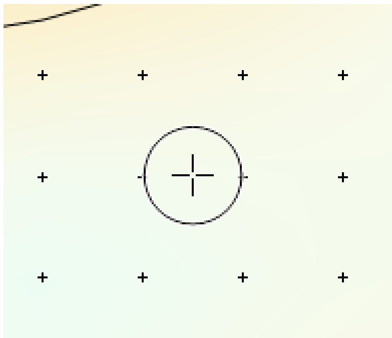
The *Brush size* and *Z value box* controls how the **Brush** tool changes the grid vales.

Brush Size

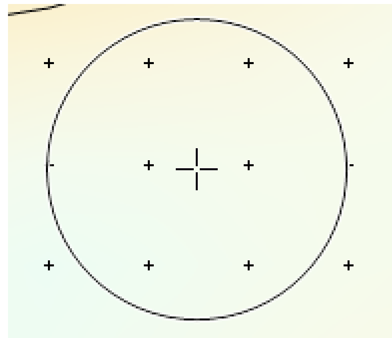
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- Click in the *Brush size* field and type the desired *Brush size*.
- Click the buttons to increment the *Brush size* by one. Click and hold the buttons to quickly scroll the *Brush size* value up or down.
- Press the [key to decrease the *Brush size*. Press the] key to increase the *Brush size*. Press and hold the [or] key to quickly decrease or increase the *Brush size*. The [and] keys increment the *Brush size* by 1 at *Brush size* values less than 10. The increment increases to 5 at a *Brush size* greater than 10 and less than 50. The increment increases to 10 at a *Brush size* greater than 50 and less than 100. The increment size continues to increase as the *Brush size* increases.

The cursor circle increases or decreases size with the *Brush size* setting and zoom level. The *Brush size* setting is saved and persistent across multiple grid editor windows.



A Brush size *of 1 makes the diameter of the brush equal to the horizontal spacing of one grid cell.*



A Brush size *of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.*

Z Value Box

The *Z value box* in the tool options bar at the top of the grid editor window displays the Z value you wish to apply to the grid nodes. Type a new value in the *Z value box* and press ENTER to change the **Brush** Zvalue. By default the *Z value box* uses the grid Z value from the most recently selected grid node.

Press and hold the "i" key to temporarily change to the Eyedropper tool mode while using the **Brush** tool. The cursor changes to an eyedropper . While holding the "i" key, click in the grid to quickly change the value in the *Z value box* to the grid Z value nearest the location you click. Release the "i" key to return to **Brush** mode.

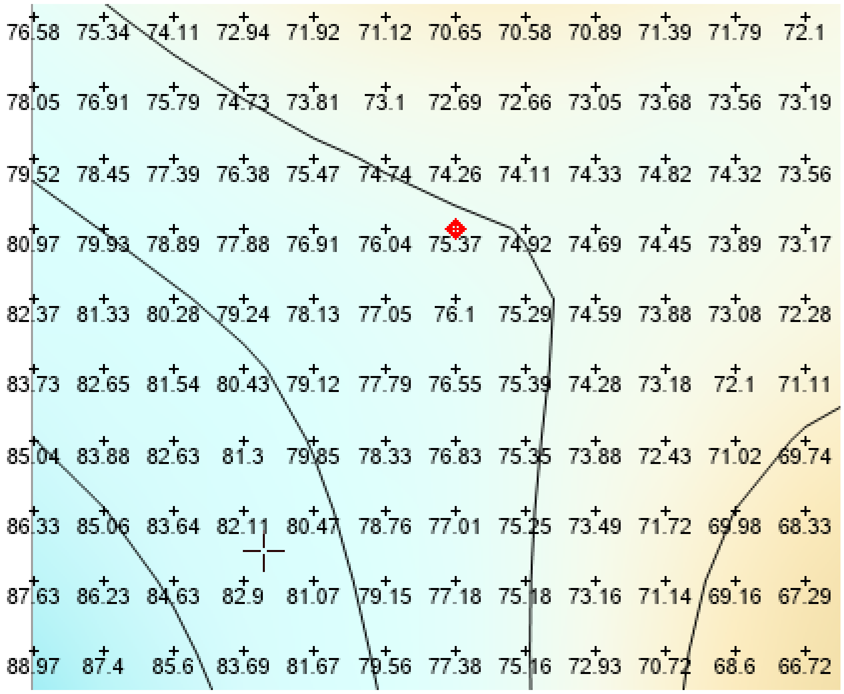
Undo

Each click is registered as one operation in the Undo levels list. For example, if you set the *Brush size* to 1 and click once on five individual grid nodes, you must use the **Undo** command five times to undo all the changes. However, if you click and drag once across all five grid nodes, you must only use the **Undo** command once to undo all the changes.

Brush Grid Values Example

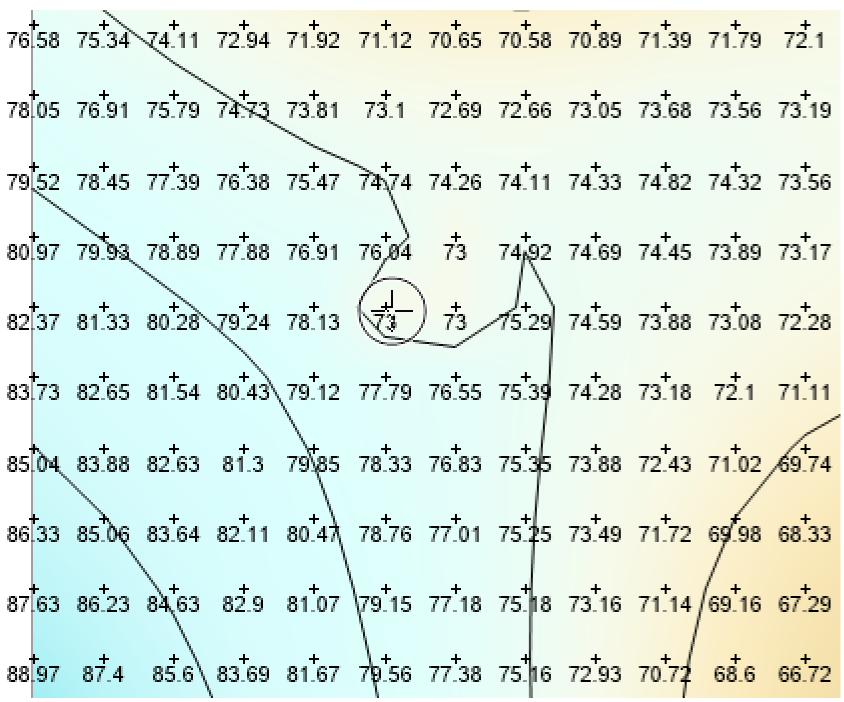
The following example will demonstrate editing the *Demogrid.grd* sample file with the **Brush** tool:

In this example, we will assume the gridding algorithm smoothed over a feature we wish to represent in the grid near node (29,6) in grid coordinates.



We will add a depression near the active grid node.

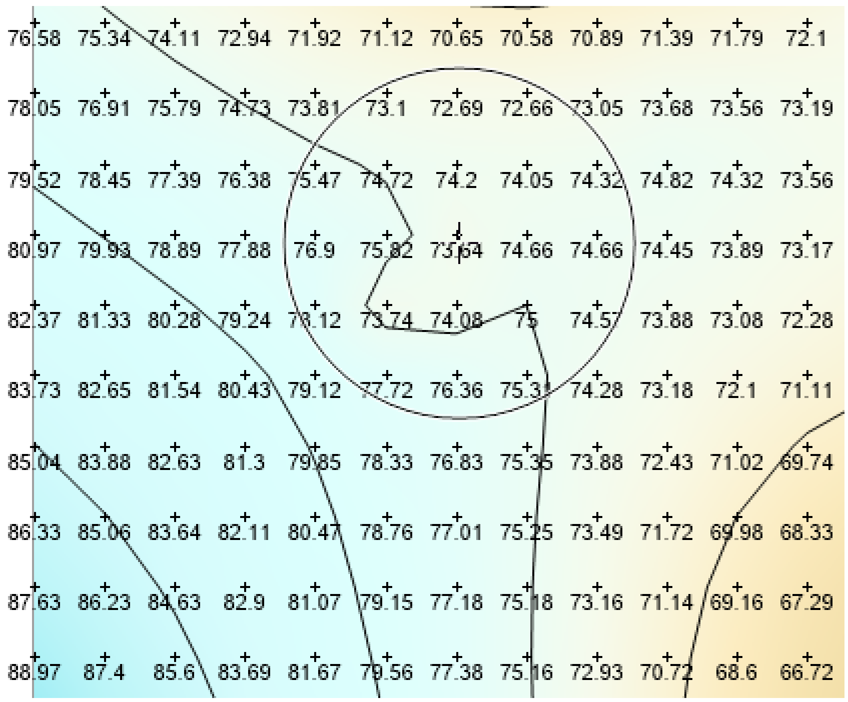
1. Click the **Grid Editor | Tools | Brush** command.
2. Set the *Brush size* to 1 by typing *1* in the *Brush size* field.
3. Set the Z value to 73 by typing *73* in the *Z value box*.
4. Click near the grid node at (29,6) and drag one row down and one column left.



*The **Brush** applied the Z value (73) to three grid nodes.*

We've now added the feature back to the grid. Now we wish to smooth the grid in the area we edited.

1. Click the Grid Editor | Tools | Smooth command.
2. Set the *Brush size* to 5 by typing 5 in the *Brush size* field.
3. Click near the grid node at (29,6). The grid is smoothed within the brush region.
4. Click twice more near the grid node at (29,6).



The edited area was then smoothed with the ***Smooth*** *tool.*

Now the depression feature has been added back to the grid and the surface has been smoothed to give a natural appearance.

Warp

The **Grid Editor | Tools | Warp** command in the grid editor enables the **Warp** tool mode. Use the **Warp** tool mode to drag grid values and "sculpt" the grid.

Click the **Grid Editor | Tools | Warp** command or the button to start warp mode. The cursor changes to a crosshair with a circle to visually indicate the *Brush size*. The **Warp** command button is highlighted in yellow to indicate warp mode as well.

Click and drag to pull grid values in the direction that you drag the cursor. The grid Nodes, Labels, Color Fill, and Contours update automatically as you use the **Warp** command.

Press ESC or click another **Grid Editor | Tools** command to end **Warp** mode.

NoData Nodes and Warp

When dragging from a region with data into a NoData region, the NoData values are replaced by values from neighboring nodes. The **Warp** tool can be used to remove NoData regions from the grid. However, NoData values are not dragged out from NoData regions by the warp tool. If you wish to assign the NoData value to nodes, use the Eraser.

Tool Options

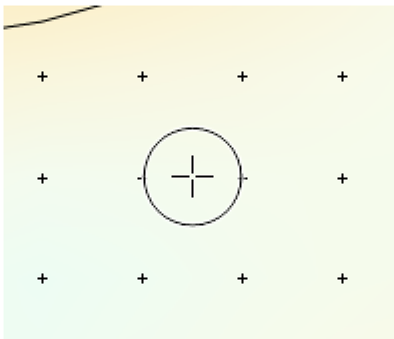
The *Brush size*, *Density*, and *Pressure* control how the **Warp** tool drags and changes the grid vales.

Brush Size

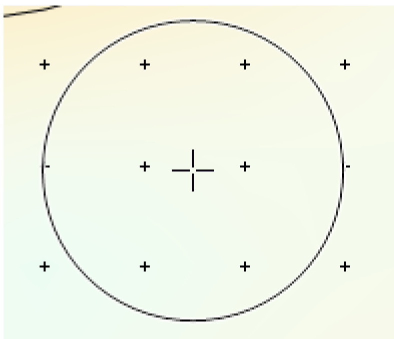
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- Click in the *Brush size* field and type the desired *Brush size*.
- Click the buttons to increment the *Brush size* by one. Click and hold the buttons to quickly scroll the *Brush size* value up or down.
- Press the [key to decrease the *Brush size*. Press the] key to increase the *Brush size*. Press and hold the [or] key to quickly decrease or increase the *Brush size*. The [and] keys increment the *Brush size* by 1 at *Brush size* values less than 10. The increment increases to 5 at a *Brush size* greater than 10 and less than 50. The increment increases to 10 at a *Brush size* greater than 50 and less than 100. The increment size continues to increase as the *Brush size* increases.

The cursor circle increases or decreases size with the *Brush size* setting and zoom level. The *Brush size* setting is saved and persistent across multiple grid editor windows.



A Brush size *of 1 makes the diameter of the brush equal to the horizontal spacing of one grid cell.*



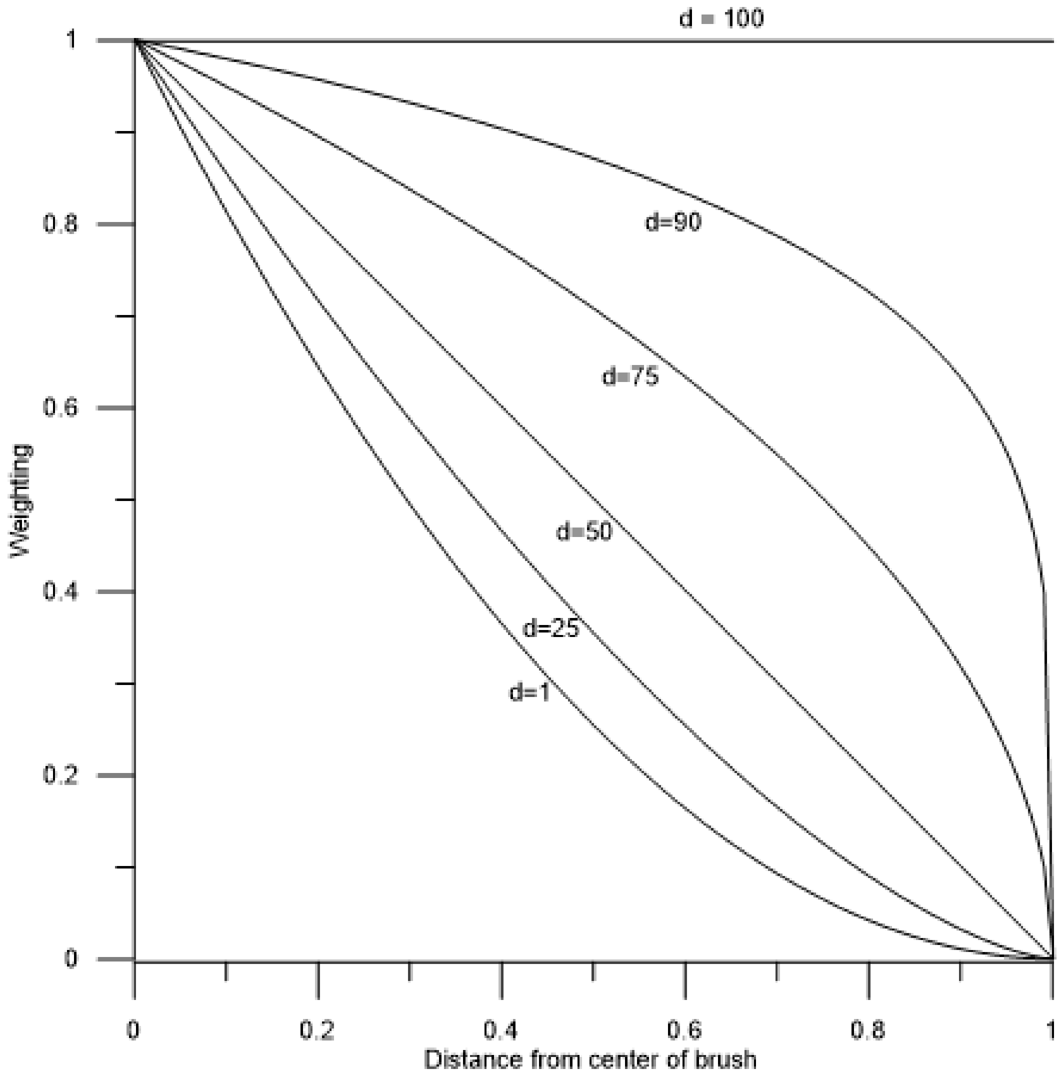
A Brush size *of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.*

Density

The *Density* tool option controls the weighting of the tool effect radially from the center of the brush. The cursor location (where the crosshair ┼ is located) receives the maximum effect of the selected tool mode. The *Density* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Density*:

- Click in the *Density* field and type the desired *Density*.
- Click the buttons to increment the *Density* by one. Click and hold the buttons to quickly scroll the *Density* value up or down.

The *Density* setting can be any value between 1 and 100. A *Density* setting of 1 has maximum "feathering" or blending between the center of the brush and values just outside the brush. A *Density* setting of 100 applies the tool mode effect to all nodes within the brush equally. The *Density* setting is saved and persistent across multiple grid editor windows. The default *Density* setting is 25.



These line plots show how Density *affects the weighting of the tool mode effect*
(y-axis) as distance from the center of the brush increases (x-axis)

Pressure

The *Pressure* tool option controls the speed at which tool effects are applied as the brush moves. The *Pressure* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Pressure*:

- Click in the *Pressure* field and type the desired *Pressure*.
- Click the buttons to increment the *Pressure* by one. Click and hold the buttons to quickly scroll the *Pressure* value up or down.

The *Pressure* setting can be any value between 1 and 100. A *Pressure* setting of 1 applies the tool effect the minimum amount with each click or drag. A *Pressure* setting of 100 applies the tool effect the maximum amount with each click or drag. The *Pressure* setting is saved and persistent across multiple grid editor windows. The default *Pressure* setting is 25.

Smooth

The **Grid Editor | Tools | Smooth** command in the grid editor smooths grid node values within the brush with a moving 3x3 average filter. Click the **Grid Editor | Tools | Smooth** command or the button to enable **Smooth** mode. The cursor changes to a crosshair with a circle to visually indicate the *Brush size*. The **Smooth** command button is highlighted in yellow to indicate smooth mode as well.

Click to smooth values within the brush circle. Click and drag to continuously smooth grid values as the cursor moves across the grid. The grid Nodes, Labels, Color Fill, and Contours update automatically as you use the **Smooth** command.

Press ESC or click another **Grid Editor | Tools** command to end **Smooth** mode.

Tool Options

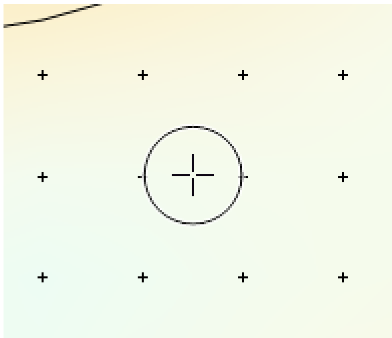
The *Brush size*, *Density*, and *Pressure* control how the **Smooth** tool smooths the grid vales.

Brush Size

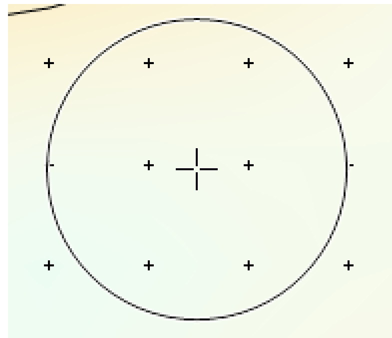
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- Click in the *Brush size* field and type the desired *Brush size*.
- Click the buttons to increment the *Brush size* by one. Click and hold the buttons to quickly scroll the *Brush size* value up or down.
- Press the [key to decrease the *Brush size*. Press the] key to increase the *Brush size*. Press and hold the [or] key to quickly decrease or increase the *Brush size*. The [and] keys increment the *Brush size* by 1 at *Brush size* values less than 10. The increment increases to 5 at a *Brush size* greater than 10 and less than 50. The increment increases to 10 at a *Brush size* greater than 50 and less than 100. The increment size continues to increase as the *Brush size* increases.

The cursor circle increases or decreases size with the *Brush size* setting and zoom level. The *Brush size* setting is saved and persistent across multiple grid editor windows.



A Brush size *of 1 makes the diameter of the brush equal to the horizontal spacing of one grid cell.*



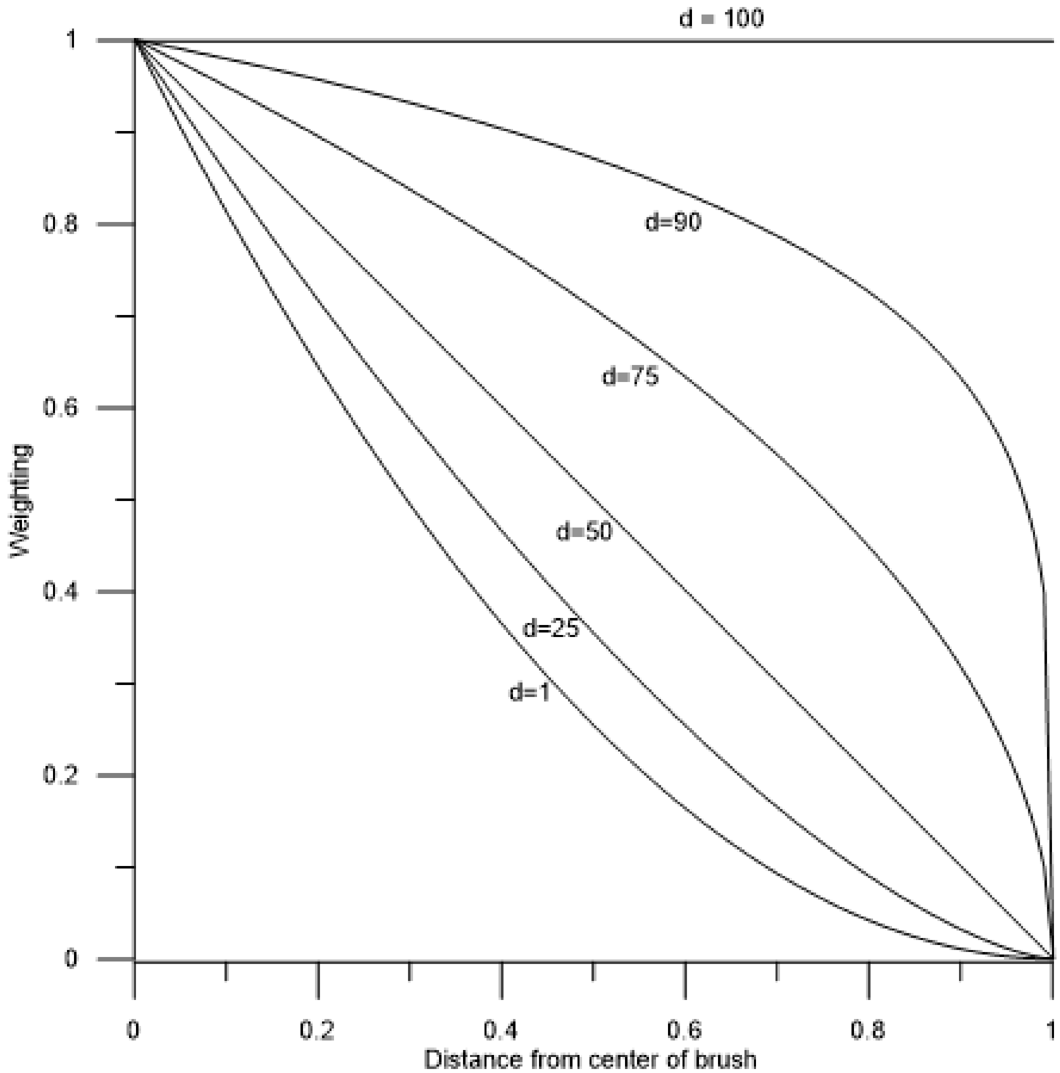
A Brush size *of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.*

Density

The *Density* tool option controls the weighting of the tool effect radially from the center of the brush. The cursor location (where the crosshair is located) receives the maximum effect of the selected tool mode. The *Density* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Density*:

- Click in the *Density* field and type the desired *Density*.
- Click the buttons to increment the *Density* by one. Click and hold the buttons to quickly scroll the *Density* value up or down.

The *Density* setting can be any value between 1 and 100. A *Density* setting of 1 has maximum "feathering" or blending between the center of the brush and values just outside the brush. A *Density* setting of 100 applies the tool mode effect to all nodes within the brush equally. The *Density* setting is saved and persistent across multiple grid editor windows. The default *Density* setting is 25.



These line plots show how Density *affects the weighting of the tool mode effect*
(y-axis) as distance from the center of the brush increases (x-axis)

Pressure

The *Pressure* tool option controls the speed at which tool effects are applied as the brush moves. The *Pressure* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Pressure*:

- Click in the *Pressure* field and type the desired *Pressure*.
- Click the buttons to increment the *Pressure* by one. Click and hold the buttons to quickly scroll the *Pressure* value up or down.

The *Pressure* setting can be any value between 1 and 100. A *Pressure* setting of 1 applies the tool effect the minimum amount with each click or drag. A *Pressure* setting of 100 applies the tool effect the maximum amount with each click or drag. The *Pressure* setting is saved and persistent across multiple grid editor windows. The default *Pressure* setting is 25.

Undo

Each click is registered as one operation in the Undo levels list. For example, if you set the *Brush size* to 1 and click once on five individual grid nodes, you must use the **Undo** command five times to undo all the changes. However, if you click and drag once across all five grid nodes, you must only use the **Undo** command once to undo all the changes.

Push Down

The **Grid Editor | Tools | Push Down** command in the grid editor decreases grid node values within the brush. Click the **Grid Editor | Tools | Push Down** command or the button to enable **Push Down** mode. The cursor changes to a crosshair with a circle to visually indicate the *Brush size*. The **Push Down** command button is highlighted in yellow to indicate **Push Down** mode as well.

Click once to decrease values within the brush circle. Click and drag to continuously decrease grid values as the cursor moves across the grid. The grid Nodes, Labels, Color Fill, and Contours update automatically as you use the **Push Down** command.

Press ESC or click another **Grid Editor | Tools** command to end **Push Down** mode.

Tool Options

The *Brush size* controls the area around the cursor that is affected by the **Push Down** tool. The *Density* controls how the **Push Down** tool decreases the grid vales in the brush area. The *Pressure* controls how much the grid values are affected.

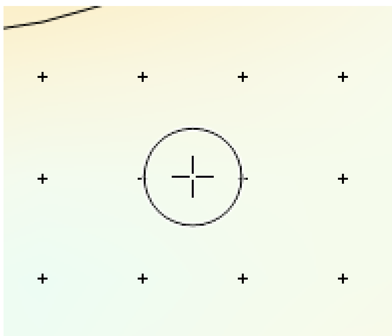
Brush Size

The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

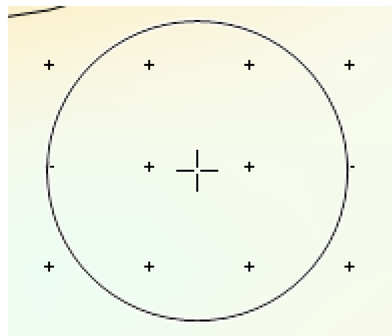
- Click in the *Brush size* field and type the desired *Brush size*.
- Click the buttons to increment the *Brush size* by one. Click and hold the buttons to quickly scroll the *Brush size* value up or down.

- Press the [key to decrease the *Brush size*. Press the] key to increase the *Brush size*. Press and hold the [or] key to quickly decrease or increase the *Brush size*. The [and] keys increment the *Brush size* by 1 at *Brush size* values less than 10. The increment increases to 5 at a *Brush size* greater than 10 and less than 50. The increment increases to 10 at a *Brush size* greater than 50 and less than 100. The increment size continues to increase as the *Brush size* increases.

The cursor circle increases or decreases size with the *Brush size* setting and zoom level. The *Brush size* setting is saved and persistent across multiple grid editor windows.



A Brush size *of 1 makes the diameter of the brush equal to the horizontal spacing of one grid cell.*



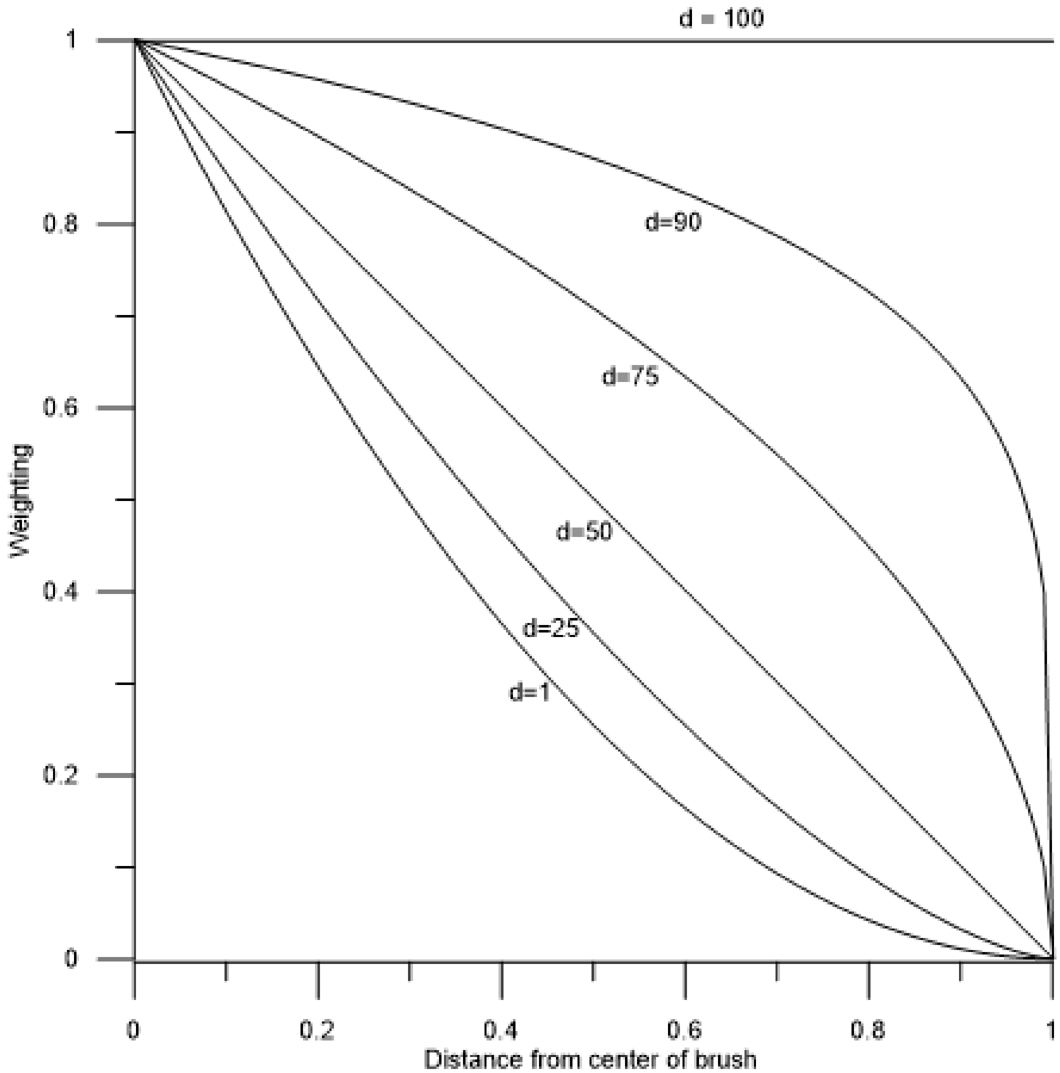
A Brush size *of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.*

Density

The *Density* tool option controls the weighting of the tool effect radially from the center of the brush. The cursor location (where the crosshair is located) receives the maximum effect of the selected tool mode. The *Density* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Density*:

- Click in the *Density* field and type the desired *Density*.
- Click the buttons to increment the *Density* by one. Click and hold the buttons to quickly scroll the *Density* value up or down.

The *Density* setting can be any value between 1 and 100. A *Density* setting of 1 has maximum "feathering" or blending between the center of the brush and values just outside the brush. A *Density* setting of 100 applies the tool mode effect to all nodes within the brush equally. The *Density* setting is saved and persistent across multiple grid editor windows. The default *Density* setting is 25.



These line plots show how Density *affects the weighting of the tool mode effect*
(y-axis) as distance from the center of the brush increases (x-axis)

Pressure

The *Pressure* tool option controls the speed at which tool effects are applied as the brush moves. The *Pressure* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Pressure*:

- Click in the *Pressure* field and type the desired *Pressure*.
- Click the buttons to increment the *Pressure* by one. Click and hold the buttons to quickly scroll the *Pressure* value up or down.

The *Pressure* setting can be any value between 1 and 100. A *Pressure* setting of 1 applies the tool effect the minimum amount with each click or drag. A *Pressure* setting of 100 applies the tool effect the maximum amount with each click or drag. The *Pressure* setting is saved and persistent across multiple grid editor windows. The default *Pressure* setting is 25.

The **Push Down** tool decreases the Z value by 1% of the Z-range of the grid at the cursor location when *Pressure* is 100. When the grid is planar, the **Push Down** tool decreases the Z value by 1% of the diagonal distance of the grid at the cursor location when *Pressure* is 100. The *Pressure* changes this amount proportionally. For example when the *Pressure* is 50, the amount the grid changes under the cursor location is 0.5% of the grid Z range (0.01 x 0.5 = 0.005)

Undo

Each click is registered as one operation in the Undo levels list. For example, if you set the *Brush size* to 1 and click once on five individual grid nodes, you must use the **Undo** command five times to undo all the changes. However, if you click and drag once across all five grid nodes, you must only use the **Undo** command once to undo all the changes.

Pull Up

The **Grid Editor | Tools | Pull Up** command in the grid editor increases grid node values within the brush area. Click the **Grid Editor | Tools | Pull Up** command or the button to enable **Pull Up** mode. The cursor changes to a crosshair with a circle to visually indicate the *Brush size*. The **Pull Up** command button is highlighted in yellow to indicate **Pull Up** mode as well.

Click once to increase values within the brush circle. Click and drag to continuously increase grid values as the cursor moves across the grid. The grid Nodes, Labels, Color Fill, and Contours update automatically as you use the **Pull Up** command.

Press ESC or click another **Grid Editor | Tools** command to end **Pull Up** mode.

Tool Options

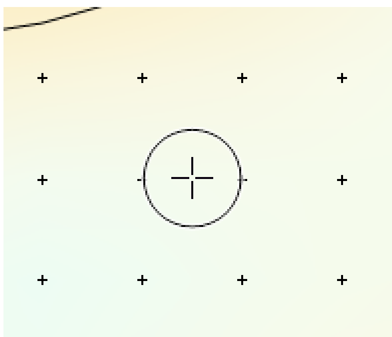
The *Brush size* controls the area around the cursor that is affected by the **Pull Up** tool. The *Density* controls how the **Pull Up** tool increases the grid vales in the brush area. The *Pressure* controls how much the grid values are affected.

Brush Size

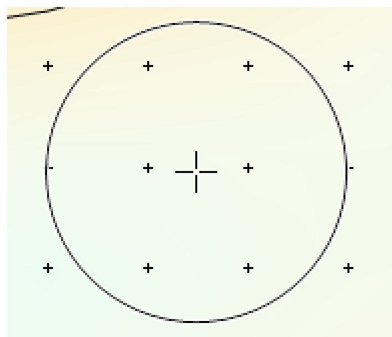
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- Click in the *Brush size* field and type the desired *Brush size*.
- Click the buttons to increment the *Brush size* by one. Click and hold the buttons to quickly scroll the *Brush size* value up or down.
- Press the [key to decrease the *Brush size*. Press the] key to increase the *Brush size*. Press and hold the [or] key to quickly decrease or increase the *Brush size*. The [and] keys increment the *Brush size* by 1 at *Brush size* values less than 10. The increment increases to 5 at a *Brush size* greater than 10 and less than 50. The increment increases to 10 at a *Brush size* greater than 50 and less than 100. The increment size continues to increase as the *Brush size* increases.

The cursor circle increases or decreases size with the *Brush size* setting and zoom level. The *Brush size* setting is saved and persistent across multiple grid editor windows.



A Brush size *of 1 makes the diameter of the brush equal to the horizontal spacing of one grid cell.*



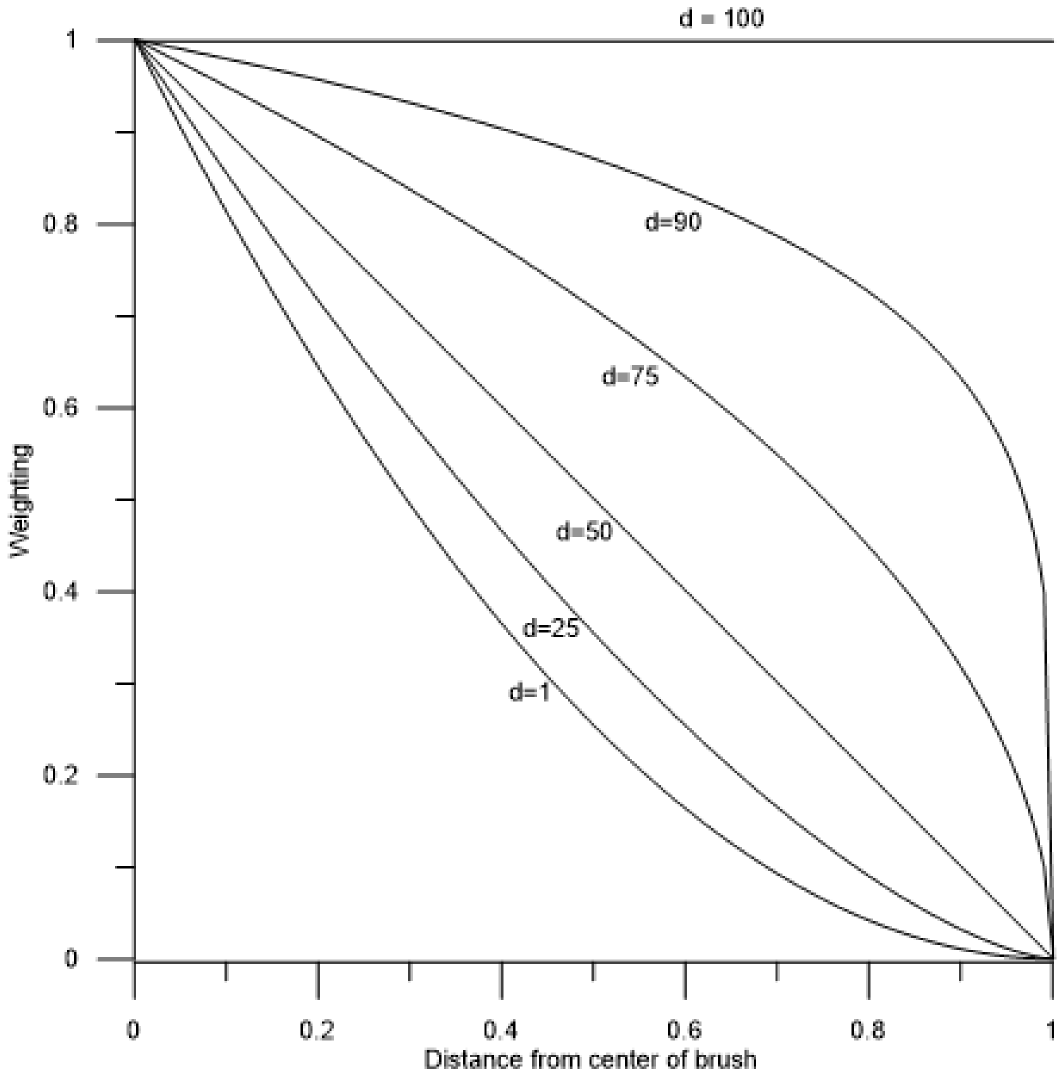
A Brush size *of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.*

Density

The *Density* tool option controls the weighting of the tool effect radially from the center of the brush. The cursor location (where the crosshair is located) receives the maximum effect of the selected tool mode. The *Density* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Density*:

- Click in the *Density* field and type the desired *Density*.
- Click the buttons to increment the *Density* by one. Click and hold the buttons to quickly scroll the *Density* value up or down.

The *Density* setting can be any value between 1 and 100. A *Density* setting of 1 has maximum "feathering" or blending between the center of the brush and values just outside the brush. A *Density* setting of 100 applies the tool mode effect to all nodes within the brush equally. The *Density* setting is saved and persistent across multiple grid editor windows. The default *Density* setting is 25.



These line plots show how Density *affects the weighting of the tool mode effect*
(y-axis) as distance from the center of the brush increases (x-axis)

Pressure

The *Pressure* tool option controls the speed at which tool effects are applied as the brush moves. The *Pressure* setting is located in the tools options bar at the top of the grid editor window. There are two methods for changing the *Pressure*:

- Click in the *Pressure* field and type the desired *Pressure*.
- Click the buttons to increment the *Pressure* by one. Click and hold the buttons to quickly scroll the *Pressure* value up or down.

The *Pressure* setting can be any value between 1 and 100. A *Pressure* setting of 1 applies the tool effect the minimum amount with each click or drag. A *Pressure* setting of 100 applies the tool effect the maximum amount with each click or drag. The *Pressure* setting is saved and persistent across multiple grid editor windows. The default *Pressure* setting is 25.

The **Pull Up** tool increases the Z value by 1% of the Z-range of the grid at the cursor location when *Pressure* is 100. When the grid is planar, the **Pull Up** tool increases the Z value by 1% of the diagonal distance of the grid at the cursor location when *Pressure* is 100. The *Pressure* changes this amount proportionally. For example when the *Pressure* is 50, the amount the grid changes under the cursor location is 0.5% of the grid Z range (0.01 x 0.5 = 0.005)

Undo

Each click is registered as one operation in the Undo levels list. For example, if you set the *Brush size* to 1 and click once on five individual grid nodes, you must use the **Undo** command five times to undo all the changes. However, if you click and drag once across all five grid nodes, you must only use the **Undo** command once to undo all the changes.

Eraser

The **Grid Editor | Tools | Eraser** command in the grid editor assigns the NoData value (1.70141e+038) to the grid nodes within the brush. Click the **Grid Editor | Tools | Eraser** command or the button to start eraser mode. The cursor changes to a crosshair with a circle to visually indicate the eraser *Brush size*. The **Eraser** command button is highlighted in yellow to indicate eraser mode as well.

Click to erase values within the circle. Click and drag to continuously erase grid values as the cursor moves across the grid. The grid Nodes, Labels, Color Fill, and Contours update automatically as you use the **Erase** command.

Press ESC or click another **Grid Editor | Tools** command to end **Eraser** mode.

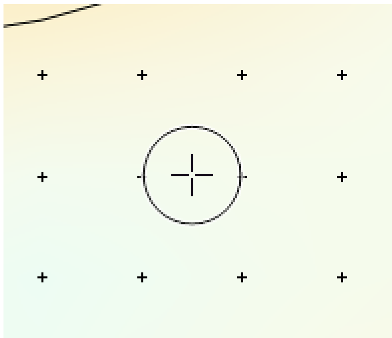
Brush Size

The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

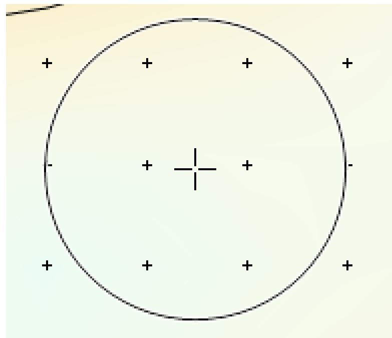
- Click in the *Brush size* field and type the desired *Brush size*.
- Click the buttons to increment the *Brush size* by one. Click and hold the buttons to quickly scroll the *Brush size* value up or down.
- Press the [key to decrease the *Brush size*. Press the] key to increase the *Brush size*. Press and hold the [or] key to quickly decrease or increase the

Brush size. The [and] keys increment the *Brush size* by 1 at *Brush size* values less than 10. The increment increases to 5 at a *Brush size* greater than 10 and less than 50. The increment increases to 10 at a *Brush size* greater than 50 and less than 100. The increment size continues to increase as the *Brush size* increases.

The cursor circle increases or decreases size with the *Brush size* setting and zoom level. The *Brush size* setting is saved and persistent across multiple grid editor windows.



A Brush size *of 1 makes the diameter of the brush equal to the horizontal spacing of one grid cell.*



A Brush size *of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.*

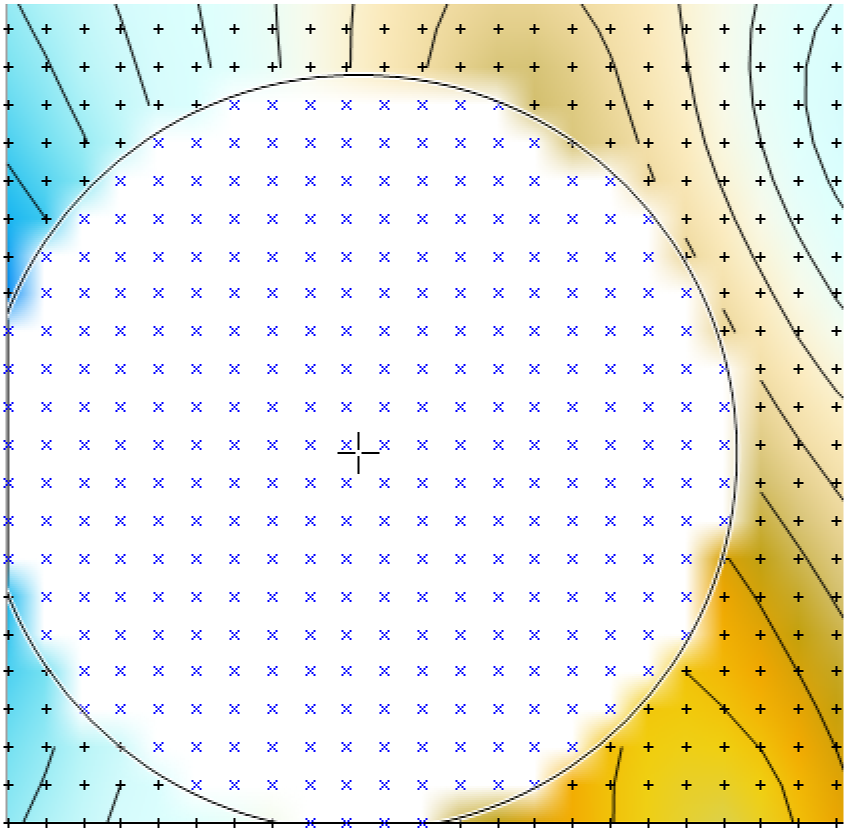
Undo

Each click is registered as one operation in the Undo levels list. For example, if you set the *Brush size* to 1 and click once on five individual grid nodes, you must use the **Undo** command five times to undo all the changes. However, if you click and drag once across all five grid nodes, you must only use the **Undo** command once to undo all the changes.

Erasing Grid Values Example

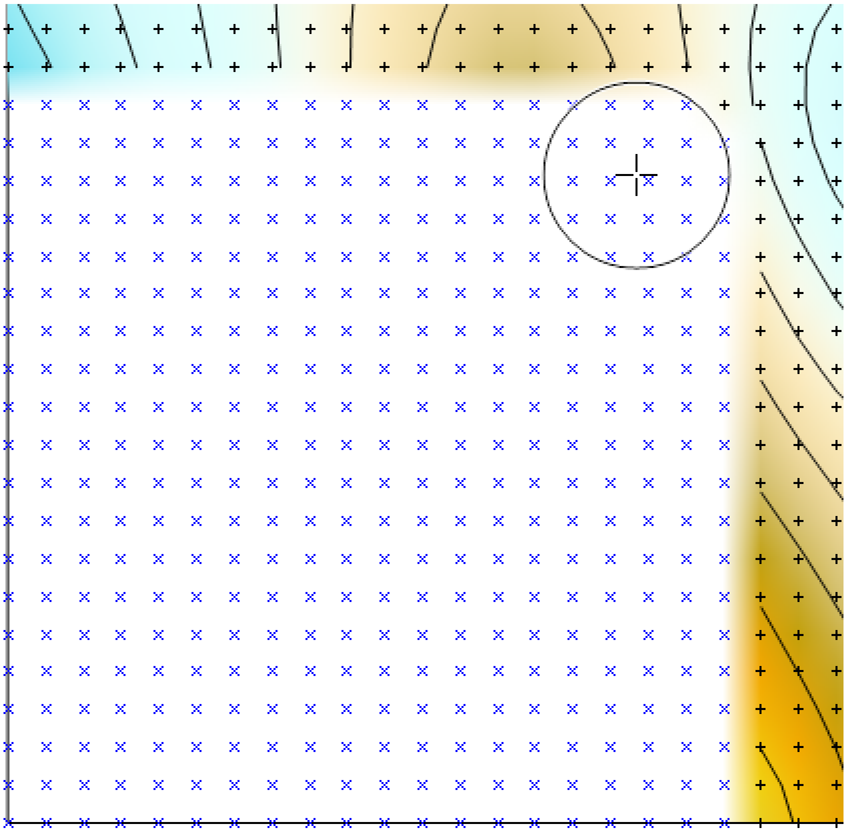
The following example will demonstrate using the **Erase** command to assign the NoData value to a 20x20 node section of the *Demogrid.grd* sample file grid with the **Erase** command,

1. Click the **Grid Editor | Tools | Eraser** command.
2. Set the *Brush size* to 20 by typing *20* in the *Brush size* field.
3. Click once in the grid near node (9,9). This is approximately (1.6,1.6) in map coordinates. The cursor map coordinates are displayed in the status bar.



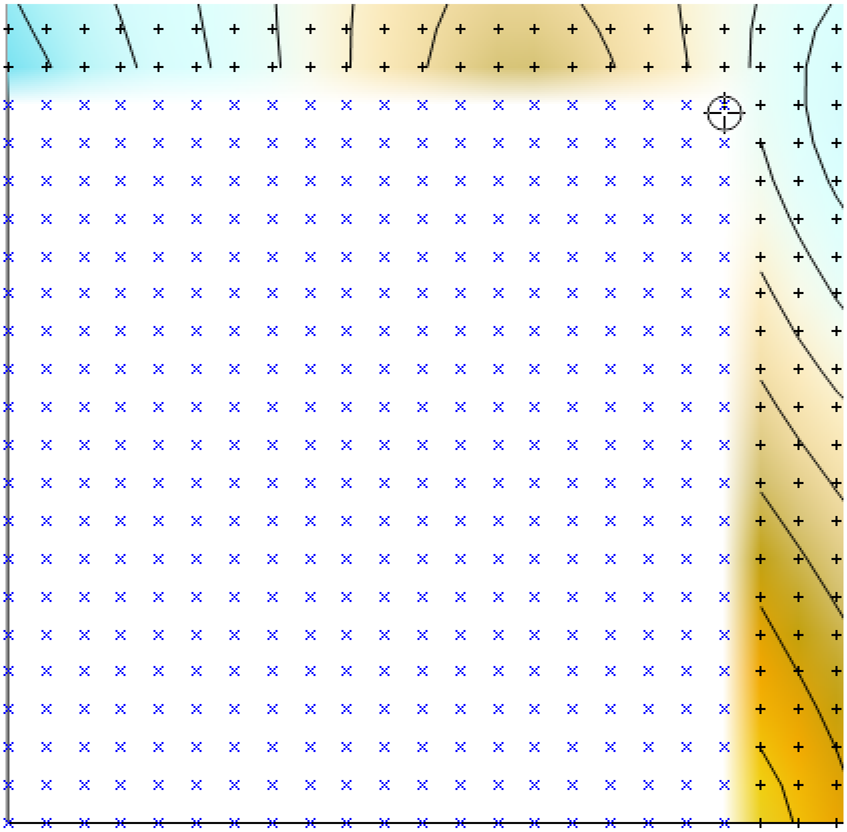
Many nodes can be erased in a single click when the Brush size *is large.*

4. Press and hold the [key to reduce the *Brush size* to 5.
5. Click and drag around the NoData region to create a rectangle.



*Click and drag with **Erase** to create any shape NoData region.*

6. A *Brush size* of 5 is too large to easily erase the final node. Change the *Brush size* to 1.
7. Click the node at (19,19) in grid coordinates to create the 20x20 node NoData rectangle.



Erasing single nodes is easier when the brush size is small.

Eyedropper

Click the **Grid Editor | Tools | Eyedropper** command or the button to pull grid Z values by clicking in the grid. The cursor changes to an eyedropper when the **Eyedropper** command is clicked. The **Grid Editor | Tools | Eyedropper** command button is highlighted in yellow to indicate eyedropper tool mode.

Click on the grid with the eyedropper to place the nearest grid node's Z value in the *Z value box* .

Press ESC or click another **Grid Editor | Tools** command to end **Eyedropper** mode.

Z Value Box

The *Z value box* in the tool options bar at the top of the grid editor window displays the Z value from the nearest grid node when you click the grid.

Eyedropper and Select Tool

After pressing ESC or clicking the Grid Editor | Tools | Select command, the grid node nearest your most recent click is made the active grid node.

Eyedropper and Brush Tool

You can use the **Eyedropper** tool to set the *Z value box* value for the Brush tool with two different methods:

- Click the *Grid Editor* | *Tools* | *Eyedropper* command, and click on the desired grid node to place its value in the *Z value box*. Next, click the Grid Editor | Tools | Brush command.
- After clicking **Grid Editor | Tools | Brush** to enable brush tool mode, press and hold the "i" key to temporarily enable **Eyedropper** mode. While holding the "i" key, click on the desired grid node to place its value in the *Z value box* . Release the "i" key to return to **Brush** tool mode.

Zoom In - Grid Editor

Click the **Grid Editor | View | Zoom In** command or the  button to immediately increase the magnification of the grid editor.

Zoom with A Wheel Mouse

You can use a wheel mouse to zoom realtime and pan in the plot window or grid editor. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out. Hold down the wheel button straight down, and the cursor will turn to a closed hand. When the cursor is a  , drag the mouse to pan the plot window. Be sure to click straight down with the scroll wheel. The zoom is changed so that the cursor location remains on the screen.

Zoom with the Keyboard

You can use keyboard commands to zoom in and out of the plot window. The default commands are CTRL+EQUAL SIGN (=) to zoom in, and CTRL+HYPHEN (-) to zoom out.

Zoom Out - Grid Editor

Click the **Grid Editor | View | Zoom Out** command or the  button to immediately decrease the magnification of the grid editor window.

Zoom with A Wheel Mouse

You can use a wheel mouse to zoom realtime and pan in the plot window or grid node editor. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out. Hold down the wheel button straight down, and the cursor will turn to a closed hand. When the cursor is a , drag the mouse to pan the plot window. Be sure to click straight down with the scroll wheel. The zoom is changed so that the cursor location remains on the screen.

Zoom with the Keyboard

You can use keyboard commands to zoom in and out of the plot window. The default commands are CTRL+EQUAL SIGN (=) to zoom in, and CTRL+HYPHEN (-) to zoom out.

Grid Info

The **Grid Editor | Options | Grid Info** command or the button in the Grid Editor displays the file name, grid size in rows x columns, and grid X, Y, and Z minimums and maximums. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report. This is the same basic information as displayed in the plot window with the **Grids | Info | Grid Info** command.

The Open Grid dialog also displays the *Grid Info* of the selected grid. The **Open Grid** dialog displays the following *Grid Info*: *Name, Format, Size, xMax, yMax, zMax, xMin, yMin,* and *zMin.*

Update Layer

Click the **Grid Editor | Options | Update Layer** command or the button to update the map layer from which the grid was opened. The grid must by opened by selecting the map layer and clicking the **Map Tools | Edit Layer | Grid** command or right-clicking the map layer and clicking **Edit Grid** in the context menu to use the **Update Layer** command. The **Update Layer** command is disabled when viewing a grid opened with the File | Open or Grids | Editor | Grid Editor command.

To edit a map layer,

1. Select the map layer you wish to edit in the plot document **Contents** window.
2. Click **Map Tools | Edit Layer | Grid** in the plot window.
3. Use the **Grid Editor | Tools** commands to make the desired adjustments to the grid.

4. When you are done editing the grid, click the **Grid Editor | Options | Update Layer** command to update the map layer in the plot document with your grid.
5. If you wish to save your edits to a file, click File | Save As to create a new grid file. Click File | Save to overwrite the existing grid file. It is necessary to save your edits to a file with **Save** or **Save As** if you wish to update all layers in your map to use the edited grid.
6. To close the grid editor window, click the **File | Close** command or click the X in the grid editor document tab. To view an existing window and keep the grid editor window open, click on another document tab.

After clicking the **Update Layer** command, view the map layer by clicking the document tab for the plot window that contains the original map layer. Notice that only the map layer that was selected in step 1 was changed. The **Update Layer** command does not overwrite the grid file used to create the map layer. The **Update Layer** command does not save a new grid file. The **Update Layer** command only changes the grid for the original map layer.

Undo Update Layer

The **Update Layer** command changes the map layer in the plot document. Therefore to undo the **Update Layer** command, click the document tab for the plot window that contains the original map layer. Next click the Home | Undo | Undo command or press CTRL+Z. The map layer will return to the unedited state. The grid in the grid editor can be reapplied to the map layer by clicking Home | Undo | Redo in the plot document or **Grid Editor | Options | Update Layer** in the grid editor.

If you did not overwrite the original grid file, you can also revert back to the original grid by clicking File | Reload Map Data.

You can discard the edited grid by closing the grid editor. Click *No* in the **Surfer** dialog when asked to save the grid.

Saving the Edited Grid

When you have finished making changes to the grid you can overwrite the original grid file or save a new grid file. You must save the grid file to use the edited grid in other map layers. To overwrite the original file, click the File | Save command in the grid editor. To save the edited grid to a new file, click the File | Save As command in the grid editor.

If you closed the grid editor without saving changes, but you decide later you wish to save the grid, click the layer that was updated with the **Update Layer** command. Next, click **Map Tools | Edit Layer | Grid** to open the grid in the grid editor. The edited grid will be opened in the grid editor and you can overwrite the original grid file or create a new grid file.

Updating All Map Layers

Only the originally selected map layer is updated when you click the **Grid Editor | Options | Update Layer** command, even when multiple map layers use the same grid file in the plot document. You must save the grid to a file to use the grid in new or existing map layers.

To update the other map layers after overwriting the original grid with the **File | Save** command,

1. Return to the plot document by clicking the document tab for the plot window.
2. Click the File | Reload Map Data command.

All map layers that were created from the same grid as the one that was edited are automatically updated to the new grid.

To update the other map layers after creating a new grid file,

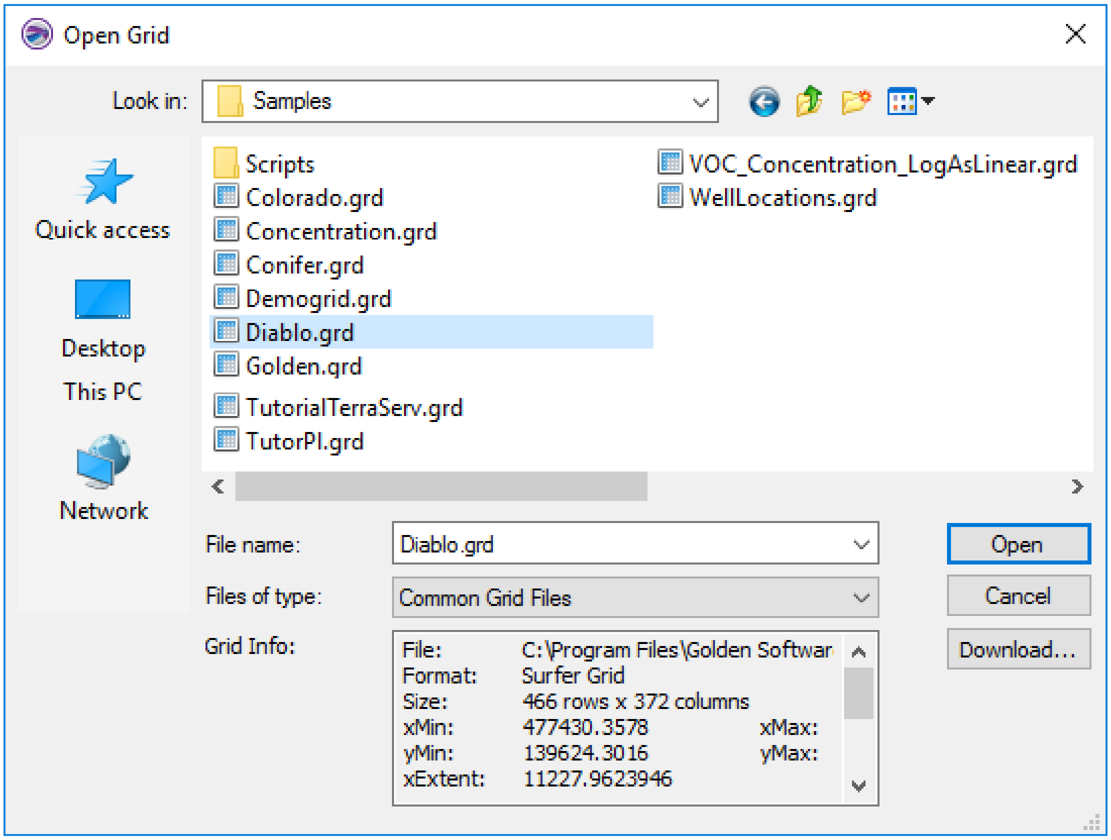
1. Return to the plot document by clicking the document tab for the plot window.
2. Click one of the map layers that is still using the original grid file in the Contents window.
3. Click the **General** tab in the Properties window.
4. Click the button in the *Grid file* property field.
5. Locate and click the new grid file in the Open Grid dialog.
6. Click *Open* in the **Open Grid** dialog.
7. Repeat steps 2 through 6 for each map layer you wish to update.

All the maps layers that were created from the same grid as the one that was edited are now updated to the new grid.

Open Grid

Contour maps, color relief maps, vector maps, watershed maps, 3D wireframes, 3D surfaces, grid values maps, and viewsheds all require grids for their generation in **Surfer**. A grid is a regular, rectangular array of values. The **Home | Grid Data | Grid Data** command provides you with several methods for generating a grid file from your XYZ data. In addition to the grid files that **Surfer** creates, it can also import many common grid files directly. The **Grid** menu contains many commands for grid operations. A grid with all NoData nodes cannot be saved.

The **Open Grid** dialog is used to open one or more grid files when creating new grid-based maps or layers or when using a grid command (i.e. **Grids | Edit | Convert**). When multiple files are selected, one map is created for each input file.



Specify the file to open in the ***Open Grid*** *dialog.*

Look In

The *Look in* field shows the current directory. Click the down arrow to see the directory structure and click on the folders to change directories.

Creating New Folders and Changing the View

The buttons to the right of the *Look in* field allow you to create new folders and change the view of the file list.

File List

The file list displays files in the current directory. The current directory is listed in the *Look in* field. The *Files of type* field controls the display of the file list. For example, if *Golden Software Data (*.DAT)* is listed in the *Files of type* field only *.DAT files appear in the files list.

Double-click on a file to open it or single-click the file and then click the *Open* button. To open more than one file, select the files you wish to use by holding CTRL while clicking, and then click *Open*. When adding grid-based maps, each file is added as a separate map.

Specify a File Name

The *File name* field shows the name of the selected file, or type a path and file name into the box to open a file.

Specify a File Type

The *Files of type* field shows the file format to be opened. To change the file format, click the down arrow and select the file type from the list. *All Files (*.*)* display all files in a directory.

The *Common Document Files (*...)* format type is selected by default. This displays all the common file formats in the navigation pane. If a different format type is selected, **Surfer** will remember the setting until the end of the current session. When **Surfer** is restarted, the default format type will be used.

To see all files in the directory, choose *All Files (*.*)* from the *Files of type* list. Double-click on a file to open it or single click the file and then click the *Open* button. The *All Files (*.*)* option shows all of the file formats in the current directory, even if the file type is not appropriate for the action chosen. For example, a .BLN file may be displayed, even though a .BLN file cannot be used to create a grid base map.

Open Files

Surfer tracks the files being used by maintaining a list of opened files for new maps and the files used in saved maps. The file(s) used in existing maps are shown in a compiled list in the *Open Grids* field of the **Open Grid(s)** dialog.

Grid Info

The selected grid file information is listed in the *Grid Info* section. The specific location, name, format, size, xMin, xMax, yMin, yMax, zMin, and zMax values are displayed. Use the scroll bar to the right if necessary to see all of the information. Use the grip at the lower right corner of the dialog to expand the size of the dialog if the grid information is cut off on the right.

Download Online Grids

Click the *Download* button to open the Download Online Grids dialog. Use the **Download Online Grids** dialog to retrieve a grid file from a WMS server.

File Types

ADF Arc/Info Binary Grid
AM Amira Mesh
AN? ACR-NEMA Medical Image
ASC Arc/Info ASCII Grid
ASI Amira Stacked Images

BIL/BIP/BSQ Banded Grid
BMP Windows Bitmap
CPS-3 Grid Format
DICOM3 Medical Image
DDF SDTS DEM
DEM USGS DEM
DTD DTED
E00 ASCII Grid Format
ECW ERMapper
ERS ER Mapper Grid Format
FLD AVS Field
FLT Esri Float Grid Format
GIF Image
GLOBE DEM
GRD Surfer Grid
GRD Geosoft Binary Grid
GXF Grid eXchange Format
HDF Hierarchical Data Format
HDR GTOPO-30
IMG Analyze 7.5 Medical Image
IMG Idrisi Raster Format
INFO Leica Confocal Raw Slices
JPG Compressed Bitmap
LAT Iris Explorer
PNG Portable Network Graphics
PNM/PPM/PGM/PBM Image
RAW Raw Binary Grid
RGB SGI-RGB Image
SID LizardTech MrSID Image
STK Metamorph
SUN Sun Raster Image
TAR SDTS DEM
TAR.GZ SDTS DEM
TGA Targa (TrueVision)
TGZ SDTS DEM
TIF Tagged Image
VTK Visualization Toolkit
X AVS X-Image
ZIP SDTS DEM
ZMap Plus Grid Format

Open Grids

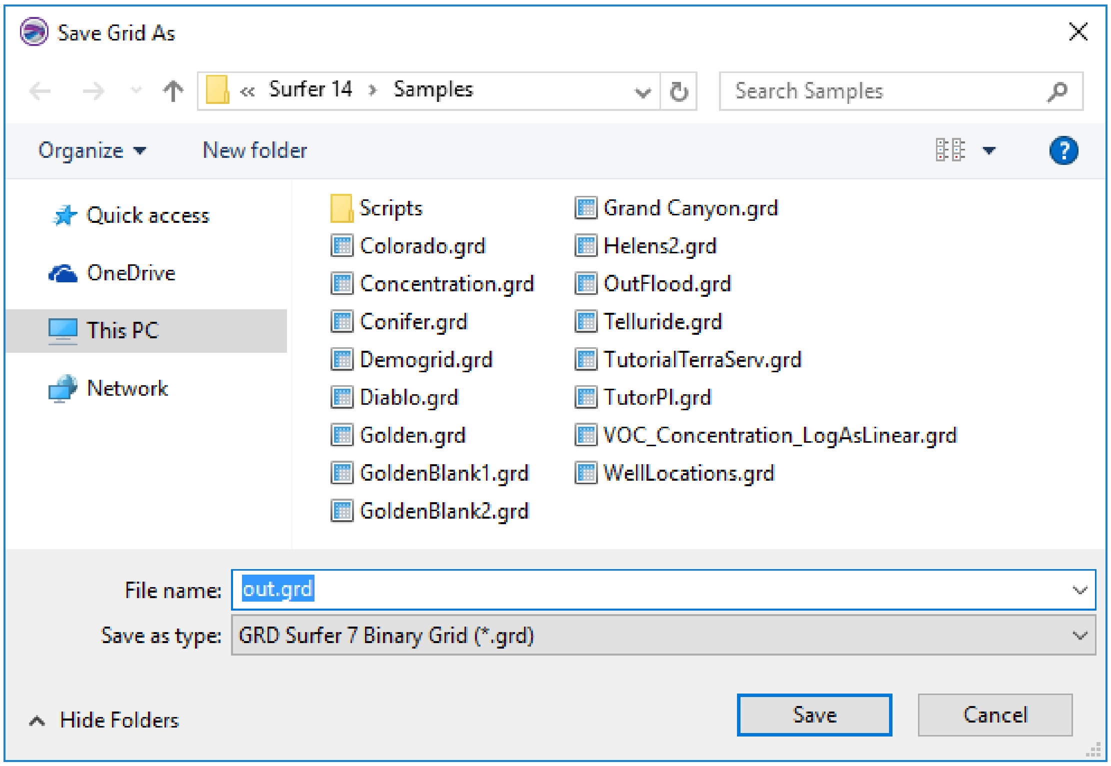
Surfer tracks the files being used by maintaining a list of opened grids for new maps and the grids used in saved maps. The grids(s) used to build maps are shown in a compiled list in the *Open Grids* field of the Open Grid dialog. The only exception is that vector data between base layers will not be shown.

Save Grid As

The **Save Grid As** dialog allows you to specify a location and grid type to save a grid file.

Save Grid As Dialog

Use the **File | Save As** command in the grid node editor or one of the **Grid** menu commands in the plot document to open the **Save Grid As** dialog.



*Specify the save location, file name, and file type in the **Save Grid As** dialog. This graphic may look different in another operating system.*

Save In

The *Save in* field shows the current directory. Click the down arrow to see the directory structure and click on the folders to change directories.

Creating New Folders and Changing the View

The buttons to the right of the *Look in* field allow you to create new folders and change the view of the file list.

File List

The file list displays files in the current directory. The current directory is listed in the *Save in* field. The *Save as type* field controls the display of the file list.

Specify a File Name

The *File name* field shows the name of the selected file, or type a path and file name into the box to open a file.

Save As Type

The *Save as type* field controls the display of the file list.

The *GRD Surfer 6 Binary Grid (*.grd)* format type is selected by default. This displays all the common file formats in the navigation pane. If a different format type is selected, **Surfer** will remember the setting until the end of the current session. When **Surfer** is restarted, the default format type will be used.

To see all files in the directory, choose *All Files (*.*)* from the *Save as type* list. Double-click on a file to open it or single click the file and then click the *Open* button. The *All Files* shows all of the file formats even if the file type is not appropriate for the action chosen (i.e. displaying a data file when creating a grid based map that requires a grid file).

In the **Save As Grid** dialog, select a grid file format from the *Save as types* list. The following file formats are supported:

- ADF Arc/Info Binary Grid (*.adf)
- AM Amira Mesh (*.am, *.col)
- ASC Arc/Info ASCII Grid
- BIL/BIP/BSQ Banded Grid (*.bil, *bip, *bsq)
- CPS-3 Grid Format (*.cps, *.cps3, *.asc, *.dat, *.grd)
- DAT XYZ (*.dat)
- DEM USGS DEM (*.dem)
- ERS ER Mapper Grid Format (*.ers)
- FLD AVS Field (*.fld)
- FLT Esri Float Grid Format (*.flt)
- GRD Surfer 6 Text Grid (*.grd)
- GRD Surfer 6 Binary Grid (*.grd)
- GRD Surfer 7 Binary Grid (*.grd)
- GRD Geosoft Binary Grid (*.grd, *.ggf)
- GXF Grid eXchange Format (*.gxf)
- HDF Hierarchical Data Format (*.hdf)
- IMG Analyze 7.5 Medical Image (*.img)
- LAT Iris Explorer (*.lat)
- netCDF Network Common Data Format (*.nc)
- RAW Binary Grid (*.raw, *.bin)
- STL 3D Mesh (*.stl)

- TIFF Image as Grid (*.tif, *.tiff, *.stk)
- VTK Visualization Toolkit (*.vtk)
- Z-Map Plus Grid Format (*.asc, *.dat, *.grd, *.xyz, *.zmap, *.zyc, *.zycor)

Some **Save As Grid** dialog *Save as type* lists may differ slightly, depending on the command that invoked the dialog.

Chapter 7 - Grid Operations

Grids Tab Commands

There are many ways to manipulate grid files in **Surfer**. The **Grids** ribbon contains commands used to assign the NoData value, convert, create, extract, filter, mosaic, slice, smooth, and transform grid files. In addition, volume calculations, variogram generation, calculus operations, cross section creation, and residual calculations can be performed using the commands on the **Grids** ribbon. Click on the **Grids** tab to access the commands on the ribbon.

Grid Data	Create a grid from irregularly spaced XYZ data
Grid from Server	Create a grid from downloaded DEM data
Grid from Contours	Create a grid from contour lines
Function	Create a grid from a user-specified function
Variogram	Create a new variogram and export a variogram
Assign NoData	Assign NoData to grid nodes inside or outside a boundary
Filter	Filter an existing grid by applying a moving matrix filter
Convert	Convert between various grid formats
Spline Smooth	Smooth an existing grid using cubic splines
Assign Coordinate System	Assign the coordinate system of a grid file and save the information to an external file
Project	Create a grid in a different coordinate system
Calculus	Perform calculus operations on an existing grid file
Volume	Compute the volume under or over a grid surface
Math	Provide grid-to-grid and grid-to-constant math operations
Transform	Scale, offset, mirror, or rotate an existing grid file
Slice	Compute a cross section of data through a grid surface
Residuals	Compute the difference between XYZ data and a grid surface
Point Sample	Compute the Z value at specified XY locations on the grid surface
Contour Volume and Area	Compute the volume and area above, below, and between contour lines
Isopach	Create an isopach or isochore map
Mosaic	Combine a series of compatible input grids into a single output grid
Extract	Extract a subset of a grid from an existing grid file

Grid Editor	Change the values of nodes in a grid or edit the grid with various tools
Grid Info	Display information about the grid

Assign NoData

The **Grids | Edit | Assign NoData** command is used to remove grid node data from a grid in areas not supported by original data or in areas where you do not want to display contours on a map. The **Assign NoData** command assigns the NoData value (1.70141e+038) to specified groups of grid nodes in a grid file. Grids used with **Assign NoData** contain the same number of grid nodes and the same grid limits as the original grid file. The grid nodes in the new grid are identical to the values in the original grid except in those locations where the NoData values are assigned.

NoData polygon boundaries are defined in a vector file or in a base layer. The blanking file format can be used and is a special ASCII format file containing the X, Y coordinates defining the NoData polygon boundary. Blanking *.BLN files can be made in the **Surfer** worksheet. Blanking *.BLN files can also be created with the Digitize command. The NoData values can be assigned to areas inside or outside the polygon or 3D polygon boundary. Regions can be assigned the NoData value using the Eraser command in the Grid Editor window.

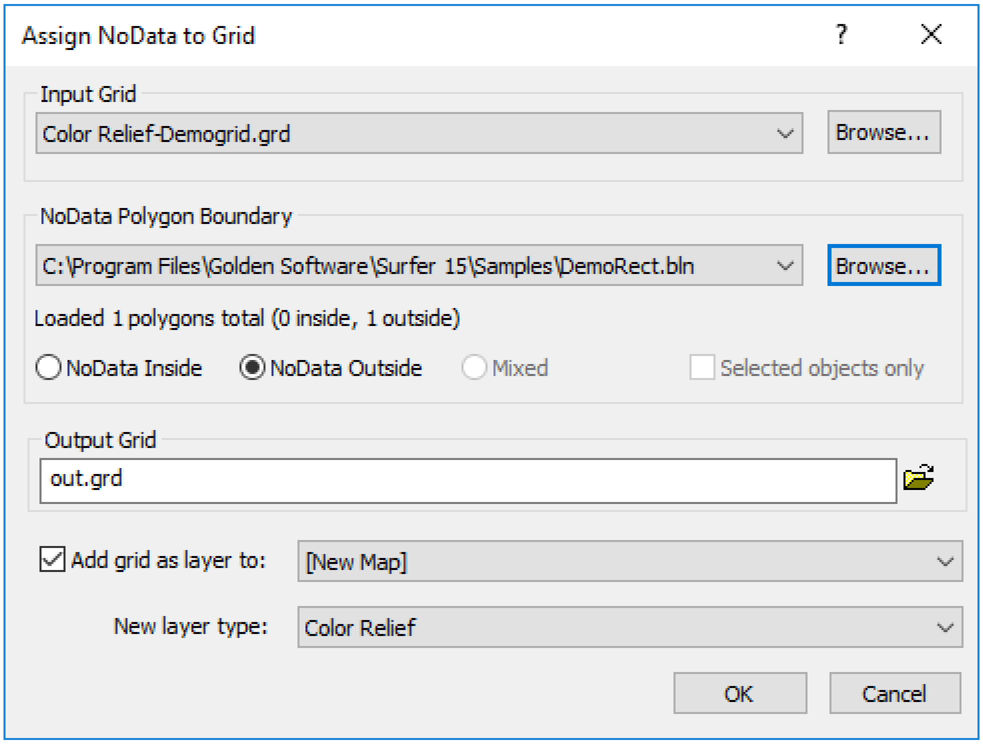
Nearly any feature type can be used when specifying boundaries with a base layer: polygon, range ring, rectangle, rounded rectangle, or ellipse. A polyline or spline polyline can also be used, but unexpected behavior may occur if the line is open. The most common feature for defining NoData polgyon and 3D polygon boundaries is the polygon or 3D polygon. The features must be in a base layer to be used with the **Assign NoData** command. Features in paper space cannot be used. Create new features in an empty base layer if you wish to assign NoData values with features created in **Surfer**.

To automatically assign the NoData value to areas outside the original data, click the Grids | New Grid | Grid Data command. In the **Grid Data** dialog, check the *Assign NoData outside convex hull of data.* Any grid nodes outside the boundary created by connecting the original data is automatically assigned the NoData value.

In Surfer 14 and prior versions, the **Assign NoData** command was referred to as *Grid Blank*.

Assign NoData to Grid Dialog

The **Assign NoData to Grid** dialog is displayed by clicking the **Grids | Edit | Assign NoData** command or the  button. Specify the input grid layer or file, vector file or layer, output grid file, and optional output map layer in the **Assign NoData to Grid** dialog.



Assign the NoData value to regions of a grid file or map layer with the ***Assign NoData to Grid*** *dialog.*

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

Polygon Boundary

Specify the region or regions to be assigned the NoData value and whether or not to use only selected objects in the *NoData Polygon Boundary* section. Select either a map layer or vector file in the *NoData Polygon Boundary* section:

- Click the current selection and select a base layer from the list. Only base layers that contain at least one polygon, polyline, 3D polygon or 3D polyline will be included in the list. The base layer must use the same source coordinate system as the grid.
- Click *Browse* to load a vector file with the Open dialog. The file must use the same coordinate system as the grid.

The number of polygons and 3D polygons and vertices is displayed below the *NoData polygon boundary* once a file or map layer has been selected. If the boundaries have blanking flags or BLN_Flag attributes, the total number of *inside* and *outside* flags is displayed.

Polyline Boundaries

Polylines and 3D polylines can be used for NoData polygon boundaries. The polylines in the base layer or vector file will be treated as polygons and 3D polylines treated as 3D polygons while assigning NoData values. The **Assign NoData** command is not recommended with open polylines as unexpected results may occur. Before clicking **Grids | Edit | Assign NoData**, consider converting the object to another type better suited to the operation you wish to perform with one of the Change To commands, and edit features with the Reshape command.

If the layer you wish to use contains both polygons and polylines, but you only wish to use some or all of the polygons, select the objects you wish to use before clicking **Grids | Edit | Assign NoData** and select the *Selected objects only* option. If the file you wish to use contains both polylines and polygons, first load the file as a base layer, and then use the **Assign NoData** command with the *Selected objects only* option.

Inside, Outside, or Mixed

Select *Inside* to assign the NoData value to the region inside the NoData polygon/3D polygon boundary or boundaries. Select *Outside* to assign the NoData value to the region outside the NoData polygon/3D polygon boundary or boundaries. Select *Mixed* to use the blanking flag or BLN_Flag attribute values from the file or layer. The *Mixed* option is only available when the layer or file contains both blanking flags or BLN_Flag attributes: assign NoData inside (1) and assign NoData outside (0). If all blanking flags or BLN_Flag attributes are the same, the *NoData Inside* or *NoData Outside* option is selected automatically, and the *Mixed* option is not available.

Selected Objects Only

Select the *Selected objects only* option to use only the selected objects in the base layer to assign NoData values to the grid. When the *Selected objects only* box is checked, the *Loaded polygons*and*vertices* values are updated. Select a base layer in the *NoData Polygon Boundary* field to use the *Selected objects only* option. The *Selected objects only* option is not available when the *NoData Polygon Boundary* is a vector file. The polygon or polygons must be selected before clicking the **Grids | Edit | Assign NoData** command.

Supported Vector Formats

The following file formats can be used to assign NoData values to a grid:

BLN Golden Software Blanking

BNA Atlas Boundary
DDF SDTS TVP
DLG USGS Digital Line Graph
DXF AutoCAD Drawing
E00 Esri AcrInfo Export Format
EMF Windows Enhanced Metafile
GSI Golden Software Interchange
KML Google Earth Keyhole Markup Language
MIF MapInfo Interchange Format
PLT Golden Software PlotCall
PLY Stanford Polygon
SHP Esri Shapefile
STL Stereo Lithograph

GML Geo Markup Language
GSB Golden Software Boundary
WMF Windows Metafile

Output Grid

In the *Output Grid* group click the  button to open the Save Grid As dialog. Specify a path or file name for the grid file to be created and click *Save*. A variety of output grid file types can be specified. Alternatively, type a file path and name, including the file type extension, in the *Output Grid* field.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

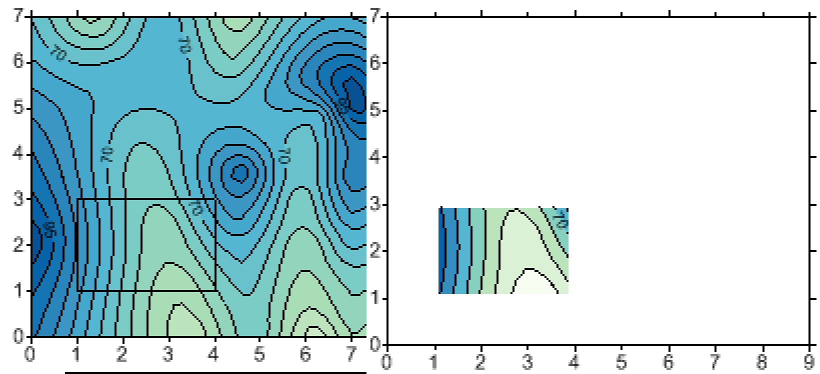
Assign NoData Example

In this example, some of the nodes in the *Demogrid.grd* sample file will be assigned the NoData value with the *Demorect.bln* sample file.

1. Click the **Home | New Map | Contour** command.
2. In the **Open Grid** dialog, select the *Demogrid.grd* sample file and click *Open*.
3. Click on the contour map in the plot window.
4. Click the **Home | Add to Map | Layer | Base** command.
5. In the **Import** dialog, select the *DemoRect.bln* sample file and click *Open* .
6. In the plot window, click the **Grids | Edit | Assign NoData** command.
7. In the **Assign NoData to Grid** dialog, click *<None>* in the *Input Grid* field and select *Contours-Demogrid.grd* from the list.
8. Click *<None>* in the *NoData Polygon Boundary* field and select *Base-DemoRect.bln* from the list.
9. Click the button next to *NoData Outside*.
10. Specify a file name and path in the *Output Grid* field.
11. Verify the *Add grid as layer to* check box is checked.
12. Verify *[New Map]* is displayed in the *Add grid as layer to* field.
13. Click the current selection in the *New layer type* field and select *Contour*.
14. Click *OK* in the **Assign NoData to Grid** dialog.

A new grid is created and saved to the file name specified in step 10. A new contour map is created and displayed in the plot window.

Before Assign NoData: *After Assign NoData:*



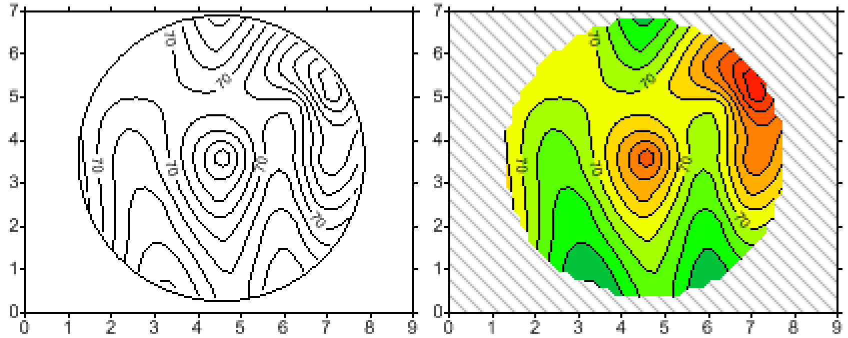
*The Blanking *.BLN file is displayed as a base map layer on the contour map.*

This is a contour map created from the new grid file.

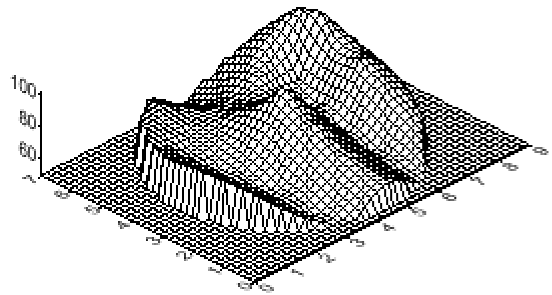
If the edges of the NoData region are jagged, the original grid file can be recreated with a higher grid density (i.e. more rows and columns). Use the more dense grid file and follow the steps to assign the NoData values to the grid file again. Assigning NoData values marks areas of a grid or map as "no data" areas. In NoData areas of a grid:

- Contours are not drawn.
- A separate fill color can be assigned to NoData regions/missing data in contour maps and color relief maps.
- 3D wireframe maps display a flat surface at the minimum Z value in the grid file.
- Volumes are not calculated and areas are calculated separately.
- Grids | Calculate | Math operations produce a NoData node if one of the grids contains a NoData node at a given location, unless a *Remap to* is selected for the grid.

Example 2



The left map shows an unfilled contour map with a base map layer displaying the circular NoData polygon boundary. Contour lines are truncated within one grid cell width of the boundary. The right map shows a filled contour map with a forward slash fill pattern assigned to NoData areas.



NoData regions appear as low flat regions on a 3D wireframe. This wireframe is produced from the same grid file used to produce the contour maps in the example above .

Assigning NoData to Areas within a Grid

There are several ways to assign the NoData value to areas within a grid.

Starting with a data file, the **Grids | New Grid | Grid Data** command assigns the NoData value to grid nodes:

- where the search criteria are not satisfied.
- outside the convex boundary of the data set if the triangulation or natural neighbor gridding methods are used.
- outside the convex boundary of the data set with any gridding method if the *Assign NoData grid outside convex hull of data* option is checked.
- within an area delineated by a fault polygon if there are no data values within that area.

Starting with a grid file, nodes are assigned NoData if:

- the **Grids | Edit | Assign NoData** command is used to assign NoData values to the area inside or outside a boundary defined by a blanking file or base

layer.
- the Eraser tool is used to assign NoData values to grid nodes in the grid editor.
- the Grids | Edit | Filter is used to assign NoData values to filtered grid nodes.

Assign NoData and .GSR2 Files

When the input .GRD file for a **Grids | Edit | Assign NoData** command has a defined .GSR2 file with coordinate system information, this information is used for the output .GRD file. The .BLN file must be in the same coordinate system as the input .GRD file, otherwise the file is not assigned the NoData value. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the ***GS Reference (Version 2) file*** if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system.

Golden Software Blanking .BLN File Description

Surfer imports and exports Golden Software Blanking Files .BLN.

Golden Software Blanking File .BLN is an ASCII format file used to store geographic information including polygons, polylines, and points. Spatial information is only concerned with the location of objects in space (i.e., their coordinates) and not with their attributes (i.e. line or fill style, marker symbol used, text labels, etc.). Even though the primary use of GS Blanking files is to indicate regions of a grid file to be assigned the NoData value, they can also be used for simple boundaries and decorative illustrations.

File Format

The general format of the file is:

```
length,flag "Pname 1"
x1,y1
x2,y2
...
xn,yn
length,flag "Pname 2"
x1,y1
x2,y2
...
xn,yn
```

Length

The *length* value is an integer which indicates the number of X, Y coordinate pairs that follow.

Flag

The *flag* value is 1 if the region inside areas is to be assigned the NoData value and 0 if the region outside areas is to be assigned the NoData value.

Pname

Pname is optional and is the name of a primary ID to be associated with the object. The primary ID is used to link the object to external data.

X, Y Coordinates

Following lines contain the actual X, Y coordinate pairs that make up the object. These can be integers or real numbers, and are stored 1 pair per line.

Type of Object

The type of object is determined as follows:

- If the *type/length* field is 1, the object is considered a point. One coordinate pair follows.
- If the *type/length* field is greater than 1 and the first and last coordinate pairs are equal, the object is considered a simple closed polygon. Otherwise, the object is considered a polyline.

Attributes

The first ID attribute for all polyline, polygon, and symbol objects are automatically exported to all .BLN files. For contour maps, the elevation is exported as the "STD_ID1" attribute for all polylines in the contour map. If a different attribute other than the first listed attribute for objects is desired, rename the desired attribute to "STD_ID1". This named attribute will be used instead of the first attribute listed on the Info tab. The color, size, symbol shape, width, and other properties are not exported.

Multiple Polygons and Polylines

The sequence may be repeated any number of times within the same file to define multiple polygons and polylines.

Blanking Flag

The blanking flag parameter is used to determine whether the NoData value is assigned inside (1) or outside (0) the polygon. The XY coordinates used in the blanking file are in the same units used for the XY coordinates in the grid file. When creating a .BLN with the Digitize command, the default blanking flag is a 1. If you need to assign NoData values outside the polygon, click **Options | NoData Inside Region** in the **Digitized Coordinates** window to clear the **NoData Inside Region** option.

With breaklines and faults, the blanking flag is not needed and can be left empty.

Optional Z Coordinate

A Z coordinate may be specified after the X and Y coordinates when using a .BLN file for breaklines. The Z value is required for breaklines. Faults do not require a Z coordinate.

Examples

This example shows a simple .BLN file, with a single polygon:

```
5 0
1 1
1 3
4 3
4 1
1 1
```

This example shows a single complex polygon .BLN file, with an island. Note the 48,99 coordinate pair starts the first polygon (line 2), closes the first polygon (line 8), and closes the entire complex polygon (line 14). The 40,70 coordinate pair starts (line 9) and closes (line 13) the sub-object within the complex polygon. Each sub-object in a complex polygon is followed by the first polygon coordinate pair:

```
13 0
48 99
52 20
57 19
56 8
29 0
27 71
48 99
40 70
50 60
48 55
34 40
40 70
48 99
```

The following example defines two breaklines for input into the gridding module:

```
2, 1
3.0, 4.5, 1.0
0.5, 4.5, 2.0
3, 1
4.5, 0.0, 1.1
8.5, 0.0, 1.2
8.5, 1.5, 1.4
```

The following example defines one fault for input into the gridding module:

```
2, 1
1.5, 1.0
7.5, 8.5
```

Loading a BLN

Use the **File | Open** or the **Open Data** dialog (i.e. **Home | New Map | Base**, **Home | New Map | Post**) to load a .BLN file.

Saving a BLN

Use **File | Export** to save a BLN file from the plot window. BLN files are also created via the Digitize command and various grid operations.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Grid Filter

The **Grids | Edit | Filter** command applies methods of digital image analysis to grids. This includes a broad suite of smoothing (low-pass) filters, as well as contrast enhancement filters, edge enhancement filters, edge detection filters, general high-pass filters, etc. **Surfer** also includes the capability for user-defined, general linear filters.

The Neighborhood

When filtering a grid, each node of the output grid is computed as a function of the corresponding node, and its neighbors, in the input grid. The concept of the neighborhood is used in grid filtering. The neighborhood of an output grid node is a rectangular sub-array of nodes in the input grid that is centered on the corresponding input grid node. A neighborhood has a non-zero width, and a non-zero height. Since the neighborhood is centered on a node, the width and height must both be odd numbers. For example, if the width and the height of the neighborhood are both three, the neighborhood of the output grid node at (21, 35) is the following rectangular sub-array of input grid nodes:

(20, 36) (21, 36) (22, 36)
(20, 35) (21, 35) (22, 35)
(20, 34) (21, 34) (22, 34)

If the height of the neighborhood is represented by S and the width of the neighborhood is represented by T, then the number of nodes in the neighborhood equals $S \times$. Furthermore, the nodes in the neighborhood can be enumerated as:

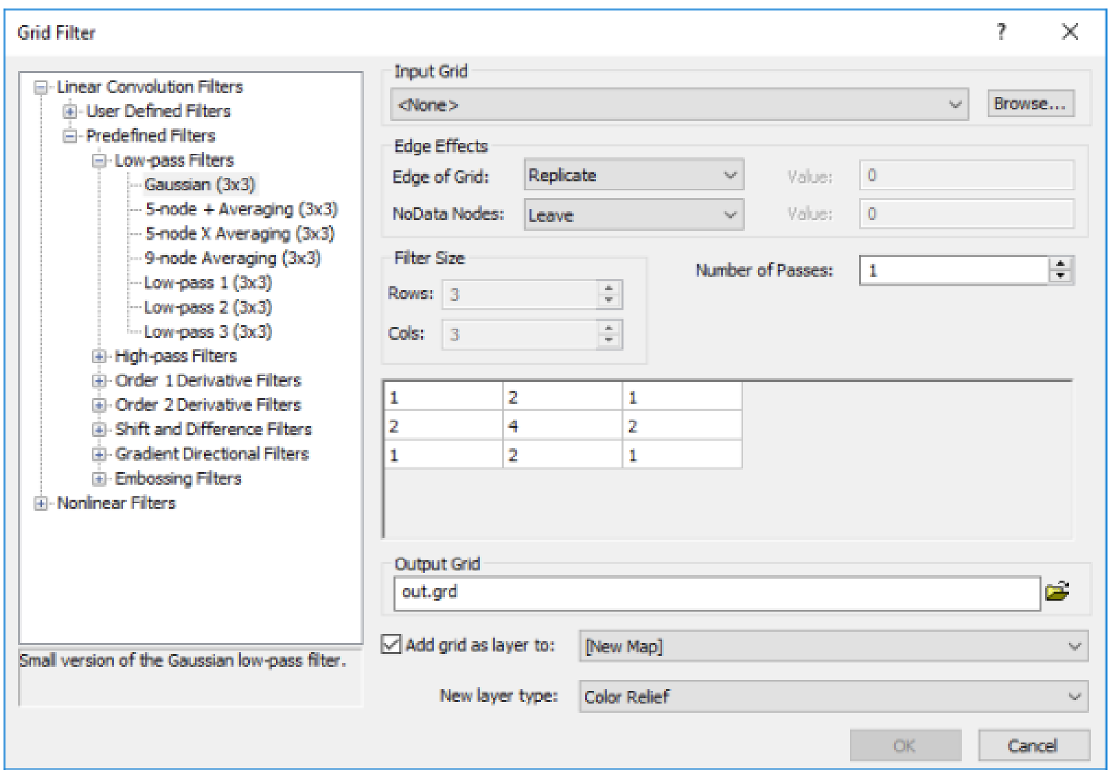
$$\left\{ Z\left(r+i, c+j\right) \text{ where } i = -\left\lfloor \frac{S}{2} \right\rfloor, \ldots, \left\lfloor \frac{S}{2} \right\rfloor \text{ and } j = -\left\lfloor \frac{T}{2} \right\rfloor, \ldots, \left\lfloor \frac{T}{2} \right\rfloor \right\}$$

where $\lfloor A \rfloor$ is the largest integer less than or equal to A.

The box in the lower-right part of the dialog displays the neighborhood size, based on the number of *Rows* and *Cols*, along with the weights for each grid node in the neighborhood. Each element of the matrix is used to weight the grid node that lies "below" it. The products are then summed, normalized, and assigned to the value below the center node. The filter is then "moved" to the next node and the process is repeated until all nodes have been processed.

Grid Filter Dialog

Click the **Grids | Edit | Filter** command or the  button to open the **Grid Filter** dialog.



*Specify filtering options in the **Grid Filter** dialog.*

Filter Categories

There are two main categories of filters: Linear Convolution Filters and Nonlinear Filters. Both of these general types are real space filtering methods.

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

Edge Effects

There are two settings for edge effects, how to calculate the *Edge of Grid* nodes and how to calculate neighborhoods with *NoData Nodes*.

Filter Size

Filter Size is available with user-defined linear convolution filters and some nonlinear filters. You can use *Filter Size* to determine the size of the neighborhood.

Number of Passes

Set the number of times the filter is applied with the *Number of Passes* box.

Output Grid

Digital filtering computes each node of the output grid as a function of the corresponding node, and its neighbors, of the input grid. Choose a path and file name for the grid in the *Output Grid* section. Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the button and specify the path and file name for the grid file in the Save Grid As dialog.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

Grid Filter and .GSR2 Files

When the input .GRD file for a **Grids | Edit | Filter** command has a defined .GSR2 file with coordinate system information, this information is used for the output .GRD file. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system automatically.

Linear Convolution Filters

All linear convolution filters compute weighted averages of the neighboring input grid nodes. The only differences between the various linear convolution filters are the size and shape of the neighborhood, and the specific weights used. Consider the computation of the output grid value at row r and column

$$\text{Numerator} = \sum_{i=-\left\lfloor \frac{S}{2} \right\rfloor}^{\frac{S}{2}} \sum_{j=-\left\lfloor \frac{T}{2} \right\rfloor}^{\frac{T}{2}} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ W(i,j)Z(r+i,c+j) & \textit{otherwise} \end{cases}$$

$$\text{Denominator} = \sum_{i=-\left\lfloor \frac{S}{2} \right\rfloor}^{\left\lfloor \frac{S}{2} \right\rfloor} \sum_{j=-\left\lfloor \frac{T}{2} \right\rfloor}^{\left\lfloor \frac{T}{2} \right\rfloor} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ W(i,j) & \textit{otherwise} \end{cases}$$

where *W(i,* are the weights defined for the specified filter. The output grid node value is then

$$Z_{out}(r,c) = \begin{cases} \dfrac{\text{Numerator}}{\text{Denominator}} & \textit{if } |\text{Denominator}| > 0 \\ \text{Numerator} & \textit{otherwise} \end{cases}$$

User Defined Filters

There are two types of user defined filters, *Low-pass Filters* and *General User-defined*. With these filters, you can specify the height and width of the filter neighborhood.

Low-pass Filters

A low-pass filter removes the high frequency noise with the resulting output being a smoother grid. There are four user-defined low-pass filters. Each of these four filters allows you to specify the size of the neighborhood. The width and height of the filter neighborhood must both be positive, odd numbers. Let the neighborhood height be S and width be T,

Moving Average (mxn)	In the *Moving Average (mxn)* filter the weights are all equal to one. $W(i,j)=1 \qquad \forall_i = -\left\lfloor \frac{S}{2} \right\rfloor,\ldots,\left\lfloor \frac{S}{2} \right\rfloor \forall_j = -\left\lfloor \frac{T}{2} \right\rfloor,\ldots,\left\lfloor \frac{T}{2} \right\rfloor$
Distance Weight-ing (mxn)	With the *Distance Weighting (mxn)* filter the weights fall-off with increased distance. The distance weighting function is $W(i,j)=\left(1-\max\left(\frac{2\lvert i\rvert}{S+1},\frac{2\lvert j\rvert}{T+1}\right)\right)^p \qquad \forall_i = -\left\lfloor \frac{S}{2} \right\rfloor,\ldots,\left\lfloor \frac{S}{2} \right\rfloor \forall_j = -\left\lfloor \frac{T}{2} \right\rfloor,\ldots,\left\lfloor \frac{T}{2} \right\rfloor$ where *p* is the specified *Power*. The higher the power the more rapidly the weights fall-off with distance. The resulting iso-weight contour lines are concentric rectangles.
Inverse Distance (mxn)	With the *Inverse Distance (mxn)* filter, the weights fall-off with increased distance. With a neighborhood height *S* and width *T*, the distance weighting function is $W(i,j)=\begin{cases} W_c & \text{if } i=0 \text{ and } j=0 \\ \left(\frac{1}{i^2+j^2}\right)^{p/2} & \textit{otherwise} \end{cases} \qquad \forall_i = -\left\lfloor \frac{S}{2} \right\rfloor,\ldots,\left\lfloor \frac{S}{2} \right\rfloor \forall_j = -\left\lfloor \frac{T}{2} \right\rfloor,\ldots,\left\lfloor \frac{T}{2} \right\rfloor$ where WCis the specifiedCentral Weightandpis thePower. The higher the powerp, the more rapidly the weights fall-off with distance. The resulting
Gaussian Low-pass (mxn)	With the *Gaussian Low-pass (mxn)* filter, the weights fall-off with increased distance. With a neighborhood height *S* and width *T*, the distance weighting function is $W(i,j)=\exp\left(-\alpha\left(\left(\frac{i}{S}\right)^2+\left(\frac{j}{T}\right)^2\right)\right) \qquad \forall_i = -\left\lfloor \frac{S}{2} \right\rfloor,\ldots,\left\lfloor \frac{S}{2} \right\rfloor \forall_j = -\left\lfloor \frac{T}{2} \right\rfloor,\ldots,\left\lfloor \frac{T}{2} \right\rfloor$ where α is the *Alpha* value (positive). This weight function takes the form of half the common bell-shaped curve. The *Alpha* para-meter controls how quickly the weights fall-off with distance. The resulting iso-weight contour lines are concentric ellipses. The lower the Alpha value, the more weight neighborhood points have on the grid value and the slower the weight drops off. Con-versely, the higher the Alpha value, the more weight the center point has on the grid value and the faster the weight of the other points drops off.

General User-defined Filter

The General User-Defined (MXN)	The *General User-defined (mxn)* linear filter allows you to specify the height and width of the filter neighborhood and any combination of weights. The box in the lower-right part of the dialog displays the neighborhood size, based on the number of *Rows* and *Cols*, along with the weights for each grid node in the neighborhood. Click in a cell in the box to change the node's weight. The grid matrix can be selected and copied using CTRL+C and pasted into the user-defined matrix. This allows a pre-defined matrix to be used as a base and then be modified.

Predefined Filters

The predefined filters section is a large collection of 3×3 filters defined in the grid filter references.

Low-Pass Filters	*Low-pass Filters* are also known as smoothing or blurring filters. These filters remove the high frequency variation.
High-Pass Filters	*High-pass Filters* are also known as sharpening or crispening filters. They have the opposite effect of blurring. They tend to remove the background variation and emphasize the local details.
Order 1 Derivative Filters	*Order 1 Derivative Filters* are used to find horizontal and vertical edges.
Order 2 Derivative Filters	*Order 2 Derivative Filters* are another set of edge enhancement filters.
Shift and Difference Filters	*Shift and Difference Filters* are the two simplest horizontal and vertical differential operators.
Gradient Directional Filters	*Gradient Directional Filters* compute and return the directional derivatives in each of the eight compass directions.
Embossing Filters	*Embossing Filters* identify and enhance edges aligned in one of the eight compass directions.

Nonlinear Filters

The *Nonlinear Filters* are not weighted averages of the neighboring input grid values; however, they are simple functions of the neighboring input grid values.

Order Statistics Filters

Minimum (mxn)	The output grid node value equals the minimum of the neighboring values.

Lower Quartile (mxn)	The output grid node value equals the twenty-five percentile of the neighboring values.
Median (mxn)	The output grid node value equals the fifty percentile of the neighboring values.
Upper Quartile (mxn)	The output grid node value equals the seventy-five percentile of the neighboring values.
Maximum (mxn)	The output grid node value equals the maximum of the neighboring values.
Range (mxn)	The output grid node value equals the maximum minus the minimum of the neighboring values.

Moment Statistics Filters

Standard Deviation (mxn)	The output grid node value equals the standard deviation of the neighboring values. The standard deviation is the square root of the variance.
Variance (mxn)	The output grid node value equals the variance of the neighboring values. The computation involves three steps. First, count the number of non-NoData (i.e. non-blank) input grid node values in the neighborhood. With a neighborhood height *S* and width *T*, compute $$N = \sum_{i=-\lfloor S/2 \rfloor}^{\lfloor S/2 \rfloor} \sum_{j=-\lfloor T/2 \rfloor}^{\lfloor T/2 \rfloor} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ 1 & \text{otherwise} \end{cases}$$ Second calculate the local average, $\overline{Z}$, $$\overline{Z} = \frac{1}{N} \sum_{i=-\lfloor S/2 \rfloor}^{\lfloor S/2 \rfloor} \sum_{j=-\lfloor T/2 \rfloor}^{\lfloor T/2 \rfloor} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ Z(r+i, c+j) & \text{otherwise} \end{cases}$$ Third, compute the local variance as $$\text{Variance} = \frac{1}{N} \sum_{i=-\lfloor S/2 \rfloor}^{\lfloor S/2 \rfloor} \sum_{j=-\lfloor T/2 \rfloor}^{\lfloor T/2 \rfloor} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ (Z(r+i, c+j) - \overline{Z})^2 & \text{otherwise} \end{cases}$$
Coef. of Variation (mxn)	The output grid node value equals the coefficient of variation of the neighboring values. The coefficient of variation is the standard deviation divided by the average. $$\text{Coefficient of Variation} = \frac{\text{Standard Deviation}}{\overline{Z}}$$ If the average (the denominator) is too small, then the coefficient of variation is arbitrarily set equal to 0.

Other Nonlinear Filters

Median Difference (mxn)	For each output grid node, (r, c), identify the set of non-NoData (non-blank), neighboring input grid node values. Compute the median of these neighboring values. Let B represent this median value. Then the output grid node value is set equal to $Z_{out}(r,c) = Z(r,c) - B$ The effect of this filter is to emphasize outliers in the grid.
Threshold Averaging (mxn)	For each output grid node, (r, c), identify the set of non-NoData (non-blank), neighboring input grid node values. Compute the average of these neighboring values, excluding the value of the corresponding input grid node. Let A represent this average value. Then the output grid node value is set equal to $Z_{out}(r,c) = \begin{cases} Z(r,c) & \text{if } \lvert Z(r,c) - A \rvert \leq \text{Threshold} \\ A & \text{if } \lvert Z(r,c) - A \rvert > \text{Threshold} \end{cases}$ where *Threshold* is the specified value. The effect of this filter is to eliminate outliers in the grid.

Compass Gradient Filters

Compass gradient filters find edges in the eight different compass orientations (east, northeast, north, northwest, west, southwest, south, and southeast). First, eight different linear convolution filters designed to detect edges in each of the eight directions is applied. The output grid node value is the maximum absolute value resulting from these eight convolutions. The *Prewitt (3X3)*, *Kirsch (3X3)*, *Robinson 3-level (3X3)*, and *Robinson 5-level (3X3)* are four of the more common edge detection convolution filters in the literature. For details see Crane (1997, p.88-90).

Edge of Grid

When computing the Grids | Edit | Filter values of the output nodes near the edge of the grid, the specified neighborhood may extend outside the range of the grid. This presents an issue because the output grid nodes are then functions of non-existent input grid nodes. There are three general ways of dealing with this issue: assign the NoData value to the output grid nodes near the edge, modify the defining function near the edge, or generate artificial input grid nodes beyond the edge upon which the defining function operates. The following methods for handling the edge are as follows:

Assign NoData

Any grid node, for which its neighborhood overlaps one or more edge, is assigned the NoData value. For example, if the neighborhood size is 3×3 then every application of the filter assigns the NoData value to one line of nodes on each edge. Every application of the filter shrinks the active grid by two rows and two columns: one row on the top, one row on the bottom, one column on the left edge, and one column on the right edge. Similarly, a 5×5 filter neighborhood Assigns the NoData value to four rows and four columns every application.

Ignore

Ignore is the default setting. In this case, the filter function is modified by truncating the neighborhood at the edge of the grid. For example, in the Neighborhood section, even though the general neighborhood is specified to be 3×3, the effective neighborhood of node (21, 0), on the left edge of the grid, is only 3×2:

(22, 0) (22, 1)

(21, 0) (21, 1)

(20, 0) (20, 1)

This approach to handling the edges applies an essentially different function near the edges than in the middle of the grid. This can cause unexpected, visually apparent, artifacts in some extreme cases.

Replicate

The edge grid node value is copied. For a grid with M rows and

$$Z(r,c) = Z(a,b)$$

where

$$a = \begin{cases} 0 & \text{if } r < 0 \\ r & \text{if } 0 \le r < M \\ M-1 & \text{if } r \ge M \end{cases} \qquad b = \begin{cases} 0 & \text{if } c < 0 \\ c & \text{if } 0 \le c < N \\ N-1 & \text{if } c \ge N \end{cases}$$

Mirror

The grid node value pattern at the edge is mirrored. For a grid with M rows and

$$Z(r,c) = Z(a,b)$$

where

$$a = \begin{cases} -r & \text{if } r < 0 \\ r & \text{if } 0 \le r < M \\ 2M - 2 - r & \text{if } r \ge M \end{cases} \qquad b = \begin{cases} -c & \text{if } c < 0 \\ c & \text{if } 0 \le c < N \\ 2N - 2 - c & \text{if } c \ge N \end{cases}$$

Cyclic Wrap

The grid is wrapped in two dimensions. If you go off the grid on the right edge, you come back on the left; if you go off on the top you come back on the bottom, etc. For a grid with M rows and

$$Z(r,c) = Z(a,b)$$

where

$$a = \begin{cases} r + M & \text{if } r < 0 \\ r & \text{if } 0 \le r < M \\ r - M & \text{if } r \ge M \end{cases} \qquad b = \begin{cases} c + N & \text{if } c < 0 \\ c & \text{if } 0 \le c < N \\ c - N & \text{if } c \ge N \end{cases}$$

Fill

Fill the edge with a specified constant. A common value for the fill is the arithmetic average of the grid.

$$Z(r,c) = \begin{cases} Z(r,r) & \text{if } 0 \le r < M \text{ and } 0 \le c < N \\ A & \textit{otherwise} \end{cases}$$

where *A* is the user-specified constant. Enter a number into the *Value* box to the right when using this option.

NoData Nodes - Grid Filter

There are several ways to handle NoData nodes in a neighborhood in filtering. (If all nodes in a neighborhood contain the NoData value, the output node is also assigned NoData.) NoData nodes must be exactly equal to the NoData value.

Expand

Expand the NoData regions. If the neighborhood of an output grid node contains a NoData node in the input grid, then the output node is assigned the NoData value. Like assigning NoData on the edge, this approach leads to NoData areas that grow with every application of the filter.

Leave

Leave the NoData regions. This is the default setting. Every output grid node is assigned the NoData value for which the corresponding input grid node is NoData. When the corresponding input grid node contains a value, but a neighboring grid node contains NoData, modify the filter to ignore the NoData node; essentially, remove the NoData node from the neighborhood. For example, if the filter called for computing the median value in the neighborhood, the NoData values would not be considered when determining the median. This keeps the NoData areas constant, but can cause internal artifacts for some filter types.

Ignore

Ignore the NoData nodes by filtering across them. The NoData nodes are removed from the neighborhood. For example, if the filter called for computing the median value in the neighborhood, the NoData values would not be considered when determining the median. This option is similar to *Leave*, however, this option does not assign NoData to the output grid nodes corresponding to NoData input grid nodes. This is essentially a simultaneous filtering and interpolation. Every application of the filter would see a shrinking of the NoData regions, since the only NoData output grid nodes are those with neighborhoods of only NoData values.

Fill

Fill NoData grid nodes with a user-specified constant prior to filtering. When using this option, enter the *Fill* value into the *Value* field.

Grid Filter References

Crane, R. (1997) *A Simplified Approach to Image Processing: Classical and Modern Techniques in C*, Prentice Hall PTR, Upper Saddle River, NJ, 317 pp. ISBN: 0-13-226416-1.

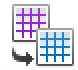
Pitas, I. (2000)Digital Image Processing Algorithms and Applications, John Wiley and Sons, New York, 419 pp. ISBN: 0-471-37739-2.

Wiggin, Ender (2001) Elementary Digital Filtering, http://www.gamedev.net/reference/articles/article1068.asp, (09/20/2001).

Grid Convert

The **Grids | Edit | Convert** command converts a grid file to another grid format, or an ASCII XYZ data file format.

To convert a grid:

1. Click the **Grids | Edit | Convert** command or the  button.
2. In the Open Grid dialog, select a grid file and click *Open*.
3. In the Save Grid As dialog, enter a file name in the *File name* box.
4. Select a file format from the *Save as type* list. Refer to the Save Grid As page for a list of available formats.
5. Click the *Save* button to create the new file.

Grid Convert and .GSR2 Files

When the input .GRD file for a **Grids | Edit | Convert** command has a defined .GSR2 file with coordinate system information, this information is used for the output file. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system.

Grid Spline Smooth

The **Grids | Edit | Spline Smooth** command uses cubic spline interpolation to compute new grid nodes. The interpolation simulates a drafting technique where a flexible strip (a spline) is used to draw a smooth curve between data points. Spline smoothing does not extrapolate beyond the edge of the original grid file. The original grid limits are used to define the grid limits for the smoothed grid. Spline smoothing does not retain faulting information.

A spline is really nothing more than the graphs of a set of contiguous (end-point adjacent) cubic polynomials with the same slopes at their endpoints. Cubic spline smoothing may increase the maximum Z value and decrease the minimum Z value from the input grid.

There are two ways to perform spline smoothing; by expanding the grid or by recalculating the grid.

Expanding a Grid

When a grid is expanded, the original grid nodes are preserved in the smoothed grid, and new grid nodes are added between existing grid nodes. The number of added grid nodes is defined in the **Spline Smooth** dialog by the *Number Nodes to Insert* boxes.

Original Grid:

o | o | o | o | o

o	o	o	o	o
o	o	o	o	o

Expanded Grid:

o	x	x	o	x	x	o	x	x	o	x	x	o
x	x	x	x	x	x	x	x	x	x	x	x	x
x	x	x	x	x	x	x	x	x	x	x	x	x
o	x	x	o	x	x	o	x	x	o	x	x	o
x	x	x	x	x	x	x	x	x	x	x	x	x
x	x	x	x	x	x	x	x	x	x	x	x	x
o	x	x	o	x	x	o	x	x	o	x	x	o

A simple three by five grid is shown on the top in the above example. Each grid node is represented by an "o". When expanding the grid, two grid nodes are inserted between each existing grid node to produce the grid on the bottom. The original grid nodes are indicated with "o", and the new grid nodes are indicated with "x." The original grid is three by five and the smoothed grid is seven by thirteen.

To Expand a Grid

1. Click the **Grids | Edit | Spline Smooth** command or the 

button.
2. In the **Spline Smooth** dialog, specify the input grid from a map layer or grid file in the *Input Grid* section.
3. Click the *Insert Nodes* option in the *Method* section.
4. In the *Number Nodes to Insert* section, enter the number of nodes to insert. Or, use the up and down arrows to change the values. As you make the changes, the number of rows and columns to be produced in the smoothed grid is indicated in the *Final Grid Size* section.
5. The *Between Rows* box specifies the number of rows to insert between the existing rows in the grid file. Rows correspond to grid nodes of constant Y.
6. The *Between Cols* box specifies the number of columns to insert between the existing columns in the grid file. Columns correspond to grid nodes of constant X.
7. If you need information on the original grid file, click the  button in the *Input Grid File* section to display the number of rows and columns, minimum and maximum X, Y, Z values, and statistics.
8. To change the name for the smoothed grid file, click the  button in the *Output Grid File* section, specify the path and file name in the **Save Grid As** dialog, and click *Save*.
9. Click *OK* and the smoothed grid is created.

Recalculating a Grid

When a grid file is recalculated, the number of rows and columns are increased or decreased relative to the original grid. The original grid values are lost unless their locations correspond exactly with the grid nodes in the output grid. The smoothed grid file will still be an accurate representation of the original data. For example, you might have a 75 x 75 grid and need a 100 x 100 grid to perform grid math with another grid. You can recalculate the grid to have exactly the number of rows and columns needed.

To Recalculate a grid:

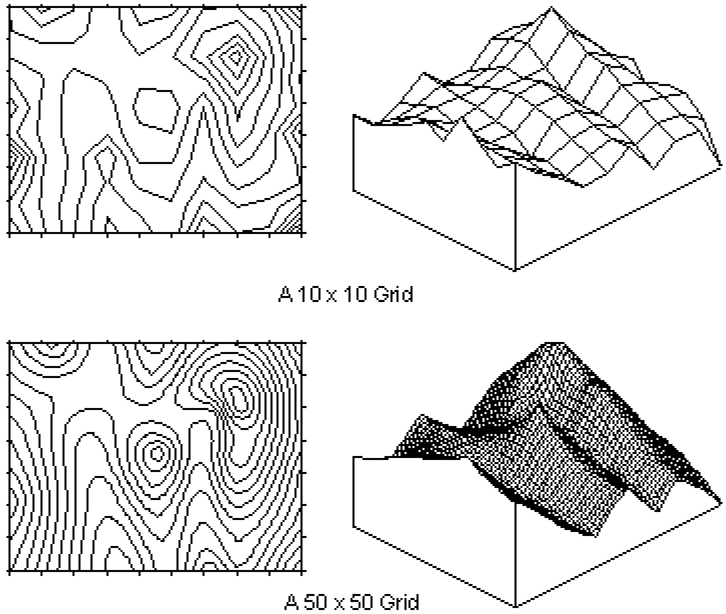
1. Click the **Grids | Edit | Spline Smooth** command.
2. In the **Spline Smooth** dialog, specify the input grid from a map layer or grid file in the *Input Grid* section.
3. Click the *Recalc Grid* option in the *Method* section.
4. In the *Final Grid Size* section, specify the number of rows and columns to produce in the smoothed grid. You can increase or decrease the number of rows and columns in the grid file. Enter the desired value, or use the up and down arrows on your keyboard to change the values for the number of rows and columns to be produced in the smoothed grid.
    - The *# Rows* box specifies the number of rows for the smoothed grid file.
    - The *# Cols* box specifies the number of columns for the smoothed grid file.
5. If you need information on the original grid file, click the  button in the *Input Grid File* section to display the number of rows and columns, minimum and maximum X, Y, Z values, and statistics.
6. To change the name for the smoothed grid file, click the  button in the *Output Grid File* group, specify the path and file name in the **Save Grid As** dialog, and click *Save*.
7. Click *OK* and the smoothed grid is created.

Reduce Grid File Density

Spline smoothing can be used to reduce grid file density. If a dense grid is created, producing a map from this grid might take a considerable amount of time. The less dense grid can be used to produce the contour map or 3D wireframe in less time. USGS GTopo30 files are quite dense and it is necessary to thin them out before attempting to plot a map of the grid.

Fill in a Sparse Grid

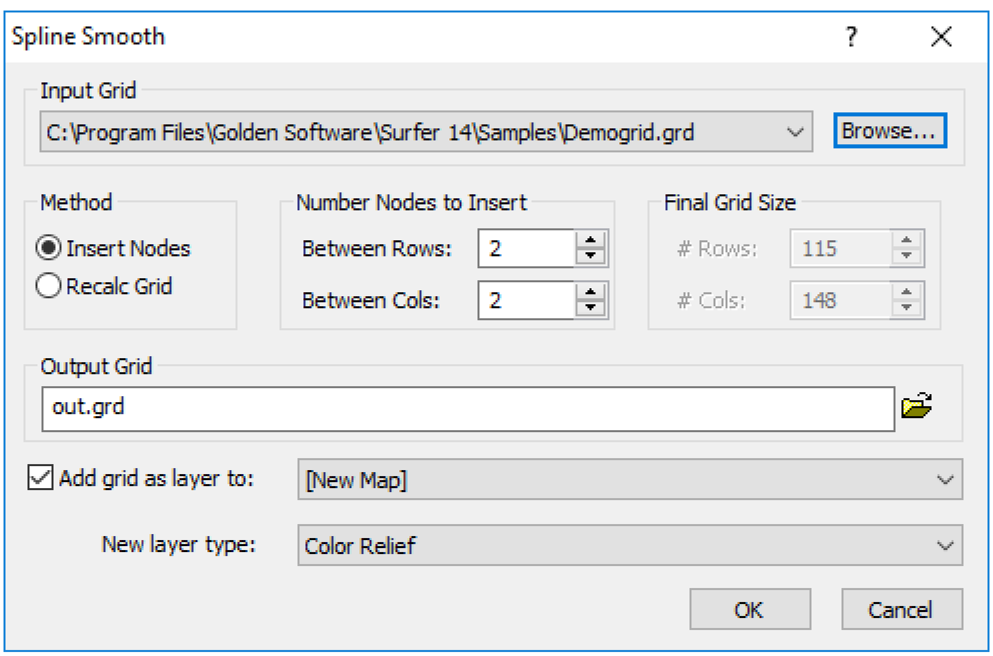
One of the purposes of spline smoothing is to fill in a sparse grid. A map produced from a sparse grid may have an angular appearance. For example, spline smoothing can be used to increase a 10 x 10 grid (a sparse grid) to a 50 x 50 grid. Denser grids produce smoother maps.



Increasing the grid density using spline smoothing increases the roundness or smoothness of the contours and the surface appearance.

The Spline Smooth Dialog

Click the **Grids | Edit | Spline Smooth** command to open the **Spline Smooth** dialog.



To expand a grid, choose the Insert Nodes *option in the* ***Spline Smooth*** *dialog. When recalculating a grid, choose the* Recalc Grid *option in the* ***Spline Smooth*** *dialog.*

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

Insert Nodes

Choose the *Insert Nodes* option in the *Method* section to activate the *Number Nodes to Insert* section.

Number of Nodes to Insert

Choose the *Insert Nodes* option in the *Method* section to activate the *Number Nodes to Insert* section. Enter the number of nodes to insert, or use the up and down arrows to change the values. As you make the changes, the number of rows and columns to be produced in the smoothed grid is indicated in the *Final Grid Size* section.

Insert Nodes Between Rows

The *Between Rows* box specifies the number of rows to insert between the existing rows in the grid file. Rows correspond to grid nodes of constant Y.

Insert Nodes Between Columns

The *Between Cols* box specifies the number of columns to insert between the existing columns in the grid file. Columns correspond to grid nodes of constant X.

Recalculate Grid

Choose the *Recalc Grid* option in the *Method* section to activate the *Final Grid Size* section. This group specifies the number of rows and columns to produce in the smoothed grid. You can increase or decrease the number of rows and columns in the grid file.

Final Grid Size

Choose the *Recalc Grid* option in the *Method* section to activate the *Final Grid Size* section. This group specifies the number of rows and columns to produce in the smoothed grid. You can increase or decrease the number of rows and columns in the grid file. Enter the *# Rows* and/or *# Cols*, or use the up and down arrows to change the values.

Rows

The *# Rows* box specifies the number of rows for the smoothed grid file. Enter the desired value, or use the up and down arrows to change the values for the number of rows to be produced in the smoothed grid.

Columns

The *# Cols* box specifies the number of columns for the smoothed grid file. Enter the desired value, or use the up and down arrows to change the values for the number of columns to be produced in the smoothed grid.

Output Grid

Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the  button and specify the path and file name for the grid file in the Save Grid As dialog.

Add New Map or Layer

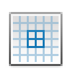
Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

Grid Spline Smooth and .GSR2 Files

When the input .GRD file for a **Grids | Edit | Spline Smooth** command has a defined .GSR2 file with coordinate system information, this information is used for the output .GRD file. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original grid file, but the new .GSR2 is required to define the coordinate system.

Assign Coordinate System - Grid

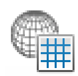
The **Grids | Edit | Assign Coordinate System** command or the  button links a grid file to a specific coordinate system. Once the coordinate system is defined for the grid file, a Golden Software Georeference .GSR2 file is created. This file contains all the relevant projection information that **Surfer** needs to load the grid in the proper projection.

Use the Data | Coordinate System | Assign Coordinate System command to link a data file to a .GSR2 file. When a .GRD file is created using the Grids | New Grid | Grid Data command, the .GSR2 file for the data is read. The projection information can be saved with the grid file using the Spatial References options. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in **Surfer**, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original data file, but the .GSR2 is required to define the coordinate system. When a map is created from either the data file or the .GRD file, the .GSR2 file is read and the map layer automatically has the correct coordinate system.

If the data file did not have an assigned coordinate system, or the grid file was provided directly from another source, the **Grids | Edit | Assign Coordinate System** can be used to link the grid with a .GSR2 file.

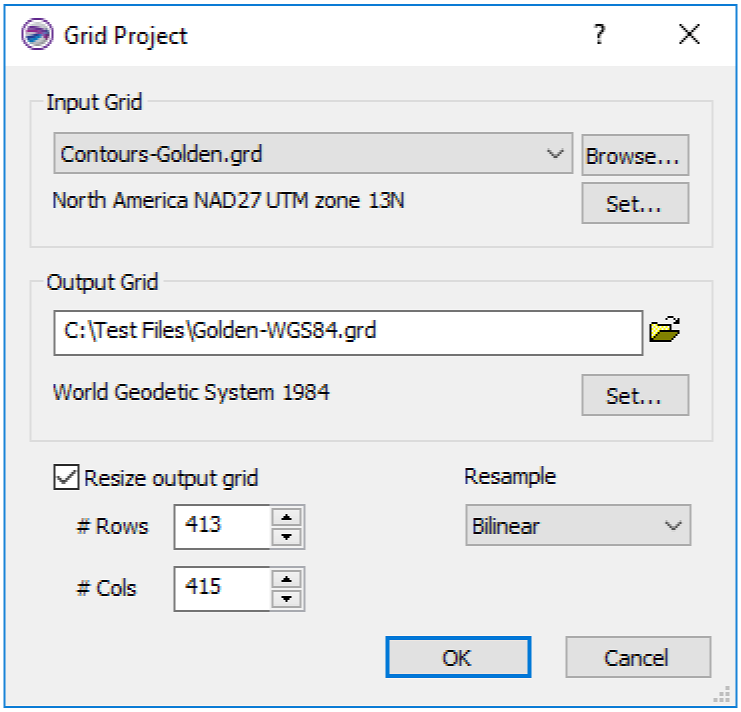
When any grid operation is performed on the referenced grid file, the **Export Options** dialog appears with the option to create a new .GSR2 file for the new output grid. For instance, if you use the Grids | Edit | Filter command and smooth the grid, the output smoothed grid will contain a .GSR2 file with the same coordinate system information as the original grid. When a map is created from the smoothed grid file, the .GSR2 file is read and the map layer automatically has the correct coordinate system. For commands that use multiple files, such as Grids | Edit | Assign NoData or Grids | Resize | Mosaic, all files should be in the same reference system, otherwise unexpected results may occur. The .GSR2 file in these cases is assigned by the first .GRD file specified.

Grid Project

Click the **Grids | Edit | Project** command or the  button to create a new grid in a different coordinate system from a grid in an existing map layer or a grid file. The **Project** command creates a grid in the desired coordinate system and then resamples the original grid to populate the new grid with Z values. You can control the output grid coordinate system, geometry, and resampling method. Click the **Grids | Edit | Project** command to open the **Grid Project** dialog.

Grid Project Dialog

The **Grid Project** dialog contains the options for creating the projected grid file.



*Specify the coordinate system, grid geometry, and resample method in the **Grid Project** dialog.*

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog. Click and drag the corner of the **Grid Project** dialog to resize the dialog if the entire *Input Grid* path and file name is not visible.

The input grid file coordinate system will be displayed below the *Input Grid File* field. The coordinate system will be updated automatically if the map layer is referenced or if the grid file includes a georeference file, such as GSR2. Define the *Input Grid* coordinate system by clicking *Set* and defining the coordinate system in the Assign Coordinate System dialog.

Output Grid

Specify a path, name, and file format for the projected grid in the *Output Grid* section. Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the  button and specify the path and file name for the grid file in the Save Grid As dialog.

Click and drag the corner of the **Grid Project** dialog to resize the dialog if the entire *Output Grid* path and file name is not visible.

Specify the desired coordinate system for the projected grid by clicking *Set* and defining the coordinate system in the Assign Coordinate System dialog.

Resize Output Grid

By default the *Resize output grid* check box is not checked. The output grid will be the same size as the input grid. The number of rows and columns will be adjusted if necessary to maintain the same or nearly the same grid spacing as the input grid file. The *# Rows* and *# Cols* display the number of rows and columns in the output grid. When *Resize output grid* is not checked, the *# Rows* and *# Cols* values are informational and cannot be changed.

Check the *Resize output grid* check box to specify the number of rows and columns in the output grid. Type a number in the *# Rows* field or click the buttons to select the number of rows. Type a number in the *# Cols* field or click the buttons to select the number of columns.

Resample

The *Resample* method determines how the Z values for the output grid are calculated. Define the resampling method by selecting a method in the *Resample* list. The output grid values can be resampled with the *Nearest Neighbor*, *Bilinear*, or *Cubic Convolution* method.

Nearest Neighbor

The *Nearest Neighbor* method applies the closest grid node value on the original grid to the grid node value in the new projected grid. When the original grid and the new grid differ in size, more than one original node may be applied to the new grid and some original grid cells may not be applied to the new grid. The *Nearest Neighbor* method is the fastest resampling method, though it can result in distorted output.

Bilinear Interpolation

The *Bilinear Interpolation* method uses a weighted average of four nodes in the original grid and applies this to the projected grid node value. The projected grid is smoothed compared to the original grid.

Cubic Convolution

The *Cubic Convolution* method uses a weighted average of 16 nodes in the original grid and applies this to the new grid node value. The projected grid is smoother than the original grid. This method is best for continuous data. This is the slowest resampling method, but it results in a sharper grid than the *Bilinear Interpolation* or the *Nearest Neighbor* methods.

OK and Cancel

Click *OK* to create the projected grid. Click *Cancel* to close the **Grid Project** dialog without creating a grid.

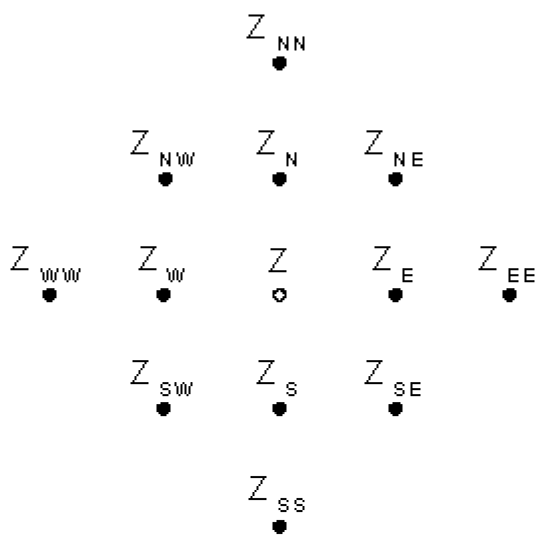
Grid Calculus

Click the **Grids | Calculate | Calculus** command or the $\sqrt{n}$ button to access tools to interpret your grid files. Grid calculus can help you define and quantify characteristics in the grid file that might not be obvious by looking at a contour or 3D wireframe of the grid.

The **Grids | Calculate | Calculus** command creates a new grid file of the generated data. Generated grid files use the same dimensions as the original grid file but might use different ranges of data depending on the type of output.

When a numerical derivative is needed, central difference formulae are used in the calculus computations in **Surfer**. Because a central difference approach is used, values on both sides of the location for which the derivative is computed are required. This leads to NoData values along the edges of the derivative grids, otherwise known as an edge effect.

Surfer uses "compass-based" grid notation to indicate the neighboring grid nodes used for many of the **Grid Calculus** operations, as illustrated below:



Using this grid notation, we can write the difference equation approximations for the necessary derivatives at location Z as follows:

$$\frac{dz}{dx} \approx \frac{Z_E - Z_W}{2\Delta x}$$

$$\frac{dz}{dy} \approx \frac{Z_N - Z_S}{2\Delta y}$$

$$\frac{d^2z}{dx^2} \approx \frac{Z_E - 2Z + Z_W}{\Delta x^2}$$

$$\frac{d^2z}{dy^2} \approx \frac{Z_N - 2Z + Z_S}{\Delta y^2}$$

$$\frac{d^2z}{dx\,dy} \approx \frac{Z_{NE} - Z_{NW} - Z_{SE} + Z_{SW}}{4\,\Delta x\,\Delta y}$$

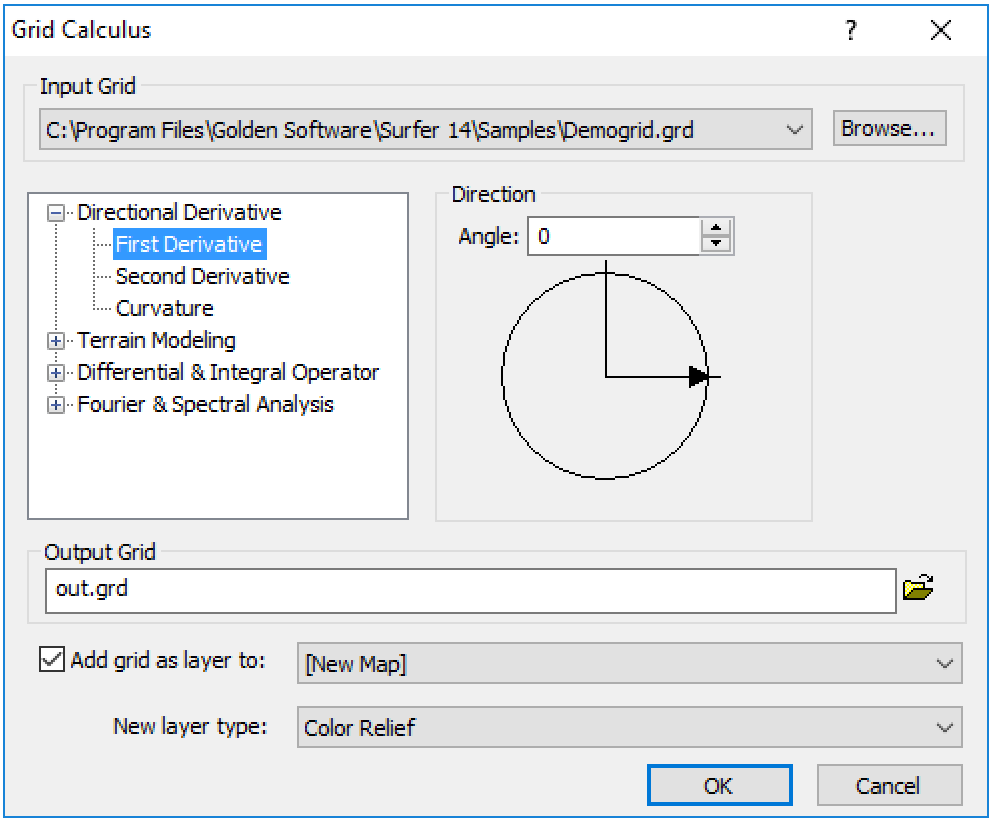
$$\frac{d^4z}{dx^4} \approx \frac{Z_{WW} - 4Z_W + 6Z - 4Z_E + Z_{EE}}{\Delta x^4}$$

$$\frac{d^4z}{dy^4} \approx \frac{Z_{NN} - 4Z_N + 6Z - 4Z_S + Z_{SS}}{\Delta y^4}$$

$$\frac{d^4z}{dx^2\,dy^2} \approx \frac{Z_{NW} - 2Z_N + Z_{NE} - 2Z_W + 4Z - 2Z_E + Z_{SW} - 2Z_S + Z_{SE}}{4\,\Delta x\,\Delta y}$$

The Grid Calculus Dialog

Click the **Grids | Calculate | Calculus** command to open the **Grid Calculus** dialog.



*The **Grid Calculus** dialog provides four sections of commands to interpret your grid file.*

The **Grid Calculus** dialog is divided into four sections:

- Directional Derivatives,
- Terrain Modeling,
- Differential and Integral Operators, and
- Fourier and Spectral Analysis.

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

Output Grid

Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the  button and specify the path and file name for the grid file in the Save Grid As dialog.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

Using Grid Calculus

To create a grid using **Grid | Calculus** :

1. Click the **Grids | Calculate | Calculus** command.
2. Select the grid map layer or grid file in the *Input Grid* section.
3. Expand the calculus type you wish to use by clicking on the ⊞ next to the type name.
4. Select a calculus operation (e.g. terrain slope).
5. If available, set the options for the operation.
6. Click the button to set the *Output Grid* path and file name.
7. Click *OK* and the new grid file is created.

Grid Calculus and .GSR2 Files

When the input .GRD file for a **Grids | Calculate | Calculus** command has a defined .GSR2 file with coordinate system information, this information is used for the output .GRD file. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system.

Directional Derivatives

The Grids | Calculate | Calculus directional derivatives provide you with information about the slope, or rate of change of slope, of the gridded surface in a specified direction. Because this takes a specified direction into account, this slope, or rate of change in slope, might not be the steepest slope at a given point. For example, if the specified direction is due East, but the gradient is due North, the directional derivative slope is zero at that point. In the specified direction, there is no slope at that point, although there is a slope to the North.

Directional derivatives are different from the values given in Terrain Modeling where the direction of the slope is defined as the gradient, or the direction of steepest ascent at a given point (i.e., straight uphill at that point). In the above example, the terrain model would report the slope in the north direction at that point. The two methods would report different values at that particular point on the grid.

Let Z be defined as a function of X and Y in a domain that includes point P. At what rate does Z change if we move from point P in a specified direction? In the X axis direction, the rate of change for Z is $\partial z / \partial x$, and the rate of change in the Y axis direction for Z is $\partial z / \partial y$.

Definition - The Directional Derivative (Schwartz, 1974)

Let f(x,

$$\gamma(h) = C_0 + S \cdot h$$,

such that the line segment PQ lies in D, and let

$$\Delta f = f(x_Q, y_Q) - f(x_P, y_P)$$.

Then

$$\lim_{\Delta s \to 0} \frac{\Delta f}{\Delta s} = \frac{df}{ds}$$

is the directional derivative of f(x,y) at P in the direction of

There are three directional derivative options available: *First Derivative*, *Second Derivative*, and *Curvature*. You can specify the angular direction for the operation.

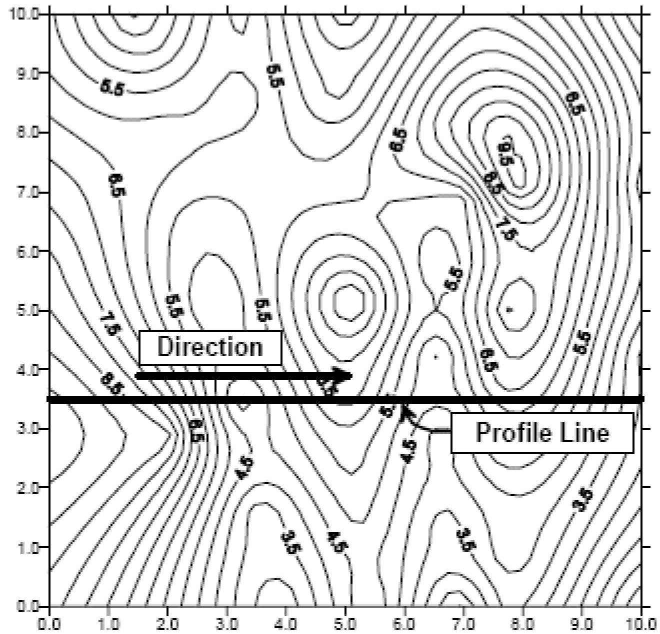
First Derivative

Second Derivative

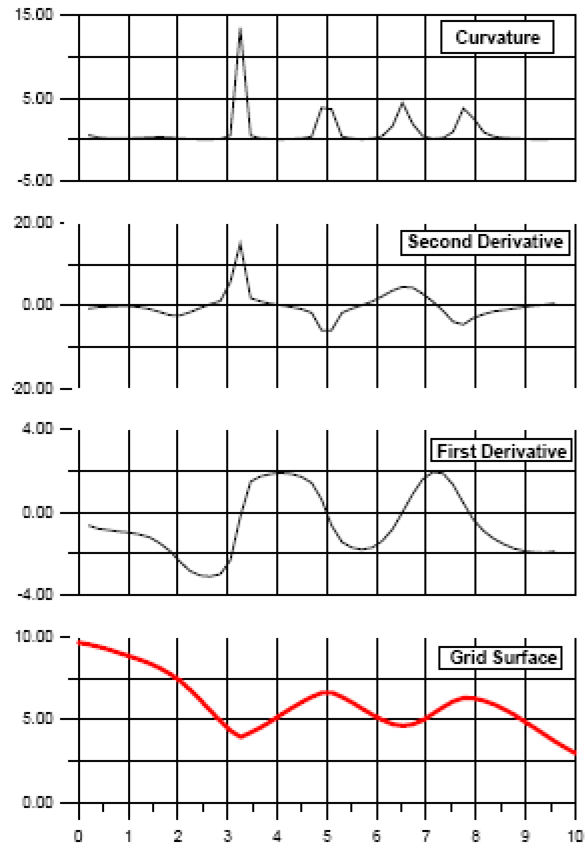
Curvature

Directional Derivative Example

This example is based on the *Demogrid.grd* file included with **Surfer**, although the limits of the grid have been extended to 10 units in both the X and Y dimensions. The map shows the contours and the direction chosen for the **Grids | Calculate | Calculus** directional derivative operations. A profile line is also included on the map. The graphs included are based on Grids | Calculate | Slice command data collected along the profile lines. Each operation was performed, and the **Slice** command was applied to the resultant grid files. The results are shown in the graphs that were produced in Golden Software, LLC's **Grapher**™ . A comparison of the map and the graphs gives a good visual analysis of the results of the directional derivative operations.



This map shows the profile line for the directional derivative and directional curvature calculations.

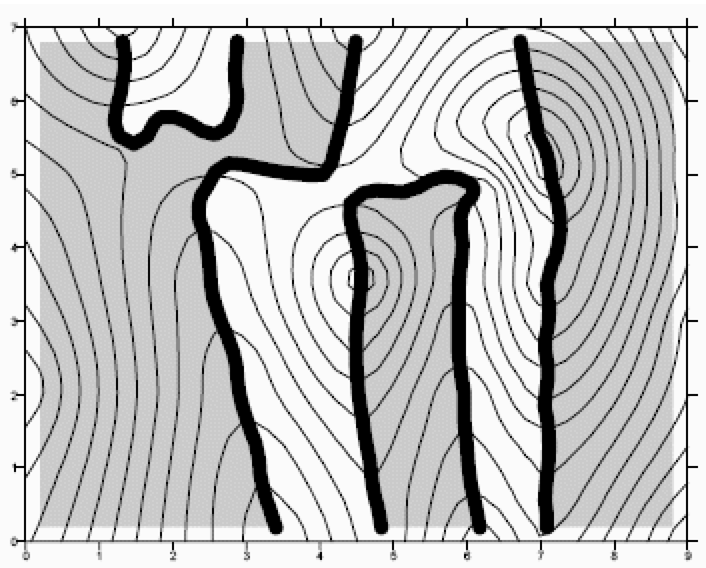


These are cross sectional views of Curvature, Second, and First Derivatives, and the original grid surface.

First Derivative

The Grids | Calculate | Calculus directional derivative, *First Derivative*, calculates the slope of the surface along a given direction. First derivative grid files produce contour maps that show isolines of constant slope along lines of fixed direction. At a particular grid node location, if the slope is uphill, the slope is positive; and if downhill, the slope is negative. Slope is reported as rise over run and can approach negative or positive infinity as the slope approaches vertical in the down or up direction.

The directional first derivative is exactly as in the definition (Schwartz, 1974). Instead of a known function "Z(X,Y)" for which we can compute the necessary limits, **Surfer** uses a grid file, so directional derivative is approximated using difference equations.



The original grid file is displayed as a contour map with thin contour lines. The first derivative map is overlaid on the contour map with negative values (downhill in the +x direction) in gray, positive values (uphill in the + x direction) in white, and the zero contour shown as a bold black line.

The directional derivative at a point is equal to the dot product between the gradient vector and a unit vector in the direction of interest:

$$\frac{df}{ds} = \vec{g} \cdot \begin{bmatrix} \cos(\alpha) \\ \sin(\alpha) \end{bmatrix}$$

$$= \begin{bmatrix} \frac{df}{dx}, \frac{df}{dy} \end{bmatrix} \cdot \begin{bmatrix} \cos(\alpha) \\ \sin(\alpha) \end{bmatrix}$$

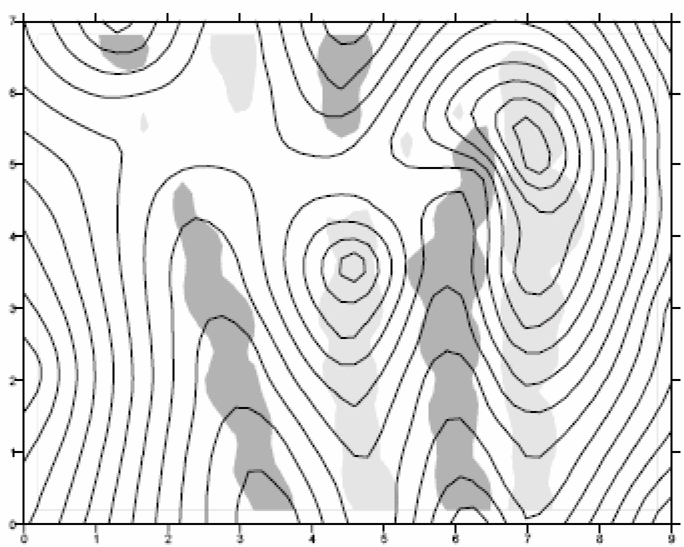
$$= \frac{df}{dx} \cdot \cos(\alpha) + \frac{df}{dy} \cdot \sin(\alpha)$$

where *a* is the user specified angle (see Schwartz, 1974, p. 785, or Tuma, 1979, p. 89). Using compass-based grid notation, the equation takes the following form:

$$\frac{dZ}{ds} \approx \frac{Z_E - Z_W}{2\Delta x} \cdot \cos(\alpha) + \frac{Z_N - Z_S}{2\Delta y} \cdot \sin(\alpha)$$

Second Derivative

The Grids | Calculate | Calculus directional derivative, *Second Derivative*, calculates the rate of change of slope along a given direction. Second derivative grid files produce contour maps that show isolines of constant rate of change of slope across the surface. If the slope is uphill, the rate of change is positive and downhill is negative.



Contour map of original grid file is overlaid on second derivative map. Areas of greatest change in slope in the X direction occur on ridges (light gray) and valleys (dark gray).

The directional second derivative is the directional derivative of the first directional derivative:

$$\frac{d^2f}{ds^2} = \frac{d\left[\frac{df}{ds}\right]}{ds}$$

$$\frac{d^2f}{ds^2} = \frac{d\left[\frac{df}{dx}\cdot\cos(\alpha) + \frac{df}{dy}\cdot\sin(\alpha)\right]}{ds}$$

$$= \frac{d^2f}{dx^2}\cdot\cos^2(\alpha) + 2\frac{d^2f}{dxdy}\cdot\cos(\alpha)\cdot\sin(\alpha) + \frac{d^2f}{dy^2}\cdot\sin(\alpha)$$

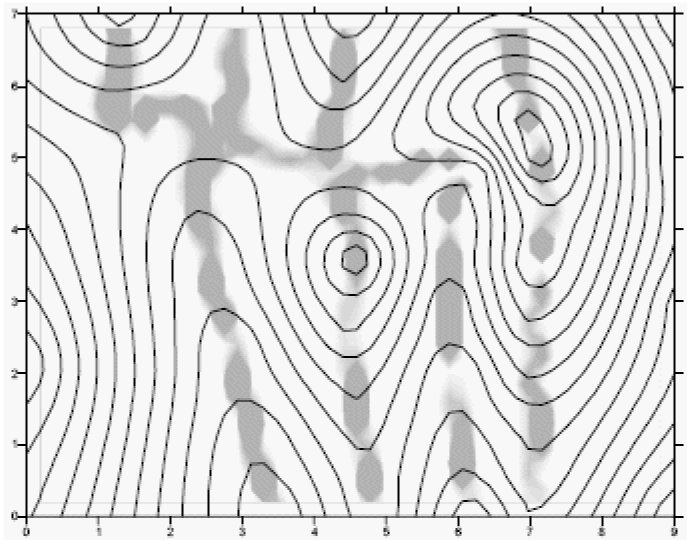
The compass-based grid notation difference equation used in **Surfer** takes the form:

$$\frac{d^2Z}{ds^2} \approx \frac{Z_E - 2Z + Z_W}{\Delta x^2}\cdot\cos^2(\alpha) + \frac{Z_{NE} - Z_{NW} - Z_{SE} + Z_{SW}}{4\Delta x\Delta y}\cdot\cos(\alpha)\cdot\sin(\alpha) + \frac{Z_N - 2Z + Z_S}{\Delta y^2}\cdot\sin^2(\alpha)$$

Curvature

The Grids | Calculate | Calculus directional derivative, *Curvature*, is a measure of the rate of change of the inclination angle of tangential planes on a profile line defined by the surface along a specified direction. *Curvature* is reported as the absolute value of the rate of change and is, therefore, a positive number. *Curvature* is similar to the Second Derivative.

The directional curvature is the absolute value of the rate of change, in a specified direction, of the inclination angle of the tangent plane.



The contour map of the original grid file is overlaid with a filled directional curvature map. The maximum curvature in the X direction (as indicated by the darker gray contour fill) occurs at valleys and ridges.

The mathematical formula for the directional curvature of a surface f(x,

$$K_s = \frac{\left|\frac{d^2f}{ds^2}\right|}{\left[1+\left(\frac{df}{ds}\right)^2\right]^{3/2}}$$

(see Schwartz, 1974, p. 592).

$$K_\alpha = \frac{\left|\frac{d^2f}{dx^2}\cdot\cos^2(\alpha)+2\frac{d^2f}{dx\,dy}\cdot\cos(\alpha)\cdot\sin(\alpha)+\frac{d^2f}{dy^2}\cdot\sin(\alpha)\right|}{\left[1+\left(\frac{df}{dx}\cdot\cos(\alpha)+\frac{df}{dy}\cdot\sin(\alpha)\right)^2\right]^{3/2}}$$

Terrain Modeling

The Grids | Calculate | Calculus terrain modeling option provides additional tools to analyze the surface geometry of a grid file. Results are based on the gradient direction (direction of steepest slope) and not a predefined direction as with Directional Derivatives. For each grid node location on a surface, **Surfer** determines the magnitude and direction of the steepest slope.

There are five terrain modeling operations:

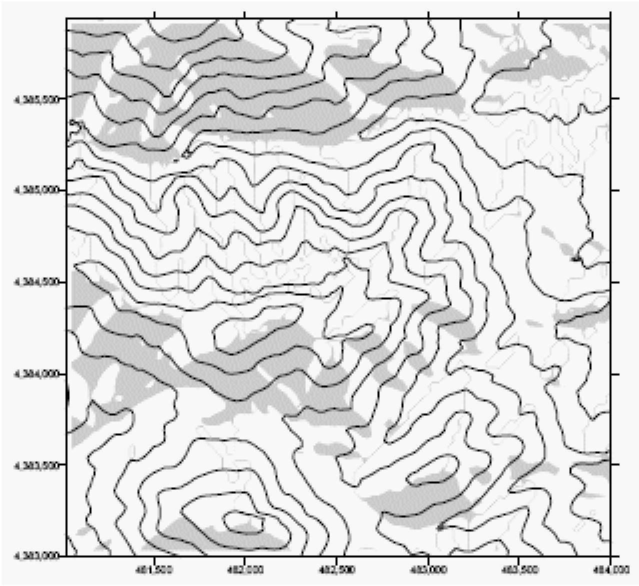
- Terrain Slope
- Terrain Aspect
- Profile Curvature
- Plan Curvature
- Tangential Curvature

Threshold

When you choose the *Terrain Aspect, Profile Curvature*, *Plan Curvature*, or *Tangential Curvature* options, you can also specify a *Threshold* value. In areas where the steepest slope approaches zero (where the surface is nearly horizontal), the gradient direction is somewhat difficult to ascertain (i.e., the definitions "downhill" and "uphill" are rather arbitrary). In these cases, it is preferable to classify the surface as flat. The *Threshold* value is the minimum value for the slope for which the aspect and curvature values are computed. In locations where the *Threshold* value is not met, the grid node value in the output grid is set to NoData. The *Threshold* value is, by default, set to a very small value.

Terrain Slope

The Grids | Calculate | Calculus terrain model, *Terrain Slope*, calculates the slope at any grid node on the surface. *Terrain Slope* is reported in degrees from zero (horizontal) to 90 (vertical). For a particular point on the surface, the *Terrain Slope* is based on the direction of steepest descent or ascent at that point (Terrain Aspect). This means that across the surface, the gradient direction can change. Grid files of the *Terrain Slope* can produce contour maps that show isolines of constant steepest slope. This operation is similar to the way the First Directional Derivative defines the slope at any point on the surface but is more powerful in that it automatically defines the gradient direction at each point on the map.



In this example, a contour map of the original DEM is overlaid with a terrain aspect map. South-facing slopes (between SE or 135° and SW or 215°) are indicated with gray fills. Gray lines on north facing slopes are contour fills drawn where the aspect varies between 0 degrees (north) and 359 degrees (1 degree west of north).

The slope, S, at a point P is the magnitude of the gradient at that point. From the definition of the gradient:

$$S = \sqrt{\left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2}$$

Using the compass-based grid notation difference equation yields:

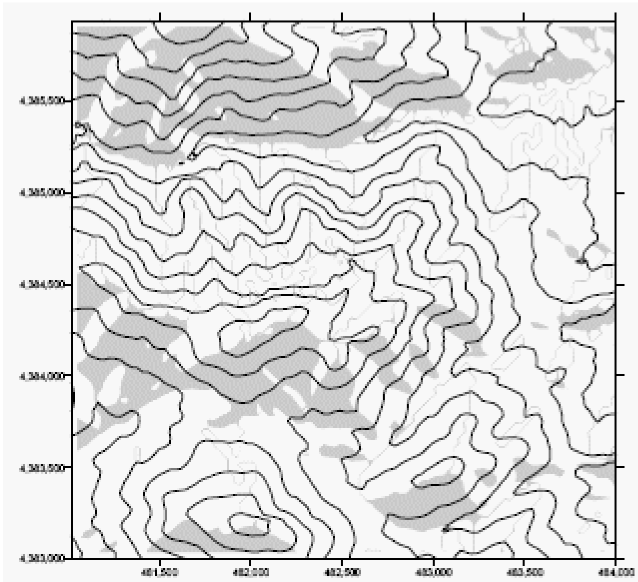
$$S \approx \sqrt{\left(\frac{Z_E - Z_W}{2\Delta x}\right)^2 + \left(\frac{Z_N - Z_S}{2\Delta y}\right)^2}$$

The terrain slope is represented as a slope angle S_T in degrees in keeping with the terrain modeling literature, such as Moore et al. (1993):

$$S_T \approx \frac{360}{2\pi} \cdot \arctan\left[\sqrt{\left(\frac{Z_E - Z_W}{2\Delta x}\right)^2 + \left(\frac{Z_N - Z_S}{2\Delta y}\right)^2}\right]$$

Terrain Aspect

The Grids | Calculate | Calculus terrain model, *Terrain Aspect*, calculates the downhill direction of the steepest slope (i.e. dip direction) at each grid node. It is the direction that is perpendicular to the contour lines on the surface, and is exactly opposite the gradient direction. *Terrain Aspect* values are reported in azimuth, where 0 degrees points due North, and 90 degrees points due East.



A contour map of the original DEM is layered with a terrain aspect map. South facing slopes (between SE or 135° and SW or 215°) are indicated with gray fills. Gray lines on north facing slopes are contour fills drawn where the aspect varies between 0 degrees (north) and 359 degrees (1 degree west of north).

In keeping with the terrain modeling literature, i.e. Moore et al. (1993), **Surfer** represents the terrain aspect, A_T, as an azimuth (in degrees, not radians):

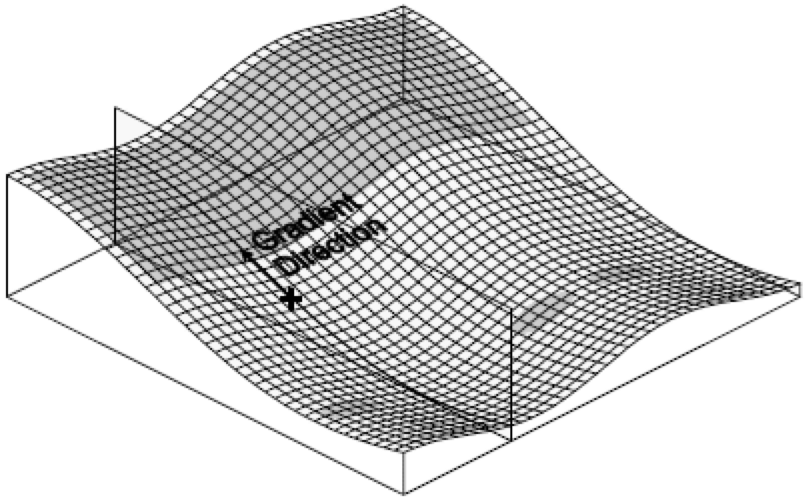
$$A_T = 270 - \frac{360}{2\pi} \cdot \text{atan2}\left[\frac{\partial z}{\partial y}, \frac{\partial z}{\partial x}\right]$$

The compass-based grid notation version of this equation is:

$$A_T \approx 270 - \frac{360}{2\pi} \cdot \text{atan2}\left[\frac{Z_N - Z_S}{2\Delta y}, \frac{Z_E - Z_W}{2\Delta x}\right]$$

Profile Curvature

The Grids | Calculate | Calculus terrain model, *Profile Curvature*, determines the downhill or uphill rate of change in slope in the gradient direction (opposite of slope aspect direction) at each grid node. Grid files of *Profile Curvature* produce contour maps that show isolines of constant rate of change of steepest slope across the surface. This operation is comparable to the Second Directional Derivative but is more powerful because it automatically determines the downhill direction at each point on the surface, and then determines the rate of change of slope along that direction at that point. Negative values are convex upward and indicate accelerated flow of water over the surface. Positive values are concave upward and indicate slowed flow over the surface.



Profile Curvature measures the curvature of the surface in the direction of gradient. Negative curvature, shown with a gray fill, indicates a convex upward surface and accelerated water flow.

The profile curvature KPis given by:

$$K_p = \frac{\left(\frac{\partial^2 z}{\partial x^2}\right)\left(\frac{\partial z}{\partial x}\right)^2 + 2\left(\frac{\partial^2 z}{\partial x \partial y}\right)\left(\frac{\partial z}{\partial x}\right)\left(\frac{\partial z}{\partial y}\right) + \left(\frac{\partial^2 z}{\partial y^2}\right)\left(\frac{\partial z}{\partial y}\right)^2}{pq^{3/2}}$$

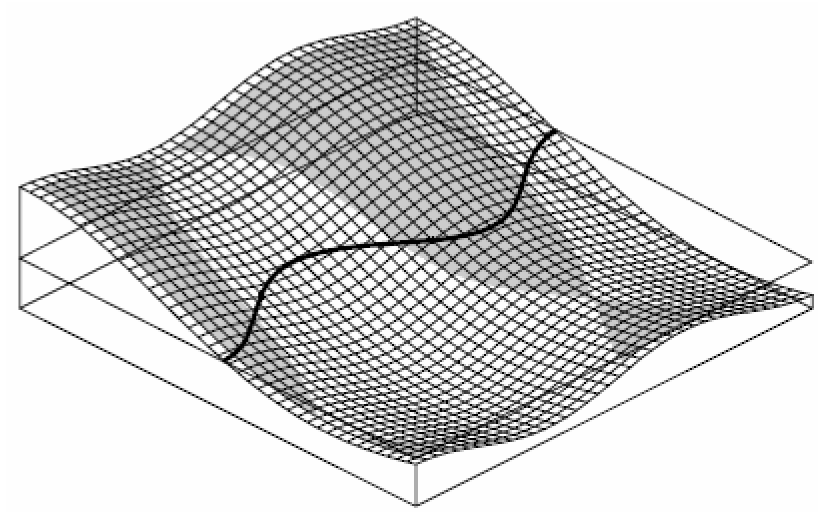
where

$$p = \left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2$$

$$q = 1 + p$$

Plan Curvature

The Grids | Calculate | Calculus terrain model, *Plan Curvature*, reflects the rate of change of the Terrain Aspect angle measured in the horizontal plane, and is a measure of the curvature of contours. Negative values indicate divergent water flow over the surface, and positive values indicate convergent flow.



Plan Curvature calculates the curvature of the surface in the horizontal plane, or the curvature of the contour. Negative curvature, shown with a gray fill, indicates areas of divergent flow.

The plan curvature KHis given by:

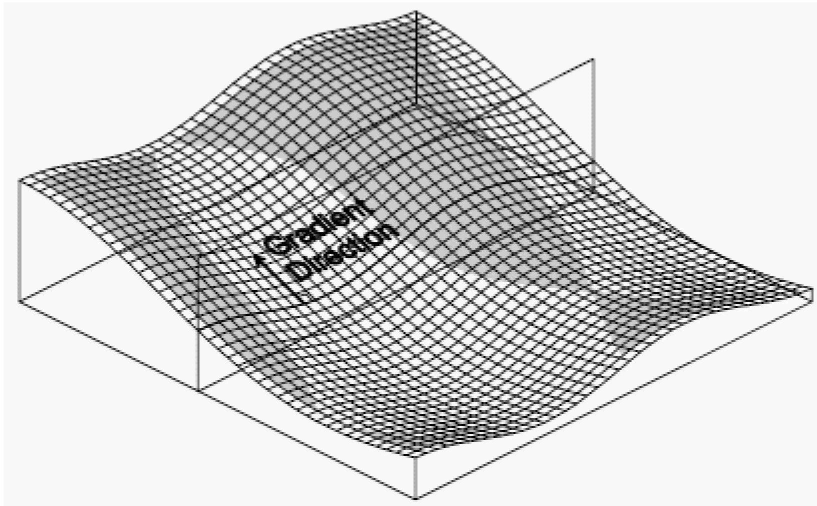
$$K_H = \frac{\left(\frac{\partial^2 z}{\partial x^2}\right)\left(\frac{\partial z}{\partial y}\right)^2 - 2\left(\frac{\partial^2 z}{\partial x \partial y}\right)\left(\frac{\partial z}{\partial x}\right)\left(\frac{\partial z}{\partial y}\right) + \left(\frac{\partial^2 z}{\partial y^2}\right)\left(\frac{\partial z}{\partial x}\right)^2}{p^{3/2}}$$

where

$$p = \left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2$$

Tangential Curvature

The Grids | Calculate | Calculus terrain model,*Tangential Curvature*, measures curvature in relation to a vertical plane perpendicular to the gradient direction, or tangential to the contour. The negative and positive areas are the same as for Plan Curvature, but the curvature values are different. *Tangential Curvature* is related to the *Plan Curvature* KHby the sine of the slope:



Tangential Curvature measures the curvature of the surface in the vertical plane perpendicular to the gradient direction. Negative curvature, displayed with gray fill, indicates divergent flow conditions.

Tangential Curvature KTis given by the equation:

$$K_T = \frac{\left(\frac{\partial^2 z}{\partial x^2}\right)\left(\frac{\partial z}{\partial y}\right)^2 - 2\left(\frac{\partial^2 z}{\partial x \partial y}\right)\left(\frac{\partial z}{\partial x}\right)\left(\frac{\partial z}{\partial y}\right) + \left(\frac{\partial^2 z}{\partial y^2}\right)\left(\frac{\partial z}{\partial x}\right)^2}{p q^{1/2}}$$

where

$$p = \left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2$$

$$q = 1 + p$$

Terrain Modeling References

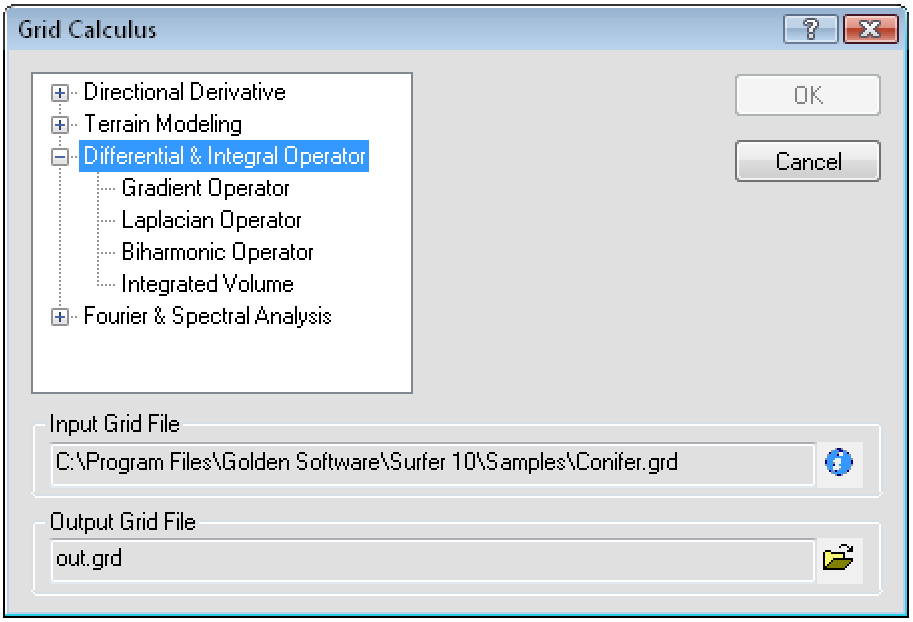
The mathematical definitions used in this implementation were taken from Mitasova and

Mitasova, Helena and

Moore, I. D., A. Lewis, and

Differential and Integral Operators

The Grids | Calculate | Calculus *Differential and Integral Operators* include the *Gradient Operator*, the *Laplacian Operator*, the *Biharmonic Operator*, and *Integrated Volume*.



Select differential and integral operators from the list on the left side of the ***Grid Calculus*** *dialog. There are no options for Differential & Integral Operators.*

Gradient Operator
Laplacian Operator
Biharmonic Operator
Integrated Volume

Gradient Operator

The Grids | Calculate | Calculus differential and integral operator, *Gradient Operator*, generates a grid of steepest slopes (i.e. the magnitude of the gradient) at any point on the surface. This is similar to the Terrain Slope operation, but the *Gradient Operator* is reported as a number (rise over run) rather than in degrees, and the direction is opposite that of the slope. The *Gradient Operator* is zero for a horizontal surface, and approaches infinity as the slope approaches vertical.

The definition of the gradient yields the following equation:

$$\|\vec{g}\| = \sqrt{\left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2}$$

Using the compass based grid notation yields the equation used in **Surfer**:

$$\|\vec{g}\| \approx \sqrt{\left(\frac{Z_E - Z_W}{2\Delta x}\right)^2 + \left(\frac{Z_N - Z_S}{2\Delta y}\right)^2}$$

Laplacian Operator

The Grids | Calculate | Calculus differential and integral operator, *Laplacian Operator*, provides a measure of discharge or recharge on a surface. In grid files generated with the *Laplacian Operator* recharge areas are positive and discharge areas are negative.

Groundwater, heat, and electrical charge are three examples of conservative physical quantities whose local flow rate is proportional to the local gradient. The Laplacian operator, $\nabla^2 Z$, is the mathematical tool that quantifies the net flow into (Laplacian > 0, or areas of recharge) or out of (Laplacian < 0, areas of discharge) a local control volume in such physical situations. The Laplacian operator is defined in multivariable calculus by.

$$\nabla^2 Z = \frac{\partial^2 Z}{\partial x^2} + \frac{\partial^2 Z}{\partial y^2}$$

The **Surfer** implementation of the *Laplacian Operator* generates a grid using a standard five-point central difference formula.

$$\nabla^2 Z(x,y) = \left(\frac{Z_E - 2Z + Z_W}{\Delta x^2}\right) + \left(\frac{Z_N - 2Z + Z_S}{\Delta y^2}\right)$$

Biharmonic Operator

Bending of thin plates and shells, viscous flow in porous media, and stress functions in linear elasticity are three examples of physical quantities that can be mathematically described using the Grids | Calculate | Calculus differential and integral operator,*Biharmonic Operator*. The *Biharmonic Operator*, $\nabla^4 Z$, is defined in multivariable calculus by

$$\nabla^4 Z = \frac{\partial^4 Z}{\partial x^4} + 2\frac{\partial^4 Z}{\partial x^2 \partial y^2} + \frac{\partial^4 Z}{\partial y^4}$$

This is comparable to applying the *Laplacian Operator* twice.

Integrated Volume

The Grids | Calculate | Calculus differential and integral operator, *Integrated Volume*, gives the accumulated volume over the grid from the southwest to the northeast corner, or over a subgrid from southwest to northeast. This method can calculate volumes over

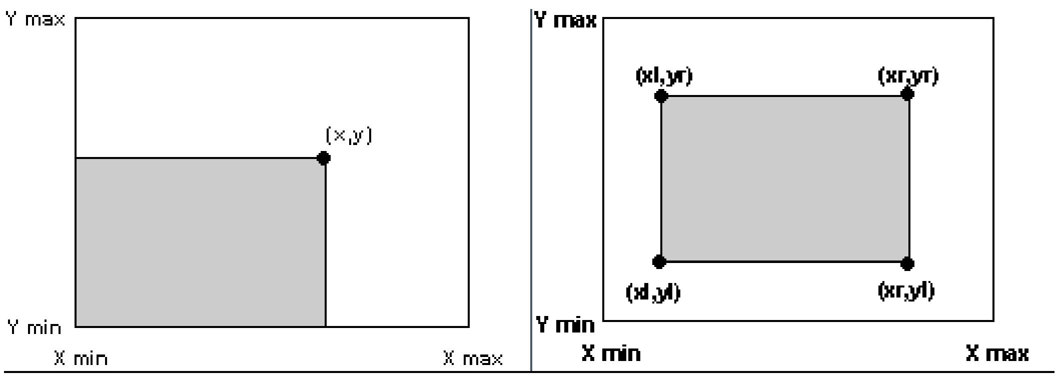
Mathematically, the integrated volume is defined by

$$V(x, y) = \int_{\alpha=x\min}^{x} \int_{\beta=y\min}^{y} z(\alpha, \beta)\, d\alpha\, d\beta$$

Consider a surface defined over a rectangular region extending from X min to X max and Y min to Y max:

V(x,

The *Integrated Volume* option enables easy computation of volumes under rectangular subgrids. For example, the volume under the shaded area in the figure below is simply V(xr,



The integral of the bilinear form over the grid cell is the average value of the four corners times the area of the grid cell. Therefore, the underlying formula is

$$V(x_i, y_i) = \frac{Z(x_i, y_i) + Z(x_{i-1}, y_i) + Z(x_i, y_{i-1}) + Z(x_{i-1}, y_{i-1})}{4} +$$

$$V(x_{i-1}, y_i) + V(x_i, y_{i-1}) - V(x_{i-1}, y_{i-1})$$

Fourier and Spectral Analysis

The correlogram and periodogram are used in a variety of fields, including hydrology, agriculture, forestry, meteorology, ecology, and sociology.

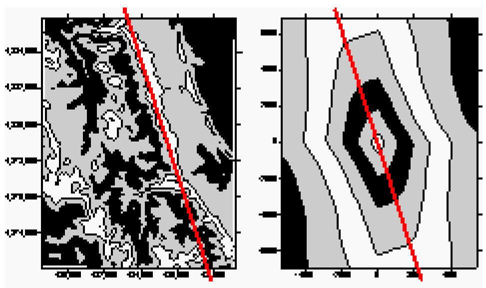
Example

See, for example, Ripley (1981, Chapter 5), and Bras and Rodriguez-Iturbe (1985, Chapter 6). The correlogram assesses spatial patterns and spatial correlation, while the periodogram identifies periodicities in the data.

Correlogram

The Grids | Calculate | Calculus Fourier and spectral analysis, *Grid Correlogram*, assesses spatial patterns and spatial correlation of a grid. Correlograms indicate how well grid values correlate across the grid. They indicate underlying trends in the grid and give a measure of the anisotropy of the grid.

Correlograms are symmetric; Z(x,



A contour map of a DEM is shown on the left and the grid correlogram is shown on the right. The heavy line illustrates the approximate directions of anistropy on both maps.

Given gridded data z(u,v), where {u = 0, ..., NX-1} and {v = 0, ..., NY-1} the grid auto-covariance,

$$C(r,s) = \frac{1}{NX \cdot NY} \sum_u \sum_v \left[z(u,v) - \bar{z} \right] \left[z(u+r, v+s) - \bar{z} \right]$$

where the summation is over *u* and *v* such that

$$0 \le u \le NX - 1 \qquad 0 \le u + r \le NX - 1$$
$$0 \le v \le NX - 1 \qquad 0 \le v + s \le NX - 1$$

and $\bar{z}$ is the arithmetic average of the grid values:

$$\bar{z} = \frac{1}{NX \cdot NY} \sum_u \sum_v z(u,v)$$

Note that the grid auto-covariance function is itself a grid.

The grid auto-covariance is the two-dimensional lattice equivalent of the time series auto-covariance function. It is comparable to the two-dimensional variogram; under the appropriate assumptions of statistical stationarity, there is a one-to-one relationship between the grid auto-covariance function and the variogram. (Beware, however, that the variogram of a raw data set - prior to gridding - is not the same as the variogram of the resulting grid.)

The grid correlogram, *corr(r,s)*, is a scaled version of the grid auto-covariance function:

$$\mathrm{corr}(r,s) = \frac{C(r,s)}{C(0,0)}$$

C(0,0) is the variance of the data in the grid.

The functions C(r,s) and

$$C(-r,-s) = C(r,s)$$

but,

$$C(-r,s) \neq C(r,s)$$

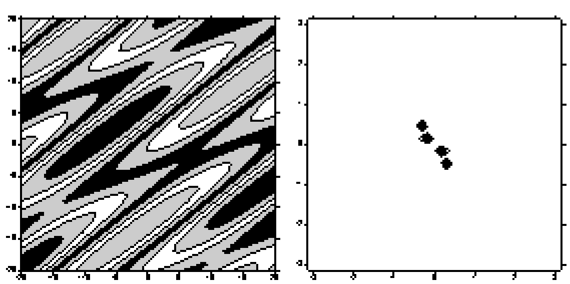
in general. As such, the grids generated by the grid correlogram option of the **Grids | Calculate | Calculus** sub-system show this symmetry.

Periodogram

The Grids | Calculate | Calculus Fourier and spectral analysis, *Grid Periodogram*, is a decomposition of the surface into the weighted sum of many two-dimensional sinusoids. This operation shows hidden periodicity or repeating patterns that might not otherwise be evident from looking at a contour map or surface.

Surfercomputes the grid periodogram for the two-dimensional *Fourier* frequencies. Grid periodograms have the same dimensions as the parent grids, but the axes represent the wave numbers associated with the *Fourier* frequencies. The grid periodogram surfaces are symmetric in the sense that Z(x,

In the current implementation of the grid periodogram there is no smoothing, no tapering, and no filtering. Such post-processing can be accomplished using the Grids | Calculate | Math capabilities.



A contour map of the function Z = sin ((x/5)-(y/7)) + cos((y/2)-(x/3)) *is on the left, and the periodogram contour map is on the right. Due to symmetry, the four apparent peaks represent two peaks, one for each of the two fundamental sinusoids.*

Given gridded data z(u,v), where {u = 0, ..., NX-1} and {v = 0, ..., NY-1} the grid periodogram,

$$I(\lambda, \mu) = \frac{N}{4\pi^2}\left|\frac{1}{N}\sum_u\sum_v\left[z(u,v) - \bar{z}\right]e^{-i\lambda u}e^{-i\mu v}\right|^2$$

where $i = \sqrt{-1}$.

The value of I(l,) is the weight or strength of the sinusoid with a two dimensional frequency of (,).

Breaking this calculation into the component real and imaginary parts yields:

$$A(\lambda, \mu) = \frac{1}{N}\sum_u\sum_v\left[z(u,v) - \bar{z}\right]\cos(\lambda r + \mu s)$$

$$B(\lambda, \mu) = \frac{1}{N}\sum_u\sum_v\left[z(u,v) - \bar{z}\right]\sin(\lambda r + \mu s)$$

Then

$$I(\lambda, \mu) = \frac{N}{4\pi^2}\cdot\left[A(\lambda, \mu)^2 + B(\lambda, \mu)^2\right]$$

where

$$-\pi < \lambda \le +\pi$$
$$-\pi < \mu \le +\pi$$

Surfer does not compute the grid periodogram, I(l,), for all possible frequencies (l,m) in this range. Rather, it computes the grid periodogram for the two-dimensionalFourier frequencies.

A two-dimensional frequency is a *Fourier frequency* if λ is an integer multiple of $2\pi/NX$, and μ is an integer multiple of $2\pi/NY$. In **Surfer** we consider the specific frequencies:

$$\lambda = \frac{2\pi j}{NX} \quad j = -\left\lfloor\frac{NX}{2}\right\rfloor, \ldots, -1, 0, 1, \ldots, \left\lfloor\frac{NX}{2}\right\rfloor$$

$$\mu = \frac{2\pi k}{NY} \quad k = -\left\lfloor\frac{NY}{2}\right\rfloor, \ldots, -1, 0, 1, \ldots, \left\lfloor\frac{NY}{2}\right\rfloor$$

The indices (j, k) are called the wave number of the associated frequency (l,m). **Surfer** follows the engineering convention (rather than the mathematicians' convention) of plotting the grid periodogram as a function of the wave numbers rather than as a function of the frequency:

wave number = 2π / wavelength (e.g. radians/meter)

The maximum wave number is the Nyquist frequency =p/

Volumes and Areas

The **Grids | Calculate | Volume** command is used to compute net volumes, cut and fill volumes, planar areas, and surface areas. The computation results are displayed in the grid volume report. The results can be saved as an ASCII text file, in Rich Text Format, or can be pasted into the clipboard.

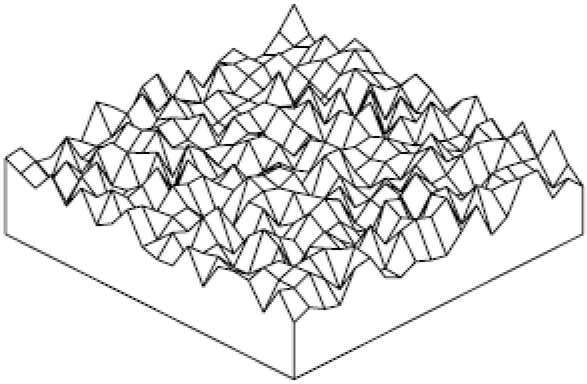
Volume calculations are performed on solids defined by an upper and lower surface or 3D polygons. The upper and lower surfaces are defined by a grid file or a plane of constant Z level. The upper surface does not have to be above the lower surface at all points; the upper surface may dip below the lower surfaces in some locations. NoData regions on either the upper or the lower surface are excluded from consideration during the volume calculations.

If the upper and lower surface grid files do not have the same number of rows and columns or the same XY limits, the selected master file determines the geometry for the volume calculations. Surfer matches the input grids by their XY coordinates and performs the calculations on the overlapping nodes.

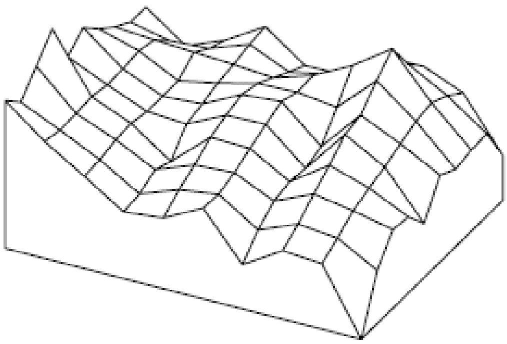
Volume results are provided in cubic units based on the units of the input grid file.

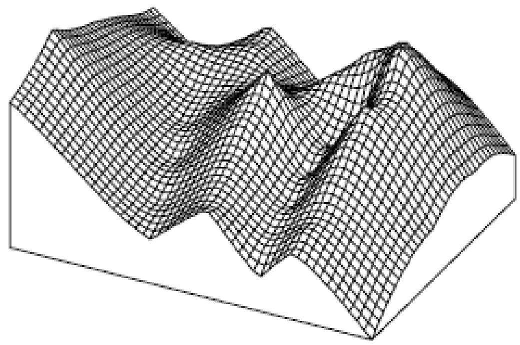
Volume Accuracy Example

Volume calculations are generated for each grid cell. In areas where the surface is tilted at the top or bottom of a grid cell, **Surfer** approximates the volume of the prism at the top or bottom of the grid cell column. For very coarse grids, the prisms can contain a significant volume. Volume calculations become more accurate as the density of the grid is increased because the relative size of the prisms is reduced compared to the size of the associated column. Grids with significant "noise" or with highly irregular surfaces are poor candidates for volume calculations.



This grid has a highly irregular surface and is a poor candidate for volume calculations.





These two grids are based on the same data but created at different grid densities. The grid on the right is a better candidate for obtaining accurate volume information because it is a much more smooth and dense grid.

To Calculate Volumes and Areas

Use the **Grids | Calculate | Volume** command to calculate net volumes, cut and fill volumes, planar areas, surface areas, and 3D polygon volumes.

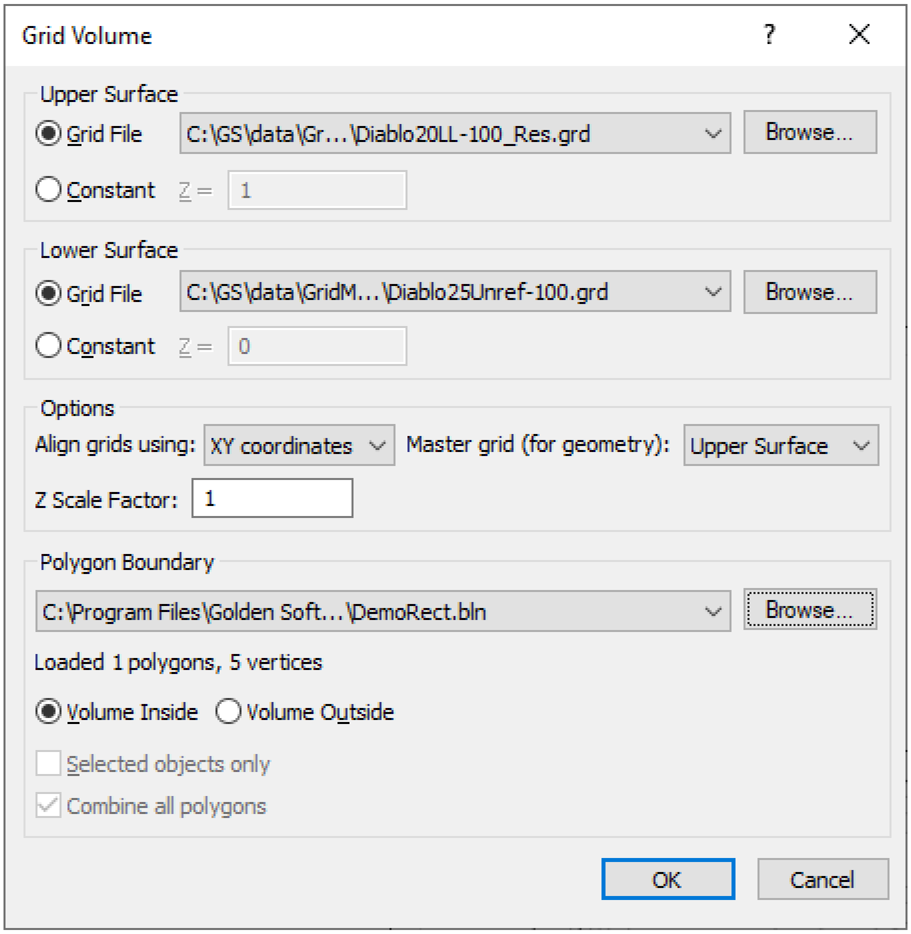
1. Click the **Grids | Calculate | Volume** command or the button.
2. Specify the *Upper Surface* and *Lower Surface* parameters, or for 3D polygon volumes specify a file in *Upper Surface* and a file for *Polygon Boundary*. See the **Grid Volume Dialog** section below for instructions on *Polygon Boundary*.
    - The *Grid File* option is used to specify a grid to use as the upper or lower surface. Specify the source map layer or grid file in the *Grid File* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Grid File* list. Click *Browse* to load a grid file with the Open Grid dialog.
    - The *Constant* option is used to specify the level of the planar surface to use as the upper or lower surface. Specify the level of the planar surface by entering the value into the *Z* = box. The specified value is in Z data units.

3. In the **Options** section *Align grids using* list, select *Node position* if the upper and lower grid files have the same number of rows and columns or select *XY coordinates* if they do not. If *XY coordinates* was selected, also select a value from the *Master grid (for geometry)* list. Select either the upper or lower grid file to be the base geometry for the calculation.
4. If the X, Y, and Z units are different, the resulting volume calculations are meaningless. If, for example, your XY values are in meters but your Z values are in feet, the volume results are square meters times feet, rather than cubic units. In the **Options** section, *Z Scale Factor* can be used to adjust for this problem. In this example, setting the *Z Scale Factor* to 0.3048 (number of meters in a foot) results in cubic meters for volume calculations. You will need to know the conversion factor to use this field.
5. Click *OK* in the **Grid Volume** dialog and the results are displayed in a report.
6. Click **File | Save As** to save the report, or you can copy the information to the clipboard and paste it into another application.

Use the **Grids | Calculate | Volume** command to calculate net volumes, cut and fill volumes, planar areas, and surface areas.

Grid Volume Dialog

Click the **Grids | Calculate | Volume** command to open the **Grid Volume** dialog.



*Specify grid volume options in the **Grid Volume** dialog.*

Upper Surface and Lower Surface

Specify the upper surface in the *Upper Surface* section, and specify the lower surface in the *Lower Surface* section. The surfaces can be grid files or constant planar surfaces.

Grid File

The *Grid File* option is used to specify a grid to use as the upper or lower surface. Specify the source map layer or grid file in the *Grid File* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Grid File* list. Map layers will be displayed from all open plot documents. Click *Browse* to load a grid file with the Open Grid dialog.

Constant Z

The *Constant* option is used to specify the level of the planar surface to use as the upper or lower surface. Specify the level of the planar surface by entering the value into the *Z* = box. The specified value is in Z data units.

Align Grids

If the input grid files have the same number of rows and columns, select the *Node position* value from the *Align grids using* list. The volume calculation will be performed in equivalent grid nodes, regardless of the X and Y values. If the input grid files do not have the same number of rows and columns, select the *XY coordinates* value from the *Align grids using* list to match the input grid files on X and Y values.

If *Align grids using* is set to *XY coordinates* and all grids are referenced but in a different coordinate system, the calculation uses the master grid's coordinate system. If an input grid files is not referenced but another grid file is referenced, Surfer displays a warning about mixing referenced and unreferenced grid files. Selecting *Yes* continues with the volume calculation, but none of the other grids will be projected to the master grid's coordinate system. If the grids are in the same coordinate system, selecting *Yes* will perform the calculation as expected. However, if a grid is not referenced and not in the same coordinate system, select *No* at the warning to cancel the calculation and return to the **Grid Volume** dialog. Select different grid files with referenced or matching coordinate systems or use the **Grids | Edit | Assign Coordinate System** to change the coordinate system of one or more of the grid files.

Master Grid

From the *Master grid (for geometry)* list, select *Upper Surface* or *Lower Surface* as the master grid to use for the base geometry for the volume calculation.

Z Scale Factor

If the X, Y, and Z units are different, the resulting volume calculations are meaningless. If, for example, your X, Y values are in meters but your Z values are in feet, the volume results are square meters times feet, rather than cubic units. *Z Scale Factor* can be used to adjust for this problem. In this example, setting the *Z Scale Factor* to 0.3048 (number of meters in a foot) results in cubic meters for volume calculations. You will need to know the conversion factor to use this field.

A grid with X, Y, and Z units in feet, the units for the results are: Net Volume = (feet * feet * feet) or ft^3.

Polygon Boundary

Calculate the volume inside or outside a specific region by specifying a *Polygon Boundary* map layer or vector file. Specify the source map layer or vector file in the *Polygon Boundary* section. Click the current selection and select a base layer from the list. Base layers will be displayed from all open documents. Click *Browse* to load a vector file with the Open dialog. When the base layer or vector file includes more than one polygon/3D polygon, the volume is calculated inside or outside all of the polygons/3D polygons in the layer or file. The number of polygons and number of vertices in the base layer or vector file is displayed below the *Polygon Boundary* field.

The following vector file types are supported in the *Polygon Boundary* field:

BLN Golden Software Blanking	GSB Golden Software Boundary
BNA Atlas Boundary	GSI Golden Software Interchange
DDF SDTS TVP	KML Google Earth
DLG USGS Digital Line Graph	MIF MapInfo Interchange Format
DXF AutoCAD Drawing	SHP Esri Shapefile
E00 Esri ArcInfo Export Format	WMF Windows Metafile
EMF Windows Enhanced Metafile	

Volume Inside or Volume Outside

Click *Volume Inside* to calculate the volume inside of the polygon(s) or 3D polygon(s) in the base layer or vector file. Click *Volume Outside* to calculate the volume outside of the polygon(s) or 3D polygon(s) in the base layer or vector file.

Selected Objects Only

Check the *Selected objects only* check box to use only the selected objects in the base layer to calculate volumes. The *Loaded polygons* and *vertices* values are updated when the *Selected objects only* box is checked. Select a base layer in the *Polygon Boundary* field to use the *Selected objects only* option. The *Selected objects only* option is not available when the *Polygon Boundary* is a vector file. The polygon or polygons must be selected before clicking the **Grids | Calculate | Volume** command.

Combine All Polygons

Select *Combine all polygons* to add all the statistics for the total number of polygons. If there is only one polygon in the map, the box is checked by default and *Combine all polygons* is not active. When *Combine all polygons* is unchecked, the report presents statistics for each individual polygon.

Grid Report

Click *OK* in the **Grid Volume** dialog and the results are displayed in the Grid Volume Report. Use **File | Save As** to save the report, or you can copy the information to the clipboard and paste it into another application.

Introduction to Volumes and Areas

The procedures described here are used to provide more information about the data contained in grid files. The topics discussed include volumes and areas.

The Grids | Calculate | Volume command calculates the volume and areas for the surfaces defined by grid files or a grid file and a horizontal plane. The **Grids | Calculate | Volume** computations contain the following information:

- Net volume between two grid files
- Net volume between a grid file and a horizontal plane
- Cut and fill volumes
- Surface area of a grid file above and below a specified level
- Planar area of a grid above and below a specified level
- volumes inside and outside of 3D polygons.

Grid Volume Report

When Grids | Calculate | Volume computations are performed, the results are displayed in the Grid Volume Report.

The volume computations in the **Grid Volume Report** dialog include the following:

Upper Surface and Lower Surface

These sections display the parameters that define the upper and lower surfaces.

Polygon Boundary

The *Polygon Boundary* section displays information about any polygon boundaries used in the volume calculation. The *File Name* shows the file path and name for the base layer or vector file used to specify the boundary or boundaries. The *Number of Polygons* value indicates the number of polygons used from the base layer or vector file. The *Volume* describes whether volume is calculated inside or outside the polygon boundaries.

Volumes

The volume is calculated by three methods: Extended Trapezoidal Rule, Extended Simpson's Rule, and Extended Simpson's 3/8 Rule. The reported volume is the sum of the Positive Volume (Cut) and Negative Volume (Fill). The *Z Scale Factor* is also reported in this section.

The *Positive Volume (Cut)* is the volume of material in those places where the upper surface is above the lower surface. The *Negative Volume (Fill)* is the volume of material in those places where the upper surface is below the lower surface. The *Cut Minus Fill* is the difference between the cut and fill volumes. See Cut and Fill Volumes for more information on cut and fill.

Areas

The *Areas* section reports both planar areas and surface areas. The *Positive Planar Area* represents the planar area where the upper surface is above the lower surface. The *Negative Planar Area* represents the planar area where the upper surface is below the lower surface. The *NoData Planar Area* is the sum of the areas over the NoData regions on both the upper and lower surfaces. The *Total Planar Area* represents the planar area for the entire grid.

The *Positive Surface Area* represents the area of the surface where the upper surface is above the lower surface. The *Negative Surface Area* represents the area of the surface where the upper surface is below the lower surface. Where two surfaces coincide exactly, the area of coincidence is reported as part of the *Positive Planar Area*.

Net Volume

The volume calculation determines the net volume between the upper and lower surface. The net volume is reported in the *Volumes* section of the Grid Volume Report. See Cut and Fill Volumes for more information on the upper and lower surface.

To visualize net volume, consider a construction site where the topography must be graded to a flat surface prior to the beginning of construction. The upper surface represents the current topography, and the lower surface represents the final graded site elevations. In some places, cut must be made into the current topography to remove earth to the level of the final site. In other areas, earth may be needed to fill in areas where the current topography is below the elevation of the final site. The net volume is the difference between all cuts and all fills. If the volume is positive, earth needs to be removed from the site to achieve the final level. If the volume is negative, earth needs to be hauled into the site to achieve the final planned grade for the site.

Three methods are used to determine volumes. **Surfer** approximates the necessary one-dimensional integrals using three classical numerical integration algorithms: Extended Trapezoidal Rule, Extended Simpson's Rule, and Extended Simpson's 3/8 Rule; see Press et al., 1988, Section [4.1]. The difference in the volume calculations by the three different methods measures the accuracy of the volume calculations. If the three volume calculations are reasonably close together, the true volume is close to these values. If the three values differ somewhat, a new denser grid file should be used before performing the volume calculations again. The net volume can be reported as the average of the three values.

Press, W.H., Flannery, B.P., Teukolsky, S.A., and Vetterling, W.T. (1988), Numerical Recipes in C, Cambridge University Press.

Mathematically, the volume under a function $f(x,y)$ is defined by a double integral:

$$\text{Volume} = \int_{X_{min}}^{X_{max}} \int_{Y_{min}}^{Y_{max}} f(x,y)\,dx\,dy$$

In **Surfer**, this is computed by first integrating over X (the columns) to get the areas under the individual rows, and then integrating over Y (the rows) to get the final volume. This approach is explained in detail in Press, et al., 1988, Section [4.6].

Units

When the X, Y, and Z units are the same, volume results are given in cubic units. A common situation is a grid with X, Y in latitude and longitude and Z in meters. The grid should be converted to X, Y in meters to create useful results in $(meter)^3$. Use the Project command to project the grid to a different coordinate system. Alternatively if the source data is available, you can first project the X and Y columns in a worksheet. Next create a new grid with the projected X and Y values.

Example 1

A grid with X, Y, and Z units in feet, the units for the results are $(feet)^3$.

Net Volume = (feet * feet * feet)

Example 2

A grid with X, Y, and Z units in meters, the units for the results are $(meter)^3$.

Net Volume = (meter * meter * meter)

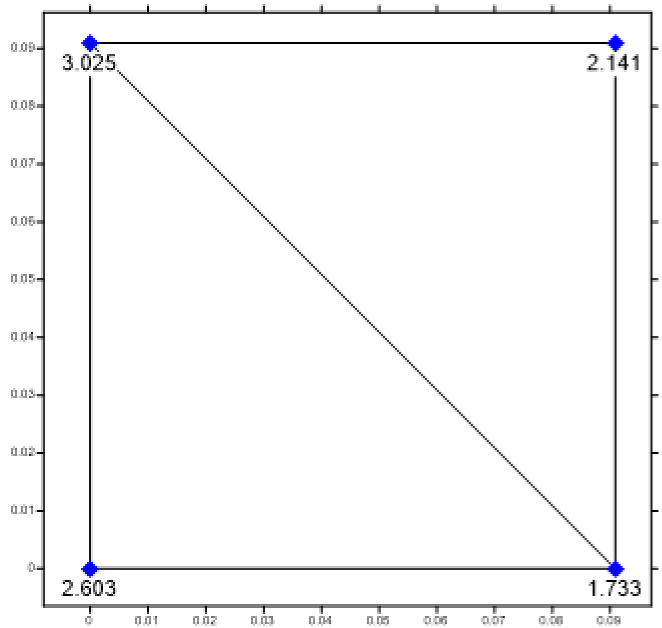
Cut and Fill Volumes

The *Cut & Fill Volumes* section of the Grid Volume Report displays the *Positive Volume (Cut)* and the *Negative Volume (Fill)*. The cut portion is the volume between the upper and lower surface when the upper surface is above the lower surface. The fill portion is the volume between the upper and lower surfaces when the upper surface is below the lower surface.

Calculating Cut and Fill Volumes

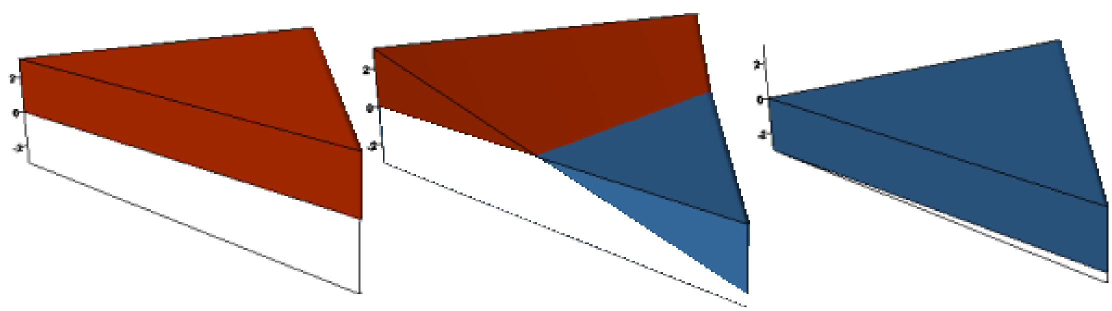
The *Cut & Fill Volumes* section uses a different calculation technique than the three methods used to define the volume, Extended Trapezoidal Rule, Extended Simpson's Rule, and Extended Simpson's 3/8 Rule. For determining cut and fill volumes, first the lower surface grid values are subtracted from the upper surface grid values. This normalizes the Z values where positive values indicate the upper surface is above the lower surface and negative values indicate the upper surface is below the lower surface.

Next each grid cell is split into two triangles. For example the grid cell defined by the first row and first column of the grid file would be split into the following triangles in grid coordinates (row, column): one triangle is Z(0,0), Z(1,0), and Z(0,1) while the other is Z(1,0), Z(0,1), and Z(1,1).



The grid cell is split into two triangles.

The triangles are compared to a horizontal plane at Z=0. This generates triangular prisms between the calculated surface and the horizontal plane. Some prisms are completely above the horizontal plane. These prisms occur where all three Z values for the upper surface are greater than the values for the lower surface. This leads to a positive volume or cut. Some prisms are completely below the horizontal plane. These prisms occur where all three Z values for the upper surface are less than the values for the lower surface. This leads to a negative volume or fill. Finally some prisms intersect the horizontal plane. This leads to one prism with a quadrilateral base and one prism with a triangular base, where one prism is a positive (cut) volume and one is a negative (fill) volume. The quadrilateral prism is between two points on the same side of the Z=0 plane. The triangular prism is opposite the quadrilateral prism.



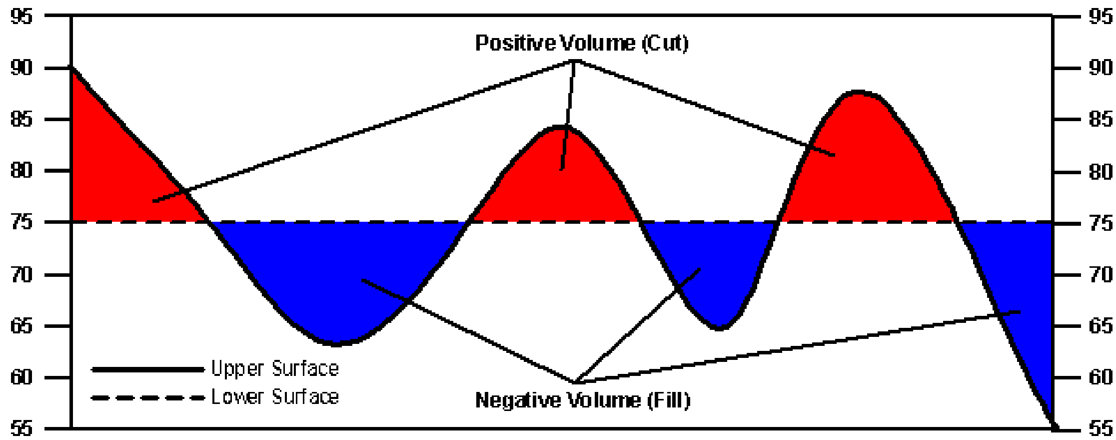
Prisms are created from the net grid values and the horizontal planar surface at Z=0. The red prisms above are positive (cut) volumes. The blue prisms are negative (fill) volumes.

Finally, the volumes are calculated for all the positive (cut) prisms and summed, and the volumes are calculated for all the negative (fill) prisms and summed. Finally, the net volume is the difference between the cut and fill volumes.

NoData regions on either the upper or lower surface are excluded from consideration during the cut and fill calculations.

Net Volume

The net volume is the positive, or cut, volume minus the negative, or fill, volume. To visualize net volume, consider a construction site where the topography must be graded to a flat surface prior to the beginning of construction. The upper surface represents the current topography, and the lower surface represents the final graded site elevations. In some places, cut must be made into the current topography to remove earth to the level of the final site. In other areas, earth may be needed to fill in areas where the current topography is below the elevation of the final site. The net volume is the difference between all cuts and all fills. If the volume is positive, earth needs to be removed from the site to achieve the final level. If the volume is negative, earth needs to be hauled into the site to achieve the final planned grade for the site.



Cross-section showing the relation between the upper and lower surfaces, and the cut and fill volumes. The lower surface is defined by Z=75.

Example

You can determine the volume of a pond for which you have generated a grid file. You can define the grid file of the pond bottom as the upper surface, and use a plane set at the level of the surface of the pond as the lower surface. The volume of the pond is the *Negative Volume (Fill)* in the *Cut & Fill Volumes* section of the *Grid Volume Report*.

Extended Trapezoidal Rule

The pattern of the coefficients is {1,2,2,2,...,2,2,1}:

$$A_i = \frac{\Delta x}{2}\left[G_{i,1} + 2G_{i,2} + 2G_{i,3} \cdots + 2G_{i,nCol-1} + G_{i,nCol}\right]$$

$$\text{Volume} \approx \frac{\Delta y}{2}\left[A_1 + 2A_2 + 2A_3 + \cdots + 2A_{nCol-1} + A_{nCol}\right]$$

where:

Δx is the grid column spacing

Δy is the grid row spacing

$G_{i,j}$ is the grid node value in row *i* and column *j*.

Extended Simpson's Rule

The pattern of the coefficients is {1,4,2,4,2,4,2,...,4,2,1}:

$$A_i = \frac{\Delta x}{3}\left[G_{i,1} + 4G_{i,2} + 2G_{i,3} + 4G_{i,4} + \cdots + 2G_{i,nCol-1} + G_{i,nCol}\right]$$

$$\text{Volume} \approx \frac{\Delta y}{3}\left[A_1 + 4A_2 + 2A_3 + 4A_3 + \cdots + 2A_{nCol-1} + A_{nCol}\right]$$

where:

Δx is the grid column spacing

Δy is the grid row spacing

$G_{i,j}$ is the grid node value in row *i* and column *j*.

Extended Simpson's 3/8 Rule

The pattern of the coefficients is {1,3,3,2,3,3,2,...,3,3,2,1}:

$$A_i = \frac{3\Delta x}{8}\left[G_{i,1} + 3G_{i,2} + 3G_{i,3} + 2G_{i,4} + \cdots + 2G_{i,nCol-1} + G_{i,nCol}\right]$$

$$\text{Volume} \approx \frac{3\Delta y}{8}\left[A_1 + 3A_2 + 3A_3 + 2A_3 + \cdots + 2A_{nCol-1} + A_{nCol}\right]$$

where:

Δx is the grid column spacing

Δy is the grid row spacing

$G_{i,j}$ is the grid node value in row *i* and column *j*.

Planar Area Calculations

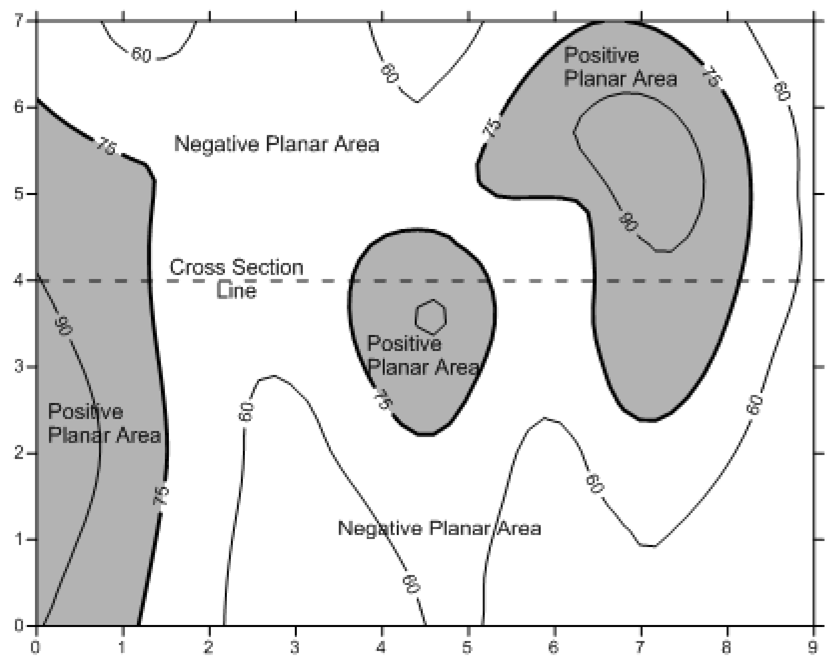
Planar area is computed by projecting the cut and fill portions of the surface onto a plane and calculating the area of the projection.

Positive Planar Area

Positive Planar Area represents the projection of the cut (map areas where the upper surface is above the lower surface) onto a horizontal plane.

Negative Planar Area

Negative Planar Area represents the projection of the fill (map areas where the upper surface is below the lower surface) onto a horizontal plane.



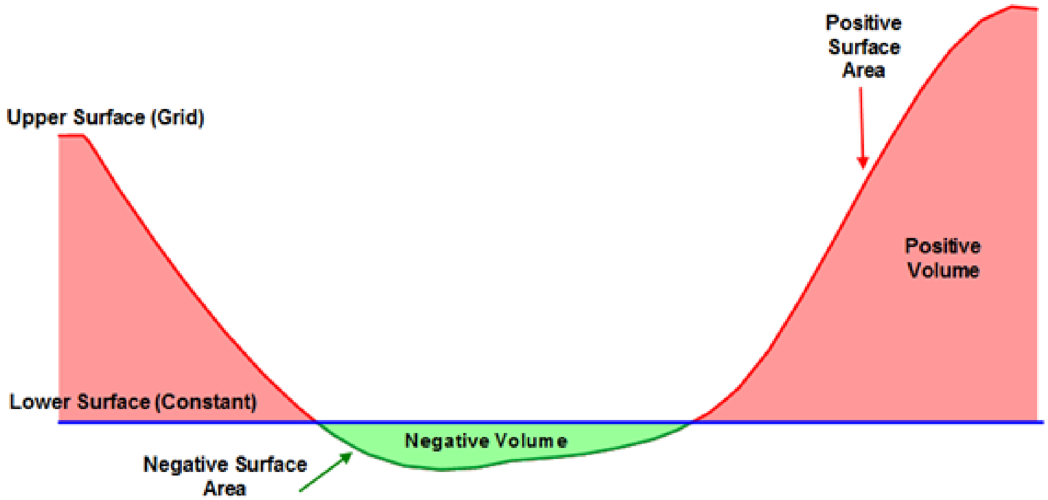
This is a contour map showing the relation between positive and negative planar area on a map. The lower surface is defined by a plane where Z = 75.

Surface Area Calculations

The surface area is the actual area of the surface. *Positive Surface Area* corresponds to the area of the surface where the upper surface is above the lower surface. *Negative Surface Area* corresponds to the area of the surface where the upper surface is below the lower surface.

Example

As an example, consider a grid file representing the bottom of a pond, with the water level at a Z value of zero. If the grid representing the pond bottom is defined as the upper surface and Z = 0 is defined as the lower surface, you can determine the surface area of the bottom of the pond. Suppose you want to drain the pond and line it with plastic. You need to know the area of the plastic necessary to line the pond bottom. In this case, the negative surface area represents the surface of the pond bottom. This would tell us precisely the amount of plastic necessary to line the pond bottom.



The green line displays the negative surface area of the pond. This value determines the amount of plastic needed to line the pond bottom in the example.

Calculating the Volume between Two Surfaces

A common problem solved with a volume calculation is determining the volume of material between two grid files. For example, you might have drill hole data that provides the elevation of the top and bottom of a coal seam, and you want to determine the volume of coal present in the seam. The elevation of the top and bottom of the coal seam defines the upper and lower surfaces, respectively. In this case, the lower surface would be entirely below the upper surface.

To find the volume between two surfaces:

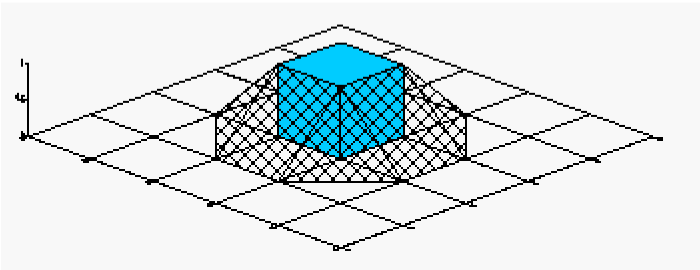
1. Create grid files of the upper and lower surfaces. In the example here, the grid files would define the elevation of the top and bottom of the coal seam. Both grids must use the same dimensions and the same number of rows and columns.
2. Choose the Grids | Calculate | Volume command and select the upper surface grid file in the **Open Grid** dialog and then click the *Open* button.
3. In the **Grid Volume** dialog, the *Grid File* option button should be selected in the *Upper Surface* group.
4. In the *Lower Surface* group, click the *Grid File* option button and click the button.

5. In the **Open Grid** dialog, select the lower surface grid file and click the *Open* button.
6. In the **Grid Volume** dialog, verify that the correct grid files are specified for the upper and lower surfaces and click *OK*. The Grid Volume Report is generated. The volume between the surfaces is reported as the *Positive Volume (Cut)*, which, in this case, is the same as the *Volume* calculation.
7. To save the volume information, use the **File | Save As** command in the report window and specify a name and file extension. Click *OK* and the information is saved.

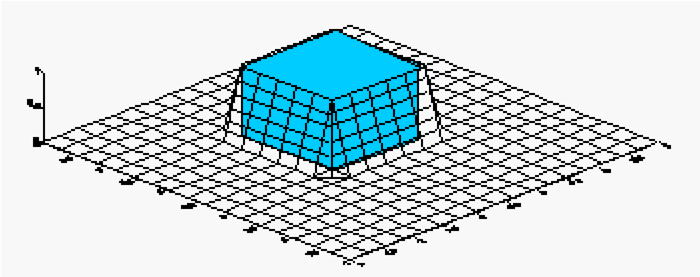
Calculating the Volume of a Grid with NoData Values

Surfer can be used to calculate the volume between two grid surfaces or between a grid surface and a horizontal plane. The accuracy of the volume calculation increases as the number of grid nodes in the grid file increases. When one of the grid surfaces contains NoData values, or the area of interest is a subset of the entire grid, additional error is introduced. This error can be minimized by specifying more grid nodes during the gridding operation.

Consider the way **Surfer** calculates the volume of a cube that is 1 foot x 1 foot x 1 foot. **Surfer** cannot generate a surface that is a perfect cube since each XY coordinate can only have a single Z value. A diagonal line is drawn from the top surface to the lower with a width of one grid unit. Using a grid with a one-foot grid line spacing, the resulting surface looks like a pyramid with the top chopped off.



A cube with a coarse grid. Volume error is increased.



A cube with a finer grid. Volume error is reduced.

Calculating the Volume of a Lake

To illustrate the capability of using the Grids | Calculate | Volume command, consider the application of calculating the volume of water within a lake. This application requires that a grid file be created which defines the lake bottom. The other surface is defined as a horizontal planar surface that defines the water surface of the lake.

To calculate the volume of a lake:

1. Create the grid file for the lake. If the lake bottom is defined as the depth of the water, use negative Z values so that the lake appears as a basin rather than a hill. It is also recommended that data points be added beyond the limits of the lake shore so that the surface is above the level of the lake in those areas.

2. Select the **Grids | Calculate | Volume** command.

3. Specify the grid file for the lake in the **Open Grid** dialog and click *Open* .

4. In the **Grid Volume** dialog, click the *Grid File* option in the *Lower Surface* group.

5. In the *Upper Surface* group, click the *Constant* option. Enter the value for the lake surface into the *Z* = box. For example, if you are using depth in your grid file as a negative value, the *Z* = value for your horizontal (lake) surface should be zero.

6. Click *OK* and the volume and area calculations are reported. The volume of the lake is reported as the *Positive Volume*.

7. To save the information, use the **File | Save As** command in the Grid Volume Report window and specify a name and file extension.

Grid Math

The **Grids | Calculate | Math** command creates a new grid file that transforms the Z values of a single grid file or combines Z values from multiple grid files. The output grid file is based on a mathematical function of the form *f(A, B, C, D)*, where *A, B, C, D* represent input grid files.

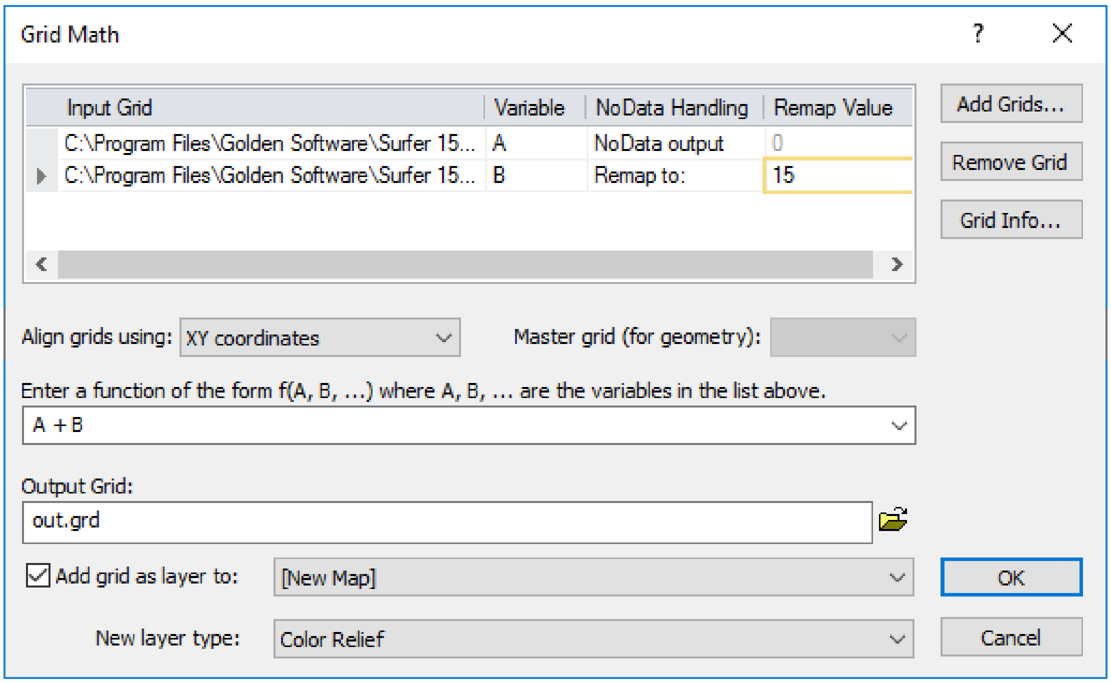
A defined math function can be performed by node position on corresponding nodes with the same row and column coordinates from the input grid files. The result of the calculation is placed at the same coordinates in the output grid file. For example, the function *log10(A)* creates an output grid file with the logarithmic base 10 values for each of the grid nodes in Grid A.

A math function can also be performed on grid files that do not have the same number of rows and columns. **Surfer** matches the grid files' X and Y values and then performs calculations on those overlapping values. If needed, input grids are projected to the master grid's coordinate system and then resampled to the master grid's geometry.

By default, if a node contains the NoData value in any input grid, it is assigned the NoData value in the output grid. However, you can change the *NoData Handling* for each grid so that NoData nodes are assigned a value. The output grid file will then have a value when the input grid contains the NoData value.

Grid Math Dialog

Click the **Grids | Calculate | Math** command to open the **Grid Math** dialog.



*Specify grid files and define a math function in the **Grid Math** dialog.*

Input Grid

The *Input Grid* column displays the selected grid files. The X and Y limits and grid size of these files define the output grid. Place the mouse over any grid file to reveal the full path and file name for the grid.

Variable

By default, **Surfer** defines each grid with a *Variable* name letter. The first grid is *A*, the second grid is *B*, and so on. To change the *Variable* name for a grid, click in the *Variable* name box next to the current *Variable* name. Delete the existing text and type a new name. The variable name is case insensitive, so *G1B* is the same as *g1b*. *Variable* names must start with an underscore (_) or letter and can only contain alphanumeric characters.

NoData Handling

The *NoData Handling* option specifies the value to use for any NoData nodes in the grid. If a NoData node is found in the grid, the *NoData output* option assigns the NoData value to the output grid file for the same node. If the *NoData Handling* is changed to *Remap to*, the NoData node in the input grid is changed to the value in the *Remap Value* column. To change the *NoData Handling* option, click on the current option and select the desired option from the list.

If *Align grids using* is set to *XY coordinates* and *NoData Handling* is set to *NoData output*, all nodes outside the overlapping area will be assigned NoData. To keep the original values of the master grid in areas where there is no data overlap, set *NoData Handling* to *Remap to*, and then set the *Remap Value* to a value that does not affect the math function (e.g., 0 for addition or subtraction).

Remap Value

When the *NoData Handling* is set to *Remap to*, the *Remap Value* box becomes available. Highlight the existing value and type in a new value to change the *Remap to* value. This sets every NoData node in the input grid to this new *Remap Value*. Each grid file can contain a different *Remap Value*.

Add Grids

Click the *Add Grids* button to open the **Open Grid(s)** dialog. To perform an operation on a single grid, select only one grid file. To perform an operation on multiple grids, select multiple grid files using the CTRL and SHIFT keys. You can select any number of grid files at once. When all grids are selected, click the *Open* button. The grid file names are displayed in the *Input Grid* list.

Remove Grid

To remove a grid from the *Input Grid* list, click once on the grid to select it. Click the *Remove Grid* button. The selected grid is removed from the list.

Grid Info

Click the *Grid Info* button to display the number of grid rows, number of grid columns, X, Y, Z minimums and maximums, and statistics for the selected grid. If the grid file is large, click *OK* in the message box that appears to create a detailed grid report or click *Cancel* to create a shorter, less detailed grid report.

Align Grids

If the input grid files have the same number of rows and columns, select the *Node position* value from the *Align grids using* list. The math function will be performed in equivalent grid nodes, regardless of the X and Y values. If the input grid files do not have the same number of rows and columns, select the

XY coordinates value from the *Align grids using* list to match the input grid files on X and Y values.

If *Align grids using* is set to *XY coordinates* and all grids are referenced but in a different coordinate system, the resulting grid file uses the master grid's coordinate system. If an input grid files is not referenced but another grid file is referenced, **Surfer** displays a warning about mixing referenced and unreferenced grid files. Selecting *Yes* continues with the math function operation, but none of the other grids will be projected to the master grid's coordinate system. If the grids are in the same coordinate system, selecting *Yes* will perform the math as expected. However, if a grid is not referenced and not in the same coordinate system, select *No* at the warning to cancel the math function operation and return to the **Grid Math** dialog. Select different grid files with referenced or matching coordinate systems or use the **Grids | Edit | Assign Coordinate System** to change the coordinate system of one or more of the grid files.

Master Grid

From the *Master grid (for geometry)* list, select the master grid to use as the base geometry for the new grid produced by the math function.

Enter a Function of the Form f(A,B, ...)

In the *Enter a function of the form f(A,B, ...) where A, B, ... are the variables in the list above* box, enter a function which describes the output grid file. The *Variable* names are used in the equation. By default, variable names are A, B, C, D, etc. These refer to the input grid files. To perform a calculation on grid file A only, you can leave any reference to additional grid files out of the specified function.

Mathematical functions and operators can be used in the *Enter a function of the form f(A,B...) where A, B... are the variables in the list above* field. For example, to add the Z values from two grids A and B, type *A + B* . Or for example to create an output grid where the values are the greater value from two grids A and B, type *IF(B>A,B,A)*.

To use a stored function, click the  next to the current function. This will display the ten most recent functions used. The functions are stored in the registry, so the equations are stored between **Surfer** sessions. You can also start typing the function in the function box. If the function is in the function history, the entire function will auto-complete.

Output Grid

Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the  button and specify the path and file name for the grid file in the Save Grid As dialog.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

To Use the Math Command

1. Click the **Grids | Calculate | Math** command or the  button.
2. In the **Grid Math** dialog, click the *Add Grids* button. Select any number of grid files and click *Open*.
3. In the *Align grids using* list, select what type of grid alignment to use.
4. In the *Output Grid* section, click the  button to specify a different path or file name for the grid file to be created.
5. In the *Enter a function of the form f(A,B, ...) where A, B, ... are the variables in the list above.* box, enter a function that represents the output grid file, and where A and B refer to the input grid files.
6. Click *OK* to create the new grid file.

Grid Math and .GSR2 Files

When the input .GRD file for a **Grid | Math** command has a defined .GSR2 file with coordinate system information, the information in the first input .GRD file is used for the output .GRD file. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in **Surfer** , as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system.

Using the Grid Math Command to Eliminate Negative Values from a Grid File

Note: This method is no longer necessary for grid files created in **Surfer**. To eliminate negative values while creating a grid file with the Grid | Data command, set the *Minimum* value to the *Data min* or a *Custom* value (such as 0) in the *Grid Z Limits* section of the **Grid Data** dialog.

Some gridding methods (i.e. Kriging, Radial Basis Function, Minimum Curvature, and Modified Shepard's Method) extrapolate beyond the Z range of the data, which can create inappropriate negative values for parameters like concentration, thickness, and magnitude. The Grids | Calculate | Math command can set negative values equal to zero with the formula *max(A,0)*.

To set negative values equal to zero:

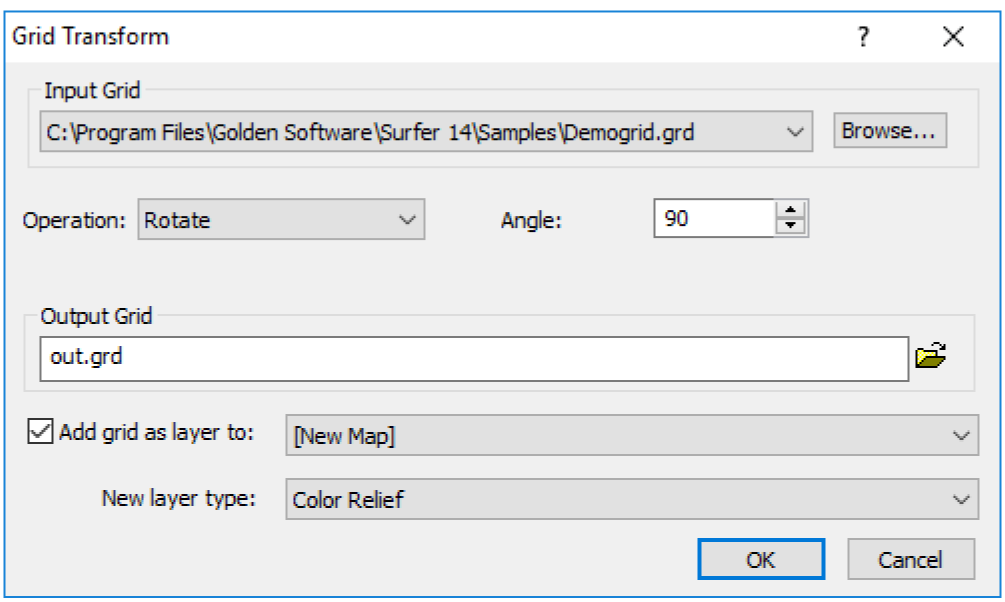
1. Click the **Grid | Math** command.
2. In the **Grid Math** dialog, click the *Add Grids* button. Select the grid file containing the unwanted negative values and click *Open*. This grid file appears in *Input Grid* column.
3. Select an output grid file name in the *Output Grid File* section by clicking on the button.
4. In the *Enter a function of the form f(A,B,...) where A, B, ... are the variables in the list above.* box, enter *max(a,0)*. This tells the program to return whichever is greater, the value in the grid or zero.
5. Click *OK* and the new grid without negative values is created.

Grid Transform

The **Grids | Calculate | Transform** command contains several options that modify the X and Y map coordinates of grid nodes within the grid file. It does not alter the Z values. The **Transform** options are used to shift, scale, rotate, or mirror grid nodes within a grid file.

The Grid Transform Dialog

Click the **Grids | Calculate | Transform** command or the button to open the **Grid Transform** dialog.



*Specify grid transform options in the **Grid Transform** dialog.*

Input Grid

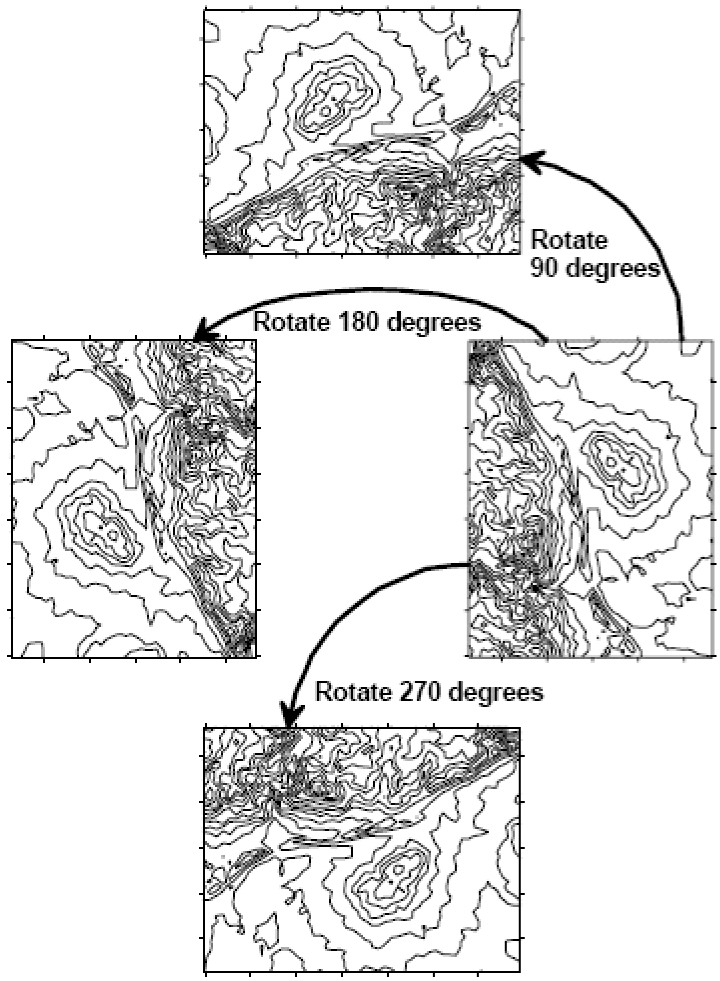
Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

Operation

The *Operation* list controls the transformation type. Click on the box or drop-down arrow and select an option from the list.

Offset	The *Offset* option adds or subtracts a number from the X and Y dimensions. Specify the *Offset* option from the *Operation* list, and specify the amount of the offset in the *X Offset* and *Y Offset* boxes in positive or negative units. For example, if the input grid ranges are X: 1 to 9 and Y: 2 to 8, and the offsets are 10.0 for both the *X Offset* and *Y Offset*, then the output grid ranges will be X: 11 to 19 and Y: 12 to 18.
Scale	The *Scale* option multiplies the X and Y ranges by a factor. Specify the *Scale* option from the *Operation* list, and specify the scaling factors in the *X Scale* and *Y Scale* boxes displayed in the **Grid Transform** dialog. The scaling origin is the lower left corner of the grid, so only the xMax and yMax values are changed. For example, if the input grid ranges are X: 1 to 9 and Y: 2 to 8 and the *X Scale* and *Y Scale* factors are both 3.0, then the output grid ranges will be X: 1 to 25 (((9 - 1) x 3.0) + 1) and Y: 2 to 20 (((8 - 2) x 3.0) + 2).

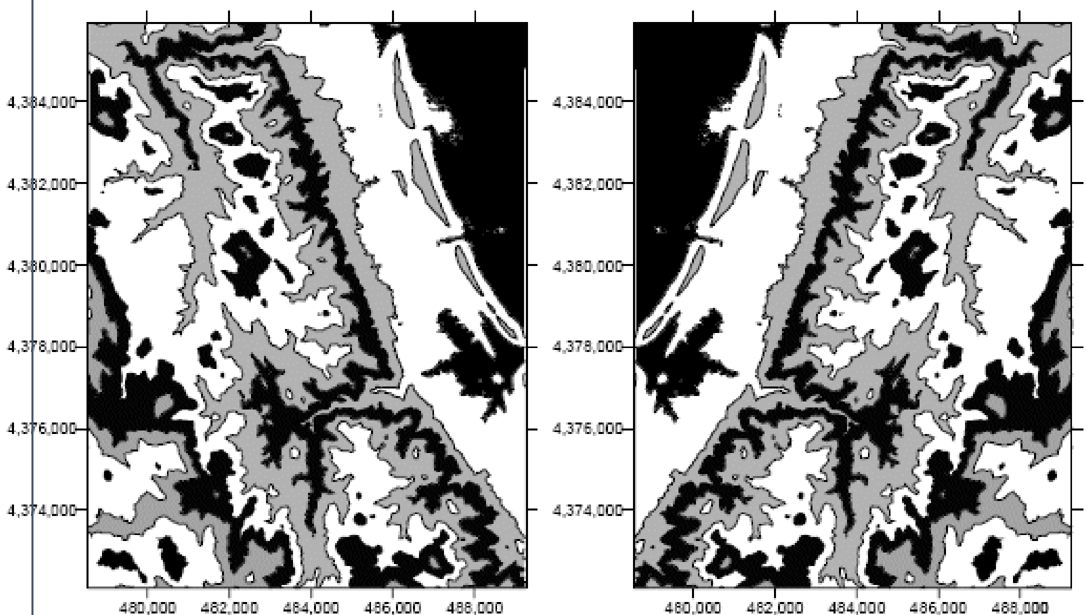
Rotat- e	The *Rotate* option rotates a grid in multiples of 90 degrees (0, 90, 180, 270, …). Select the *Rotate* option in the *Operation* list, and specify the rotation angle in the *Angle* box. The origin of the map remains in the lower left, and the axes continue to increase in the same direction, with the X axis increasing to the right, and the Y axis increasing upwards.



The Rotate option rotates the grid in multiples of 90 degrees without changing the XY origin.

Mirror X

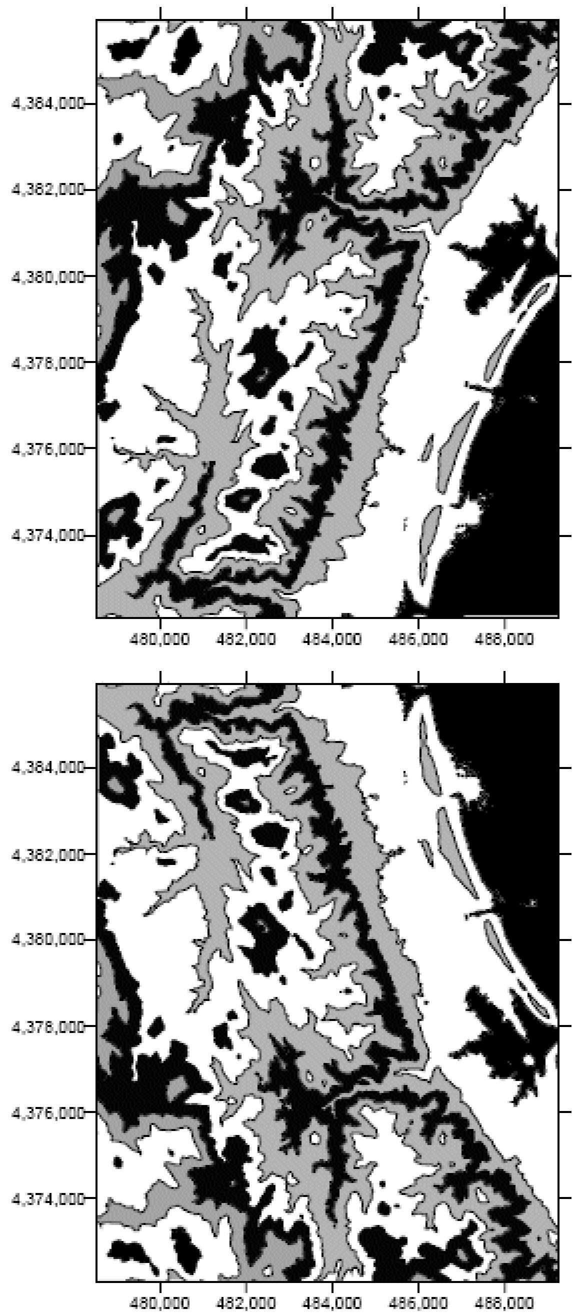
The *Mirror X* option creates a new grid file with the order of the grid nodes reversed in the X direction. The output grid file uses the same minimum and maximum X and Y coordinates as the input grid file. The origin of the grid remains in the lower left corner, with X increasing to the right and Y increasing upward. The Y order remains unchanged.



The Mirror X option creates a mirror image of the grid file with the X order reversed.

Mirror Y

The *Mirror Y* option creates a new grid file with the order of the grid nodes reversed in the Y direction. The output grid file uses the same minimum and maximum X and Y coordinates as the input grid file. The origin of the grid remains in the lower left corner, with X increasing to the right and Y increasing upward. The X order remains unchanged.



The Mirror Y option creates a mirror image of the grid file with the Y order reversed.

Output Grid

Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the button and specify the path and file name for the grid file in the Save Grid As dialog.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

To Transform a Grid File

1. Click the **Grids | Calculate | Transform** command or the button.
2. In the **Open Grid** dialog, select a grid file to transform and click *Open*.
3. In the **Grid Transform** dialog, choose an *Operation* and any associated options.
4. Click the button to display the **Save Grid As** dialog.
5. In the *Save as type* list, select the format for the output file. nter the path and *File name* for the transformed grid. Click *Save* to return to the **Grid Transform** dialog.
6. Click *OK* to create the transformed file.

Grid Transform and .GSR2 Files

When the input .GRD file for a **Grid | Transform** command has a defined .GSR2 file with coordinate system information, this information is used for the output .GRD file. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system.

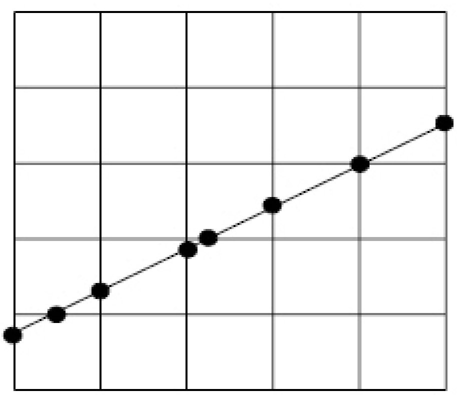
Grid Slice

The **Grids | Calculate | Slice** command or the button produces cross sectional data for 2D and 3D objects. The cross section is generated by taking a vertical slice through a gridded surface along a boundary line. Surface traces on a cross section are sometimes referred to as profile lines.

Another method to create a slice from the data is to use the Map Tools | Add to Map | Profile command. The profile automatically creates a graph showing accumulated distance and elevation. The **Grids | Calculate | Slice** command produces a data file that can show the same object, but contains additional data as well.

The surface is based on a grid file and the boundary line is based on the information in a vector file or base map. The boundary line can be defined by a polyline in a base map or by one of the following file formats:

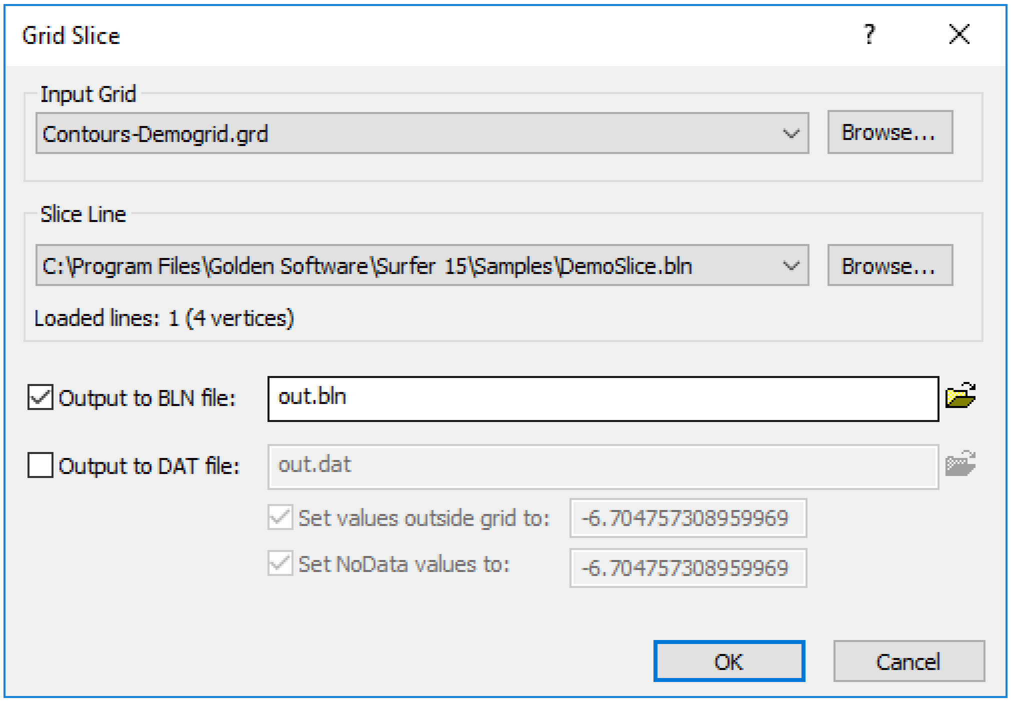
BLN Golden Software Blanking	GSB Golden Software Boundary
BNA Atlas Boundary	GSI Golden Software Interchange
DDF SDTS TVP	KML Google Earth
DLG USGS Digital Line Graph	MIF MapInfo Interchange Format
DXF AutoCAD Drawing	PLT Golden Software Plotcall
E00 Esri AcrInfo Export	PLY Stanford Polygon
EMF Windows Enhanced Metafile	SHP Esri Shapefile
GML Geo. Markup Language	STL Stereolithograph
GPX GPS Exchange Format	WMF Windows Metafile



At each point where the boundary line crosses a grid line, a cross section data point is generated.

The Grid Slice Dialog

Click the **Grids | Calculate | Slice** command or the  button. The **Grid Slice** dialog opens.



*The **Grid Slice** dialog allows you to specify the resulting .BLN and/or .DAT file name and location.*

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

Slice Line

Specify the file or base layer with the slice line in the *Slice Line* section. Load either a map layer or vector file in the *Slice Line* section:

- Click the current selection and select a base layer from the list. Only base layers that contain at least one polyline or polygon will be included in the list. The base layer must use the same source coordinate system as the grid.
- Click *Browse* to load a vector file with the Open dialog. The file must use the same coordinate system as the grid.

Once a file or map layer has been selected the number of lines and vertices is displayed below the slice line selection.

Output BLN File

A Blanking BLN file may be produced with the slicing operation by selecting the *Output to BLN file* option. Type a path and filename for the *Output BLN*, or set the path and filename in the **Save As** dialog by clicking the button next to *Output BLN*. The blanking file contains three columns of data:

Column A:	Accumulated horizontal distance along the boundary line. This distance is accumulated even if the boundary lines go outside the grid region. The distance is reset to zero for each new boundary in the blanking file.
Column B:	Z value at the boundary line and grid line (row or column) intersection.
Column C:	Boundary number, used when more than one boundary line is contained in the file. Each new boundary line in the blanking file is assigned a different number.

The output BLN file can be used to display a profile in **Surfer** as a base map. The Profile command generates a profile with more display options. However, it may be faster to generate a profile from the *Output BLN* when the surface trace already exists in a vector file format and has many vertices. To create a profile from the *Output BLN* file,

1. After creating the BLN file with the **Slice** command, click the **Home | New Map | Base** command
2. Select the output BLN file in the **Import** dialog.
3. Click *Open* and the profile is created as a base map. The X axis corresponds to accumulated distance along the trace, and the Y axis corresponds to grid Z values along the trace.
4. Select the *Map* object for the output BLN base map.
5. On the **Properties** window Scale page, clear the *Proportional XY scaling* check box.
6. On the **Properties** window Scale page, adjust the *X Scale* and *Y Scale* section *Length (page units)* property values until the profile scaling is displayed as desired.

Output DAT File

Cross section data can be written to an ASCII data .DAT file during the **Slice** operation by selecting the *Output to .DAT file* option. Type a path and filename for the *Output DAT*, or set the path and filename in the **Save As** dialog by clicking the button next to *Output DAT*. The ASCII data file consists of five columns of data. When the profile is written to the ASCII file, a new row of data is generated for each point where the boundary line intersects the grid lines in the data file. The output data file can be used to plot the profile in a graphing application, such as Grapher. The columns in the data file are arranged as follows:

Column A:	X coordinate of the boundary line and grid line (row or column) intersection.
Column B:	Y coordinate of the boundary line and grid line (row or column) intersection.

Column C:	Z value at the boundary line and grid line (row or column) intersection.
Column D:	Accumulated horizontal distance along the boundary line. This distance is accumulated even if the boundary lines go outside the grid region. The distance is reset to zero for each new boundary in the blanking file.
Column E:	Boundary number, used when more than one boundary line is contained in the file. Each new boundary line in the blanking file is assigned a different number.

Set Values Outside Grid To

The *Set values outside grid to* option assigns a special Z value to points that lie beyond the grid limits when the check box is active. If this option is not activated, points outside the grid limit are not included in the output data file. The default value is equal to the following equation:

$$(1.4 \times \text{Grid Z Minimum}) - (0.4 \times \text{Grid Z Maximum})$$

Set NoData Values To

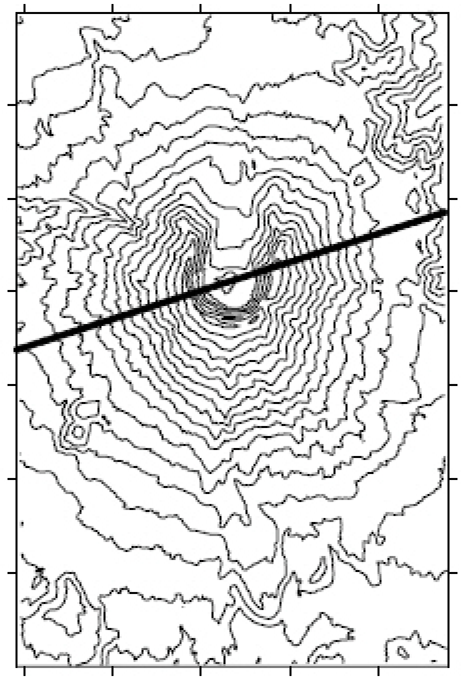
The *Set NoData values to* option assigns a special Z value to points that lie inside NoData regions when the check box is selected. If the check box is cleared, points inside the NoData areas are not included in the output data file.

OK or Cancel

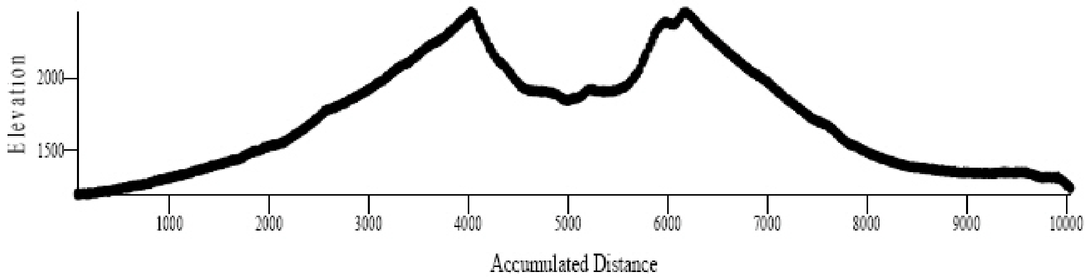
Click *OK* and a data file (and a blanking file, if specified) is created. Click *Cancel* to exit the dialog without creating the data.

To Create a Data File Containing Cross Section Data

1. Click the **Grids | Calculate | Slice** command.
2. In the **Open Grid** dialog, select a grid file and click *Open*.
3. In the **Open** dialog, select the blanking file to use when producing the profile data and then click *Open*.
4. If desired, enter a name for the *Output BLN* in the **Grid Slice** dialog by clicking the  button to open the **Save As** dialog.
5. Specify the name for the *Output DAT* by clicking the  button to open the **Save As** dialog.
6. Click *OK* and a data file (and a blanking file, if specified) is created.



You can display the profile line across the contour map. This is the profile line used to produce the following cross section.



This is the cross section taken from the profile line on the map. The cross section was created as a base map using the Output BLN*.*

Plotting Cross Sections

To display the location of the slice for the cross section on a map, use the Home | New Map | Base command and create a base map from the slice line vector file used for the cross section.

Profile Line Projected onto the X-Z Plane

In **Surfer** use the **Home | New Map | Post** command to plot the actual cross section projected onto a specific plane. To plot the cross section, column C, the Z value, is plotted as the Y coordinate. Plotting column A as the X coordinate and column C as the Y coordinate produces a profile line projected onto the X-Z plane.

Profile Line Projected onto the Y-Z Plane

In **Surfer** use the **Home | New Map | Post** command to plot the actual cross section projected onto a specific plane. To plot the cross section, column C, the Z value, is plotted as the Y coordinate. Using column B as the X coordinate and column C as the Y coordinate produces a profile line projected onto the Y-Z plane.

Profile Line Showing True Accumulated Distance

Use the *Output BLN* to create a profile with the Home | New Map | Base command. The *Output BLN* produces a profile line that shows true accumulated distance along the boundary line.

Alternatively, to create a profile from the *Output DAT* data file, in **Surfer** use the **Home | New Map | Post** command to plot the actual cross section projected onto a specific plane. To plot the cross section, column C, the Z value, is plotted as the Y coordinate. Using column D as the X coordinate and column C as the Y coordinate produces a profile line that shows the true accumulated distance along the boundary line.

To create more elaborate cross sections, we recommend Golden Software's Grapher program.

Residuals

The **Grids | Calculate | Residuals** command computes the vertical difference between the Z value in the data file and the interpolated Z value on a gridded surface. It gives a quantitative measure of how well the grid file agrees with the original data.

A residual is the difference between the Z value of a point in a data file and the interpolated Z value at the same XY location on a gridded surface. Residual values are reported as either positive or negative values. If the Z value in the data file is greater than the Z value derived from the gridded surface, the residual value is positive. A negative residual value indicates that the Z value from the data file is less than the interpolated Z value at that point. If the grid file is contains the NoData value at the data point, the residual value is left blank in the worksheet. **Surfer** uses a bilinear interpolation method to calculate Z values at points that do not coincide with grid nodes.

The formula used to compute a residual value is:

$$Z_{res} = Z_{dat} - Z_{grd}$$

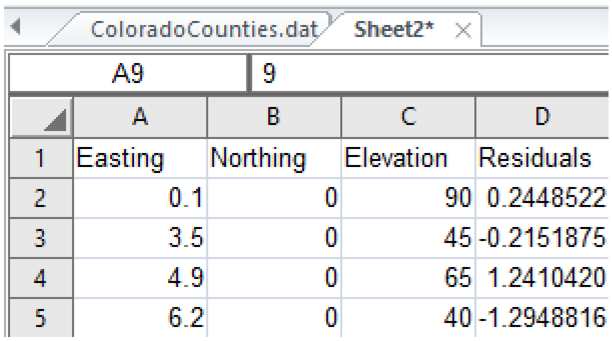
where:

Z_{res} = the residual value

Z_{dat} = the Z value in the data file

Z_{grd} = the Z value of the surface at the XY coordinate from the grid file

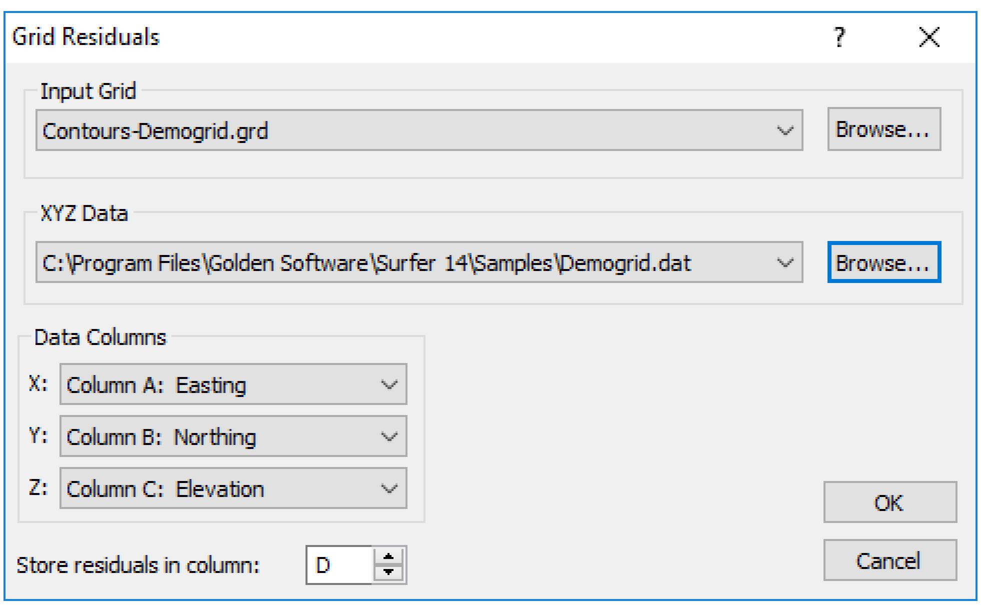
A worksheet window is automatically opened after the **Residuals** command is used. The worksheet includes the data file used for the calculation and the newly generated residual information. Use the worksheet's Data | Data | Statistics command to obtain statistical information on the calculated residuals.



*After clicking OK in the **Grid Residuals** dialog, the worksheet window opens with the original data and the new residuals column.*

The Grid Residuals Dialog

Click the **Grids | Calcluate | Residuals** command or the button to open the **Grid Residuals** dialog.



*Specify the locations for the X, Y, and Z data and the location for the calculated residuals in the **Grid Residuals** dialog.*

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

XYZ Data

Specify the data file in the *XYZ Data* section. Click the current selection and select a map layer from the list. Only post and classed post layers are displayed in the *XYZ Data* list. Click *Browse* to load the data file with the Open Data dialog.

Data Columns

Specify the location for the X, Y, and Z coordinates from the data file in the *Data Columns* section.

Store Residuals in Column

In the *Store residuals in column* box, enter the column where you would like the residuals stored. By default, **Surfer** selects the first empty column in the data file.

To Calculate Residuals

1. Click the **Grids | Calculate | Residuals** command or the  button.
2. In the **Open Grid** dialog, select the grid file to be used for the residual calculation and then click *Open*.
3. In the **Open** dialog, select the data file to be compared to the grid file, and then click *Open*.
4. In the **Grid Residuals** dialog, specify the location for the X, Y, and Z coordinates in the *Data Columns* section.
5. In the *Store residuals in column* box, enter the column where you would like the residuals stored. By default, **Surfer** selects the first empty column in the data file.
6. Click *OK* and the worksheet window opens. The worksheet window displays the data file as it was specified in the **Grid Residuals** dialog with the additional column for the residual data.

Calculating the Agreement between a Grid and the Original Data File

After performing the Grids | Calculate | Residuals command:

1. In the worksheet, click the column letter for the column containing the residual values to select the entire column. To get the statistics for just a portion of the residual values, select only those particular values.

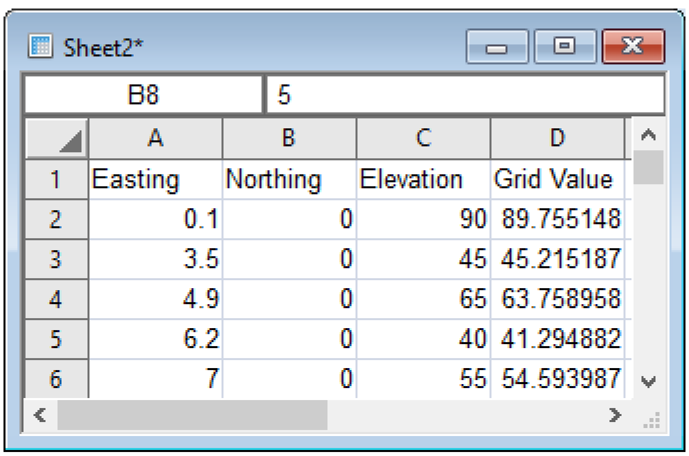
2. Choose the Data | Data | Statistics command. Specify the desired parameters to be reported.

3. If *Show in a window* is selected, the *Statistics Results* window is displayed with the statistical information on the residual data. If the *Copy to worksheet* option is selected, the residual results are placed directly into the specified columns of the worksheet.

Point Sample

The **Grids | Calculate | Point Sample** command computes the interpolated Z value on a gridded surface at given XY locations. XY locations from a data file are used to sample the grid. Bilinear interpolation is used to calculate the Z value at the XY location from the nearest four grid nodes. If a Z value cannot be calculated due to NoData nodes or if the XY location is beyond the extent of the grid, the cell is left blank in the output worksheet.

A worksheet window is automatically opened after the **Point Sample** command is used. The worksheet includes the data file used for the calculation and the newly generated point sample information. Use the worksheet's Data | Data | Statistics command to obtain statistical information on the calculated Z values.

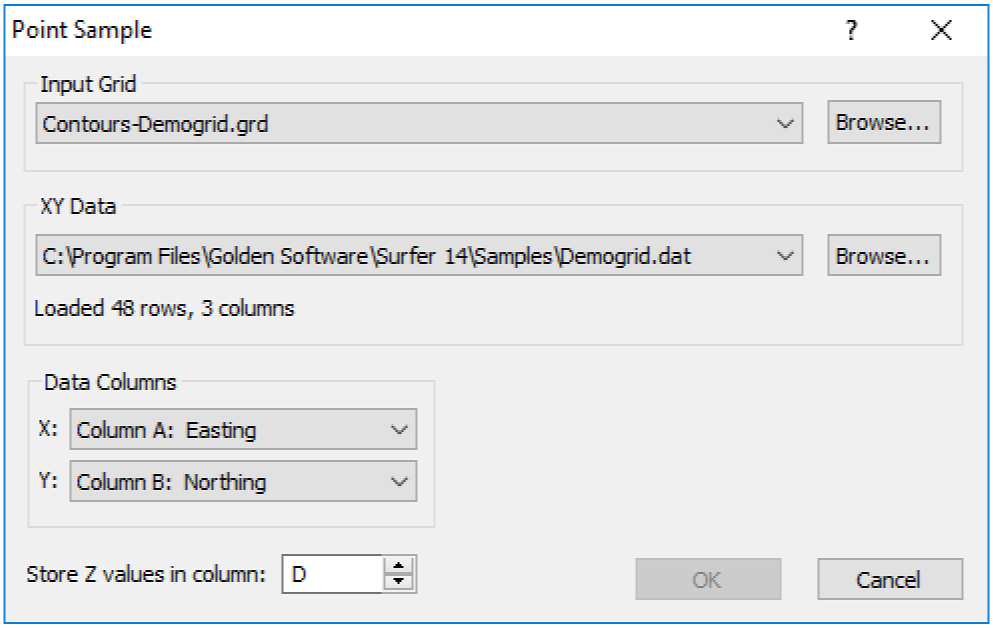


	A	B	C	D
1	Easting	Northing	Elevation	Grid Value
2	0.1	0	90	89.755148
3	3.5	0	45	45.215187
4	4.9	0	65	63.758958
5	6.2	0	40	41.294882
6	7	0	55	54.593987

*After clicking OK in the **Point Sample** dialog, the worksheet window opens with the original data and the new* Grid Value *column.*

The Point Sample Dialog

Click the **Grids | Calculate | Point Sample** command or the button to open the **Point Sample** dialog.



*Specify the grid, XY data, and the column for the sampled points in the **Point Sample** dialog.*

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

XY Data

Specify the data file in the *XY Data* section. Click the current selection and select a map layer from the list. Only post and classed post layers are displayed in the *XY Data* list. Click *Browse* to load a data file with the Open Data dialog.

Data Columns

Specify the location for the X and Y coordinates from the data file in the *Data Columns* section.

Store Z values in Column

In the *Store Z values in column* box, enter the column where you would like the sampled Z values stored. By default, **Surfer** selects the first empty column in the data file.

Calculating the Z Value of the Surface at a Point

The Grids | Calculate | Point Sample command samples Z values from a grid at any point. To perform the operation, create a data file with X and Y coordinates for the points you'd like to sample and click the **Grids | Calculate | Point Sample** command.

Example

Consider a grid file that ranges from zero to 10 in the X and Y dimensions, and you want to determine the value of the surface at the points (1,3) and (4,2) on the surface.

1. Open a new worksheet window with **File | New | Worksheet**.
2. In column A, type the X coordinates for the points you wish to find the Z value, e.g. type 1 in Row 1 and 4 in Row 2. In column B, type in the Y coordinate, e.g. type 3 in Row 1 and 2 in Row 2. Add as many points as you want into the data file by entering the coordinates for each point into a different row in the data file.
3. Use the **File | Save As** command and enter a name for the data file, for example PointSample.dat. Click *Save* and the XYZ data file is created.
4. Click the Point Sample command.
5. Specify the *Input Grid*, *XY Data*, and *Data Columns* in the **Point Sample** dialog.
6. Click *OK* and a worksheet is opened with the Z values from the points you entered.

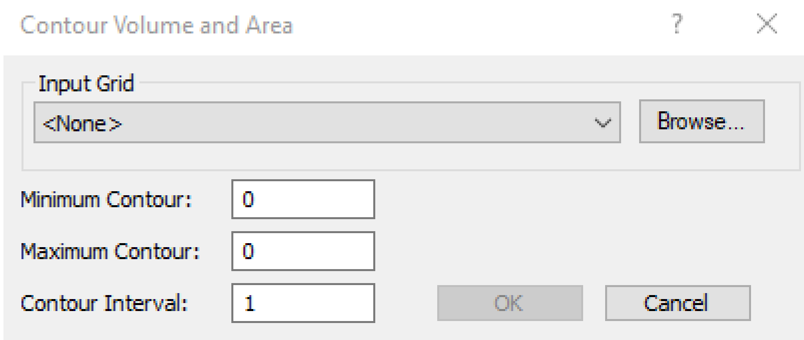
Contour Volume and Area

The **Grids | Calculate | Contour Vol/Area** command computes the volume and area above contour lines, below contour lines, and between contour lines on contour maps. Two contour lines must be defined to calculate the area and volume, otherwise the result is *n/a*. The computation results are displayed in the grid contour vol/area report. The volume and area results can be saved as an ASCII text file, in Rich Text Format, or can be copied onto the clipboard.

Area results are provided in square units and volume results are provided in cubic units based on the units of the input grid file.

Calculating Volumes and Areas

Click the **Grids | Calculate | Contour Vol/Area** command or the  button to open the **Contour Volume and Area** dialog.



Specify the minimum and maximum contour lines and contour interval for the sampled contour lines in the ***Contour Volume and Area*** *dialog.*

Input Grid

Specify the source map layer or grid file in *Input Grid*. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

The defaults for *Minimum Contour*, *Maximum Contour*, and *Contour Interval* are set on the Levels page in the Properties window for the contour layer.

Minimum Contour

Set the *Minimum contour* to the value of the first contour line to be included in the calculation. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard.

Maximum Contour

Set the *Maximum contour* to the value of the last contour line you want to be included in the calculation. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard.

Contour Interval

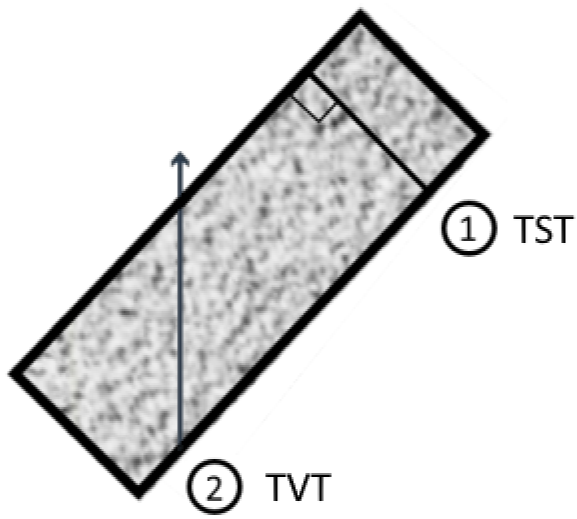
The difference between two contour lines is defined as the *Contour Interval*. The *Simple* level method is always used to define the spacing between adjacent contours when calculating the volume and area, even if another level method has been set for the contour layer in the plot.

Grid Contour Vol/Area Report

Click *OK* in the **Contour Volume and Area** dialog and the results are displayed in the Grid Contour Vol/Area Report. Use **File | Save As** to save the report, or you can copy the information to the clipboard and paste it into another application.

Isopach Map

The **Grids | Calculate | Isopach** command can be used to generate an isopach or isochore grid and map to show thickness between two surfaces. An isopach map is also referred to as a True Stratigraphic Thickness (TST) map. This type of map represents the true thickness of a stratigraphic unit. An isochore map, also known as a True Vertical Thickness (TVT) map, shows the vertical thickness of a unit, which is the thickness from the upper surface straight down to the lower surface. If the unit was horizontal, this would be the same as TST, but if units have a dip or incline, the vertical thickness is different from the stratigraphic thickness. The following image illustrates this difference:



Isopach and isochore thickness measurement example.

1. Isopach (TST) measures the actual thickness between two surfaces.
2. Isochore (TVT) measures from a point on the upper surface to a point on the lower surface, which may not be the actual thickness. This is what the **Grids | Calculate | Math** command calculates when you subtract one grid file from another.

Note: Before creating an isopach map, make sure the XY and Z data in your grid file are the same unit of measure (e.g. meters, feet, etc...). If they are not the same units, then you must convert or project them to the same units. Isochore maps can be created from data in different units.

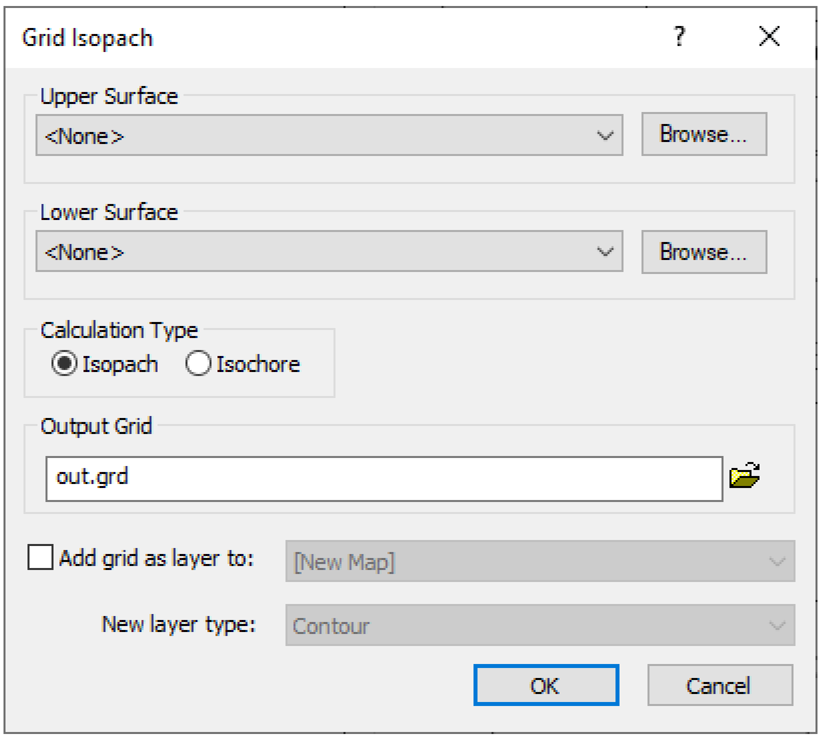
To convert or project a grid file to the same units for isopach maps:

- If the grid data is based on longitude and latitude, use the **Grids | Edit | Project** command to project the grid in a coordinate system in units that match the Z units. For example, if the Z units are meters, an appropriate coordinate system could be one of the UTM systems.

- If the map uses different linear units for XY, and Z (e.g. X and Y are in feet and Z is in meters), use the **Grids | Calculate | Math** command to convert the Z units to match the X and Y units, or use the **Grids | Calculate | Project** command to convert the X and Y units to match the Z units.

Follow these steps to create an isopach or isochore map, for example to generate a contour map representing the thickness of a coal seam using drillhole data that provides the elevation of the upper surface and lower surface of the coal seam.

1. Create grid files of the upper and lower surfaces (for example, grid files that define the elevation of the top and bottom of a coal seam). Both grids must use the same XY ranges and have the same number of rows and columns.
2. Select the **Grids | Calculate | Isopach** command.



Grid Isopach dialog

3. In the **Grid Isopach** dialog, click the *Browse* button next to *Upper Surface*. In the *Open Grid* dialog, select the grid file for the upper surface and click *Open* or click the *Download* button to download a grid file.
4. Click the *Browse* button next to *Lower Surface*. In the *Open Grid* dialog, select the grid file for the lower surface and click *Open* or click the *Download* button to download a grid file.
5. Based on the type of map that you want, click *Isopach* or *Isochore* from the *Calculation Type* section.
6. Click the  button in the *Output Grid* section, enter the name for the thickness grid file in the **Save Grid As** dialog, and click *Save*.

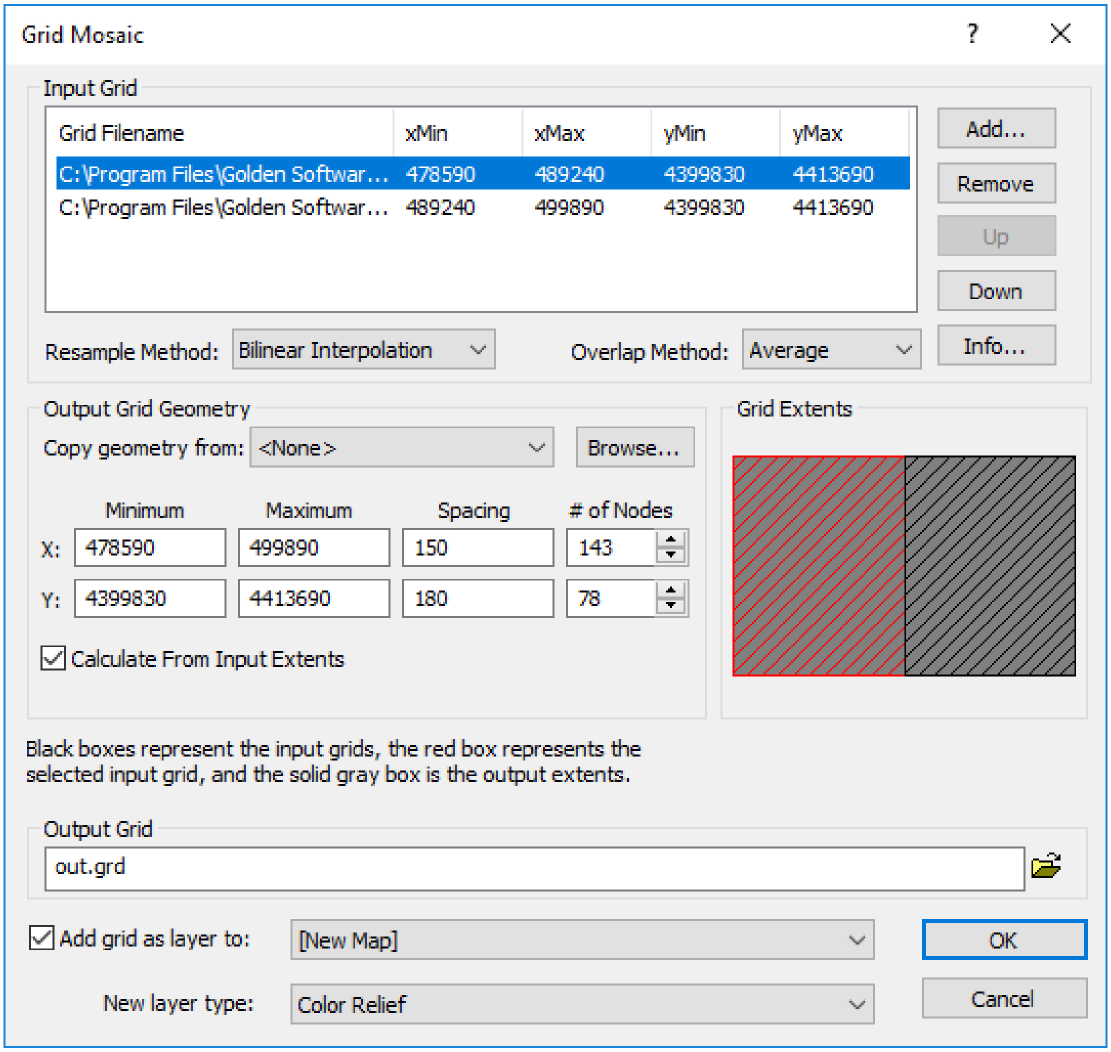
7. Check the *Add grid as layer to* check box to display the grid or leave unchecked to just create the grid file listed in *Output Grid*.
8. Select the layer type for the new grid from the *New layer type* list.
9. Click *OK* to generate the selected grid.

Grid Mosaic

The **Grids | Resize | Mosaic** command combines two or more input grids of the same coordinate system into a single output grid. For example, if you have four USGS SDTS Raster Profiles (.DEMs), you can easily combine them into one grid with **Grids | Resize | Mosaic**.

Grid Mosaic Dialog

Click the **Grids | Resize | Mosaic** command or the  button, specify the grid file to mosaic in the **Open Grid(s)** dialog, and click *Open* to display the **Grid Mosaic** dialog.



*Use the **Grid Mosaic** dialog to combine two or more input grids into a single output grid.*

Input Grid

The *Input Grid* group initially contains all the grid files selected in the **Open Grid (s)** dialog. You can *Add* or *Remove* grids to the *Input Grid* group. When a grid is selected in the *Input Grid* group, the grid is highlighted in the *Grid Extents* section.

Header Button Size and Sorting

The *Input Grid* header buttons can be dragged side to side to make the columns wider or narrower. Also, the grids can be sorted by clicking on the *Grid Filename*, *xMin*, *xMax*, *yMin*, and *yMax* headers. Double-clicking the header buttons reverses the direction.

Add

Click the *Add* button to open the **Open Grid(s)** dialog and add additional grids to the mosaic. Added grids will be listed in the *Input Grid Files* section. All grids must be in the same coordinate system. For example, this means that 30 meter DEM files must be in the same UTM zone.

Remove

Select a grid and click the *Remove* button to remove a grid from the mosaic. One grid can be removed at a time.

Up and Down Buttons

Alternatively, the grids can be moved around the list by highlighting a grid name (click on it) and then clicking the *Up* and *Down* buttons.

Info

Click the *Info* button to obtain information about the selected grid, including the grid file name and statistics. If the grid file is large, click *OK* in the message box that appears to create a detailed grid report or click *Cancel* to create a shorter, less detailed grid report.

Resample Methods

There are three Resample Methods from which to choose: *Bilinear Interpolation*, *Nearest Neighbor*, and *Cubic Convolution*.

Overlap Method

When grids overlap, choose an *Overlap Method* to determine the value of the grid node in the new grid: *Average*, *First*, *Last*, *Minimum*, *Maximum,* or *Sum*. The *First* and *Last* are the first and last grids listed in the *Input Grid Files* section.

Output Grid Geometry

Output Grid Geometry defines the grid limits and grid density. Grid limits are the minimum and maximum X and Y coordinates of all grid files. The default grid *Spacing* is set to the minimum spacing of the input grids. The *# of Nodes* in the *X* direction is the number of grid columns, and the *# of Nodes* in the *Y* direction is the number of grid rows. Increasing the grid extents will not increase the grid as the resampling methods do not extrapolate beyond the extents of the original grid.

Copy Geometry

The *Copy geometry from* option copies the grid geometry from an existing map layer or grid file. This option is useful when creating grids that will become overlaid map layers, processed with the Grid Math command, or used to calculate a volume between two surfaces. The **Math** and **Volume** commands require the input grids to have the same geometry.

To copy the geometry from an existing layer, select the layer in the *Copy geometry from* list. To copy the geometry from a grid file, click *Browse* and select the file in the Open Grid dialog. Select *<None>* to return the *Output Grid Geometry* options to their default values and to manually edit the grid geometry.

Calculate From Input Extents

Check the *Calculate From Input Extents* box to return the grid to its original size based on the input grids' minimum and maximum.

Grid Extents

The *Grid Extents* group graphically displays the *Input Grid* files. Each grid that is added to the *Input Grid* files group is displayed in the *Grid Extents* group. Black boxes represent input grids. Red boxes represent the selected input grid. The solid gray box represents the output extents. The input grid boxes can be clicked with the mouse to graphically select the grid in the *Grid Extents* group. Alternatively, select a grid file in the *Input Grid* files group to select a grid box.

Output Grid

Choose a path and file name for the grid in the *Output Grid* section. You can type a path and file name, or click the button to browse to a new path and enter a file name in the Save Grid As dialog.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

To Combine (Mosaic) Grids

1. Click the **Grids | Resize | Mosaic** command or the button.
2. Select the files you would like to combine in the **Open Grid(s)** dialog. If the grid files are in the same folder, use the CTRL and SHIFT keys while clicking on the file names to make multiple selections. If the files are not in the same folder, you will have the opportunity to select more files later. Click *Open*
3. In the **Grid Mosaic** dialog, choose the desired *Resample method* and *Overlap method*. You can also set the *Output Grid Geometry* .
4. Name the new grid in the *Output Grid* box by typing a path and file name or using the button to browse to a new path.
5. Click *OK* and the new grid is created.

Mosaic Tips

- When a NoData node overlaps a node with a value, the NoData node is ignored.
- The default output grid spacing is set to the minimum spacing of the input grids.
- By specifying *First* for the *Overlap Method*, a second overlapping grid can be used to fill in NoData values in the first grid.
- Some grid files, such as USGS DEM files, are not seamless. Occasionally, a grid node is missing along an edge. Sometimes, the reported corner points for adjacent files are slightly different, which causes the neat lines between grids to be slightly off, which causes some edge nodes to be excluded from both grids. This shows up as a NoData node along the seam.
- It is possible to resample a single grid with **Grids | Resize | Mosaic** by specifying a single grid and changing the grid spacing.

Grid Mosaic and .GSR2 Files

When the input .GRD file for a **Grids | Resize | Mosaic** command has a defined .GSR2 file with coordinate system information, the information from the first input .GRD file is used for the output .GRD file. All .GRD files should be in the same coordinate system. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system.

Resample Methods

When using Grids | Resize | Mosaic, **Surfer** uses resampling methods to assign data values in the new grid. For each grid node location in the output grid (X, Y) the Z value is located in the set of input grids. The Z value is computed according to the specified resample method.

Note that all resample methods are exact interpolators - that is, if the X, Y location falls exactly on a grid node, the grid node’s Z value is returned. If more than one grid can supply a Z value at the specified X, Y position, they are combined according to the specified *Overlap Method*.

The resampling methods do not extrapolate outside the grid limits. This means that NoData nodes may be inserted between adjacent input grids if the input grids to not each include the common boundary.

Nearest Neighbor

The *Nearest Neighbor* method applies the closest grid node value on the original grid to the grid node value in the new grid. When the original grid and the new grid differ in size, more than one original node may be applied to the new grid and some original grid cells may not be applied to the new grid.

The *Nearest Neighbor* method is the fastest resampling method, though it can result in distorted output if the original grid and new grid differ in size.

Bilinear Interpolation

The *Bilinear Interpolation* method uses a weighted average of four nodes in the original grid and applies this to the new grid node. The new grid is smoothed compared to the original grid.

Cubic Convolution

The *Cubic Convolution* method uses a weighted average of 16 nodes in the original grid and applies this to the new grid. The new grid is smoother than the original grid. This method is best for continuous data. This is the slowest resampling method, but it results in a sharper grid than the *Bilinear Interpolation* or the *Nearest Neighbor* methods.

Grid Extract

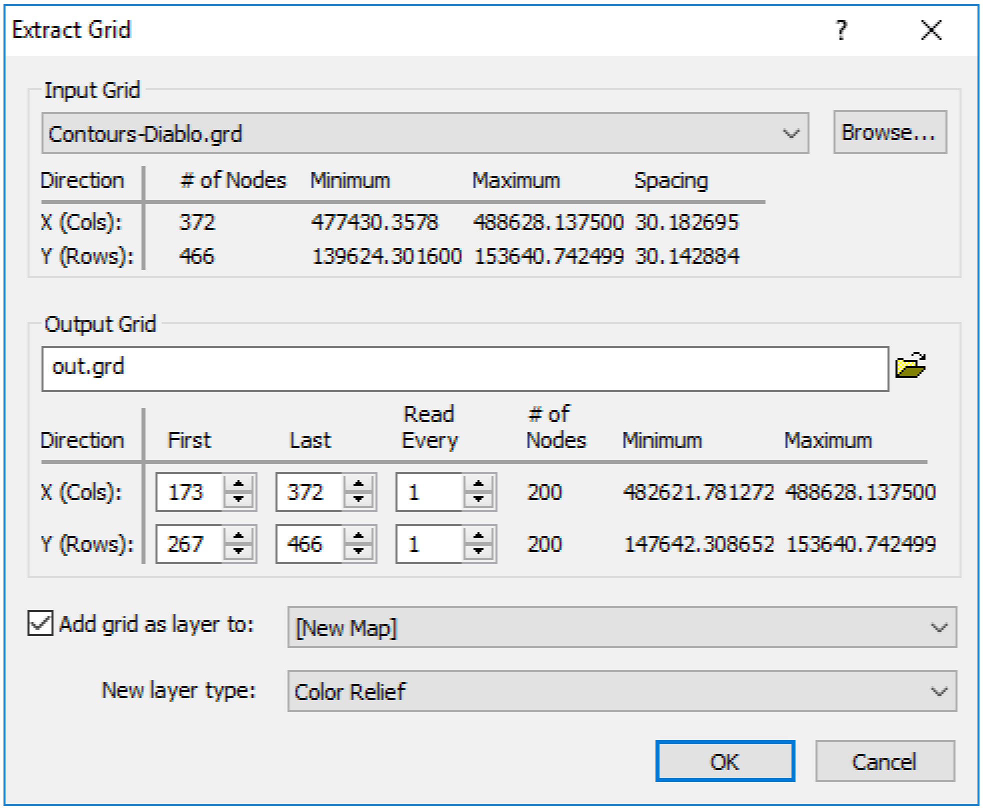
The **Grids | Resize | Extract** command creates a subset of an existing grid file. Grid files consist of rows and columns (lines of constant Y and X value) containing Z values. When you extract information from a grid file, you can specify the starting and ending rows and columns to extract. For example, you can create a new grid file that consists of only the center portion of an existing grid by specifying the middle columns and rows from the original grid.

Consider a grid file that has 100 rows and 100 columns. You might want to create a grid that contains the information from row 25 to row 75 and column 25 to column 75. In this example, specify 25 as the first row and first column to extract and 75 as the last row and last column to extract. The grid file you create in this case would consist of 51 rows and 51 columns of data (from 25 to 75 rows and columns, inclusive).

Subsets can also be based on every n^{th} row or column read from the input grid file, reducing the density of the grid. In this case, you specify a step factor that skips rows and columns when reading information from the original grid file.

Extract Grid Dialog

Click the **Grids | Resize | Extract** command or the  button to open the **Open Grid** dialog. Select a grid file and click *Open* to open the **Extract Grid** dialog.



*The **Extract Grid** dialog displays information about the input grid file, lets you specify a subset of a grid, and writes the subset to a new grid file.*

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

In the *Input Grid* section, the file name and statistics about the grid are listed. The information includes the *# of Nodes, Minimum, Maximum,* and *Spacing* in both the X and Y directions. This is provided as information only for adjusting the output grid. This information cannot be changed.

Output Grid

Type a file path and file name, including the file type extension, in the *Output Grid* field, or click the  button and specify the path and file name for the grid file in the Save Grid As dialog.

First and Last

The *First* and *Last* boxes show the first and last rows and columns from the input grid file. The number of nodes and the map coordinates are displayed to the right. As you change the *First* and *Last* values, the number of nodes and map coordinates change accordingly.

Read Every

The *Read Every* boxes specify a skip factor to "thin out" a grid. A value of one reads every row or column within the selected limit. A value of two reads every other row or column, and so on.

Add New Map or Layer

Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

Extracting a Subset of an Existing Grid File

1. Click the **Grids | Resize | Extract** command or the button.
2. In the **Open Grid** dialog, select the grid file from which you want to extract a subset and click *Open.*
3. In the **Extract Grid** dialog, specify the *First*, *Last*, and *Read Every* values in the *Rows (Y)* and *Cols (X)* boxes.
4. Click the button to display the **Save Grid As** dialog.
5. In the *Save as type* list, select the format for the output file. Specify the path and file name for the file to be created, and click *Save* .
6. In the **Extract Grid** dialog, click *OK* and the grid is created with the subset of grid information from the original grid file.

Grid Extract and .GSR2 Files

When the input .GRD file for a **Grids | Resize | Extract** command has a defined .GSR2 file with coordinate system information, this information is used for the output .GRD file. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original file, but the .GSR2 is required to define the coordinate system.

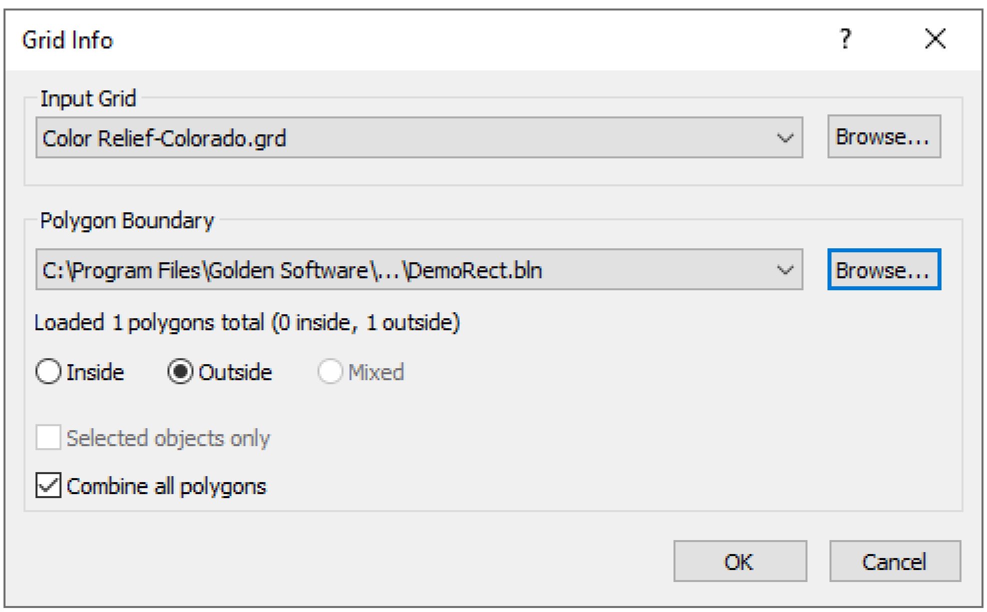
Grid Info

The **Grids | Info | Grid Info** command or the button in the plot window allow you to select a .GRD file and display the file name, grid size in rows x columns, and grid X, Y, and Z minimums and maximums. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report. This is the same basic information as displayed in the **Grid Editor** window with the Grid Editor | Options | Grid Info command.

The Open Grid dialog also displays the *Grid Info* of the selected grid. The **Open Grid** dialog displays the following *Grid Info*: *Name, Format, Size, xMax, yMax, zMax, xMin, yMin,* and *zMin.*

Grid Info Dialog

The **Grid Info** dialog is displayed after clicking the **Grid Info** command or the button.



Select the grid for calculating grid statistics.

Input Grid

Specify the source map layer or grid file in the *Input Grid* section. Click the current selection and select a map layer from the list. Only map layers created from grid files are included in the *Input Grid* list. Click *Browse* to load a grid file with the Open Grid dialog.

Polygon Boundary

Specify the region or regions from which to calculate the statistics in the *Polygon Boundary* section. Select either a map layer or vector file in the *Polygon Boundary* section:

- Click the current selection and select a base layer from the list. Only base layers that contain at least one polygon or polyline will be included in the list. The base layer must use the same source coordinate system as the grid.
- Click *Browse* to load a vector file with the Open dialog. The file must use the same coordinate system as the grid.

The number of polygons and vertices is displayed below the *Polygon boundary* once a file or map layer has been selected. If the boundaries have blanking flags or BLN_Flag attributes, the total number of *inside* and *outside* flags is displayed.

Polyline Boundaries

Polylines can be used for polygon boundaries. The polylines in the base layer or vector file will be treated as polygons while assigning regions for calculating statistics. The **Grid Info** command is not recommended with open polylines, unexpected results my occur. Consider converting polylines to polygons with the Polyline to Polygon command and editing features with the Reshape command.

If the layer you wish to use contains both polygons and polylines, but you only wish to use some or all of the polygons, select the objects you wish to use before clicking **Grids | Info | Grid Info** and select the *Selected objects only* option. If the file you wish to use contains both polylines and polygons, first load the file as a base layer, and then use the **Grid Info** command with the *Selected objects only* option.

Inside, Outside, or Mixed

Select *Inside* to use the region inside the polygon boundary or boundaries to calculate statistics. Select *Outside* to use the region outside the polygon boundary or boundaries. Select *Mixed* to use the blanking flag or BLN_Flag attribute values from the file or layer. The *Mixed* option is only available when the layer or file contains both blanking flags or BLN_Flag attributes: assign NoData inside (1) and assign NoData outside (0). If all blanking flags or BLN_Flag attributes are the same, the *Inside* or *Outside* option is selected automatically, and the *Mixed* option is not available.

Selected Objects Only

Select the *Selected objects only* option to use only the selected objects in the base layer to calculate statistics. When the *Selected objects only* box is checked, the *Loaded polygons and vertices* values are updated. Select a base layer in the *Polygon Boundary* field to use the *Selected objects only* option. The *Selected objects only* option is not available when the *Polygon Boundary* is a vector file. The polygon or polygons must be selected before clicking the **Grids | Info | Grid Info** command.

Combine All Polygons

Combine all polygons is checked by default and the statistics will be calculated for the total of all the polygons. When *Combine all polygons* is unchecked, the report presents statistics for each polygon.

Filtering Grid Files

Surfer contains filtering commands which operate on grid files, or on contours in a contour map. These commands round out angular contours, eliminate noise in a map, or change the density of a grid.

These are the smoothing methods available in **Surfer**:

- Filter contains predefined filtering options and also allows you to define your own filtering schemes.
- Spline Smooth is best for eliminating angular contours or surfaces; this operation fills in a sparse grid. For example, a 10 x 10 grid might result in very angular contours on a contour map produced from the grid. By increasing the grid density using the **Grids | Edit | Spline Smooth** command, the 10 x 10 grid can be expanded to a 50 x 50 grid. The 50 x 50 grid produces a much smoother appearing contour map than the 10 x 10 grid.
- The Smooth tool in the **Grid Editor** smooths the grid with a 3x3 averaging filter.
- Contour Smoothing smooths contour lines of a contour map by smoothing the grid file or changing the contour map properties.

Filter Data Before Gridding

To filter your data before gridding, use the data filters in Grids | New Grid | Grid Data.

Grid Operations References

Akai, Terrence J., (1994), *Applied Numerical Methods for Engineers*, John Wiley and Sons, Inc., New York, 410 pp.

Bras, R. L., and I. Rodriguez-Iturbe, (1985), *Random Functions and Hydrology*, Addison-Wesley, Reading, Massachusetts, 559 pp.

Mitasova, Helena and Jaroslav Hofierka, (1993), Interpolation by Regularized Spline with Tension: II. Application to Terrain Modeling and Surface Geometry Analysis, *Mathematical Geology*, v. 25, no. 6, p. 657-669.

Moore, I. D., R. B. Grayson, and A. R. Ladson, (1991), Digital Terrain Modeling: A Review of Hydrological, Geomorphological and Biological Applications, *Hydro. Proc.*, v. 5, no. 1, p. 3-30.

Moore, I. D., A. Lewis, and J. C. Gallant, (1993), Terrain properties: Estimation Methods and Scale Effects, *Modeling Change in Environmental Systems*, A.J. Jakeman et al. editors, John Wiley and Sons, New York.

Press, William H., Saul A. Teukolsky, William T. Vetterling, and Brian P. Flannery, (1992), *Numerical Recipes in C: The Art of Scientific Computing*, Second Edition, Cambridge University Press, New York, 994 pp.

Ripley, B. D., (1981), *Spatial Statistics,* John Wiley and Sons, New York, 252 pp.

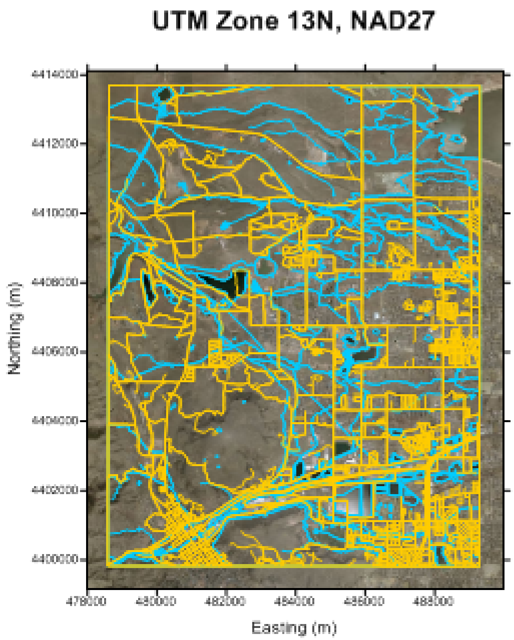
Schwartz, Abraham, (1974), *Calculus and Analytic Geometry*, 3rd edition, Holt, Rinehart, and Winston, New York, 1140 pp.

Tuma, Jan J., (1979), *Engineering Mathematics Handbook*, 2nd edition, McGraw-Hill Book Company, New York, 394 pp.

Chapter 8 - Base Maps

Base Map

Base maps display geographic and political information such as roads, streams, lakes, or state and county boundaries. Base maps can be combined with any other map type in Surfer. A vector base map or raster base map is automatically created based on the input data.



This map shows multiple base layers, depicting roads, waterways, and a USGS urban area satellite image for Golden, Colorado, USA.

Base (vector) Layers

Base (vector) layers are created when a vector data file is used to create a base map. Vector files draw objects at precise X, Y locations on a map and can contain points, polygons, polylines, and text.

Symbology

Vector base maps can also include symbology. Symbology uses symbols or colors to display statistical information about the features in the base layer. Symbology applies line, fill, and/or symbol properties to features in the base layer depending on an attribute value. The symbology can be included in a legend. The type of symbology and the symbols' appearance are controlled in the **Symbology** dialog. Click *Edit Symbology* in the Base (vector) layer **Properties** window General page to open the **Symbology** dialog. These are the symbology types in Surfer:

- *Unique Values* - Line, fill, and/or symbol properties are specified for unique values in the attribute field.
- *Unclassed Colors* - Colors from a color spectrum are applied to the features by numeric attribute value.
- *Unclassed Symbols* - Symbols are added for each polygon feature and scaled proportionally by numeric attribute value, or point features are scaled by numeric attribute value. *Unclassed Symbols* symbology is not applied to polylines.
- *Classed Colors* - Colors are applied to the features by classifying numeric attribute values.
- *Classed Symbols* - Symbols are added for each polygon feature and classified by a numeric attribute value, or point features are classified by a numeric attribute value. *Classed Symbols* symbology is not applied to polylines.
- *Pie Chart* - Attributes in the base layer are added as *Pie Chart* symbols with classified pie slices.

Editing Features

Some base maps consist of multiple objects (i.e. polygons, polylines, text, etc.). The sub-objects can be selected individually in the **Contents** window and edited in the **Properties** window. The expansion state of sub-objects in the **Contents** window is retained in the Surfer .SRF file. Use the *Expand new Contents window items* option in the Options dialog to control the expansion state of new objects in **Contents** window.

Moving Features

Features such as points, polylines, and polygons can be moved between base (vector) layers and other base (vector) layers the plot document. The Move/Copy to Layer command can be used to move or copy features. Features can also be moved in the Contents window. To move a feature to another base (vector) layer, select the feature and drag it to a new position within another base (vector) layer. To move a feature to the plot document, select the feature and drag it to a new position above, between, or below the top-level objects in the **Contents** window.

Base Layers with Vector and Raster Data

When importing a file that includes both vector and raster data, such as with some DXF or GSI files, a base (vector) layer is created. In this case, the image properties are controlled by clicking the *Image* object in the **Contents** window.

Base (raster) Layers

Base (raster) layers are created when an image file is used to create a base map. An image file is a raster data file where the cell values correspond to pixel color. Rasters files where the cell values describe other information, such as elevation, are used to create grid-based maps. Some images files are georeferenced, where the image includes real world coordinates. Base (raster) layers can be georeferenced in Surfer.

Georeferencing is necessary to place unreferenced images in the correct relative position in the coordinate system. Occasionally you may need to adjust a georeferenced image. Georeference an image by selecting the base (raster) layer and clicking **Map Tools | Layer Tools | Georeference Image**. You can also right-click the base layer in the **Contents** window or plot window and click **Georeference Image** in the context menu. Finally, you can georeference a base (raster) layer by clicking *Georeference image* in the General page for the base (raster) layer. The image must be at least 2x2 pixels to be georeferenced.

If you need to add vector features to a map with a base (raster) layer, add an empty base layer to the map.

Creating Base Maps

There are four commands for creating new base maps:

- The **Home | New Map | Base** command or the button create base maps from existing vector or image files. Use the base map properties in the Properties window to set the base map line, fill, symbol, font, and opacity properties. With some images, you can also reassign the base map coordinates. You can also set the base map coordinate system.
- The **Home | New Map | Base | Base from Data** command or the button create a base map from a data file. A data file containing XY data is used to create points in the base layer. Other data in the data file is added to the points as attribute data.
- The **Home | New Map | Base | Base from Server** command or the button allows you to download an image or vector data as a base map from an online web mapping server.
- The **Home | New Map | Base | Empty Basemap** command or the button create an empty base map with the intention of creating new geometry within the empty base map limits.

For example, the following steps are used to create a new base map from an existing file:

1. Click the **Home | New Map | Base** command, click the **Home | New Map | Base | Base from Data** command, click the **Home | New Map | Base | Base from Server** command, or click the **Home | New Map | Base | Empty Basemap** command.

2. Depending on which command was used, a dialog is respectively displayed.
   - For the **Base** command, the Import dialog is displayed with a list of files in all the format types that can be imported. Select a file in the list and click the *Open* button. Sometimes an **Import Options** dialog for the file type is displayed. Make any changes to the import options and click *OK*.
   - For the **Base from Data** command, the Base from XY Data dialog is displayed. Specify the rows to include and set the X and Y data columns. Click *OK* to create a base layer.
   - For the **Base from Server** command, the Download Online Maps dialog is displayed. Select or add the server and layer, specify the area to download and resolution, and click *OK* to download the server base map.
   - For the **Empty Basemap** command, the Base Map Limits dialog is displayed. Set the *Minimum* and *Maximum X* and *Y* values and click *OK*. A new empty base map is created.

The map is automatically created with reasonable defaults. Base maps are named with the name of the file from which the base map was created, such as *Base(vector)-CentralCalifornia.gsi*. New empty base maps are named *Base*. Base maps from servers are named with the map server name, such as Base(raster)-*Orthoimagery/USGS_EDC_Ortho_NAIP* or *Base(vector)-Interstate*. In addition, the image downloaded with the base map from server adds attribute information, including the server name and layer title.

Adding Base Layers

Select one of the four Add to Map base layer commands to add a base layer to an existing map:

- The **Home | Add to Map | Layer | Base** command adds a base map layer to the selected map.
- The **Home | Add to Map | Layer | Base from Data** command adds a base map layer from an XY data file to the selected map.
- The **Home | Add to Map | Layer | Base from Server** command allows you to add an image base map from an online web mapping server. When adding a base map layer from a server to an existing map with a defined coordinate system, the *Specify Latitude/Longitude extents* option is automatically selected. The boxes are filled in with the map limits of the current map.
- The **Home | Add to Map | Layer | Empty Base** command allows you to add an empty base map layer to an existing map.

Editing an Existing Base Map

To change the features of a base map, click once on the base map to select it. The properties are displayed in the **Properties** window.

Base (vector) Layer Properties

The vector base layer properties contains the following pages:

General
Labels
Layer
Coordinate System
Info

Individual objects in the vector base layer may have other pages, such as Line, Fill, or Text. Each object also has a Info page.

Base (raster) Layer Properties

The raster base layer properties contains the following pages:

General
Layer
Coordinate System
Info

Map Properties

The map properties contain the following pages:

View
Scale
Limits
Frame
Coordinate System
Info

Base Map from Data

Click the **Home | New Map | Base from Data** command or the button to create a new base map from an XY data file. Click the **Home | Add to Map | Layer | Base from Data** or **Map Tools | Add to Map | Layer | Base from Data** to add a base layer from an XY data file to an existing map. Each XY data point in the file is represented by a point object in the base map. Any other column data is added as attributes to the points in the base layer. If the data file includes a header row, the headers will be used for attribute names. The data file must contain at least two rows and two columns of data to create a base map.

The points in the base map can be edited, queried, and geoprocessed with the Features tab commands.

To create a base map from XY data,

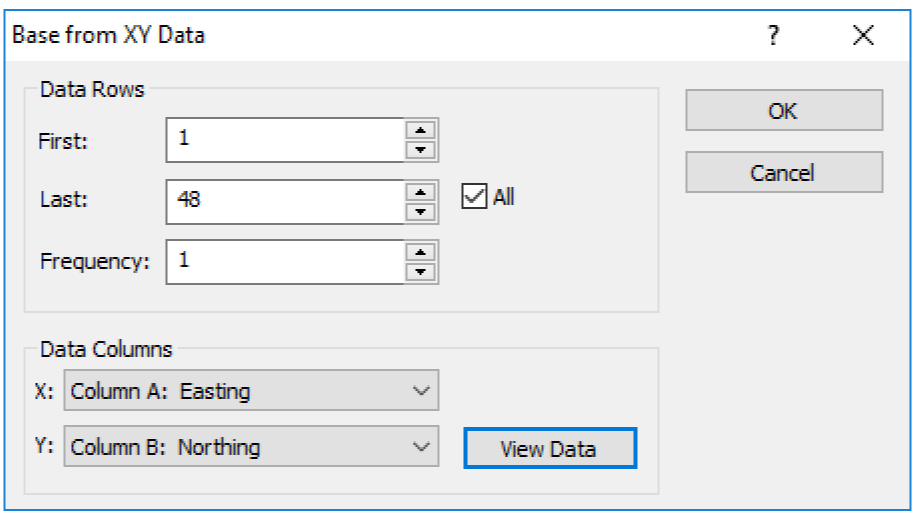
1. Click the **Home | New Map | Base | Base from Data** command.
2. Select the data file in the Open Data dialog and click *Open*.
3. The **Base from XY Data** dialog is displayed.
4. Specify the rows you wish to include in the base layer and which columns contain the X and Y coordinates.
5. Click *OK*.

Base from Data Layer vs Post Layer

The **Base from Data,** post, and classed post layer types use XY data files to create map layers. Post and classed post layers are a single object, and therefore post and classed post maps use less memory and are faster to render. Post or classed post maps should be used for large data files. Post and classed post maps have more control over labels and can be included in legends. Base layers include each point as an object. These point objects can be edited and queried after creating the base layer. The points in the base layer also retain attribute information, and therefore a symbology can be applied to the base layer.

Base from XY Data Dialog

The **Base from XY Data** dialog is displayed when creating a base layer. Specify the rows you wish to include in the base layer and which columns contain the X and Y coordinates in the **Base from XY Data** dialog.



*Specify data rows and columns in the **Base from XY Data** dialog.*

Data Rows

The *Data Rows* group controls which rows from the data file are included in the base layer.

First

Specify the first row you wish to include in the base layer in the *First* field. Type the row number in the *First* field or click the buttons to change the value.

Last

Specify the last row you wish to include in the base layer in the *Last* field. Type the row number in the *Last* field or click the buttons to change the value.

Frequency

Specify the number of rows included between the first and last row in the *Frequency* field. Use the *Frequency* field to reduce the number of points created from large data files. A *Frequency* of 1 includes every row. A *Frequency* of 2 includes every second row after the *First* row. A *Frequency* of 3 includes every third row, 4 includes every fourth row, etc. The *First* row is always included.

For example the Demogrid.dat sample data file includes a header row and 47 rows of XYZ Data. When *First* is set to 1, *Last* is set to 48, and *Frequency* is set to 2, rows 1, 3, 5, 7, ...47 are included in the base map. For Demogrid.dat row 1 is excluded because it is a header and contains text data. When *First* is set to 2, *Last* is set to 48, and *Frequency* is set to 2, rows 2, 4, 6, ...48 are included in the base map.

All

Check the *All* check box to include all data in the data file in the base map. The *First* value will change to 1, the *Last* value will change to the last row containing data, and the *Frequency* will change to 1 when the *All* check box is clicked.

Data Columns

The *Data Columns* group controls which columns are used for the X and Y coordinates when creating the base layer.

X and Y Coordinates

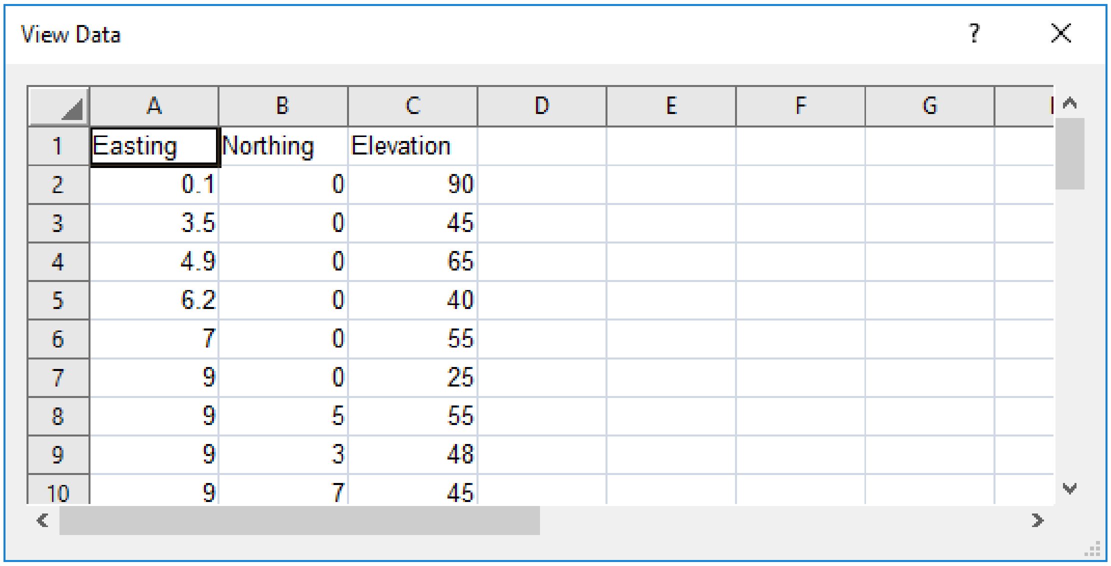
Specify the column containing X coordinates in the *X* field. Click the current selection and click the desired column in the list to change the value. Only columns containing data will be included in the *X* list. Specify the column containing Y coordinates in the *Y* field. Click the current selection and click the desired column in the list to change the value. Only columns containing data will be included in the *Y* list. If the data file contains a header row, the column names will be included next to the column letter.

View Data

Click *View Data* to view the data file in a read-only worksheet window. This window is useful if you need to see the data before determining the *X* and *Y* columns or what you want to use for *First*, *Last*, and *Frequency* values.

View Data

The *View Data* button in the Grid Data and Base from XY Data dialogs displays a preview of the data file in the **View Data** dialog. Use the **View Data**dialogto be sure you are selecting the correct X, Y, and Z data columns in the **Grid Data** or **Base from XY Data** dialog.



*The **View Data** dialog displays a preview of the data file.*

Base Map from Server

Click the **Home | New Map | Base | Base from Server** command or the  button, the **Home | Add to Map | Layer | Base from Server**command, or the **Map Tools | Add to Map | Layer | Base from Server** command to download a base map from an online web server. Clicking one of the preceding commands opens the Download Online Maps dialog. Use the **Download Online Maps** dialog to select the server, map extents, and image resolution.

When using the **Download Online Maps** dialog, a server with appropriate data must be selected. Surfer currently supports four types of servers, web mapping service (WMS), Open Street Map (OSM), web coverage service (WCS), and web feature service (WFS). WMS, OSM, WCS, and WFS servers provide different types data.

- When selecting a WMS or OSM server, an image is downloaded to be used as a base (raster) layer.
- When selecting a WCS server, data values are downloaded to be used as a grid file. WCS servers cannot be used with the **Base from Server** command. Use the Grid from Server command or the *Download* button in the Open Grid dialog instead.

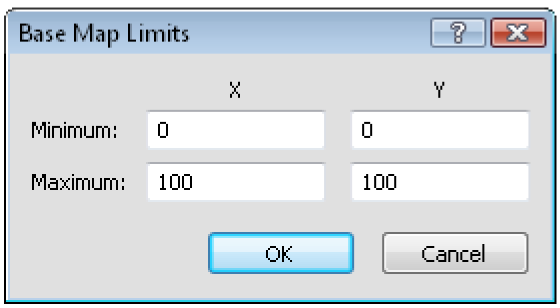
- When selecting a WFS server, vector data is downloaded with points, polylines, and/or polygons representing features in a base (vector) layer.

Empty Base Map

The **Home | New Map | Base | Empty Basemap** command or the  button create an empty base map. An empty base map is often created with the intention of creating new geometry within the base map limits. The **Home | Add to Map | Layer | Empty Basemap** command adds an empty base map layer to an existing map. When an empty base layer is added to an existing map, the base layer is created with the same coordinate system as the map frame.

Base Map Limits Dialog

The **Home | New Map | Base | Empty Basemap** command opens the **Base Map Limits** dialog.



*Use the **Base Map Limits** dialog to specify Minimum and Maximum values.*

Minimum and Maximum X and Y

Enter the limits of the new empty base map in the *Minimum* and *Maximum* boxes for the *X* and *Y* coordinates. This specifies the initial limits of the new base map, but may be overridden on the map Limits page later if needed.

Export Drawn Objects in Map Units

To export drawn objects in map units, you can either draw them on a base layer and export that layer or you can turn off the visiblity of the map and export the drawn objects in map units. To move or scale drawn objects with the map, you can either add draw them on a base layer, export the objects and import them as a new base layer, or copy/paste them into a base layer.

To draw objects on a base layer:

1. Select your existing map and click **Home | Add to Map | Layer | Empty Base**.

2. Click the **Features** tab and select the item you want to draw (text, polylines, polygons, symbols, etc).

3. Draw the objects over the map. Create as many objects as you wish.

4. If you want a separate base map file with these objects, hide the original map layer and all the axes in the **Contents** window (clear their visibility check boxes), and click **File | Export** to export the new base layer and drawn objects to a new file (e.g. BLN or DXF).

If the objects are already drawn on top of the map, and you just want to export them in map units, you can turn off the visiblity of the map and export the objects. To do this:

1. Clear the map layer visibility check box in the **Contents** window, so it is invisible. Hide any other objects, so only the objects you want to export are visible.

2. Click **File | Export**.

3. Give the file a name, choose the export format and click *Save*.

4. In the **Export Options** dialog, select *Map* as the *Scaling source*. The *File Rectangle* coordinates should update with map units.

5. Click *OK* the file is exported. You can turn on the visibility of the map.

If the objects are already drawn on top of the map, and you want to add them to be part of the map to be moved and scaled with the map, you can either export the objects (as above) and then add them as a new base layer to the map. Or, you can create a new base layer and then copy and paste them into it. If you copy and paste them, any formatting of the objects (e.g. fill color, fill pattern) will be retained. To do this:

1. Select the drawn objects to be added and click **Home | Clipboard | Copy**.

2. Select the map and click **Home | Add to Map | Layer | Empty Base**.

3. Click **Home | Clipboard | Paste**. The objects are pasted in the base layer.

4. Select the pasted objects and move them so that they are directly over the original objects. To move them, you can click and drag, arrow keys on the keyboard or type the object locations into the *X:* and *Y:* fields in the Position/Size Toolbar.

5. You can delete the original objects if you wish, or hide them by clearing their visibility check boxes in the **Contents** window.

6. If you want a separate base map file with these objects, hide the original map layer and all the axes in the **Contents** window, select the map and click **File | Export**. In the **Export** dialog, check *Selected objects only*, and export the base layer with the drawn objects to a new file (e.g. BLN or DXF).

Placing Boundaries on Other Maps

Base maps can be used to place boundary features on any type of map. To place boundary lines on a map, add a base map layer to the existing map with the **Map Tools | Add to Map | Layer | Base** command.

Coordinates

In **Surfer**, you can use map layers with different coordinate systems by setting the source coordinate system for each map separately. **Surfer** displays the map in the target coordinate system by changing the view of the individual maps. For example, consider a contour map based on latitude/longitude coordinates. To overlay a boundary map on the contour map, the boundary map must cover the same general range as the contour map. But, the coordinate system can be any coordinate system for this location. For example, the *CoordinateSystems.srf* sample file includes three layers with two different source coordinate systems.

Limits and Scale

When you overlay maps, if the new map limits exceed that of the current map limits, **Surfer** gives you the option to adjust the map limits to include all layers.

To add a boundary base map to another map:

1. Create a map with the **Home | New Map** command. The grid file should use limits that cover the same general extent as the boundary base map.

2. Select the map and click the **Map Tools | Add to Map | Layer | Base** command to add a base map layer.

3. In the **Import** dialog, select your boundary file and click the *Open* button.

4. Set any file import properties, if required, and click *OK*.

5. To clip the map to specific limits, click on the *Map* object in the Contents window.

6. Open the Limits page in the Properties window, or use the Map Tools | Map Tools | Set Limits command.

7. If necessary, set the scale on the Scale page.

The map automatically redraws to the new limits and size.

Changing Properties in a Base Map

Properties can be assigned to base map polylines, polygons, text, symbols, or images. Any properties you assign are applied to all the objects in the base map. The only properties that can be changed in images are the coordinates.

Change All Objects

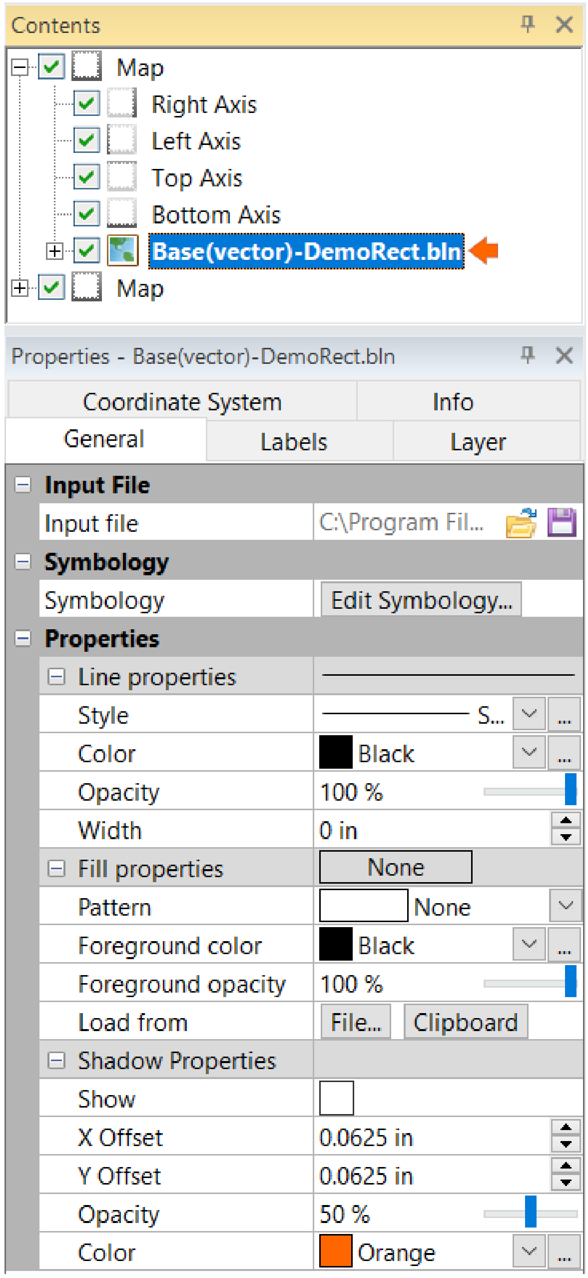
To change the properties of all objects in a base map:

1. Click on the *Base* map layer in the Contents window.

2. On the General page in the Properties window, click the ⊞ button next to the type of property you want to change (i.e. *Line Properties*, *Fill Properties, Font Properties*, or *Symbol Properties*). If the option is not present, the base map does not contain any objects of that type. For example, if the base map does not contain any polygons, the *Fill Properties* option is not available.

3. Specify the properties you want to assign to the objects in the base map.

The base map is automatically redrawn. The specified properties are applied to all the polyline, polygon, text, or symbols in the base map.



Select the Base map layer to change properties for all of the objects.

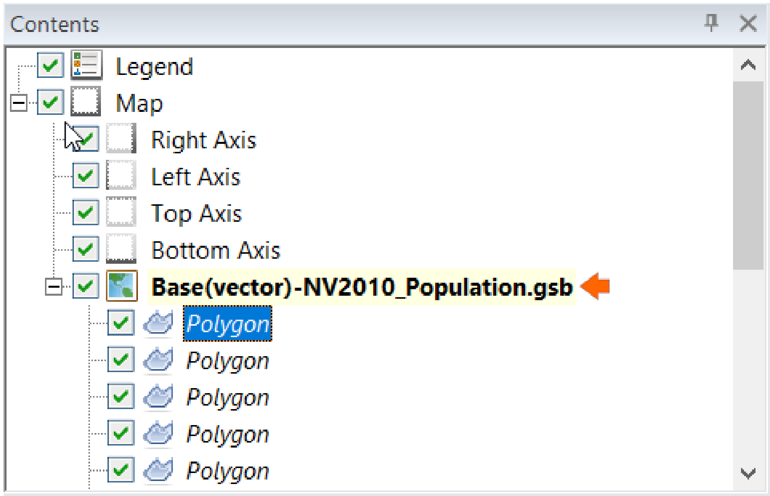
Change One Object

To change the properties of an individual object in a base map:

1. Click the object within the base map layer (i.e. Polygon, Polyline, etc.) in the **Contents** window.

2. Make changes to the properties available for that object on the appropriate page (i.e. **Line** page, **Fill** page, etc.) in the **Properties** window.

The property change is automatically shown for the individual object.



Select the sub-object to change individual properties.

Changing Multiple Objects

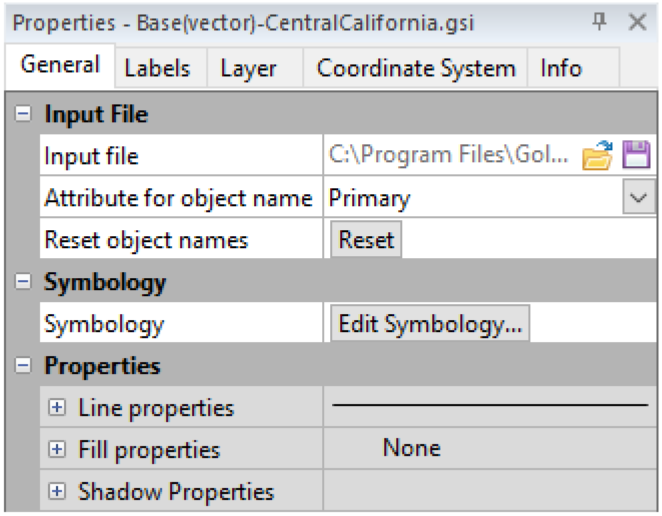
To change the properties of multiple objects in a base map:

1. Click the *Base* map layer in the **Contents** window.
2. Use the SHIFT or CTRL key to select multiple sub-objects of the base map layer.
3. The shared properties are displayed in the **Properties** window. Make changes as needed. The objects will automatically update to reflect the new properties.

Base (vector) Layer General Properties

The **General** page in the base map properties contains options for setting feature properties. The properties can be applied based on attribute values, or properties can be the same for all objects within the layer.

The base map properties **General** page contains the following options:



*The **General** page specifies the* Input file, Symbology, *and shared properties.*

Input File

The *Input File* lists the current file used in the base map.

Change File

Click the button to display the **Import** dialog. This allows you to select a new file or an updated version of the current file used to create the base map. If the file exceeds the current map limits, you will be prompted to adjust the map limits.

A vector file must be selected when changing the *Input file* for a base (vector) layer. An error message will be displayed if a raster file is selected in the **Import** dialog. If you wish to add an image to the map, select the **Home | Add to Map | Layer | Base** command instead.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

The button displays the **Export** dialog. Type a *File name* and change the *Save as type* to the desired file format. Click *Save.* For some file types, a file type specific **Export Options** dialog appears. The **Export Options** dialog Spatial

References and Scaling pages are available when saving a base layer. Check the desired file formats. It is recommended that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Set any export options and click *OK* to save the file.

Attribute for Object Name

The *Attribute for object name* shows the attribute field from the Attribute Table. The default is *None*. The names shown in the **Contents** window for the objects in the base layer correspond with their attribute value from the attribute field selected. Objects without attribute values are named according to their object type, e.g. Polygon. All objects are named according to their object type when None is selected. Changes made to the object's attribute value in the attribute table will be shown in the object's name in the **Contents** window when the attribute field is selected. Changing an object name in the **Contents** window using Rename Object does not change the object's attribute value.

Reset Object Names

To reset all custom object names, click the *Reset* button to return the object names to their attribute values.

Symbology

Symbology uses symbols or colors to display statistical information about the features in the base layer. Symbology applies line, fill, and/or symbol properties to features in the base layer depending on an attribute value. The type of symbology and the layer's appearance are controlled in the **Symbology** dialog. Click *Edit Symbology* in the Base (vector) layer **Properties** window **General** page to open the **Symbology** dialog. These are the symbology types in Surfer:

- *Unique Values* - Line, fill, and/or symbol properties are specified for unique values in the attribute field.
- *Unclassed Colors* - Colors from a color spectrum are applied to the features by numeric attribute value.
- *Unclassed Symbols* - Symbols are added for each polygon feature and scaled proportionally by numeric attribute value, or point features are scaled by numeric attribute value. *Unclassed Symbols* symbology is not applied to polylines.
- *Classed Colors* - Colors are applied to the features by classifying numeric attribute values.
- *Classed Symbols* - Symbols are added for each polygon feature and classified by a numeric attribute value, or point features are classified by a numeric attribute value. *Classed Symbols* symbology is not applied to polylines.
- *Pie Chart* - Attributes in the base layer are added as *Pie Chart* symbols with classified pie slices.

None can also be selected. When symbology is not used, the properties are controlled for all features in the *Properties* section of the **General** page for the base layer or controlled for individual features in the **Line**, **Fill** and/or **Symbol** Properties window pages.

Properties

The *Properties* section controls the properties for all objects in the base layer when no symbology is applied. When *Symbology* is applied to the objects, the *Properties* section controls the properties for any object not included in the symbology and for properties of the objects that are not determined by the *Symbology* settings. For example, when *Unclassed Colors* is used for polygon features the polygon fill properties are controlled by the symbology and the polygon line properties are controlled in the *Properties* section.

The *Properties* section is separated by property type:

- Click the ⊞ next to *Line Properties* to adjust the line properties for all objects in the base layer.
- Click the ⊞ next to *Fill Properties* to adjust the fill properties for all objects in the base layer.
- Click the ⊞ next to *Font Properties* to adjust the font properties for all objects in the base layer.
- Click the ⊞ next to *Symbol Properties* to adjust the symbol properties for all objects in the base layer.
- Click the ⊞ next to *Shadow Properties* to adjust the shadow properties for all objects in the base layer.

If no object exists in the base layer that uses a particular property type, that property type is not displayed.

Symbology

Vector base maps can include symbology. Symbology applies line, fill, and/or symbol properties to features in the base layer depending on an attribute value. The symbology can be included in a legend. The type of symbology and the layer's appearance are controlled in the **Symbology** dialog. Click *Edit Symbology* in the Base (vector) layer **Properties** window **General** page to open the **Symbology** dialog. The symbology types in **Surfer** are:

- *Unique Values* - Line, fill, and/or symbol properties are specified for unique values in the attribute field.
- *Unclassed Colors* - Colors from a color spectrum are applied to the features by numeric attribute value.
- *Unclassed Symbols* - Symbols are added for each polygon feature and scaled proportionally by numeric attribute value, or point features are scaled by numeric attribute value. *Unclassed Symbols* symbology is not applied to polylines.

- *Classed Colors* - Colors are applied to the features by classifying numeric attribute values.
- *Classed Symbols* - Symbols are added for each polygon feature and classified by a numeric attribute value, or point features are classified by a numeric attribute value. *Classed Symbols* symbology is not applied to polylines.
- *Pie Chart* - Attributes in the base layer are added as *Pie Chart* symbols with classified pie slices.

Adding Symbology to a Base Layer

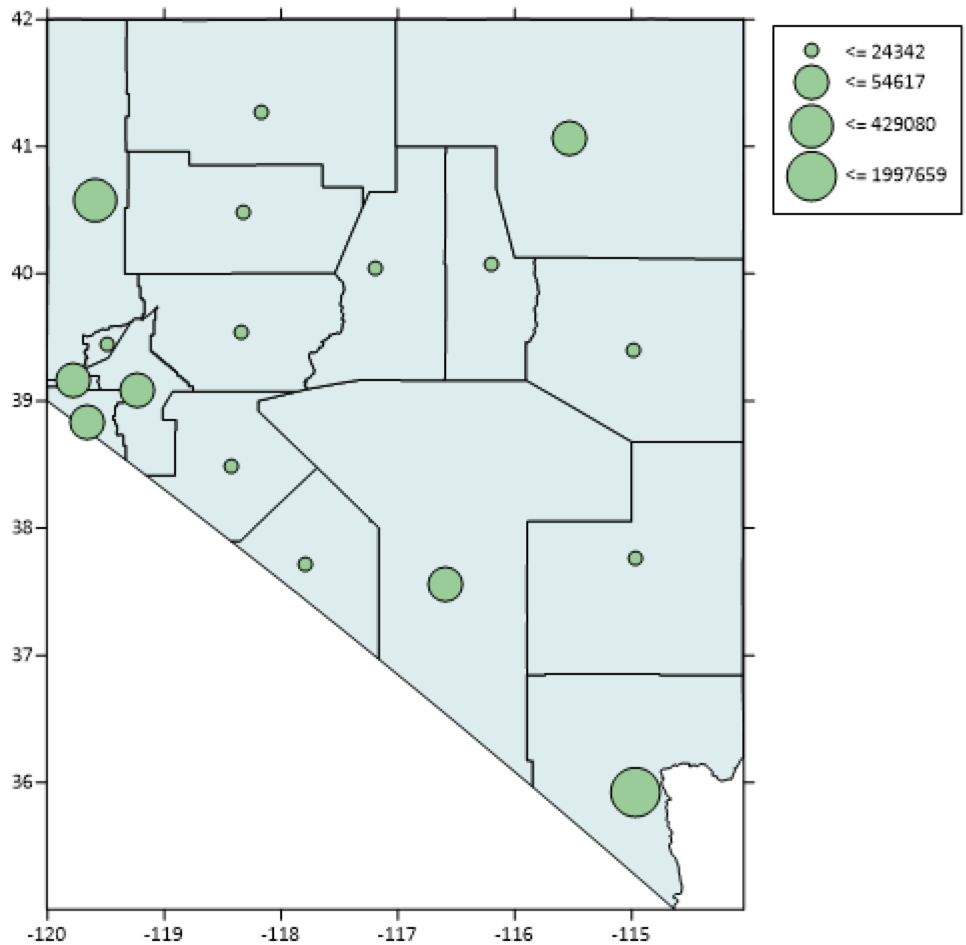
A symbology can be added to a base (vector) layer by clicking *Edit Symbology* in the **Properties** window **General** page. Select the symbology type, specify the attribute field for the symbology, and then specify the various line, fill, and/or symbol properties for the symbology in the **Symbology** dialog. Click *OK* or *Apply* to apply the symbology to the base layer.

To apply symbology, the features in the base (vector) layer must have at least one attribute field. Any of the five symbology types can be applied to an attribute field that contains numeric data. *Unique Values* symbology can be applied to text or numeric data. Add or edit attribute data in the base layer with the Attribute Table.

The following example uses the NV2010_Population.gsb sample file to create a base map with a symbology:

1. Create the base map by clicking **Home | New Map | Base**.
2. Select the *NV2010_Population.gsb* sample file in the Import dialog. The sample files are located at *C:\Program Files\Golden Software\Surfer\Samples* by default.
3. Click *Open* in the **Import** dialog. The base (vector) map is created.
4. Click *Base(vector)-NV2010_Population.gsb* in the Contents window to select the base layer.
5. Click **General** in the Properties window to view the General page.
6. Click *Edit Symbology* in the *Symbology* field of the **Properties** window **General** page. The **Symbology** dialog opens.
7. Click *Classed Symbols* on the left side of the **Symbology** dialog to display the Classed Symbols page.
8. If *Population* was not automatically selected in the *Attribute field*, click the current selection and select *Population* from the *Attribute field* list in the *General* properties section.
9. In the *Classes* properties section, select *Natural breaks* from the *Class method* list.
10. Click the button to reduce the *Class counts* to *4*.
11. In the *Symbol* properties section, select *Square root* from the *Scaling method* list.
12. Click the ⊞ next to *Symbol properties* to expand the symbol properties.
13. Change the symbol by selecting the filled circle *(Symbol 12)* in the *Symbol* property list.

14. Change the symbol *Fill color* to *Faded Green*.
15. Click *OK* and the classed symbol symbology is added to the map.
16. Click the Map Tools | Add to Map | Legend command to add a legend to the base map. The *Classed Symbols* legend shows the upper class limit for each class.

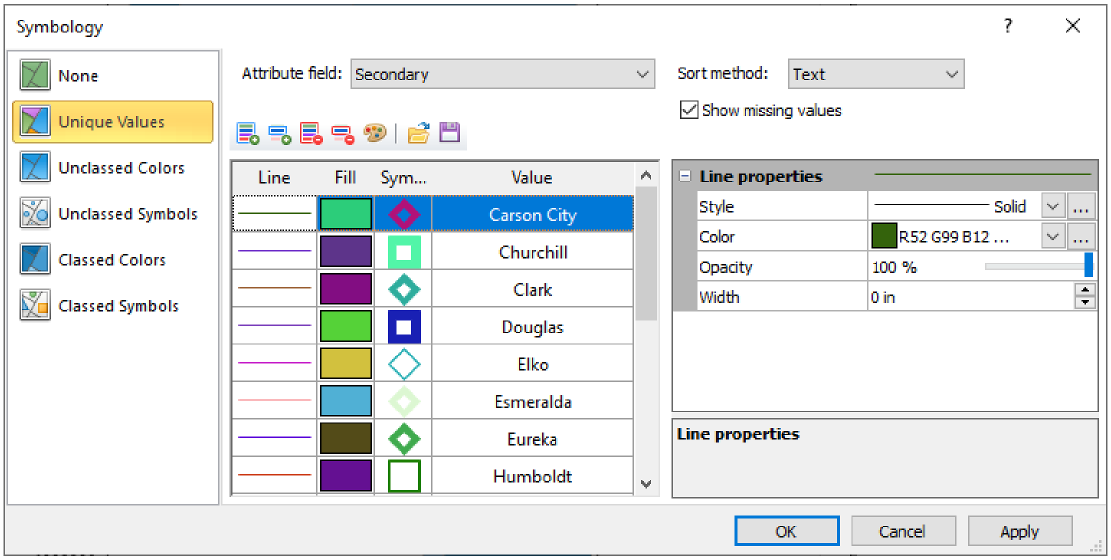


The symbology classifies the counties in Nevada by population and displays proportionally sized symbols to indicate the class. The legend indicates the maximum population for each class.

Symbology - Unique Values

The *Unique Values* symbology applies line, fill, and/or symbol properties to the features in the base layer for the values in the attribute field. Display properties for some or all of the unique values in the attribute field can be specified. Line properties are applied to polyline and polygon features. Fill properties are applied to polygon features. Symbol properties are applied to point features.

The *Unique Values* symbology does not add features to the map. For example, setting the symbol properties for a base layer without point features makes no change to the base layer appearance.



Select Unique Values *to specify properties for unique values in the* Attribute field.

Attribute Field

The *Attribute field* specifies which attribute is used to define the features' properties in the base layer. Select the desired attribute from the *Attribute field* list. When the *Attribute field* value is changed, the values in the *Values List* are removed.

The first attribute for the objects in the layer is the default selection in the *Attribute field* when a symbology is selected for the first time. The *Attribute field* value is not changed when *Unique Values* is selected and a symbology is already in use.

Sort Method

The *Sort Method* specifies the sort order for the unique values in the **Symbology** and **Add Unique Values** dialogs and the legend. The *Sort Method* automatically updates when an *Attribute field* is selected. If the first ten values in the selected *Attribute field* are numeric, then the *Sort Method* becomes *Numeric*. Otherwise, the *Sort Method* is set to *Text*.

Select *Text* to sort the unique values alphabetically from A to Z. Select *Number* to sort the unique values from low to high. Select *None* to use the same order as the features in the Contents window in the **Symbology** dialog and to sort the values alphabetically in the **Add Unique Values** dialog.

Show Missing Values

Select the *Show missing values* option to include any features that do not have an attribute value equal to one of the values in the *Values list* in the map. When the *Show missing values* option is checked, features that are not represented by a value in the *Values list* are displayed, and their properties are controlled by the base layer General page, as well as their individual Line, Fill, and/or Symbol properties.

Clear the *Show missing values* check box to hide the features that are not represented in the *Values list* from the base layer. The features will still be visible in the **Contents** window, but the features will not be included in the base layer in the plot window.

Values List

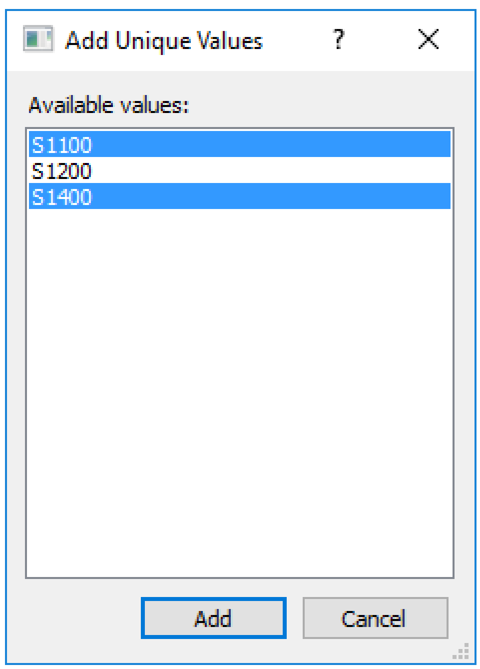
The values list displays the selected unique values and their associated properties. Values are added and removed by clicking the command buttons above the list. Click the *Line*, *Fill*, or *Symbol* property preview to display the properties in the properties pane. The properties preview is updated automatically when changes are made in the properties pane.

Add All

Click the **Add All** command to add all unique values in the *Attribute field* to the values list.

Add Values

Click the **Add Values** command to add one or more unique values to the values list. Select the desired values in the **Add Unique Values** dialog. The **Add Unique Values** dialog displays all available unique values in the *Attribute field*. Values that have already been added to the **Symbology** dialog are not displayed in the *Available values* list.



Select unique values to add to the values list .

Click a value to select the value. Hold CTRL to select multiple values. Click a value, hold SHIFT, and click a second value to select a contiguous group of values. When you have selected the desired unique values, click *Add* to add the values to the values list in the **Symbology** dialog. Click *Cancel* to close the **Add Unique Values** dialog without adding any values to the list.

Remove All

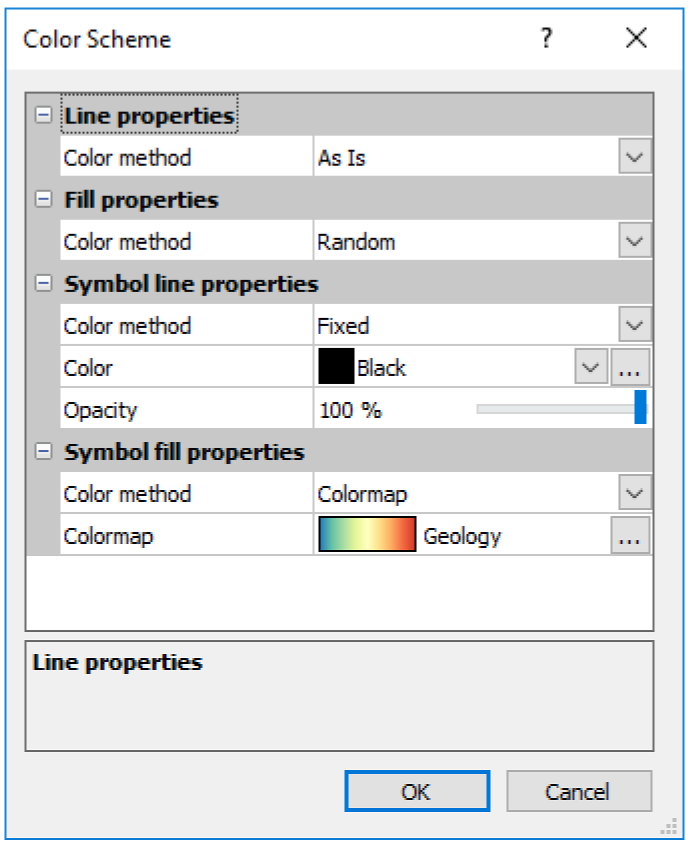
Click the **Remove All** command to remove all unique values from the values list.

Remove

Click the **Remove** command to remove the selected value from the list.

Color Scheme

Click the **Color Scheme** command to apply a color scheme to the line, fill, and/or symbol properties in the unique values list in the **Color Scheme** dialog.



*Select color scheme properties to change the color properties for every unique value in the **Symbology** dialog.*

Select the *Color method* to apply a scheme to the unique values' line, fill, symbol line, and/or symbol fill properties. The *Line properties*, *Fill properties*, *Symbol line properties*, and *Symbol fill properties* are all applied independently.

- Select *As Is* to make no changes to the current colors for the unique values. *As Is* is selected by default each time the **Color Scheme** dialog is opened.
- Select *Random* to apply random colors to the unique values.
- Select *Fixed* to apply a single *Color* and *Opacity* to all unique values.
- Select *Colormap* to apply a colormap to the unique values properties. The colormap is applied in the order the unique values appear in the **Symbology** dialog.

Save and Load Values

Click the **Save Values** command to save the unique values and their properties to a Symbology Unique Values File (*.ssuv) in the Save As dialog. The **Save Values** command is only available after at least one value has been added to the current table.

Click the **Load Values** command to load unique values with their properties from a Symbology Unique Values File (*.ssuv) in the Open dialog. Values from the file are added to the *Unique Values* table. If the file includes a value that is already listed in the *Unique Values* table, the properties of the existing value are maintained and the loaded value is ignored.

Line, Fill, and Symbol Properties

The line, fill, and symbol properties for the selected value are displayed on the right side of the **Symbology** dialog. No properties are displayed when no value is selected.

- To edit the line properties for a value, click the line preview in the row for the value. The line properties are applied to any polylines and/or polygons with this value in the *Attribute field*.
- To edit the fill properties for a value, click the fill preview in the row for the value. The fill properties are applied to any polygons with this value in the *Attribute field*.
- To edit the symbol properties for a value, click the symbol preview in the row for the value. The symbol properties are applied to any points with this value in the *Attribute field*.

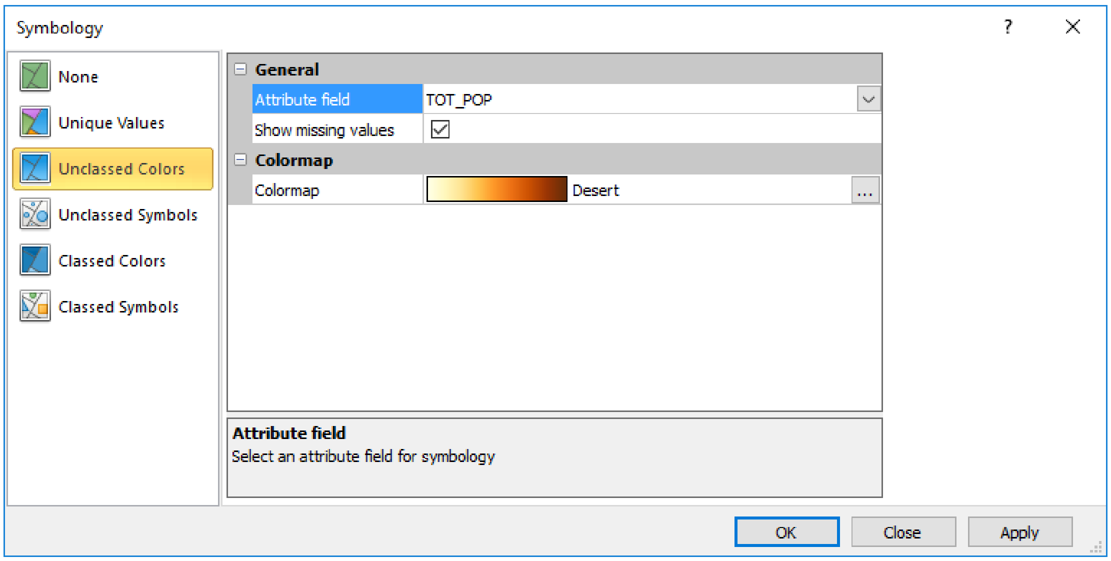
OK, Cancel, and Apply

Click *OK* to save your changes, close the **Symbology** dialog, and update the base layer symbology. Click *Cancel* to close the **Symbology** dialog without making changes to the base layer symbology. Click *Apply* to apply your changes to the base layer while keeping the **Symbology** dialog open.

Symbology - Unclassed Colors

The *Unclassed Colors* symbology applies line, fill, or symbol fill color to the features in the base layer. The color is specified by the attribute value and the colormap. When used with normalized data, population per area for example, the *Unclassed Colors* symbology creates a choropleth map.

- The attribute value and colormap determine the line color for polyline features.
- The attribute value and colormap determine the fill color for polygon features. Polygon line color is set on the base layer General page.
- The attribute value and colormap determine symbol fill color for point features. Point symbol line color is set on the base layer General page.



Select Unclassed Colors *to map a color spectrum to the features by attribute value.*

Attribute Field

The *Attribute field* specifies which attribute is used to define the features' properties in the base layer. Select the desired attribute from the *Attribute field* list. Only attributes with numeric values are displayed in the *Attribute field* list when the *Unclassed colors* symbology is selected.

The first numeric attribute for the objects in the layer is the default selection in the *Attribute field* when a symbology is selected for the first time. When a base layer symbology using a numeric attribute is already applied, the *Attribute field* value is not changed when *Unclassed Colors* is selected. The *Attribute field* changes to the first numeric attribute for the objects in the layer when *Unclassed Colors* is selected and the base layer is using a text attribute for the current symbology.

Show Missing Values

Select the *Show missing values* option to include any features that do not have an attribute value for the selected *Attribute field* in the layer. When the *Show missing values* option is checked, features that do not have an *Attribute field* value are displayed, and their properties are controlled by the base layer General page, as well as their individual Line, Fill, and/or Symbol properties.

Clear the *Show missing values* check box to hide the features that do not have an *Attribute field* value from the base layer. The features will still be visible in the **Contents** window, but the features will not be included in the base layer in the plot window.

Colormap

The *Colormap* is the color spectrum that is assigned to the features in the base layer. Select a predefined colormap from the *Colormap* list. Click the ... to modify or create a colormap in the Colormap Editor.

Beyond selecting the colors for the color spectrum, the **Colormap Editor** is used to apply linear or logarithmic scaling, use attribute value minimum and maximum or fixed values, and apply transparency to colors in the colormap. Color spectrums can be saved and/or loaded with the **Colormap Editor**.

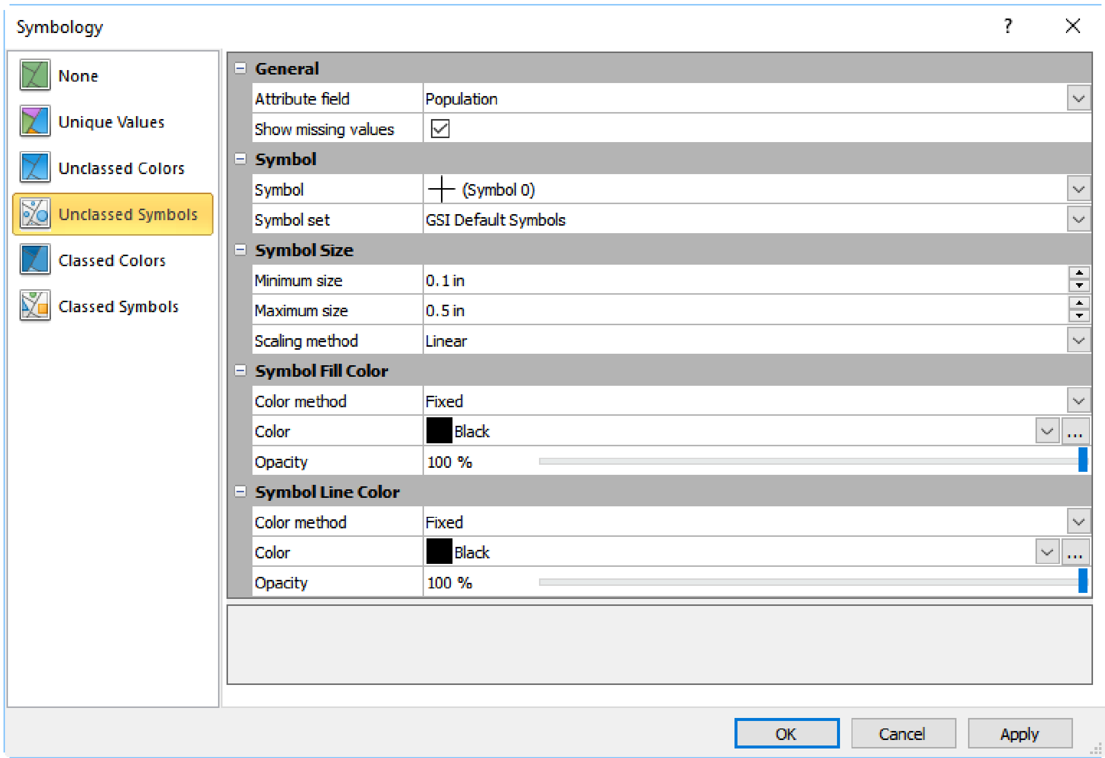
OK, Cancel, and Apply

Click *OK* to save your changes, close the **Symbology** dialog, and update the base layer symbology. Click *Cancel* to close the **Symbology** dialog without making changes to the base layer symbology. Click *Apply* to apply your changes to the base layer while keeping the **Symbology** dialog open.

Symbology - Unclassed Symbols

The *Unclassed Symbols* symbology adds proportionally scaled symbols to the map. The proportional symbol map is useful for indicating the relative difference in a value between areas.

When the *Unclassed Symbols* symbology is applied to polygon features, symbols are created in addition to the polygon features. When the *Unclassed Symbols* symbology is applied to point features, the point feature symbol properties are controlled by the symbology. *Unclassed Symbols* symbology is not applied to polyline features.



Select Unclassed Symbols *to add proportionally scaled symbols to the map.*

General

The *General* section includes the *Attribute field* and *Show missing values* properties.

Attribute Field

The *Attribute field* specifies which attribute is used to scale the symbols. Select the desired attribute from the *Attribute field* list. Only attributes with numeric values are displayed in the *Attribute field* list when the *Unclassed symbols* symbology is selected.

The first numeric attribute for the objects in the layer is the default selection in the *Attribute field* when a symbology is selected for the first time. When a base layer symbology using a numeric attribute is already applied, the *Attribute field* value is not changed when *Unclassed Symbols* is selected. The *Attribute field* changes to the first numeric attribute for the objects in the layer when *Unclassed Symbols* is selected and the base layer is currently using a text attribute for the current symbology.

Show Missing Values

Select the *Show missing values* option to include any features that do not have an attribute value for the selected *Attribute field* in the layer. When the *Show missing values* option is checked, features that do not have an *Attribute field* value are displayed, and their properties are controlled by the base layer General page, as well as their individual Line, Fill, and/or Symbol properties.

Clear the *Show missing values* check box to hide the features that do not have an *Attribute field* value from the base layer. The features will still be visible in the **Contents** window, but the features will not be included in the base layer in the plot window.

Symbol

The properties in the *Symbol* section control the appearance of the symbols, including the *Symbol* and *Symbol set*.

Symbol Size

The *Symbol Size* section includes properties for scaling the symbols.

Minimum and Maximum Size

Specify the smallest and largest symbol sizes in the *Minimum size* and *Maximum size* fields. The feature with the minimum attribute value is created with the *Minimum size*. The feature with the maximum attribute value is created with the *Maximum* size. The remaining features' symbols are scaled proportionally between the *Minimum size* and *Maximum size* by the *Scaling method*.

Scaling Method

The *Scaling method* specifies which scaling type is used to determine symbol sizes between the minimum and maximum. Select *Linear* or *Square root* from the *Scaling method* list. The interpolation equations for the scaling methods are as follows:

Linear:	$H_x = [((Z_x - Z_1)/(Z_2 - Z_1)) * (H_2 - H_1)] + H_1$
Square Root:	$H_x = \left[\sqrt{(Z_x - Z_1)/(Z_2 - Z_1)} * (H_2 - H_1)\right] + H_1$

Square root scaling is commonly used with solid symbols to offset the fact that the area increases as a function of the symbol height squared. Square root scaling is essentially making the area of the symbol proportional to the Z value, rather than making the size of the symbol proportional to the Z value.

Symbol Fill and Line Colors

The *Symbol Fill Color* section specifies the symbol fill color method and color. The *Symbol Line Color* section specifies the symbol line color method and color. Symbol fill and line colors are set separately.

Color Method

The *Color method* property sets whether the symbols use a single color or the symbols are colored by their attribute value. Select *Fixed* to use a single color for all symbols. Select *Colormap* to color the symbols by their attribute values.

Color

Specify the symbol color when the *Color method* is *Fixed* in the *Color* field. Select a color from the color palette.

Opacity

Specify the symbol opacity when the *Color method* is *Fixed* in the *Opacity* field. Select a value between 100% (completely opaque) to 0% (completely transparent).

Colormap

Select a predefined colormap in the *Colormap* field when the *Color method* is *Colormap* . Create a custom colormap in the Colormap Editor by clicking the ... button.

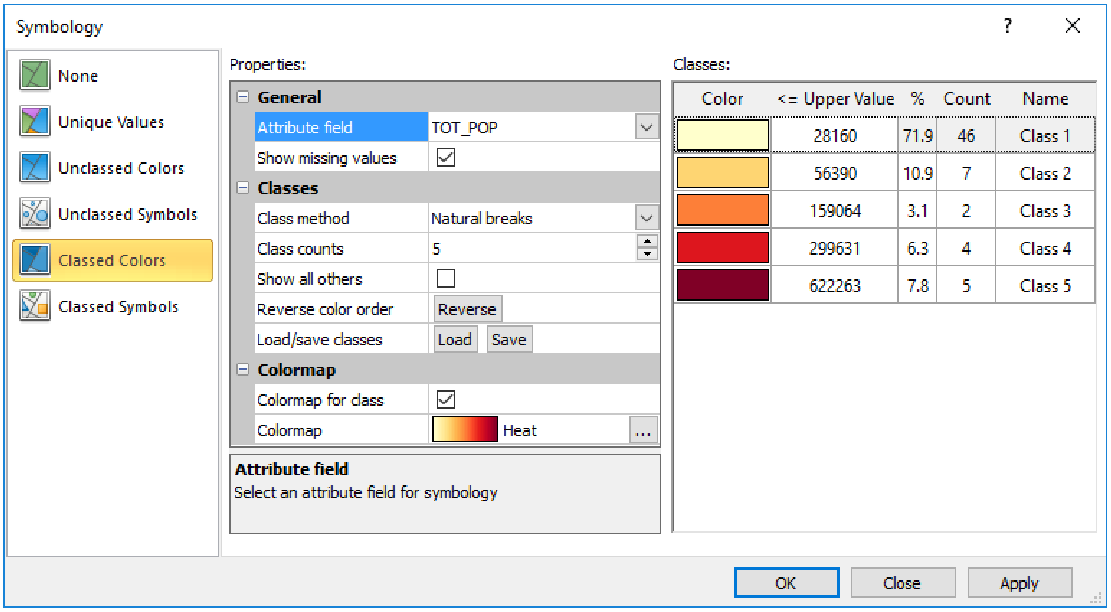
OK, Cancel, and Apply

Click *OK* to save your changes, close the **Symbology** dialog, and update the base layer symbology. Click *Cancel* to close the **Symbology** dialog without making changes to the base layer symbology. Click *Apply* to apply your changes to the base layer while keeping the **Symbology** dialog open.

Symbology - Classed Colors

The *Classed Colors* symbology applies line, fill, or symbol fill color to the features in the base layer by class. The features in the base layer are classified by the selected attribute. The colors for each class can be assigned one at a time or a colormap can be used to assign graduated colors across the classes. When used with normalized data, population per area for example, the *Classed Colors* symbology creates a choropleth map.

- The class color is applied to the line color for polyline features.
- The class color is applied to the fill color for polygon features. Polygon line color is set on the base layer General page.
- The class color is applied to symbol fill color for point features. Point symbol line color is set on the base layer General page.



Select Classed Colors *to color features by a classified attribute value.*

Properties

The *Properties* pane includes the properties for selecting the attribute and defining the classes. The *Properties* pane is in the middle of the **Symbology** dialog.

Attribute Field

The *Attribute field* specifies which attribute is used to classify the features in the base layer. Select the desired attribute from the *Attribute field* list. Only attributes with numeric values are displayed in the *Attribute field* list when the *Classed colors* symbology is selected.

The first numeric attribute for the objects in the layer is the default selection in the *Attribute field* when a symbology is selected for the first time. When a base layer symbology using a numeric attribute is already applied, the *Attribute field* value is not changed when *Classed Colors* is selected. The *Attribute field* changes to the first numeric attribute for the objects in the layer when *Classed Colors* is selected and the base layer is currently using a text attribute for the current symbology.

Show Missing Values

Select the *Show missing values* option to include any features that do not have an attribute value for the selected *Attribute field* in the layer. When the *Show missing values* option is checked, features that do not have an *Attribute field* value are displayed, and their properties are controlled by the base layer General page, as well as their individual Line, Fill, and/or Symbol properties.

Clear the *Show missing values* check box to hide the features that do not have an *Attribute field* value from the base layer. The features will still be visible in the **Contents** window, but the features will not be included in the base layer in the plot window.

Features that have an *Attribute field* value outside the class limits are controlled by the *Show all others* option. When *Show all others* is cleared, features with an *Attribute field* value outside the class limits are treated the same as missing values. When *Show all others* is selected, features with an *Attribute field* value outside the class limits are controlled by the *All others* class properties.

Class Method

The *Class method* specifies the method used to calculate the limits of the classes. There are five options available: *Equal number, Equal interval*, *Natural breaks* , *Standard deviation* , and *User defined*. To change the *Class method*, click on the current option and select the desired option from the list.

- *Equal number* assigns the class ranges such that approximately equal numbers of features are included in each class. Normally in this case, the interval of each class is different.
- *Equal interval* assigns the class ranges so the interval between the minimum and maximum values are equal for each class. Normally in this case, a different number of features is assigned to each class.
- *Natural breaks* uses the Fisher-Jenks algorithm to calculate the ideal natural breaks in the data. Typically these natural breaks are ideal beginnings and endings for data classes. The algorithm is based on Fisher's statement that any optimal classification of data consists of the sum of optimal classes of subsets of the data. To learn more about Jenks' natural breaks data classification method, please refer to Slocum, Terry A. *Thematic Cartography and Visualization.* New Jersey: Prentice-Hall, 1999.
- *Standard deviation* generates data classes based on the standard deviation of the data.
- *User Defined* allows you to set the *>=Minimum* value and the *<Maximum* value for each class individually. This allows you to specify your own ranges for the classes. Ranges defined in this way do not have to be contiguous. To change the *>=Minimum* or *<Maximum* value, double-click the values in the class list.

Class Counts

The *Class counts* value specifies the number of classes there will be on the map. When the *Class count* value is changed, the classes are automatically updated to reflect the change. To change the number of classes, click the button to increase or decrease *Class count* or highlight the existing number and type in a new value from 1 to 1000 classes.

Show All Others

Select the *Show all others* option to include a color for features where the attribute value does not fall within an existing class. Clear the *Show all others* check box to remove the *All others* class.

Features that do not have an *Attribute field* value are controlled by the *Show missing values* option. When *Show all others* is not selected, features where the *Attribute field* value is outside the class limits are treated the same as missing values.

Reverse Color Order

Click *Reverse* in the *Reverse color order* field to reverse the colors in the *Classes* list.

Load/Save Classes

Click *Load* to load an existing *Symbology Color Class (*.colorclass)* file. Click *Save* to save the current classes to a *Symbology Color Class (*.colorclass)* file. The class limits, names, and colors are saved in the *Symbology Color Class (*.colorclass)* file.

Colormap for Class

Select the *Colormap for class* option to apply the *Colormap* color spectrum colors to the classes. Changing the color of any class in the *Classes* list automatically clears the *Colormap for class* check box.

Colormap

The *Colormap* is the color spectrum that is assigned to the classes. Select a predefined colormap from the *Colormap* list. Click the ... to modify or create a colormap in the Colormap Editor.

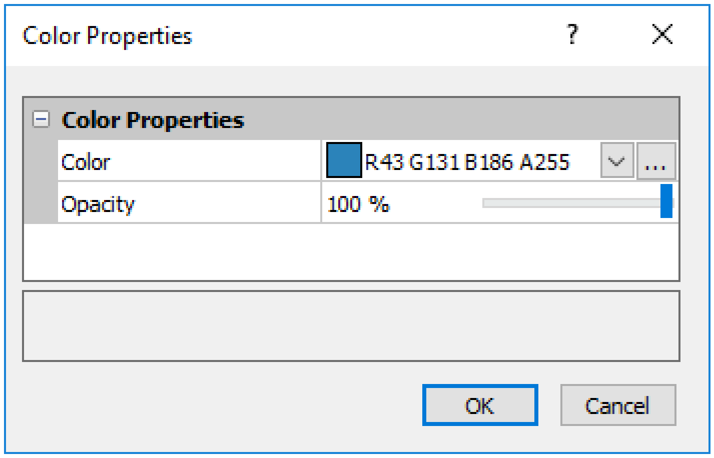
The *Colormap* property applies colors to the classes based on class number, not the data value. Therefore the *Data* section of the **Colormap Editor** does not have any effect on the class colors by default. We recommend you do not change the *Data* properties in the **Colormap Editor**. The first class is always assigned the minimum value color, and the last class is always assigned the maximum value color. Remaining classes are distributed equally across the colormap. The *Data* properties may change the class colors when *Logarithmic scaling* is used: The first class is always assigned the minimum value color, and the last class is always assigned the maximum value color. The remaining classes are distributed logarithmically across the colormap.

Classes

The *Classes* list displays the class color (*Color*), upper limit (*<= Upper Value*), proportion (%), count (*Count*), and class name (*Name*) for the classes. The *Classes* listis on the right side of the **Symbology** dialog.

Changing the Class Color

The *Color* column displays the colors for the classes. Click the color preview in the *Color* column to change the color for the class. The **Color Properties** dialog is displayed.



Specify the class Color *and* Opacity *in the* ***Color Properties*** *dialog.*

Specify the class color by selecting a predefined color from the color palette in the *Color* field, or click ... to select a color in the Colors dialog. Set the class opacity by clicking and dragging the slider or typing a value between 0 and 100 in the *Opacity* field. An *Opacity* of 100% is completely opaque. An *Opacity* of 0% is completely transparent.

Changing the Class Limits

The *<= Upper Value* column displays the upper limit for each class. The first class in the *Classes* list has no lower limit. Subsequent classes use the prior class upper limit for a class minimum. For example, in the image above *Class 2* includes values greater than 28160 (Class 1's upper limit) and less than or equal to 159064.

The class limits are automatically calculated when the *Class method* is *Equal number*, *Equal interval*, *Natural breaks*, or *Standard deviation*. The class limits must be individually defined when the *Class method* is *User defined*. Changing a class *<= Upper Value* automatically updates the *Class method* to *User defined*.

Type the desired class maximum in the *<= Upper Value* column to change the upper limit for a class. The proportion and count statistics are automatically updated when the *<= Upper Value* is changed.

Statistical Information about the Features in Each Class

The *Classes* list displays summary statistics for each class. The *%* and *Count* columns display statistical information about the class. The *%* column indicates the percentage of features in the particular class. The *Count* column indicates the number of features included in each class. These values cannot be directly edited and are for informational purposes only.

Changing the Class Name

The *Name* column displays the class names. Specify a class name in the *Classes* list to display class names in a legend. By default, classes are named *Class 1*, *Class 2*, etc. Type a name in the *Name* column to change the class name.

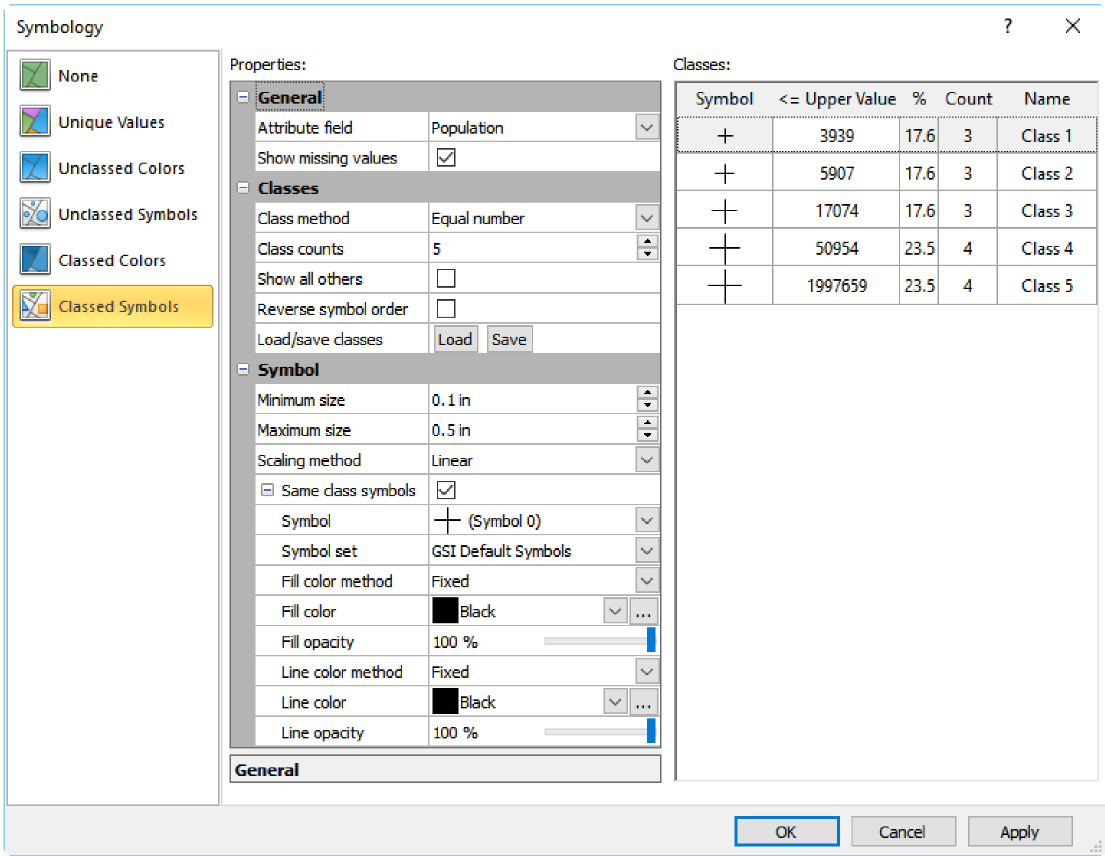
OK, Cancel, and Apply

Click *OK* to save your changes, close the **Symbology** dialog, and update the base layer symbology. Click *Cancel* to close the **Symbology** dialog without making changes to the base layer symbology. Click *Apply* to apply your changes to the base layer while keeping the **Symbology** dialog open.

Symbology - Classed Symbols

The *Classed Symbols* symbology adds proportionally sized symbols to the features by class. The features in the base layer are classified by the selected attribute. The symbols for each class can be assigned individually or graduated symbols can be assigned across the classes.

When the *Classed Symbols* symbology is applied to polygon features, symbols are created in addition to the polygon features. When the *Classed Symbols* symbology is applied to point features, the point feature symbol properties are controlled by the symbology. *Classed Symbols* symbology is not applied to polyline features.



Select Classed Symbols *to add symbols to the map by a classified attribute value.*

Properties

The *Properties* pane includes the properties for selecting the attribute and defining the classes. The *Properties* pane is in the middle of the **Symbology** dialog.

Attribute Field

The *Attribute field* specifies which attribute is used to classify the features in the base layer. Select the desired attribute from the *Attribute field* list. Only attributes with numeric values are displayed in the *Attribute field* list when the *Classed colors* symbology is selected.

The first numeric attribute for the objects in the layer is the default selection in the *Attribute field* when a symbology is selected for the first time. When a base layer symbology using a numeric attribute is already applied, the *Attribute field* value is not changed when *Classed Symbols* is selected. The *Attribute field* changes to the first numeric attribute for the objects in the layer when *Classed Symbols* is selected and the base layer is currently using a text attribute for the current symbology.

Show Missing Values

Select the *Show missing values* option to include any features that do not have an attribute value for the selected *Attribute field* in the layer. When the *Show missing values* option is checked, features that do not have an *Attribute field* value are displayed, and their properties are controlled by the base layer General page, as well as their individual Line, Fill, and/or Symbol properties.

Clear the *Show missing values* check box to hide the features that do not have an *Attribute field* value from the base layer. The features will still be visible in the **Contents** window, but the features will not be included in the base layer in the plot window.

Features that have an *Attribute field* value outside the class limits are controlled by the *Show all others* option. When *Show all others* is cleared, features with an *Attribute field* value outside the class limits are treated the same as missing values. When *Show all others* is selected, features with an *Attribute field* value outside the class limits are controlled by the *All others* class properties.

Class Method

The *Class method* specifies the method used to calculate the limits of the classes. There are five options available: *Equal number, Equal interval*, *Natural breaks*, *Standard deviation*, and *User defined*. To change the *Class method*, click on the current option and select the desired option from the list.

- *Equal number* assigns the class ranges such that approximately equal numbers of features are included in each class. Normally in this case, the interval of each class is different.
- *Equal interval* assigns the class ranges so the interval between the minimum and maximum values are equal for each class. Normally in this case, a different number of features is assigned to each class.
- *Natural breaks* uses the Fisher-Jenks algorithm to calculate the ideal natural breaks in the data. Typically these natural breaks are ideal beginnings and endings for data classes. The algorithm is based on Fisher's statement that any optimal classification of data consists of the sum of optimal classes of subsets of the data. To learn more about Jenks' natural breaks data classification method, please refer to Slocum, Terry A. *Thematic Cartography and Visualization.* New Jersey: Prentice-Hall, 1999.
- *Standard deviation* generates data classes based on the standard deviation of the data.
- *User Defined* allows you to set the *>=Minimum* value and the *<Maximum* value for each class individually. This allows you to specify your own ranges for the classes. Ranges defined in this way do not have to be contiguous. To change the *>=Minimum* or *<Maximum* value, double-click the values in the class list.

Class Counts

The *Class counts* value specifies the number of classes there will be on the map. When the *Class count* value is changed, the classes are automatically updated to reflect the change. To change the number of classes, click the  button to increase or decrease *Class count* or highlight the existing number and type in a new value from 1 to 1000 classes.

Show All Others

Select the *Show all others* option to include a symbol for features where the attribute value does not fall within an existing class. Clear the *Show all others* check box to remove the *All others* class.

Features that do not have an *Attribute field* value are controlled by the *Show missing values* option. When *Show all others* is not selected, features where the *Attribute field* value is outside the class limits are treated the same as missing values.

Reverse Symbol Order

Select the *Reverse symbol order* option to reverse the symbols in the *Classes* list. When the *Reverse symbol order* option is selected, the class with the smallest <= *Upper Value* is sized by the *Maximum size* and the class with the largest <= *Upper Value* is sized by the *Minimum size*. Clear the *Reverse symbol order* check box to return the symbols to the original order and scaling.

Load/Save Classes

Click *Load* to load an existing *Symbology Symbol Class (*.symbolclass)* file. Click *Save* to save the current classes to a *Symbology Symbol Class (*.symbolclass)* file . The class limits, names, symbol set, symbol fill and line colors, symbol indexes, and symbol sizes are saved in the *Symbology Symbol Class (*.symbolclass)* file.

Symbol Size

Specify the smallest and largest symbol sizes in the *Minimum size* and *Maximum size* fields. By default, the class with the smallest <= *Upper Value* is created with the *Minimum size*. The class with the largest <= *Upper Value* is created with the *Maximum* size. The remaining features' symbols are scaled proportionally between the *Minimum size* and *Maximum size* by the *Scaling method*. Select the *Reverse symbol order* option to reverse the symbol sizing.

Scaling Method

The *Scaling method* specifies which scaling type is used to determine symbol sizes between the minimum and maximum. Select *Linear* or *Square root* from the *Scaling method* list. The interpolation equations for the scaling methods are as follows:

Linear:	$H_x = \left[\left((Z_x - Z_1)/(Z_2 - Z_1)\right) * (H_2 - H_1)\right] + H_1$

Square Root:	$H_x = \left[\sqrt{(Z_x - Z_1)/(Z_2 - Z_1)} * (H_2 - H_1)\right] + H_1$

Square root scaling is commonly used with solid symbols to offset the fact that the area increases as a function of the symbol height squared. Square root scaling is essentially making the area of the symbol proportional to the Z value, rather than making the size of the symbol proportional to the Z value.

Same Class Symbols

Select the *Same class symbols* option to use the same symbol, fill color and opacity, and line color and opacity for the symbols in each class. When *Same class symbols* is checked, only symbol size differentiates symbols between the classes. The *Same class symbols* check box is automatically cleared after the symbol for any class is changed.

The *Same class symbols* section controls the symbol properties for the classes when the *Same class symbols* option is selected. The *Symbol properties* are unavailable when the *Same class symbols* check box cleared. The *Same class symbols* section also includes *Fill color method* and *Line color method* properties. When the color method is set to *Fixed*, all symbols use the same fill or line color. When the color method is *Colormap*, the fill or line colors are applied to the class symbols from a colormap. Select the colormap in the *Fill colormap* and *Line colormap* fields or create a custom colormap in the Colormap Editor by clicking the ... button.

Classes

The *Classes* list displays the class symbol (*Symbol*), upper limit (*<= Upper Value*), proportion (*%*), count (*Count*), and class name (*Name*) for the classes. The *Classes* list is on the right side of the **Symbology** dialog.

Changing the Class Symbol

The *Symbol* column displays the symbols for the classes. Click the symbol preview in the *Symbol* column to change the symbol for the class. The Point Properties dialog is displayed.

Changing the Class Limits

The *<= Upper Value* column displays the upper limit for each class. The first class in the *Classes* list has no lower limit. Subsequent classes use the prior class upper limit for a class minimum. For example, in the image above *Class 2* includes values greater than 28160 (Class 1's upper limit) and less than or equal to 56390.

The class limits are automatically calculated when the *Class method* is *Equal number*, *Equal interval*, *Natural breaks*, or *Standard deviation*. The class limits must be individually defined when the *Class method* is *User defined*. Changing a class *<= Upper Value* automatically updates the *Class method* to *User defined*.

Type the desired class maximum in the <= *Upper Value* column to change the upper limit for a class. The proportion and count statistics are automatically updated when the <= *Upper Value* is changed.

Statistical Information about the Features in Each Class

The *Classes* list displays summary statistics for each class. The *%* and *Count* columns display statistical information about the class. The *%* column indicates the percentage of features in the particular class. The *Count* column indicates the number of features included in each class. These values cannot be directly edited and are for informational purposes only.

Changing the Class Name

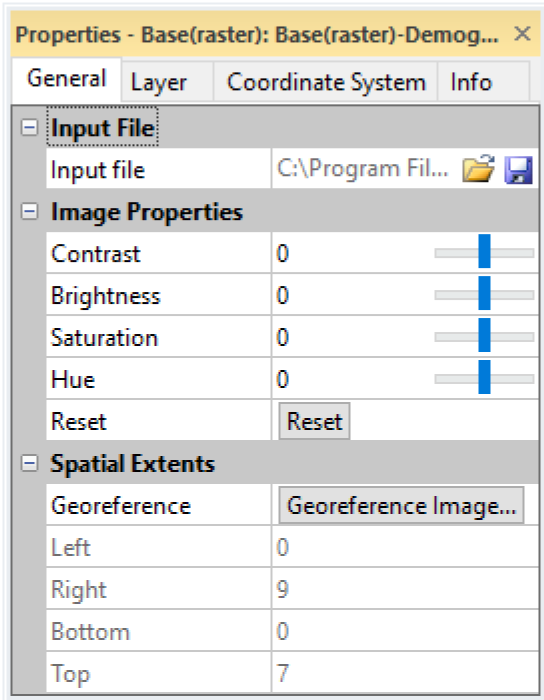
The *Name* column displays the class names. Specify a class name in the *Classes* list to display class names in a legend. By default, classes are named *Class 1*, *Class 2*, etc. Type a name in the *Name* column to change the class name.

OK, Cancel, and Apply

Click *OK* to save your changes, close the **Symbology** dialog, and update the base layer symbology. Click *Cancel* to close the **Symbology** dialog without making changes to the base layer symbology. Click *Apply* to apply your changes to the base layer while keeping the **Symbology** dialog open.

Base (raster) Layer General Properties

Image files can be loaded into **Surfer** as a base (raster) layer with the **Home | New Map | Base** or **Home | Add to Map | Layer | Base** command. Click on the *Base (raster) Layer* object in the Contents window to open the base layer properties in the Properties window. If you wish to import the image without creating a map layer, use the **Home | Insert | Graphic** command instead.



*The **General** page controls the image appearance and* Spatial Extents.

Input File

The *Input File* lists the current file used in the base map.

Change File

Click the button to display the **Import** dialog. This allows you to select a new file or an updated version of the current file used to create the base map. If the file exceeds the current map limits, you will be prompted to adjust the map limits.

A raster file must be selected when changing the *Input file* for a base (raster) layer. An error message will be displayed if a vector file is selected in the **Import** dialog. If you wish to add a base (vector) layer to the map, select the **Home | Add to Map | Layer | Base** command instead.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

The button displays the **Export** dialog. Type a *File name* and change the *Save as type* to the desired file format. Click *Save.* For some file types, a file type specific **Export Options** dialog appears. The **Export Options** dialog Spatial References and Scaling pages are available when saving a base layer. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Set any export options and click *OK* to save the file.

The georeference information is not saved for the image when exporting from the **Properties** window **General** page. To export the image in the map coordinate system, click the File | Export command and select *Selected objects only* option in the **Export** dialog.

Image Properties

The properties in the *Image Properties* section on the **General** page transform the image appearance: *Opacity* , *Contrast* , *Brightness* , *Saturation* , and *Hue*. These image transformations are applied to the display of the image or base layer, not to the image file. However, the image transformations are saved to the exported file when exporting the image or map.

Opacity

Change the *Opacity* of an image by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the to change the opacity percentage.

Contrast

Contrast is the difference in luminance or color of the objects in the image. Light and dark objects appear to stand out from one another when contrast is high. The difference in luminance between objects is difficult to see when contrast is low. The *Contrast* value is relative in **Surfer**. The original image is given a *Contrast* value of 0. Positive values increase the image contrast. Negative values decrease the image contrast. Type a value between -100 and 100 in the *Contrast* field or click and drag the to adjust the image contrast.

Brightness

Brightness is the perception of emitted or reflected light. Brightness can also be considered the average of the Red, Green, and Blue values for the pixels in the image. As brightness increases objects become lighter until they begin to "wash out" or turn white. Objects in the image get darker until they become black as brightness decreases. The *Brightness* value is relative in **Surfer**. The original image is given a *Brightness* value of 0. Positive values increase the image brightness. Negative values decrease the image brightness. Type a value between -

100 and 100 in the *Brightness* field or click and drag the  to adjust the image brightness.

Saturation

Saturation is a combination of light intensity and distribution across different wavelengths. High saturation relates to vivid, bright colors, i.e. high intensity across a narrow wavelength band or even single wavelength. This can also be considered very pure color. Low saturation leads to muted or gray colors, i.e. low intensity across a wide band of wavelengths. No saturation transforms the image to grayscale. The *Saturation* value is relative in **Surfer**. The original image is given a *Saturation* value of 0. Positive values increase the image saturation. Negative values decrease the image saturation. Type a value between -100 and 100 in the *Saturation* field or click and drag the  to adjust the image saturation.

Hue

Hue describes the colors in the image. The *Hue* property shifts the colors in the image around the color wheel. The *Hue* value is relative in **Surfer**. The original image is given a *Hue* value of 0. Positive values shift the colors in the red to orange, then yellow, green, blue and finally magenta direction. Negative values shift colors in the opposite direction, from red to magenta then blue, green, yellow and finally orange. Type a value between -180 and 180 in the *Hue* field or click and drag the  to adjust the image hue.

Reset

Click *Reset* to return the image to its original appearance. This sets *Opacity* to 100% and *Contrast*, *Brightness*, *Saturation*, and *Hue* to 0.

Georeference

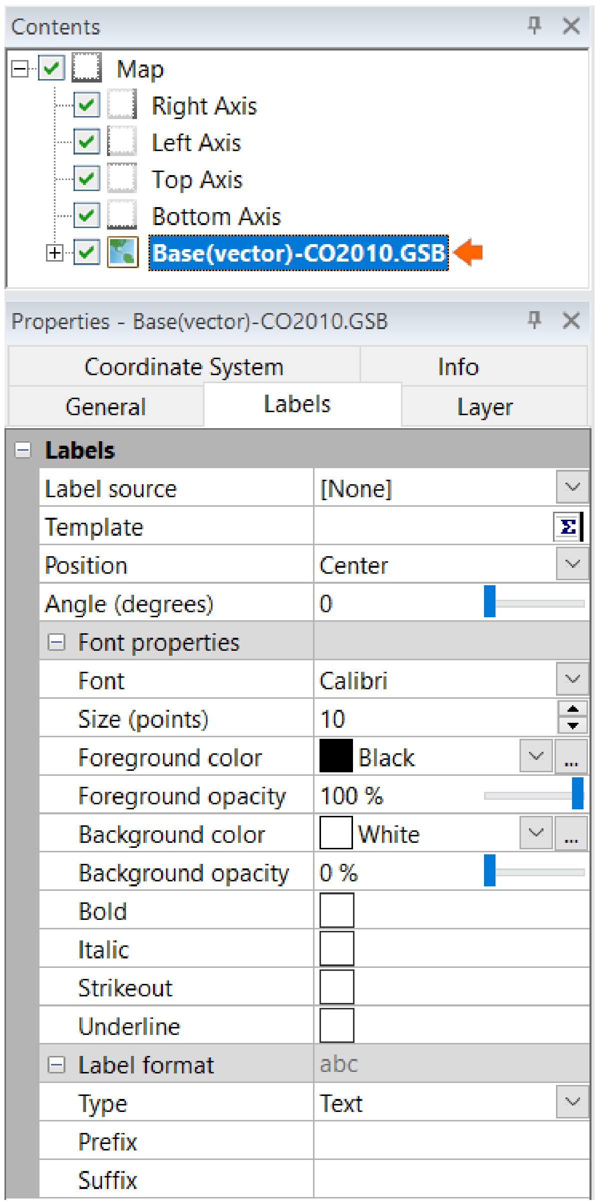
Click *Georeference Image* in the *Georeference* field to apply control points to the image and reference it to your map's coordinate system. Clicking *Georeference Image* opens the Georeference Image window. The **Georeference Image** command is only available when the image is a base (raster) layer. The image must be at least 2x2 pixels to be georeferenced.

Spatial Extents

The *Left, Right, Bottom,* and *Top* values in the *Spatial Extents* group displays the current extents of the image. These values are read-only. Click the *Georefence Image* button to change the spatial extents.

Base Map Labels Properties

After assigning attributes to polylines, polygons, and symbols, attribute labels can be added. The attribute, position, and font properties can also be specified on the **Labels** page in the base map properties. Attributes for rectangles, rounded rectangles, and ellipses do not appear as labels. To add labels from attributes to a base map, click once on the base map layer to select it. In the Properties window, click on the **Labels** tab.



*Change base map label properties on the **Labels** page in the **Properties** window.*

Labels Properties

The properties in the *Labels* section control the content, position, and appearance of the base (vector) layer labels.

Label Source

The *Label source* determines whether or not labels are displayed and which attribute should be displayed. To change the *Label source*, click on the existing option and select the desired option from the list. Available options are *[None]*, *[Template]*, and then a list of all attribute names in the file. To display no labels, set the *Label source* to *[None]* . To display a single attribute, select the attribute name from the list. To display multiple attribute names or an attribute name and additional text, select *[Template]* . Then, set the *Template* option to the desired text and attributes.

Template

The *Template* option allows multiple attributes or attributes and text to be displayed for the label. Attribute names must be listed exactly as they appear in the *Label source* list. Attribute names must have square brackets around them, such as *[Attribute Name]* in order to be recognized. Attribute names are case sensitive, so there is a difference between *[name]* and *[Name]*. Any number of attributes may be added to the *Template* section.

The square brackets are mandatory. Math text options (such as \up50) must appear outside of the square brackets. If the font or font properties are changed for the attribute, highlight the entire text including the square brackets before changing the properties in the Text Editor.

Click the Σ button to open the **Text Editor** where multiple lines of text can be created. Attribute names must appear in the **Text Editor** with square brackets around the name.

Position

The *Position* controls the offset of the label from the object center. To change the position, click on the existing position and select the new position in the list. Specify *Center*, *Left*, *Right*, *Above*, *Below*, or *User defined*. *User defined* allows you to specify the exact offset (in page units) in the *X offset* and *Y offset* boxes. The posted labels are all placed in the same position relative to the associated object. Positive values in the *X offset* shift the label position toward the positive direction (to the right for normal axes, to the left for reversed axes). Negative values in the *X offset* shift the label position toward the negative direction (to the left for normal axes, to the right for reversed axes). Positive values in the *Y offset* shift the label position toward the positive direction (upward for normal axes, downward for reversed axes). Negative values in the *Y offset* shift the label position toward the negative direction (downward for normal axes, upward for reversed axes). The *X offset* and *Y offset* values are numbers between -4 and +4 inches (-10.16 and +10.16 centimeters). To change the *X offset* and *Y offset* values, highlight the existing value and type a new value or click the buttons to increase or decrease the offset.

Angle (degrees)

The *Angle (degrees)* box specifies the angle in degrees to rotate the labels. Positive angles rotate the labels counterclockwise. To change the *Angle (degrees),* highlight the existing angle value. Type a new value or click and drag the to the desired angle position.

Font

Click the ⊞ next to *Font Properties* to display the Font Properties section. The font properties are applied to all labels.

Format

Click the ⊞ next to *Label Format* to display the Label Format section. The label format properties are applied to all labels. If the values in the *Label source* attribute are not numeric, the label will be displayed as unformatted text.

Examples

The following examples demonstrate adding labels to base layer objects.

Attribute Labels

Create a base map from a .SHP file. When the .SHP file is loaded, the attributes are also imported. To label all of the polygons with the Primary ID:

1. Click on the *Base* object in the **Contents** window to select it.
2. In the **Properties** window, click on the **Labels** tab.
3. Change the *Label source* to *Primary ID*. The *Primary ID* will appear next to all polylines, polygons, and symbols in the .SHP file.

Template Labels

Create a base map from a file that contains attributes. After the file is loaded, the attributes appear for each of the individual polylines, polygons, and symbols. To create complex labels for each object:

1. Click on the *Base* object in the **Contents** window to select it.
2. In the **Properties** window, click on the **Labels** tab.
3. Change the *Label source* to *[Template]*.
4. Next to *Template,* type the desired attribute names, surrounded by square brackets. For instance, if you wanted to display the name of a property and the owner's name, your *Template* may look like:

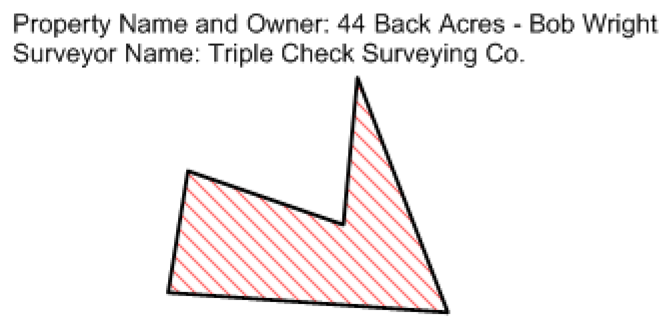
    [Name] - [Owner]

5. To create multiple lines, click the button.
6. In the Text Editor, type the attribute names, surrounded by square brackets. Add any additional text. To add a new line, press ENTER on the keyboard. You might have:

   Property Name and Owner: [Name] - [Owner]

   Surveyor Name: [Surveyor]

7. When all edits are complete, click *OK* to close the **Text Editor**. The labels appear on the screen.

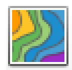


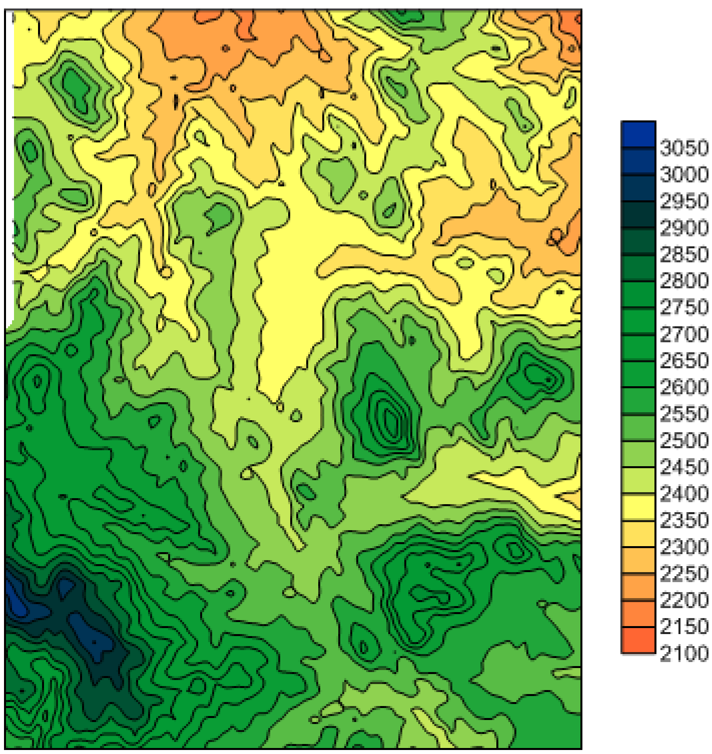
The label appears in the desired location, substituting the attribute value for the attribute names in square brackets.

Chapter 9 - Contour Maps

Contour Map

The **Home | New Map | Contour** command creates a contour map from a grid file. A contour map is a two-dimensional representation of three-dimensional data. The first two dimensions are the X and Y coordinates. The third dimension (Z) is represented by lines of equal value. The relative spacing of the contour lines indicate the relative slope of the surface. The area between two contour lines contains only grid nodes having Z values within the limits defined by the two enclosing contours. The difference between two contour lines is defined as the contour interval.

The **Home | Add to Map | Layer | Contour** , the  button, or **Map Tools | Add to Map | Layer | Contour** command adds a contour map layer to the selected map.



This example contour map shows filled color contours and a color scale bar.

Creating a Contour Map

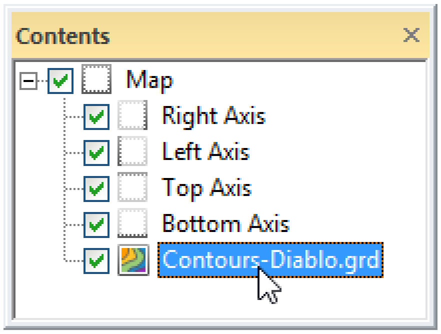
To create a new contour map:

1. Click **Home | New Map | Contour** command.
2. Select a grid file in the Open Grid dialog. Click *Open*.

The map is automatically created with reasonable defaults.

Editing an Existing Contour Map

To change the features of the contour map, click once on the contour map in the Contents window or plot window. When the contour map is selected, the contour map properties are displayed in the Properties window.



Click on the Contours *map layer to select it.*

Contour Layer Properties

The contour layer properties contains the following pages:

General
Levels
Layer
Coordinate System
Info (Grids)

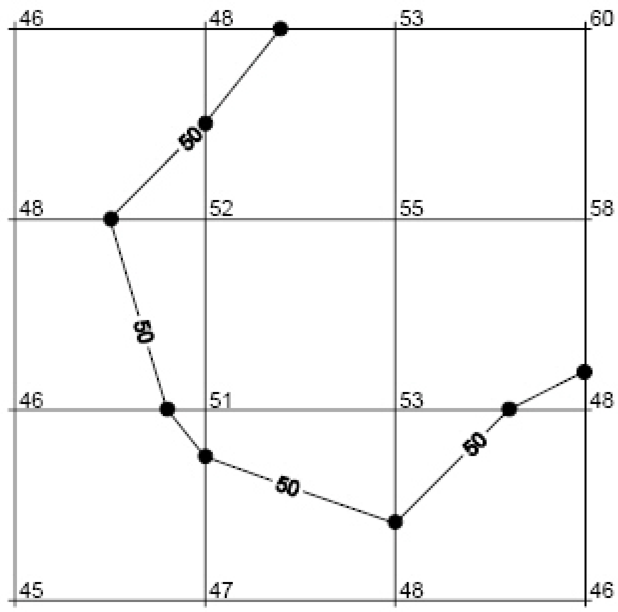
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info (Objects)

Drawing Contours

When you create a contour map, the grid file is read into the plot window as an internal array of X, Y, and Z grid nodes. The grid nodes consist of rows and columns of Z values. The rows contain grid nodes with the same Y coordinate, and the columns contain grid nodes with the same X coordinate. Grid file columns and rows are sometimes referred to as X grid lines and Y grid lines, respectively. The intersection of a row and column is defined as a grid node. Grid files define the X, Y location of each grid node over the extent of the map, and the interpolated Z value at each node.



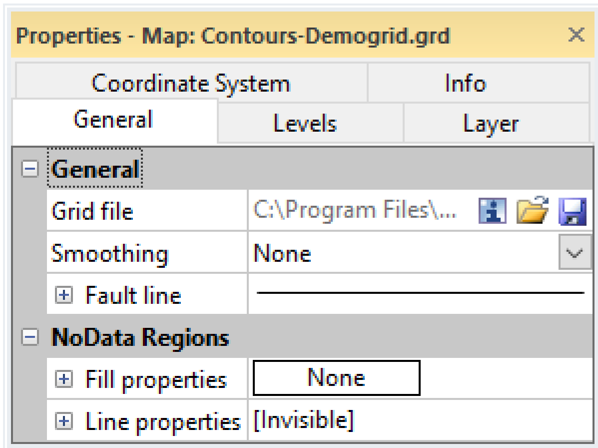
The path of a contour line (Z = 50) through a grid is defined by interpolating between grid nodes. This example shows the straight-line segments that define the contour line. The dots are added to show the ends of each line segment.

When **Surfer** creates a contour map, the contour lines are drawn as a series of straight line segments between adjacent grid lines. The point where a contour line intersects a grid line is determined by interpolation between Z values at adjacent grid nodes.

The grid limits define the extent of contour maps. Once a grid file is created you cannot produce a contour map larger than the extent of the grid file. However, you can use the Limits page to specify a subset of the grid used to create the contour map.

Contour Map General Properties

The contour map properties **General** page controls the *Grid file, Smoothing, Fault line* properties*,* and *Blanked Regions* for a contour map. To open the **General** page, click on a contour map to select it. In the Properties window, click on the **General** tab.



Change contour map properties in the ***Properties*** *window on the* ***General*** *page.*

Input Grid File

The *Grid file* lists the current grid file used in the contour map. The path and file name are the location of the grid file when the map was created or the grid file was most recently changed.

Grid Information

The button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

The button displays the **Open Grid** dialog. This allows you to select a new grid file, or an updated version of the grid file used to create the contour map. Select a grid file and click *Open*. If the Z range for the new grid is outside the old contour levels, you will be prompted to continue the operation. If you click-Cancel, the grid file is not replaced. If you clickOK, the grid file is replaced, but no contour lines are drawn. You can update the contour levels being displayed on theLevelstab.

If the new grid exceeds the current map limits, another warning will appear asking you to adjust the map limits. If you click *Yes*, the limits are automatically adjusted to fit the new grid. If you click *No*, the limits are not automatically adjusted. The map may not be displayed. To change the map limits, click on the Map object in the **Contents** window and the **Limits** tab in the **Properties** window.

You may also see a warning message that the current map scale may result in an un-viewable map. Clicking *OK* allows the map scale to automatically be adjusted.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

The button displays the **Save Grid As** dialog. Type a *File name* and change the *Save as type* to the desired grid file format. Click *Save.* If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, an **Export Options** dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Smoothing

The *Smoothing* option allows you to smooth the contours displayed on the map. Click on the existing value (*None*, by default) and select a new smoothing value from the list. If smoothing is enabled, contour lines may cross. The *Smoothing* option in the contour map properties applies a constrained spline smoothing algorithm to interpolate additional contour vertices. Each line is separately calculated, thus the lines can cross if too much smoothing is applied.

A better method to smooth contours is to regrid the data using the Grids | New Grid | Grid Data command, use the Grids | Edit | Spline Smooth or Grids | Resize | Mosaic commands to create a new grid with smaller grid node spacing. See Smoothing Contours for more information on smoothing.

Fault Line

If your grid file contains faulting information, you can set the fault line color, style, opacity, and thickness of the fault line drawn on the contour map by clicking on the next to *Fault line* and changing the appropriate line property options.

NoData Regions

The *NoData Regions* section allows you to choose fill and line color properties for areas containing nodes assigned the NoData value.

Click on the ⊞ next to Fill Properties to set fill properties for NoData areas, or to load a custom fill pattern. If no colors appear after setting these options, click on the **Levels** tab and check the box next to the *Fill contours* option.

Click on the ⊞ next to Line Properties to set the line properties that outline the NoData area.

Contour Map Levels Properties

The contour map properties **Levels** page controls the display of contours, contour labels, and color fill on the contour map. The levels can be set using a simple, logarithmic, equal area, or advanced method. All methods are described below. The advanced option is useful when more control over individual contour line properties is desired.

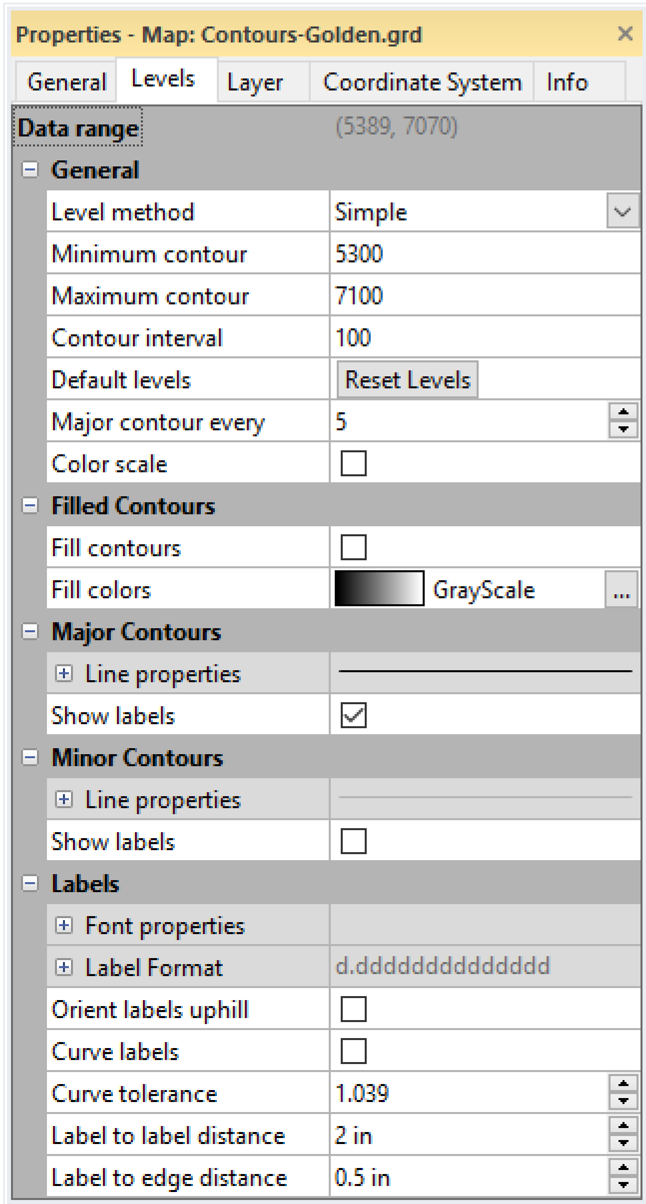
Display Simple, Logarithmic, or Equal Area Options

To use the simple options, click on the contour map to select it. In the Properties window, click on the **Levels** tab. Change the *Level method* to *Simple*. To use logarithmic options, change the *Level method* to *Logarithmic*.

Switching Between Level Methods

When switching the *Level method* to *Advanced*, all current *Simple*, *Logarithmic*, or *Equal area* options are initially shown in the **Levels for Map** dialog. However, when switching the *Level method* from *Advanced* to *Simple*, *Logarithmic*, or *Equal area* it is likely that some level customizations will be lost.

Some settings require the *Level method* be set to *Advanced*. For example, if you wish to load a .LVL file, you must use the *Advanced* setting.



*Adjust the contour map contour line properties on the **Levels** page in the **Properties** window.*

Data Range

The *Data range* displays the Z minimum and maximum values read from the grid file.

General

The *General* section controls the method used to create the contour lines, the minimum contour value, maximum contour value, contour interval, how many minor contours are in a major contour interval, and the display of the color scale bar.

Level Method

The *Level method* determines which options are available for setting the contour map contour line properties. Available options are *Simple, Logarithmic, Equal area*, and *Advanced*. *Simple* allows basic options to be set. *Logarithmic* uses a log(10) interval to display the contours. *Equal area* separates the map into approximately equal area bins and creates contour lines at the bin boundaries. *Advanced* allows individual lines to have labels, hachures, fill, and line properties set individually. *Advanced* options are explained on the Contour Map Advanced Levels Dialog page. To set the *Level method*, click on the existing method and select the desired method from the list.

When the *Level method* is set to *Logarithmic*, the colormap for the filled contours automatically changes to use a logarithmic scaling, as well.

Minimum Contour

Set the *Minimum contour* to the value of the first contour line you want displayed on the contour map. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard or click another command in the **Properties** window for the value to be accepted. Values are in map Z units. When the *Level method* is set to *Logarithmic* , the *Minimum contour* value must be greater than 0.

Maximum Contour

Set the *Maximum contour* to the value of the last contour line you want displayed on the contour map. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard or click another command in the **Properties** window for the value to be accepted. Values are in map Z units.

Contour Interval

The *Contour interval* option is available when the *Level method* is set to *Simple* . The *Contour interval* defines the spacing units between adjacent contour lines. This will affect how many contours appear on the contour map and how close those contours are to one another. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard or click another command in the **Properties** window for the value to be accepted. Values are in map Z units.

Surfer uses the **Contour interval**, the **Minimum contour** value, and the **Maximum contour** value to determine how many contour lines are created on the map. **Surfer** will issue a warning message if the number of contour lines increases above 5000. Click *OK* in the warning message and alter the **Contour interval**, the **Minimum contour** value, or the **Maximum contour** value so that the total number of lines is less than 5000.

Default Levels

The *Default levels* option is available when the *Level method* is set to *Simple*. If the *Minimum contour*, *Maximum contour*, or *Contour interval* has changed, you can return to the **Surfer** default values by clicking the Reset Levels button next to *Default levels*. **Surfer** returns the *Minimum contour*, *Maximum contour*, and *Contour interval* to reasonable values. This is similar to clicking the *Use Defaults* button on the Contour Levels dialog when using the advanced *Level method*.

Number of Contours

The *Number of contours* property is available when the *Level method* is set to *Equal area*. The *Number of contours* property defines how many contour lines, and therefore equal area intervals, are displayed on the map. The number of intervals is equal to *Number of contours* plus one. In the rare case the grid has fewer unique values than necessary for the specified *Number of contours*, the number of contours on the map may be less than the *Number of contours* value.

Area per Interval

The *Area per interval* property is a read-only value that indicates the approximate area in each interval. **Surfer** calculates *Area per interval* by the area of the grid multiplied by the ratio of non-NoData to total grid nodes divided by number of levels plus one. The actual area of each interval will likely be very near, but not exactly equal, the *Area per interval* value. As the *Number of contours* increases, the *Area per interval* decreases. The *Area per interval* value is in layer units.

Major Contour Every

The *Major contour every* option is available when the *Level method* is set to *Simple* or *Equal area*. The *Major contour every* option sets the frequency for how often a major contour line appears on the map. Major contour lines and minor contour lines can have different line properties. By default, the first line is the *Minimum contour*. The *Major contour every* value tells the program how many contour lines to skip before the next major contour line. If the *Major contour every* value is set to 1, every contour line is a major contour line. If the value is 2, every other line is a major contour line. If the value is 3, every third line is a major contour line, and so on.

Minor Levels Per Decade

The *Minor levels per decade* option is available when the *Level method* is set to *Logarithmic.* The *Minor levels per decade* controls how many minor contour lines appear between major contour lines. When the *Level method* is set to *Logarithmic*, the major contour lines occur at multiples of 10 (0.001, 0.01, 0.1, 1, 10, 100, 1000, 10000, etc.). The number of contour lines between each major contour line is set by the *Minor levels per decade*. To change the *Minor levels per decade*, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the  to increase or decrease the value. The default value is 8.

If the value is 8, minor contour lines appear at 2, 3, 4, 5, 6, 7, 8, and 9 between 1 and 10 or 20, 30, 40, 50, 60, 70, 80, and 90 between 10 and 100. 8 is the most common number of minor contour lines per major. Any value between 0 and 500 can be specified, though. Some common levels are defined below, with the values of the contour lines that would appear.

Minor Levels per Decade	Values of Contour Lines
8	1, 2, 3, 4, 5, 6, 7, 8, 9
7	2.125, 3.250, 4.375, 5.500, 6.625, 7.750, 8.875
5	2.5, 4.0, 5.5, 7.0, 8.5
4	2.8, 4.6, 6.4, 8.2
3	3.25, 5.5, 7.75
2	4, 7
1	5.5
0	no minor lines are displayed

Contour Levels

When the *Level method* is set to *Advanced*, the *Contour levels* option is available. Click the *Edit Levels* button to open the Levels for Map dialog, where the advanced level options are set.

Color Scale

Check the box next to the *Color scale* option to display a color scale bar object. The color scale bar will be displayed in the plot window and the **Contents** window. The color scale bar can show the line properties for the contour lines, and the color scale bar shows the fill colors for a filled contour map. See color scale bar for more information on the color scale properties.

Filled Contours

The *Filled Contours* section controls the display of color fill between the contour lines.

Fill Contours

Check the box next to *Fill contours* to fill the space between each contour line with color. See Adding Color Fill between Contours for more information on filled contours.

Fill Colors

The *Fill colors* option defines the colormap used to fill the contour map. To fill contours, check the box next to the *Fill contours* option. Then, change the color by clicking the existing color bar next to *Fill colors*. Select the new colormap from the list. If the desired color map is not listed, click the ... button to the right of the color. The Colormap Editor appears. Make any changes and click *OK* to see the change on the map. If the *Level method* is set to *Advanced*, the colors are set in the **Levels for Map** dialog.

When changing the *Level method* to *Logarithmic* , the colormap scaling is automatically changed to logarithmic. When changing the *Level method* to *Equal area* , the colormap is not changed. However, when using *Equal area* level method, consider using the *Equalize* option in the *Stretch* field of the **Colormap Editor**.

Major Contours

Click on the ⊞ next to *Major Contours* to set the line properties for the major contour lines. You can also turn label display on or off.

Line Properties

Click on the ⊞ next to Line Properties to set the major contour line properties. Line *Style, Color, Opacity* , and *Width* can be set.

Show Labels

Check the box next to the *Show labels* option to display contour labels along major contour lines. When checked, all major contour lines contain labels. When unchecked, no major contour lines contain labels. If you want to set label visibility on an individual contour line basis, set the *Level method* to *Advanced* and use the options in the Levels page. Label font and format are controlled in the *Labels* section.

Minor Contours

Click on the ⊞ next to *Minor Contours* to set the line properties for the minor contour lines. You can also turn label display on or off.

Line Properties

Click on the ⊞ next to Line Properties to set the minor contour line properties. Line *Style, Color, Opacity*, and *Width* can be set. By default, the minor lines use a 30% Black (light gray) color.

Show Labels

Check the box next to the *Show labels* option to display contour labels along minor contour lines. When checked, all minor contour lines contain labels. When unchecked, no minor contour lines contain labels. If you want to set label visibility on an individual contour line basis, set the *Level method* to *Advanced* and use the options in the Levels dialog. Label font and format are controlled in the *Labels* section.

Labels

Click on the ⊞ next to *Labels* to set the major and minor contour label properties. Font properties, label format, and label properties can be changed.

Font Properties

Click on the ⊞ next to Font Properties to set the text properties for the contour line labels.

Label Format

Click on the ⊞ next to Label Format to set the label format for the contour labels.

When the *Type* is set to *Date/time*, if *Invalid date* text is displayed instead of the actual date/time format, the value for the label is outside the defined date/time range.

Label Orientation

The *Orient labels uphill* check box displays the labels so they are always oriented uphill. If this box is checked, the "tops" of the labels point uphill. If this option is unchecked, the labels always are oriented right-side up on the page.

Curve Labels

The *Curve labels* option creates contour labels that curve along the path of the contour lines. Select the *Curve labels* option to curve the contour labels. Clear the *Curve labels* option to create straight contour labels, regardless of contour line curvature.

Curve Tolerance

Curve Tolerance specifies the maximum amount of contour curvature allowed when placing labels on contour lines. Curve tolerance is calculated by dividing the actual distance along the contour line by the straight-line distance between the end points of the contour label. Highly curved lines might not be labeled automatically. You can increase the curve tolerance value to allow labels on highly curved contour lines, although contour labels might be hard to read. The default *Curve Tolerance* value of 1.015 should be acceptable in most cases.

Label to Label Distance

Label to Label Distance specifies the minimum distance (in inches or centimeters) between labels along the contour line. **Surfer** searches for the next suitable location for a label after moving the minimum distance specified by the *Label to Label Distance*. The next label is not drawn until a segment of the necessary length, within the curve tolerance limits, is found. As the *Label to Label Distance* is increased, fewer labels are drawn on the contours.

Label to Edge Distance

Label to Edge Distance specifies the minimum distance (in inches or centimeters) from the label to the edge of the map. This feature controls label placement so labels do not overwrite the map borders or axes.

Contour Map Advanced Levels Properties

The contour map properties **Levels** page controls the display of contours, contour labels, and color fill on the contour map. The levels can be set using a simple, logarithmic, or equal area method, which uses the same options for all major and minor contour lines. Or, an advanced option can be set that allows more control over individual contour line properties.

Display Advanced Options

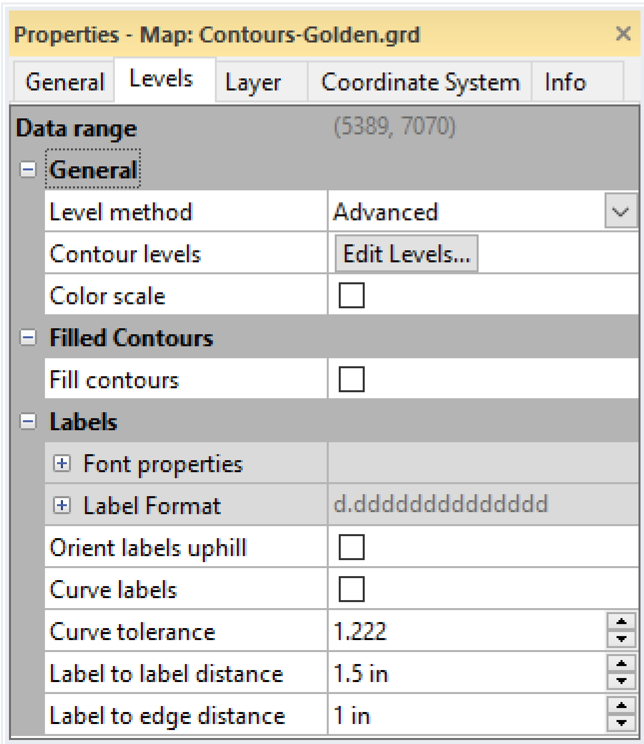
To use the advanced options, click on the contour map to select it. In the Properties window, click on the Levels tab. Change the *Level method* to *Advanced*. To open the advanced **Levels for Map** dialog, click the *Edit Levels* button next to *Contour levels.*

The **Levels** page in the contour map **Levels for Map** dialog controls the display of contours, contour labels, color fill, and hachures on the contour map. The contour level list shows the contour levels to be displayed on the map. Contour level parameters can be set as a group or individually.

Switching Between Level Methods

When switching the *Level method* to *Advanced*, all current *Simple*, *Logarithmic,* or *Equal area* options are initially shown in the **Levels for Map** dialog. However, when switching the *Level method* from *Advanced* to *Simple*, *Logarithmic*, or *Equal area* it is likely that some level customizations will be lost.

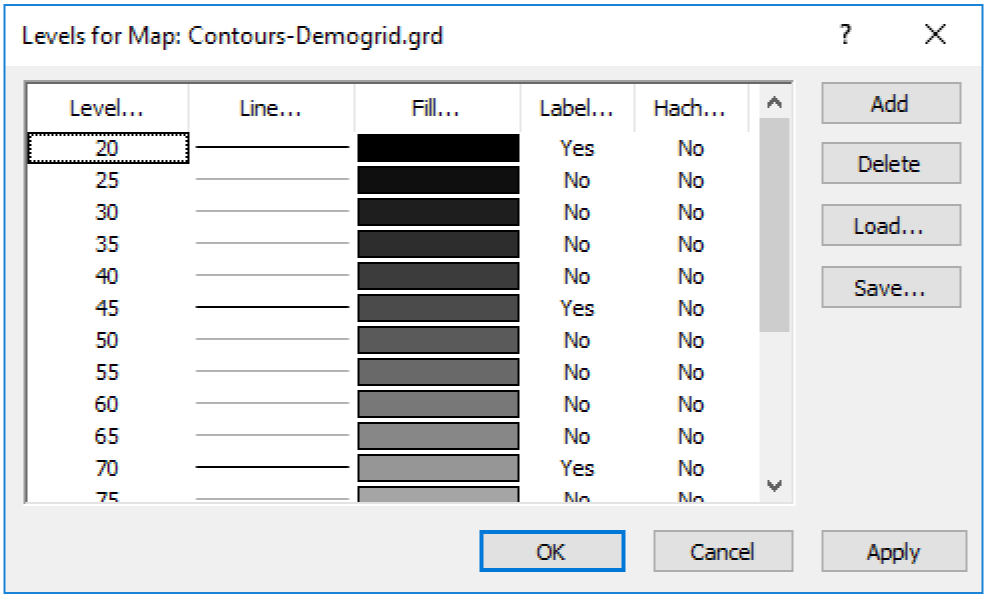
Some settings require the *Level method* be set to *Advanced*. For example, if you wish to load a .LVL file, you must use the *Advanced* setting.



Change the Level method to Advanced to control individual contour lines.

Levels for Map Dialog

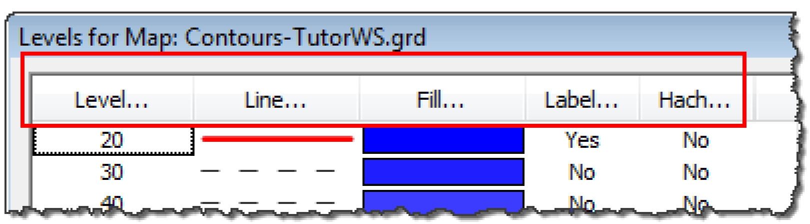
Control advanced settings for the *Level, Line, Fill, Label,* and *Hach* properties of the contour map in the **Levels for Map** dialog. Properties can be adjusted for all contours at once (by clicking on the column buttons), or for individual contours (by double-clicking on the specific contour level). Click the *Edit Levels* button in the *Contour levels* field to edit the advanced levels.



*The **Levels for Map** dialog appears when the Level method is set to Advanced. This allows increased control for displaying the contour map levels.*

Level Parameters For All Levels

To apply level parameters to all defined levels, click the Level, Line, Fill, Label, or Hach buttons at the top of the contour level list. For example, to define the contour range and the interval between contours, click the *Level* button and specify these parameters in the Contour Levels dialog.



Clicking the Level, Line, Fill, Label, and Hach buttons allow you to apply parameters to multiple levels at once.

Level Parameters For One Level

To set parameters for a specific contour level, double-click any of the elements in the list. For example, to change one contour level line's properties, double-click on the specific line to open the line properties. Make any changes and click *OK*. To edit one contour level's value, double-click on the individual *Level* to open the Z Level dialog, enter a new value, and click *OK*.

To change the display of an individual level label or hachure, double-click on the *Yes* or *No* value for that level. The option will change.

Load and Save

The *Load* and *Save* buttons allow you to load and save .LVL level files containing contour level settings. If you want to retain contour map settings such as contour intervals, contour labels, line color, and color fill you can save the setting in a level .LVL file. Once you have modified the settings click the *Save* button on the **Levels** page in the **Levels for Map** dialog. Level files can only be used with maps of similar Z ranges.

If you want to save only the color definitions, you can do this by clicking the *Fill* button. In the dialog that appears, click the color bar next to *Foreground Color*. In the Colormap Editor that appears, click the *Save* button to save the color definitions as a .CLR file. The .CLR file can be used with any Z range, because only the relative locations of the colors are saved.

Add and Delete

Use the *Add* and *Delete* buttons to add or delete specific contour levels.

Editing Advanced Contour Options

1. Click on a contour map to select it.
2. In the **Properties** window, click on the **Levels** page.
3. Change the *Level method* to *Advanced*.
4. To open the advanced levels dialog, click the *Edit Levels* button next to *Contour levels.*
5. Click on an existing contour level value to select it.
6. Click the *Add* button to add a contour level with a value halfway between the selected level and the next lower level. Adding levels creates irregularly spaced contours.
7. Click the *Delete* button to remove the selected level from the list.
8. To modify an existing level, double-click the level value. Enter a new value for the contour level in the Z Level dialog. Click *OK* and the contour level list is updated.
9. When you are finished changing the contour levels, click *OK* and the map is displayed with your changes.
10. To add color to the contours, check the box next to *Fill contours* in the **Properties** window on the **Levels** page.

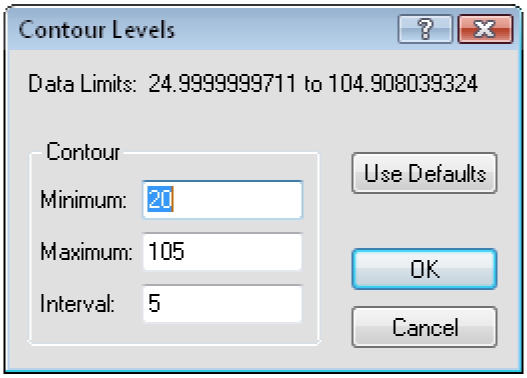
Specifying Contour Levels

The **Contour Levels** dialog is used to assign regularly spaced contour levels or color zones (i.e. 3D wireframe). For example, assume the smallest Z value is defined as 50, the largest Z value is 100, and the interval between contours or zones is 10. In this case, contour lines or color zones are drawn at Z levels of 50, 60, 70, 80, 90, and 100. If the range of grid Z values goes beyond the defined minimum and maximum contour levels, the contours or color zones outside the levels are not drawn.

You can manually change the Z value of any single contour level by double-clicking on the level value on the Levels page in the Levels for Map dialog when the *Level method* is set to *Advanced*.

Contour Levels Dialog

Click the *Level* button in the Levels for Map dialog to open the **Contour Levels** dialog.



*Customize contour levels in the **Contour Levels** dialog.*

Data Limits

The actual *Data Limits* of the grid file are listed for reference. These values cannot be changed.

Minimum, Maximum, Interval Contour Values

Enter values for the *Minimum* and *Maximum* contours, and the *Interval* value that defines the spacing in Z units between adjacent contour lines.

Surfer uses the *Interval*, the *Minimum* contour value, and the *Maximum* contour value to determine how many contour lines are created on the map. **Surfer** will issue a warning message if the number of contour lines increases above 5000. Click *OK* in the warning message and alter the *Interval*, the *Minimum* contour value, or the *Maximum* contour value so that the total number of lines is less than 5000.

Use Defaults

The *Use Defaults* button can be used to calculate a reasonable set of parameters based on the range of Z values within the grid.

OK and Cancel buttons

Click *OK* and the contour level list is updated with the selections made in the *Contour*group. Click *Cancel*to close the dialog with making any changes.

To Create Contour Levels at Regular Intervals

The easiest way to create regular contour intervals is to use the simple level method:

1. Click on a contour map to select it.
2. In the Properties window, click on the **Levels** tab.
3. Set the *Level method* to *Simple*.
4. Set the values for the *Minimum contour* and *Maximum contour.*
5. Set the *Contour interval* value that defines the spacing in Z units between adjacent contour lines.

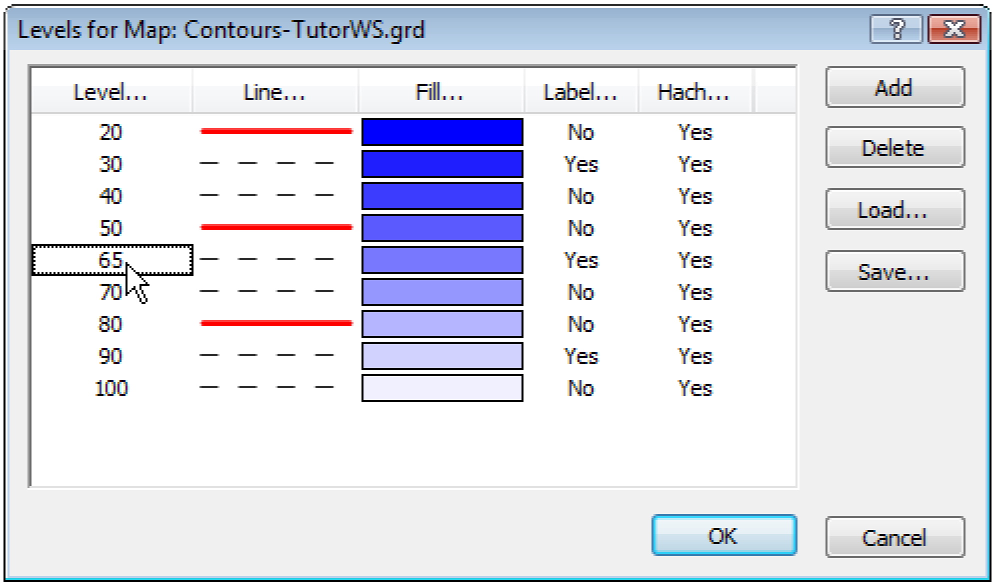
Regularly spaced contours are automatically created.

Using advanced options allows the possibility of controlling additional features for each contour line separately. Use these steps for advanced contour lines:

1. Click on a contour map to select it.
2. In the Properties window, click on the **Levels** tab.
3. Set the *Level method* to *Advanced*.
4. Click the *Edit Levels*button next to*Contour levels*to open the advanced options**Levels for Map**dialog.
5. To assign evenly spaced contour levels, click the *Level* button and the **Contour Levels** dialog is displayed.
6. Set the values for the *Minimum* and *Maximum* contours.
7. Set the *Interval* value that defines the spacing in Z units between adjacent contour lines.
8. The *Use Defaults* button can be used to calculate a reasonable set of parameters based on the range of Z values within the grid.
9. Click *OK* and the contour level list is updated.
10. Click *OK* in the dialog to update the map.

Z Level

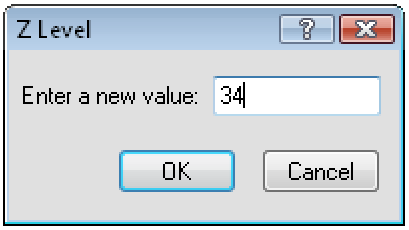
Individual contour levels can be adjusted by double-clicking on an individual *Level* in the Levels column of the **Levels for Map** dialog when the *Level method* is set to *Advanced*.



*Double-click a contour level to open the **Z Level** dialog. This example shows the 65 level contour being adjusted.*

New Value

Enter a new value into the *Enter a new value* box, and click *OK* to add the new value to the dialog.



*Change individual contour lines with the**Z Level** dialog.*

Level Files - Contour

For contour maps, .LVL level files contain information about contour levels, including the line properties, color fill, label frequency, and hachure information. The level file contains all the information contained on the **Levels** page in the advanced contour map Levels for Map dialog. After defining custom contour levels and colors on a map, you can save the level and color information in a level file. A level file can be recalled for any other contour map or 3D wireframe. Not all of the information in a level file can be used in a 3D wireframe. If you use a level file created in the contour map properties dialog in a wireframe, the color fill, contour label, and hachure information are ignored.

Creating Level Files

The easiest way to create a level file is from the advanced contour map properties or 3D wireframe properties dialogs. You can also create a level file in the worksheet or in an ASCII editor, using the .LVL file format.

To create a .LVL level file containing contour level information in the advanced contour map level properties:

1. Click once on a contour map to select it. The contour map properties will be shown in the Properties window.
2. Click on the **Levels** tab. Change the *Level method* to Advanced.
3. Click the *Edit Levels* button next to *Contour levels*. The Levels for Map dialog opens.
4. Change any of the contour parameters, including the contour levels, line properties, fill properties, contour labels, and hachure information.
5. After changing the settings, click *Save*.
6. In the **Save As** dialog, type the name of the level file and click the *Save* button. The level file is saved with a .LVL extension.
7. Click *OK* to close the dialog.

Using Level Files

To use an existing .LVL file with any contour map:

1. Click once on a contour map to select it. The contour map properties will be shown in the **Properties** window.
2. Click on the **Levels** tab. Change the *Level method* to Advanced.
3. Click the *Edit Levels* button next to *Contour levels*. The **Levels for Map** dialog opens.
4. Click *Load*.
5. Select the .LVL file and click *Open*.
6. Click OK and the contour map updates.

Note: All contour maps using the level file must have comparable Z data ranges, otherwise contour lines and fill will not appear on the map.

Creating Level Files in the Worksheet

You can create simple level files in the **Surfer** worksheet or an ASCII editor. The minimum amount of information a level file contains is the elevation data telling **Surfer** which contour lines to place on a contour map. For more information on the level file format, see the level file format.

Creating Level Files from the Worksheet

1. Enter level values into column A.
2. Add any other information you wish to include for the levels (i.e. line color) according to the level file format.

3. After entering the level information, click **File | Save As**.
4. In the *File name* field, enter the level file name and .LVL extension enclosed in quotes ("colors.lvl").
5. Choose *DAT Data (*.dat)* from the *Save as type* field and click *Save*.
6. In the **Data Export Options** dialog, choose *Space* as the *Delimiter* and *Double Quote* as the *Text Qualifier*. Choosing any other delimiter in the **Data Export Options** dialog results in an invalid level file since the file must be space delimited.
7. Click *OK* to create the level file.

Contour Lines

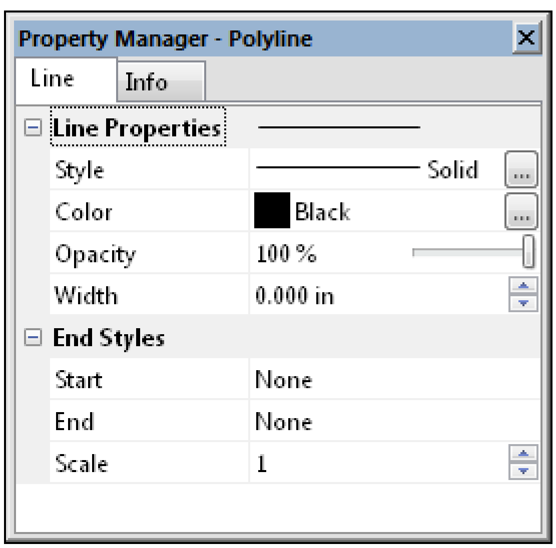
The following sections describe the various methods for adjusting the contour line appearance.

Line Properties

Use the **Line** page in the Properties window to change line properties for selected lines in the document. You can set default line properties in the **Options** dialog Defaults page by clicking on *Line*.

Most line properties are edited in the **Properties** window in a *Line Properties* section. When changing line properties for a selected object, the *Line Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Occasionally, objects will open a Line Properties dialog to access the line properties.



Specify individual line properties in the ***Properties*** *window in a* Line Properties *section.*

Sample

The sample of the line is displayed next to *Line Properties*. The sample shows the line style, color, opacity, and width options.

If the *Line Properties* section is closed, click the ⊞ next to *Line Properties* to open the section.

Style

Click the line next *Style* to open the line style palette. Click on a style to use it for the selected line. The line style sample updates to show the new selection. Click on the ... button at the right of the line style to open the **Custom Line** dialog, where you can specify a custom line style.

Complex line styles can be selected for most object types. Wireframe map layers and surface mesh lines do not support complex line styles. When a complex line style is selected, the line *Width* automatically increased to 0.03125 inches (0.079 cm). Most complex line styles require lines with the *Width* set to something larger than 0.03125 inches to fully distinguish the line style.

Color

Click the color next to *Color* to open the color palette. Click on a color to use it for the selected line. The color box and the sample line update to show the new selection. Click on the ... button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Opacity

The *Opacity* is the amount of transparency of the line. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click and drag the slider to change the opacity percentage.

Width

The *Width* controls the thickness of the line in page units. The value can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide. To change the *Width*, highlight the existing number and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the spin buttons to increase or decrease the width.

End Styles

The *End Styles* section is unique to polylines. The ends of the polylines can have arrowheads on them as defined in the *End Styles* section. To open the *End Styles* section, click the ⊞ next to *End Styles*.

The *Start* style is placed at the first vertex of the polyline. The *End* style is placed at the last vertex of the polyline. To change the *Start* or *End* style, click on the current option and select the desired option from the list.

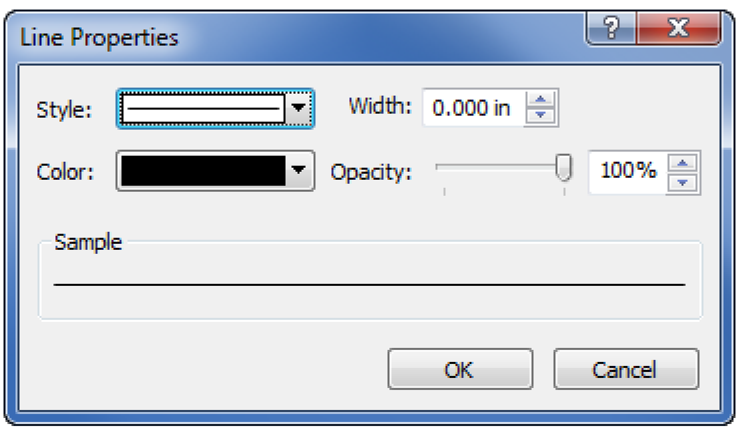
The *Scale* determines the scale factor of the arrowhead. To change the size, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the values. A value of 1 makes the arrow the default size.

Line Properties Dialog

Use the **Line Properties** dialog to change line properties for selected lines in the document. You can set default line properties in the **Options** dialog Defaults page by clicking on the *Line option*.

Most line properties are edited in the Properties window in a Line Properties section. When changing line properties for a selected object, the *Line Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Some objects use a **Line Properties** dialog. This dialog is accessed differently, depending on the type of object selected. For example, with the *Level method* set to *Advanced* for a contour map, the *Line Properties* dialog appears when you double-click on an individual line in the **Levels for Map** dialog.



*Specify individual line properties in the **Line Properties** dialog.*

Style

Click the button next *Style* to open the line style palette. Click on a style to use it for the selected line. The line style sample updates to show the new selection. Click on the *Custom* button at the bottom of the line style palette to specify a custom line style.

Complex line styles can be selected for most object types. Wireframe map layers and surface mesh lines do not support complex line styles. When a complex line style is selected, the line *Width* automatically increased to 0.03125 inches (0.079 cm). Most complex line styles require lines with the *Width* set to something larger than 0.03125 inches to fully distinguish the line style.

Color

Click the button next to *Color* to open the color palette. Click on a color to use it for the selected line. The color box and the sample line update to show the new selection. Click on the *Custom* button at the bottom of the color palette to choose a custom color.

Width

Change the line *Width* by typing a new number into the box or by using the buttons to the right of the box to increase or decrease the value. The line width can be 0 to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide.

Opacity

The *Opacity* is the amount of transparency of the line. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value, use the buttons to the right of the box to increase or decrease the value, or click and drag the to change the opacity percentage.

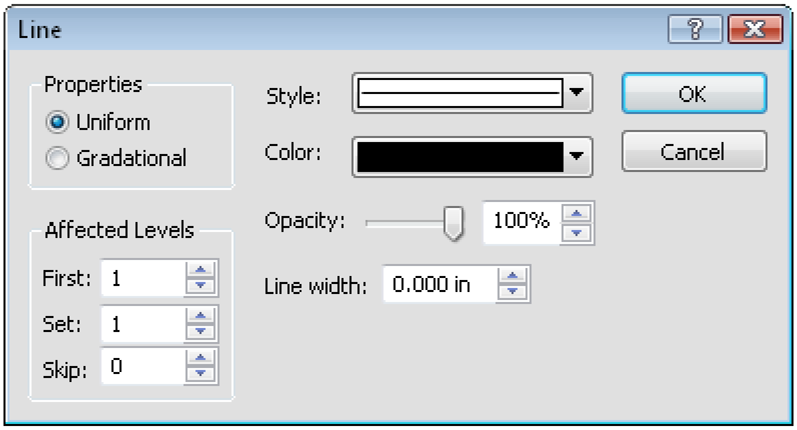
Sample

The sample of the line is displayed in the *Sample* section. The sample shows the line style, color, opacity, and width options.

Line Dialog

The **Line** dialog contains methods to assign line styles, colors, widths to specific line levels when advanced levels are used to create a contour map. Refer to the contour level settings for simple contour maps.

Click the *Line* button on the Levels page of the Levels for Map dialog to open the **Line** dialog.



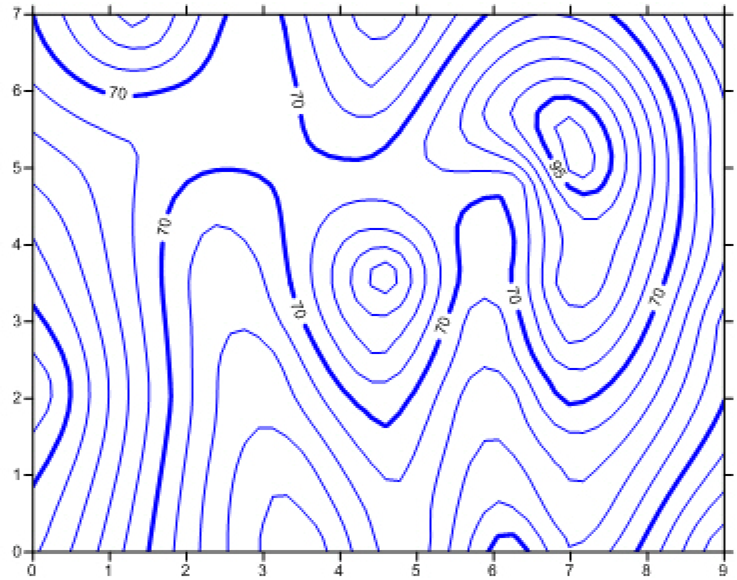
Specify line properties for multiple contour lines in the ***Line*** *dialog.*

Properties

In the *Properties* group, select *Uniform* to apply the same line properties to all lines. Select *Gradational* to apply a color spectrum to the lines.

Affected Levels

The *Affected Levels* group is used to set line color on a frequency basis.



You can set the frequency of the line properties in the Affected Levels group. This example has an index contour of a thicker width every fifth contour line.

Style

Click the button next *Style* to open the line style palette. Click on a style to use it for the lines. The line style sample updates to show the new selection. Click on the *Custom* button at the bottom of the line style palette to specify a custom line style.

Color

If you have *Uniform* selected, clicking the *Color* button opens the color palette. Click on a color to use it for the selected line. The color box and the sample line update to show the new selection. Click on the *Custom* button at the bottom of the color palette to choose a custom color.

If you have *Gradational* selected, clicking the *Color* button opens the Colormap Editor. Set the properties in the **Colormap Editor** and click *OK* to return to the **Line** dialog.

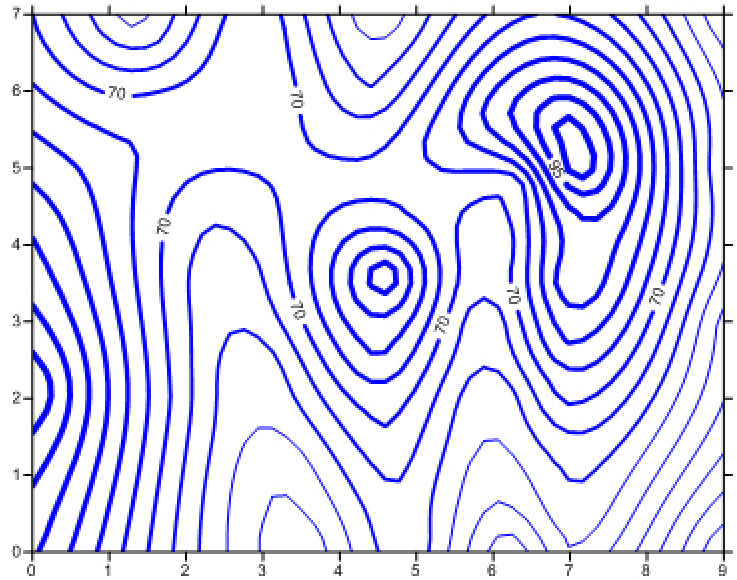
Opacity

The *Opacity* is the amount of transparency of the lines. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value, use the buttons to the right of the box to increase or decrease the value, or click and drag the to change the opacity percentage.

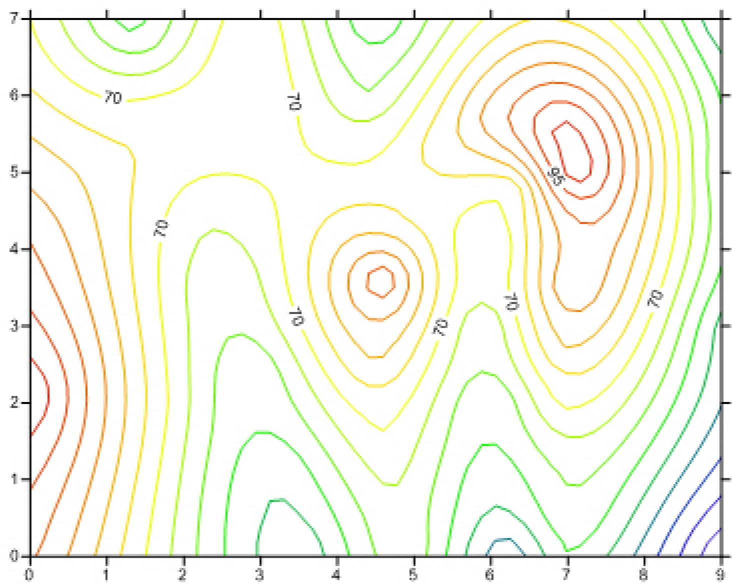
Line Width

If you have *Uniform* selected, change the *Line width* by typing a new number into the box or by using the buttons to the right of the box to increase or decrease the value. The line width can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide.

If you have *Gradational* selected, enter a *Min width* and *Max width* value to create a gradational line width for the contour map. The line width can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide. To set the *Min width* or *Max width*, type a new number into the box or use the buttons to the right of the box to increase or decrease the value. The line width can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide.



This is a contour map using gradational line widths.



This is a contour map using gradational line colors.

Assigning Gradational Line Properties

Gradational line properties on a contour map can be done with the advanced contour level method. To create gradational lines,

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Advanced*.

4. Click the *Edit Levels* button next to *Contour levels* to open the advanced levels dialog.
5. Click the *Line* button to display the Line dialog.
6. In the **Line** dialog, select *Gradational* contour lines.
7. Click the line *Style* button to assign a specific style to all of the contour lines.
8. Click the *Color* button to assign a color spectrum to the contour lines.
9. To assign varying line thickness, enter numbers into the *Min width* and *Max width* edit boxes. These line widths are applied to the minimum and maximum contour levels respectively. All intermediate line widths are interpolated proportionally.
10. If the *Skip* box in the *Affected Levels* section is set to a number other than zero, the contour lines are not all assigned the colors or widths set in the **Line** dialog.
11. Click *OK* in the **Line** dialog to return to the advanced levels dialog. The selected line colors and widths are shown under *Line*. Be aware that some video cards and monitors are not capable of generating smooth gradations from one color to another, so the line spectrum might consist of ranges of one color, resulting in "color zones" rather than a smooth line color spectrum.
12. Click *OK* in the **Levels for Map** dialog and the contour map is drawn with the specified color and width contour lines.

Assigning the Same Line Properties to all Contour Levels

The easiest way to assign the same line properties to all contour levels is to use the simple levels method:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Simple*.
4. Set the *Major contour every* value to one. This means that all contour lines are major contour lines.
5. Open the Line Properties section under *Major Contours*. Set the *Style*, *Color*, *Width*, and *Opacity*.

If you have customizations on individual levels, you can still set the same line properties from the advanced levels dialog. To assign the same line properties to all contour levels:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced levels dialog.
5. Click the *Line* button to display the Line dialog.
6. In the **Line** dialog, choose *Uniform* contour lines.
7. Click the *Style* and *Color* buttons to open the line style and color palettes. Select the style and color to use for all of the contour lines. To eliminate the contour lines from the contour map, set the line style to "Invisible."

8. Enter a number into the *Line width* box to set the line width. A *Line width* of 0.00 is one pixel wide.
9. Set the *First* value to 1. This is the first contour line that will have the properties applied.
10. Set the *Set* to 1. This tells the program to set only one line.
11. Set the *Skip* to 0. The zero tells the program to skip no lines. This means all lines will have the same properties.
12. Click *OK* to return to the **Levels for Map** dialog.
13. Click *OK* in the advanced levels dialog to create a contour map with the same line properties for all contours on the map.

Setting Line Properties on a Frequency Basis

If the contour map should have a two different line types at a regular interval, the easiest way to create this interval would be to use the simple contour levels method. To set line properties on a frequency basis with simple contour levels, you need to set a major and minor contour level:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Simple*.
4. Change the *Major contour every* value to the desired frequency for major contour lines. Setting this value to one means every contour line is a major contour line. Setting this value to two means every other value is a major contour line. Setting this value to three means every third line is a major contour line, and so on.
5. Open the Line Properties section under *Major Contours*. Set the *Style*, *Color*, *Width*, and *Opacity*.
6. Open the Line Properties section under *Minor Contours*. Set the *Style*, *Color*, *Width*, and *Opacity*. This will set the line properties for all non-major contour lines.

If you have additional properties to change for an individual level, you can set line properties on a frequency basis using the advanced contour levels method. To set line properties for advanced contour levels:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced levels dialog.
5. Click the *Line* button to display the Line dialog.
6. Set the *Style, Color, Width,* and select *Uniform* or *Gradational* for the line.
7. Set the *First* value to the first contour line position that will have the properties applied.
8. Set the *Set* value to the number of contour lines to apply this property.
9. Set the *Skip* value to the number of contour lines to not apply this property.
10. Click *OK* to return to the **Levels for Map** dialog.

11. Click *OK* in the **Levels for Map** dialog to create a contour map with the same line properties for all contours on the map.

Affected Levels

The *Affected Levels* settings are used to modify the contour levels on a frequency basis. These settings can be found by clicking the Line button on the Levels page in the contour map advanced levels dialog. The *First*, *Set*, and *Skip* fields allow you to specify which contour levels are modified.

First

The *First* field indicates the first contour level affected by the change. The number refers to the contour level position in the list (starting with 1) and not the actual contour value.

Set

The *Set* field indicates the number of levels to apply the changes.

Skip

The *Skip* field indicates the number of contour levels to skip when assigning the specified properties.

Apply Multiple Sets of Affected Levels

You can set the *Affected Levels* as many times as you want before leaving the dialog. For example:

1. In the **Line** dialog, enter First=one,Set=one,Skip=four and click theOKbutton. This applies the properties to the first contour level, skip four levels, set the next level, and so on.
2. In the **Line** dialog, enter *First* =two, *Set* =four, *Skip* =one and then click the *OK* button. This applies the new properties starting at the second level. Four consecutive levels are modified, one is skipped, four are modified, one is skipped, etc. This combination of *Affected Levels* applies one set of properties to one group of levels and another set of properties to a second group of levels. Combinations of settings for line properties are limitless.

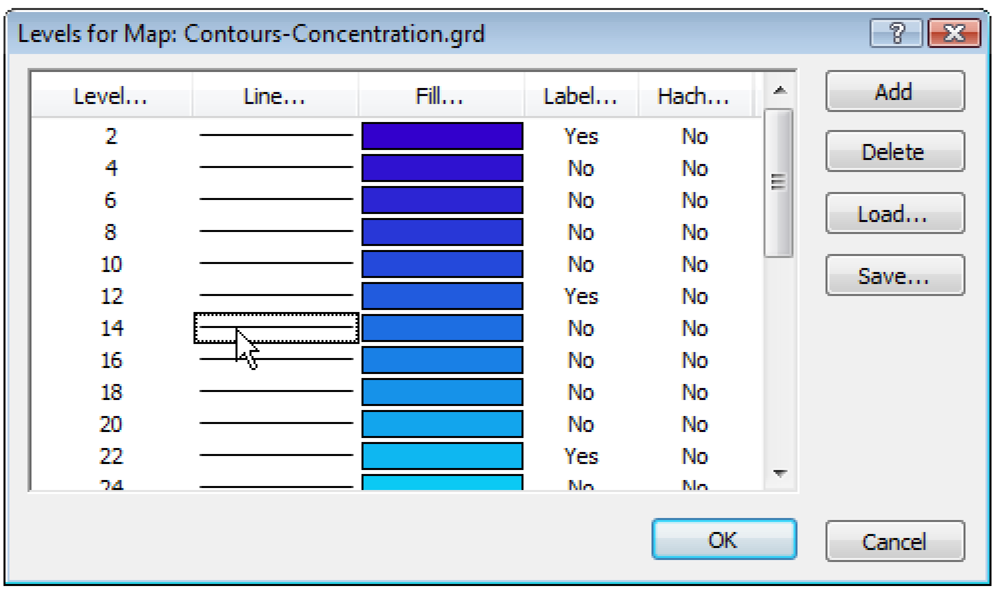
Levels	First = 1 Set = 1 Skip = 4	First = 2 Set = 4 Skip = 1
0	Yes	No
10	No	Yes
20	No	Yes
30	No	Yes
40	No	Yes

50	Yes	No
60	No	Yes
70	No	Yes
80	No	Yes
90	No	Yes
100	Yes	No

Assigning Line Properties to Specific Contour Levels

To assign line properties to specific contour levels, you must use the advanced contour levels method.

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels* to open the **Levels for Map** dialog.
5. Double-click the line sample for the contour line you want to modify.

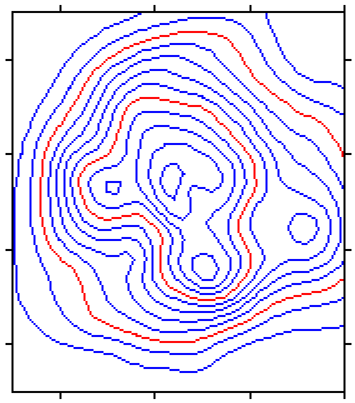


In this example, the line sample for the 14 Level *is selected.*

6. The Line Properties dialog is displayed, allowing you to select the line style, color, width, and opacity.
7. Click *OK* in the **Line Properties** dialog and repeat the procedure for any other contour lines you want to change.
8. When you are done, click *OK* and the map is drawn with the specified line properties.

Index (Major) Contours

Index contours are lines of different colors, styles, or widths that appear at a regular frequency. With simple and logarithmic contour levels, this is called *Major Contours*. You can create index contours in several ways: use *Major Contours* with the simple contour levels. With advanced contour levels, you can set individual contour line properties, or use the *Affected Levels* group in the Line dialog to automatically assign the index contours.



This is an example of a map with red index contours.

To set index line properties with simple contour levels, you need to set a major and minor contour level:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Simple*.
4. Change the *Major contour every* value to the desired frequency for major contour lines. This is the interval for the index contours. To have a red index line every fifth line, set this value to 5.
5. Open the Line Properties section under *Major Contours*. This is the index contour properties. Set the *Color* to Red.
6. Open the Line Properties section under *Minor Contours*. This will set the line properties for all non-major contour lines. Set the *Color* to Blue.

To set index line properties with logarithmic contour levels, you need to set the number of minor levels per decade:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Logarithmic*.
4. Change the *Minor levels per decade* value to the desired frequency for minor contour lines.
5. Open the Line Properties section under *Major Contours*. This is the index contour properties. Set the *Color* to Red. When using the logarithmic contour

level method, the decades are automatically defined to be multiples of 10 (0.001, 0.01, 0.1, 1, 10, 100, 1000, etc).

6. Open the Line Properties section under *Minor Contours*. This will set the line properties for all non-major contour lines. Set the *Color* to Blue.

To create index contours with advanced contour levels, you can use the affected levels options:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced **Levels for Map** dialog.
5. Click the *Line* button to display the Line dialog.
6. Click the line *Color* button and change the color to red.
7. Set the index contour line properties in the *Affected Levels* group in the **Line** dialog.
8. In the *Affected Levels* group, set *First* to one, *Set* to one, and *Skip* to four. This means, starting with the first contour level, set one contour level to red, skip four contour levels, and then set the fifth contour level to red, skip four more levels, etc.
9. Click *OK*.
10. Next, set the line style and color used on the intermediate levels. Click the *Line* button on the Levels page.
11. Click the line *Color* button and change the line color to blue.
12. In the *Affected Levels* group, set *First* to two, *Set* to four, and *Skip* to one. This means, starting with the second contour level (the first level is a red index contour) change the line color to blue for the next four contour levels, skip one contour level, set the next four contour levels to blue, etc.
13. Click *OK* in the **Line** dialog to return to the contour map properties dialog. The first contour level line appears as red, the next four are blue, followed by one red line, etc.
14. Click *OK* in the contour map properties dialog to draw the map with the specified contour lines.

The Affected Levels group contains *First*, *Set*, and *Skip* fields. This group allows you to apply the selected line properties to a repeating series of levels. For example, use the *Affected Levels* options to create four blue contour lines, one red contour line, followed by four blue contour lines, etc. The *Affected Levels* group exists in the **Line**, **Fill**, **Labels**, and **Hachures** dialogs.

Adding Color Fill between Contours

Surfer allows you to fill the areas between contour lines with color fill. The color fill can gradationally change from the minimum to maximum contours, or specific fill properties can be assigned to individual contour levels.

Gradational colors are assigned in the Properties window for predefined color spectrums. Or, gradational colors can be defined in the Colormap Editor. For example, if the color red is assigned to the minimum value, and the color blue to the maximum value, the resulting color spectrum gradually changes from red to blue.

Individual fill colors for specific contour levels can be assigned from the advanced contour levels dialog by double-clicking the fill sample for a contour value on the contour map advanced levels dialog. There is no gradational change for fill patterns.

Fill Properties - Contours

Contour maps can be filled using a simple or logarithmic contour level method or an advanced contour level method.

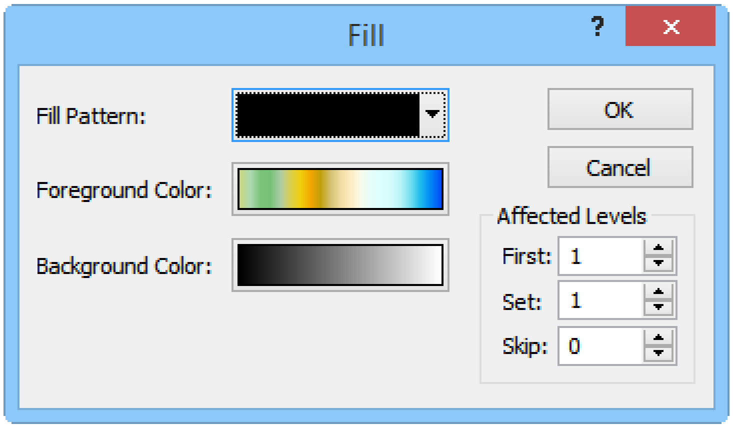
Use the *Fill colors* option in the Properties window to change the fill color properties for a simple level contour map.

1. Click on the contour map to select it.
2. In the **Properties** window, open the Levels tab.
3. Set the *Level method* to *Simple* or *Logarithmic*.
4. Check the box next to *Fill contours*.
5. Click the color bar next to *Fill colors.* Select the desired predefined colormap.
6. If a different colormap is desired, click the ... button to the right of the current color bar. The Colormap Editor appears, where you can make any customizations.

Fill Dialog

With an advanced level contour map, the **Fill** dialog will open, allowing you to set additional properties. To open the **Fill** dialog:

1. Click on the contour map to select it.
2. In the **Properties** window, open the Levels tab.
3. Check the box next to *Fill contours*.
4. Set the *Level method* to *Advanced*.
5. Click the *Edit Levels* button next to *Contour levels.*
6. In the advanced contour Levels for Map dialog, click the *Fill* button. The **Fill** dialog opens.



*Specify methods to assign fill patterns and colors in the **Fill** dialog.*

Fill Pattern

Change the *Fill Pattern* by selecting a pattern from the pattern palette. Open the pattern palette by clicking the fill pattern button. Any pattern, including solid filled patterns, can be transparent by setting the *Opacity* in the Colormap Editor.

Foreground Color

Foreground Color is the color of the pattern lines or pixels. With the *Fill Pattern* set to *Solid*, the *Foreground Color* is the solid color. Click on the *Foreground Color* button to open the Colormap Editor. Foreground color *Opacity* can be controlled in the **Colormap Editor**.

Background Color

Background Color is the color behind the pattern. All raster (pixel) patterns must have a background color. Click on the *Background Color* button to open the **Colormap Editor**. Background color *Opacity* can be controlled in the **Colormap Editor**.

Affected Levels

The *Affected Levels* settings are used to modify the contour levels on a frequency basis. The *First*, *Set*, and *Skip* fields allow you to specify which contour levels are modified.

First

The *First* field indicates the first contour level affected by the change. The number refers to the contour level position in the list (starting with 1) and not the actual contour value.

Set

The *Set* field indicates the number of levels to apply the changes.

Skip

The *Skip* field indicates the number of contour levels to skip when assigning the specified properties.

Assigning Color Fill Based on a Fill Spectrum

The Levels page properties can be used to apply a color fill to the contour layer levels.

To assign color fill based on a fill spectrum for a simple level contour map, use the *Fill Colors* option:

1. Click on the contour map to select it.
2. In the Properties window, open the Levels tab.
3. Set the *Level method* to *Simple*.
4. Check the box next to *Fill contours*.
5. Click the color bar next to *Fill colors*. Select the desired predefined colormap.
6. If a different colormap is desired, click the ... button to the right of the selected colormap. The Colormap Editor appears, where you can make any customizations.

To assign color fill based on a fill spectrum for a logarithmic level contour map, use the *Fill Colors* option:

1. Click on the contour map to select it.
2. In the Properties window, open the Levels tab.
3. Set the *Level method* to *Logarithmic*.
4. Check the box next to *Fill contours*.
5. Click the color bar next to *Fill colors*. Select the desired predefined colormap.
6. Click the ... button to the right of the selected colormap. The Colormap Editor appears, where you can make any customizations. Check the box next to *Logarithmic scaling*, if it is not already checked.

To assign color fill based on a fill spectrum for an advanced level contour map:

1. Click on the contour map to select it.
2. In the Properties window, open the Levels tab.
3. Check the box next to *Fill contours*.
4. Set the *Level method* to *Advanced*.
5. Click the *Edit Levels* button next to *Contour levels*.
6. Click the *Fill* button to display the Fill dialog.
7. Click the *Fill Pattern* button to display the pattern palette. The fill pattern is constant for all levels.
8. Click the *Foreground Color* or *Background Color* buttons to open the Colormap Editor. The foreground color is used for the pattern. The background color is used for areas behind the pattern. The *Background Mode* can

be set to *Opaque* or *Transparent* with stock Windows patterns (except *Solid*). With image fills, the background is always set to *Opaque*.

9. Click *OK* in the **Fill** dialog to return to the **Levels for Map** dialog. The specified fill properties are displayed under the *Fill* button.
10. Click *OK* and the contour map is drawn with colors filling the areas between the contour lines.

Assigning Color Fill to Specific Contour Levels

To assign color fill to specific contour levels, you need to use the advanced contour level method.

1. Click on the contour map to select it.
2. In the Properties window, open the Levels tab.
3. Set the *Level method* to *Advanced*.
4. Check the box next to *Fill contours* .
5. Click the *Edit Levels* button next to *Contour levels*.
6. Double-click the fill sample for the contour level you want to modify.
7. Select the fill properties from the Fill Properties dialog.
8. Click *OK* in the **Fill Properties** dialog and repeat the procedure for all of the contour levels you want to change.
9. Click *OK* and the contour map is drawn with colors filling the areas between the contour lines.

Creating a Filled Contour Map Containing NoData Areas

When a filled contour map contains regions with NoData values, the NoData area has a default color assigned to it. You can select the fill for the NoData areas as well as the line surrounding the NoData areas on the General page in the contour map properties. NoData areas are typically created with the Grids | Edit | Assign NoData command. NoData areas can also be created due to insufficient data during the gridding process.

To assign fill and line properties to NoData areas:

1. Click on the contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Check the box next to *Fill contours*.
4. Click on the General tab.
5. In the *NoData Regions* section, click the ⊞ next to Fill Properties. Choose the *Pattern*, *Foreground Color*, *Foreground Opacity*, *Background Color*, and *Background Opacity* for the NoData areas. Use the *Load From* if you want the *Pattern* to use an image on your computer.
6. Click the ⊞ next to Line Properties. Choose the line *Style*, *Color*, *Opacity,* and *Width* for the line surrounding NoData areas.

The contour map automatically updates to show the NoData area properties.

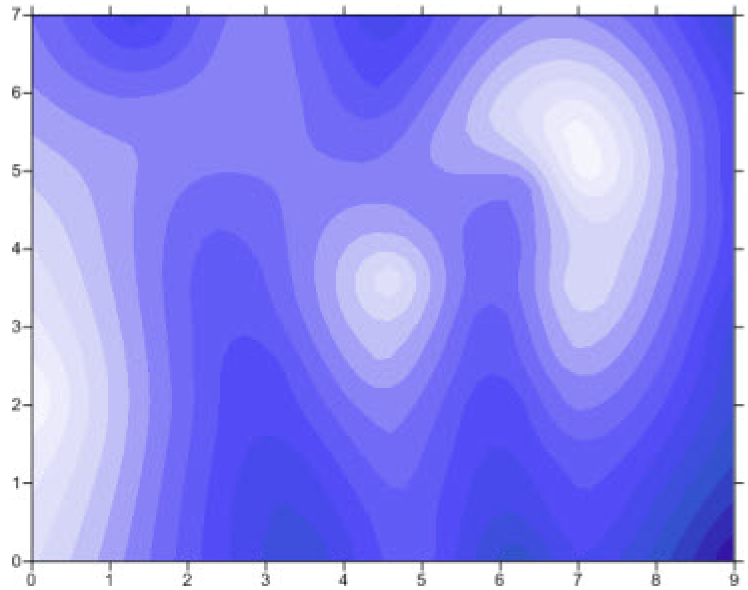
Displaying a Filled Contour Map without Contour Lines

To create a filled contour map without contour lines using a simple or logarithmic level contour map:

1. Click on the contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Set the *Level method* to *Simple* or *Logarithmic*.
4. Check the box next to *Fill contours*.
5. In the *Major Contours* section,
    a. Click the ⊞ next to *Line Properties.*
    b. Set the *Style* to *Invisible*.
    c. Uncheck the box next to *Show labels*.
6. In the *Minor Contours* section,
    a. Click the ⊞ next to *Line Properties*.
    b. Set the *Style* to *Invisible.*
    c. Uncheck the box next to *Show labels*.

To create a filled contour map without contour lines using an advanced level contour map,

1. Click on the contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Set the *Level method* to *Advanced*.
4. Check the box next to *Fill contours*.
5. Click the *Edit Levels* button next to *Contour levels.*
6. Click the *Line* button to open the **Line** dialog.
7. In the Line dialog, click the line *Style* palette and choose the *Invisible* line style.
8. In the *Affected Levels* group, set *First* to one, *Set* to one, and *Skip* to zero.
9. Click *OK* in the **Line** dialog to return to the **Levels for Map** dialog.
10. Click *OK* in the **Levels for Map** dialog, and the filled contour map is displayed without contour lines.



This contour map (Demogrid.grd) has invisible contour lines.

Setting Fill Properties on a Frequency Basis

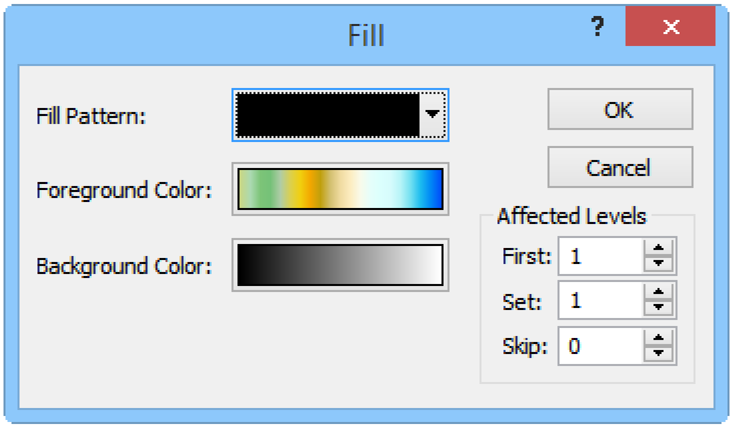
If the contour map should have a two different fill types at a regular interval, the easiest way to create this interval would be to use the advanced contour levels method. To set fill properties for advanced contour levels:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced levels dialog.
5. Click the *Fill* button to display the Fill dialog.
6. Set the *Fill Pattern, Foreground Color, Background Color,* and *Background Mode* for the fill.
7. Set the *First* value to the first contour fill position that will have the properties applied.
8. Set the *Set* value to the number of contour intervals to apply this property.
9. Set the *Skip* value to the number of contour intervals to not apply this property.
10. Click *OK* to return to the **Levels for Map** dialog.

11. Click *OK* in the **Levels for Map** dialog to create a contour map with the same fill properties for all contours on the map.

Fill Dialog

Click the Fill button on the Levels page in the contour map properties dialog to open the **Fill** dialog.



The First, Set, *and* Skip *fields in the* **Fill** *dialog specify which contour levels are modified.*

Affected Levels

The *Affected Levels* settings are used to modify the contour levels on a frequency basis. These settings can be found by clicking the Fill button on the Levels page in the contour map advanced levels dialog. The *First*, *Set*, and *Skip* fields allow you to specify which contour levels are modified.

First

The *First* field indicates the first contour level affected by the change. The number refers to the contour level position in the list (starting with 1) and not the actual contour value.

Set

The *Set* field indicates the number of levels to apply the changes.

Skip

The *Skip* field indicates the number of contour levels to skip when assigning the specified properties.

The *Affected Levels* settings are used to modify the contour levels on a frequency basis. To use *Affected Levels* with fills

1. Click on the contour map to select it.

2. In the **Properties** window, click on the **Levels** tab.

3. Set the *Level method* to *Advanced*.

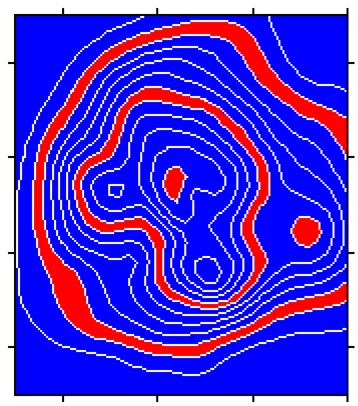
4. Click the *Edit Levels* button next to *Contour levels*.

5. Click the *Fill* button.

6. Set the *First* contour level to apply the properties to, the number of levels to be set, and the number of levels to skip.

7. You can set the *Affected Levels* as many times as you want before leaving the dialog. For example, entering First=one,

Levels	First = 1 Set = 1 Skip = 4	First = 2 Set = 4 Skip = 1
0	Yes	No
10	No	Yes
20	No	Yes
30	No	Yes
40	No	Yes
50	Yes	No
60	No	Yes
70	No	Yes
80	No	Yes
90	No	Yes
100	Yes	No



This is an example of a map using multiple fill styles.

Labels - Contour

Contour labels indicate the value of the contour line. Labels are regularly spaced along the contour line subject to curvature restrictions. Contour labels can use any text properties and numeric format, but all contour labels in a given map must use the same properties.

Contour labels are controlled from the contour map properties. Labels can be controlled when using simple, logarithmic, or advanced level contour maps. To show labels when using simple or logarithmic contour maps,

1. Click on the contour map to select it.

2. In the Properties window, open the Levels tab.

3. Set the *Level method* to *Simple*or*Logarithmic*.

4. Check the box next to *Show labels* in the *Major Contours* section to show labels along the major contour lines.

5. Check the box next to *Show labels* in the *Minor Contours* section to show labels along the minor contour lines.

6. Click the ⊞ next to *Labels* to set the properties for the labels.

To show labels on advanced contour level maps,

1. Click on the contour map to select it.

2. In the Properties window, click on the Levels tab.

3. Set the *Level method* to *Advanced*.

4. Click the *Edit Levels* button next to *Contour levels.*

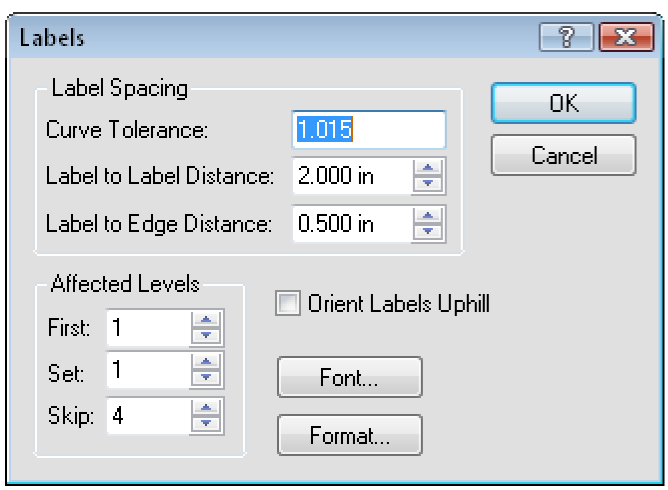
5. To turn the display of individual labels on or off, double-click on the word *Yes* or *No*. In the *Label* column, the labeled contour levels are indicated by the word *Yes*.

6. To change properties for all labels, click the *Label* button. The **Labels** dialog is displayed.

7. Make any changes and click*OK*in both dialogs. The change is shown on the map.

Labels Dialog

Click the *Label* button to open the **Labels** dialog.



*Customize label spacing in the***Labels***dialog.*

Label Spacing

The *Label Spacing* group controls where labels are placed on the contour lines.

Curve Tolerance

Curve Tolerance specifies the maximum amount of contour curvature allowed when placing labels on contour lines. Curve tolerance is calculated by dividing the actual distance along the contour line by the straight-line distance between the end points of the contour label. Highly curved lines might not be labeled automatically. You can increase the curve tolerance value to allow labels on highly curved contour lines, although contour labels might be hard to read. The default *Curve Tolerance* value of 1.015 should be acceptable in most cases.

Label to Label Distance

Label to Label Distance specifies the minimum distance (in inches or centimeters) between labels along the contour line. **Surfer** searches for the next suitable location for a label after moving the minimum distance specified by the *Label to Label Distance*. The next label is not drawn until a segment of the necessary length, within the curve tolerance limits, is found. As the *Label to Label Distance* is increased, fewer labels are drawn on the contours.

Label to Edge Distance

Label to Edge Distance specifies the minimum distance (in inches or centimeters) from the label to the edge of the map. This feature controls label placement so labels do not overwrite the map borders or axes.

Affected Levels - Label Frequency

The *Affected Levels* settings are used to modify the contour levels on a frequency basis. The *First*, *Set*, and *Skip* fields allow you to specify which contour levels are modified.

First

The *First* field indicates the first contour level affected by the change. The number refers to the contour level position in the list (starting with 1) and not the actual contour value.

Set

The *Set* field indicates the number of levels to apply the changes.

Skip

The *Skip* field indicates the number of contour levels to skip when assigning the specified properties.

Label Orientation

The *Orient Labels Uphill* check box displays the labels so they are always oriented uphill. If this box is checked, the "tops" of the labels point uphill. If this option is unchecked, the labels always are oriented right-side up on the page.

Font Properties

Click the *Font* button to display the Font Properties dialog. Choose the *font*, *points*, *style*, *color*, and *opacity* to use for the contour labels in this dialog.

Label Format

Click the *Format* button to display the Label Format dialog. This dialog allows you to specify the numeric format to use for the contour labels (i.e. number of decimal places).

When the *Type* is set to *Date/time*, if *Invalid date* text is displayed instead of the actual date/time format, the value for the label is outside the defined date/time range.

Edit Contour Labels

Contour labels can be interactively moved, added, and deleted. To enter contour label edit mode, select a contour map by clicking on it in the plot window or by clicking on it in the Contents window. Next, click the **Map Tools | Edit Layer | Contour Labels** command or the  button. Alternatively, right-click on the selected map and select *Edit Contour Labels*. Each contour label is displayed with a rectangular outline.

Move Individual Contour Label

To move a label, select it with the mouse or keyboard and drag it to a new location. To drag with the mouse, hold down the left mouse button and move the cursor to a new location. To drag with the keyboard, position the cursor on top of the label, hold down the SPACEBAR, and press the ARROW keys.

Cancel and Undo Label Movement

To cancel label movement and to return the label to its original position, press the ESC key before releasing the left mouse button or SPACEBAR.

Use the Undo command to undo a label move, label deletion, or label insert.

Add a Contour Label

To insert a contour label, hold down the CTRL key and left-click with the mouse on a contour line. A label is not added if the contour is too small or too curved for a label to fit.

Delete a Contour Label

To delete a label, click once on the label and press the DELETE key.

Move Around the Plot Window in Edit mode

If you are zoomed in on the contours, use the scroll bars to move to off window locations. Alternatively, click and hold the scroll button of a wheel mouse to pan the plot window.

Exit the Edit mode

To leave contour label edit mode, press the ESC key or click the **Map Tools | Edit Layer | Contour Labels** command again.

Reset Contour Labels to Default Positions

If you make any changes to the contour map, changes to the contour labels are deleted. **Surfer** displays a warning message that the custom labels will be lost in this case.

Setting Label Properties on a Frequency Basis

If the contour map should have a two labels on a frequency basis, the easiest way to create this interval would be to use the simple contour levels method. To set label frequency with simple contour levels, you need to set a major and minor contour level:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Simple*.
4. Change the *Major contour every* value to the desired frequency for major contour labels. Setting this value to one means every contour line is a major contour line. Setting this value to two means every other value is a major contour line. Setting this value to three means every third line is a major contour line, and so on. Only the major contour lines will be labeled.
5. Open the Line Properties section under *Major Contours*. Check the *Show labels* box.
6. Open the Line Properties section under *Minor Contours*. Uncheck the *Show labels* box.

If you have additional properties to change for an individual level, you can set label properties on a frequency basis using the advanced contour levels method. To set label properties using advanced contour levels:

1. Click on an existing contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Change the *Level method* to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced **Levels for Map** dialog.
5. Click on the *Label* button to open the Labels dialog.
6. Set the *First* contour level affected by the setting, the *Set* number indicating the number of levels to set with the specified properties, and the *Skip* number indicating the number of levels to skip assigning the specified properties.

Labels can only have one set of *Affected Levels*. If the *Affected Levels* are set a second time, they override the changes made by the first.

Levels	First = 1 Set = 1 Skip = 4
6000	Yes
6100	No
6200	No
6300	No
6400	No
6500	Yes
6600	No
6700	No
6800	No
6900	No
7000	Yes

Removing All Labels

To avoid displaying contour labels on the contour map when the *Level method* is set to *Simple*:

1. Click on the contour map to select it.
2. In the Properties window, click on the Levels tab.

3. In the *Major Contours* section, uncheck the *Show labels* box. All major labels are now removed from the contour map.

4. In the *Minor Contours* section, uncheck the *Show labels* box. All minor labels are now removed from the contour map.

To avoid displaying contour labels on the contour map when the *Level method* is set to *Advanced,* follow these steps:

1. Click on the contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Set the *Level method* to Advanced.
4. Click the *Edit Levels* button next to *Contour levels*.
5. Click the *Label* button.
6. In the *Affected Levels* section of the Labels dialog, change the *First* to 1, the *Set* to 0, and the *Skip* to 0. Click *OK* in the **Labels** dialog to return to the contour map **Levels for Map** dialog.
7. Click *OK* in the contour map **Levels for Map** dialog to remove all labels.

Hachures

Hachures are small tick marks placed along contour lines to indicate the direction of slope. In **Surfer**, hachures can either point upslope or downslope.

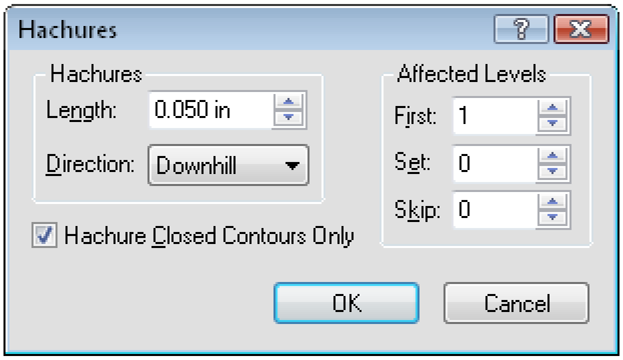
To display hachures on a contour map:

1. Click on the contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Set the *Level method* to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels.*
5. Click the *Hach* button to open the **Hachures** dialog. Set any properties. This controls the hachures for the entire map. Click *OK*.
6. In the *Hach* column, the hachured contour levels are indicated by the word *Yes*. Contours not hachured are indicated with the word *No*. To control the display of hachures for a particular level, double-click the words *Yes* or *No*.
7. Click *OK* in the dialog to make the changes on the map.

If you do not want to display hachures on any contour lines, specify a *Set* value of zero in the **Hachures** dialog.

Hachures Dialog

Click the *Hach* column button to open the **Labels** dialog.



*Use the **Hachures** dialog to customize hachure settings.*

Hachure Length

Use the *Length* box to control the length of the hachures. Hachure length can be greater than zero and up to one inch. To change the length, highlight the existing value and type a new value or click the  buttons to increase or decrease the value.

Hachure Direction

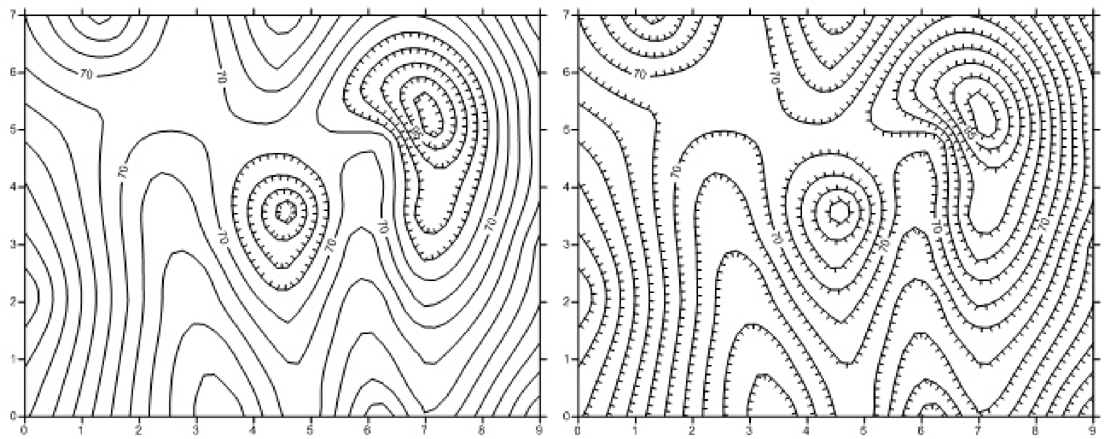
The *Direction* box controls whether hachures point in an upward or downward direction. For example, to point hachures towards contours of lower value, choose the *Downhill* option. To change the *Direction*, click on the existing option and select the desired option from the list.

Hachure Closed Contours Only

If the*Hachure Closed Contours Only* check box is checked, then only closed contour lines use hachures. If a contour line intersects the border of the map, hachures are not displayed on that contour line. If the *Direction* is set to *Uphill*, only peaks are hachured. If the *Direction* is set to *Downhill*, only depressions are hachured. If the *Hachure Closed Contours Only* option is not checked, then all contours are hachured relative to the *Affected Levels*.

Affected Levels

The *Affected Levels* group controls which levels are hachured. For more information on the use of *First*, *Set*, and *Skip* see Setting Hachures on a Frequency Basis.



These are two examples showing different hachuring parameters. The map on the left displays hachures on the uphill side of contours on closed contours only. The map on the right displays hachures on both open and closed contours, and the hachures are placed on the downhill side of the contours.

Setting Hachures on a Frequency Basis

The *Affected Levels* settings are used to modify the contour levels on a frequency basis. These settings can be found by clicking the *Hach* button on the Levels page in the contour map properties dialog. The *First*, *Set*, and *Skip* fields allow you to specify which contour levels are modified.

- The *First* field indicates the first contour level affected by the change. The number refers to the contour level position in the list (starting with one) and not the actual contour value.
- The *Set* field indicates the number of levels to apply the changes to.
- The *Skip* field indicates the number of contour levels to skip when assigning the specified properties.

To use *Affected Levels* with hachures:

1. To display hachures on a contour map:
2. Click on the contour map to select it.
3. In the Properties window, click on the Levels tab.
4. Set the *Level method* to *Advanced*.
5. Click the *Edit Levels* button next to *Contour levels.*
6. Click the *Hach* button to open the Hachures dialog.

7. Set the *First* contour level affected by the setting, the *Set* number indicating the number of levels to set with the specified properties, and the *Skip* number indicating the number of levels to skip assigning the specified properties.

8. Click *OK*.

9. Click *OK* in the dialog to make the changes on the map.

Hachures can only have one set of *Affected Levels*. If the *Affected Levels* are set a second time, they override the changes made by the first.

Example

As an example, consider a map displaying contour lines every ten feet from 50 to 100. Within the Hachures dialog, if you specify 1 (corresponding to the first contour line in the levels list) as the *First* hachured contour line, and a *Skip* value of one, then hachures appear on the contour lines at 50, 70, and 90 (every second contour line on the map).

Masking Portions of a Contour Map with a Base Map

To mask areas of a contour map, you can overlay a base map or filled shape on the contour map. The base map must contain polygons that can be filled with patterns or colors that obscure the underlying contour map.

To overlay a masking base map:

1. Create a contour map of your grid file by clicking the **Home | New Map | Contour** command.
2. Select the contour map.
3. Click the Map Tools | Add to Map | Layer | Base command and select a vector base map. This will add a base map layer to your contour map and create a single multi-layer map. Note: The base map must contain polygons (areas).
4. Right-click on the base map in the Contents window and select Order Objects | Move to Front. The base map moves to the top of the list.
5. When the base map is selected, the base map properties appear in the Properties window. Click on the **General** tab.
6. Click the ⊞ next to *Fill Properties* to specify the fill properties to apply to the boundaries from the base map. The map is automatically updated to show the boundaries that mask the underlying contour lines.

To add an empty base map and use the drawing tools:

1. Create a contour map of your grid file by clicking the **Home | New Map | Contour** command.

2. Select the contour map. Click the Map Tools | Add to Map | Layer | Empty Base command. This will add an empty base map on top of the contour layer.
3. Click the Home | Insert | Polygon command to draw a polygon over the contour map. You could use any of the **Draw** menu commands. By drawing objects inside an empty base map, the drawn objects become map objects.
4. When the base map is selected, the base map properties are displayed in the **Properties** window. Click on the **General** tab.
5. Click the ⊞ next to *Fill Properties* to specify the fill properties to apply to the boundaries from the base map. The map is automatically updated to show the boundaries that mask the underlying contour lines.

To assign the NoData value to regions of grid-based maps:

1. Create a grid file using the Grids | New Grid | Grid Data command.
2. Create a .BLN file. The .BLN file should have identical first and last points. This closes the polygon.
3. Click the Grids | Edit | Assign NoData command.
   a. Select the .GRD file created in step 1 as the *Input Grid*.
   b. Select the .BLN file created in step 2 as the *NoData Polygon Boundary.*
   c. Type a *File name* and click *OK*. A new .GRD file is created.
4. Create a contour map using the Home | New Map | Contour command. The NoData portion will not appear on the map.

Smoothing Contours

Contour smoothness controls the angularity, or roundness, of the contour lines. Smoothness in a contour map can be controlled in several ways:

- By using Grids | New Grid | Grid Data to produce a denser grid file from your original data. As a rule, contour maps made from high-density grids are visibly smoother than contour maps made from low-density grid files. For example, a 10 X 10 grid file (10 rows and 10 columns) results in more angular contours than a 50 X 50 grid file.

- By using the Grids | Edit | Spline Smooth command to insert additional rows and columns into an existing grid file.

- By using the Smooth tool in the **Grid Editor**.

- By using the Grids | Resize | Mosaic command to resample an existing grid file.

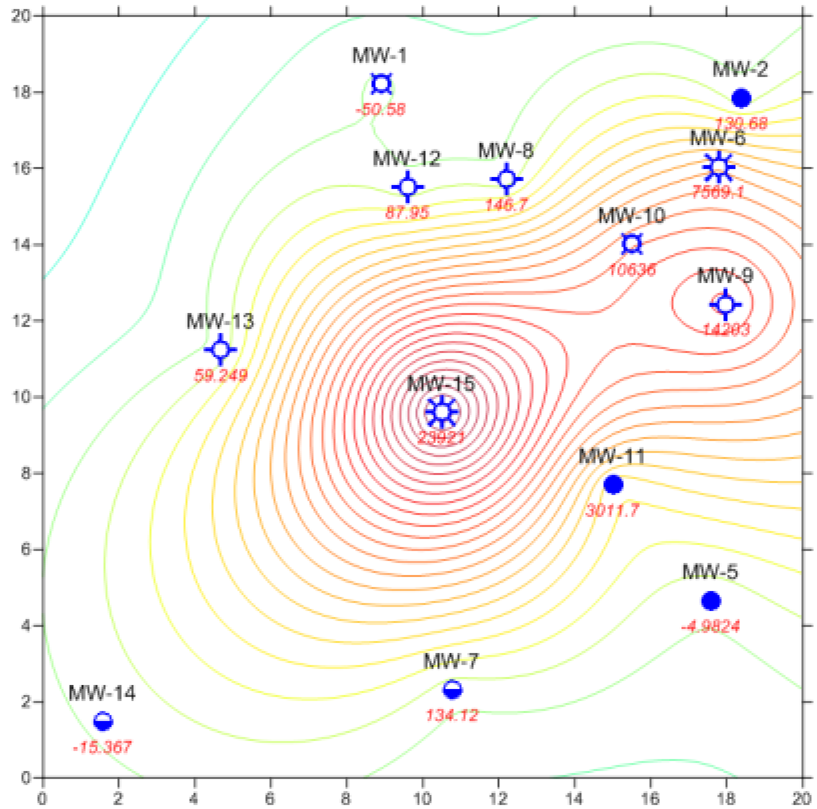
- By changing the *Smoothing* option in the contour map general properties. If smoothing is enabled, contour lines may cross. The *Smoothing* option in the contour map properties applies a constrained spline smoothing algorithm to interpolate additional contour vertices. Each line is separately calculated, thus the lines can cross if too much smoothing is applied.
- By using Grids | New Grid | Grid Data to change the gridding method or gridding method parameters. Some methods produce smoother grids than others, and some have smoothing parameters that can be utilized during gridding. See Creating Grid Files for more information on gridding.

If you perform the smoothing by creating a new grid, click on the contour map to select it. In the Properties window, click on the **General** tab. Click the button next to *Grid file* to replace the old grid in the contour map. Use the **Grid Editor | Options | Update Layer** command to update the grid in the contour map if you perform the smoothing with the grid editor.

Chapter 10 - Post and Classed Post Maps

Post Map

Post maps indicate X, Y locations with symbols and labels. Post maps display a fixed or proportionally sized symbol at each data location. Posting data points on a map can be useful for determining the spatial distribution and density of your data, as well as placing data or text information at specific locations on the map. You can specify the symbol type, size, color, and angle for the data points. Associated data values or text strings may be placed next to the posted point. The size, angle, color, and typeface for the label can also be specified. Multiple columns can be displayed as labels around each symbol.



This post map shows varied symbols and labels. A contour map is overlaid.

Post maps can be used to show the spatial distribution of the original data when overlaid on a grid-based map, such as a contour map. This is often an excellent means of presenting a qualitative measure for the accuracy of the contour lines on the map.

The limits of a post map are based on the limits of the X and Y coordinates contained within the associated data file. These limits can be changed if necessary.

Data Files

Post maps are created from data files containing X and Y coordinates. These coordinates are used to determine the locations for symbols on the post map. The data files can contain additional information to size the symbols, determine the symbol angle, set the symbol color, or post text associated with the point location. To create a post map, you must first create a data file in the format described in the Data Files Used for Posting topic.

Creating a Post Map

To create a post map:

1. Click the **Home | New Map | Post** command or the button.
2. Select a data file in the **Open Data** dialog and click *Open*.

The map is automatically created with reasonable defaults.

If the post map does not have any symbols in it, the map may not have the correct worksheet columns specified or may have text instead of numerical values in the worksheet columns. To determine if this is the case, click on the post map to select it. In the Properties window, click on the General tab. Open the *Worksheet Columns* section, if necessary. Adjust the *X coordinates* and *Y coordinates* columns. If the columns are specified correctly, check the data file to verify that the columns contain numeric data. If the data columns are correct, the post map limits may be incorrect, preventing the data from being displayed. Click on the *Map* in the Contents window to select it. Check the values on the **Limits** tab.

Editing an Existing Post Map

To change the features of the post map, open the post map properties by clicking on the post map in the plot window or clicking on the post map name in the **Contents** window. The properties are displayed in the **Properties** window.

Adding a Map Layer

When post maps are created, they are independent of other maps in the plot window. For example, creating a post map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the post map data points on the contour map, select both maps by clicking the Home | Selection | Select All command. Overlay the maps using the **Overlay Maps** command.

Alternatively, you can add the post map directly to the existing contour map by creating the post map using the **Home | Add to Map | Layer | Post** or **Map Tools | Add to Map | Layer | Post** command. This automatically adds the post map to the contour map axes.

Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the post map using the **Home | New Map | Post** command. This creates two separate maps. Click on the post map, hold down the left mouse button, and drag the post map into the contour map. A single map with two map layers, using one set of axes and scaling parameters is created.

You can combine any number of post maps with any other map.

Post Layer Properties

The post layer properties contains the following pages:

General
Symbol
Labels
Layer
Coordinate System
Info

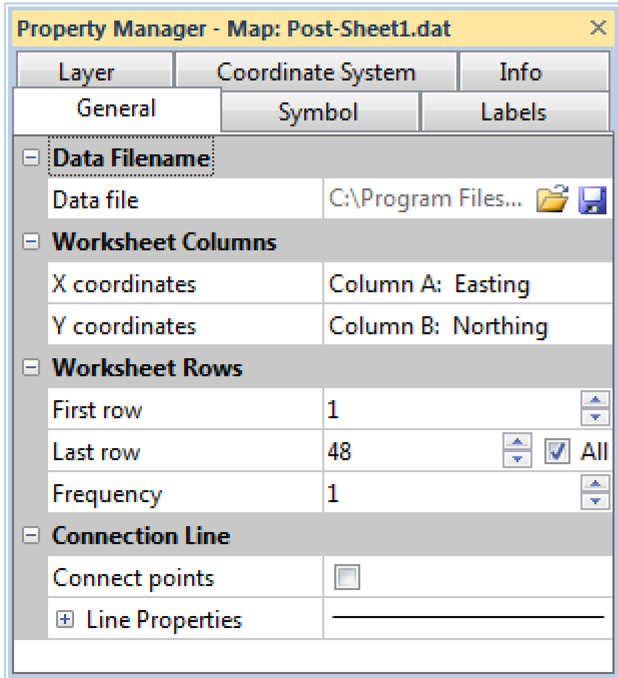
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info

Post Layer General Properties

To edit a post map, click once on the post map to select it. In the Properties window, click on the **General** tab. The post map properties **General** page contains the following options:



*Change post map properties in the **Properties** window on the **General** page.*

Data File

The *Data file* displays the current file used in the post map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Change File

Click the button to display the **Open Data** dialog. This allows a new or updated data file to be specified for the post map. Select the new data file and click *Open* to reference the new file.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

Click the button to display the **Save As** dialog. This allows the data file used for the post map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save*. If a coordinate system has been defined on the Coordinate

System tab in the **Properties** window, the Export Options dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Worksheet Columns

The *Worksheet Columns* section specifies the columns to be used from the data file.

X and Y Coordinate Columns

Set *X coordinates* and *Y coordinates* to the columns containing the X and Y coordinates, respectively. If an empty cell is encountered in either of these columns, the corresponding point is excluded from the map. To change the column, click on the existing column name and select the new column from the list.

Worksheet Rows

The *Worksheet Rows* section specifies the rows to be used from the data file.

First Row

Set the *First row* to the first row in the worksheet for which you want a posted symbol to be displayed. The default value is 1, which will start at the first numeric row of data in the worksheet. To change the first row, highlight the existing row number and type a new row number. Alteratively, click the  to increase or decrease the first row value.

Last Row

Set the *Last row* to the last row in the worksheet for which you want a posted symbol to be displayed. The default is the last row of data in the worksheet. To change the last row, highlight the existing value and type a new value. Alternatively, click the  to increase or decrease the last row value.

To return to showing all of the rows in the worksheet, check the *All* box. If the *First row* is set to 1 and the *Last row* is set to *All*, all of the rows of data in the worksheet will be used.

Frequency

Use the *Frequency* setting to control how often data points are posted. A frequency of 1 posts every point. A frequency of 2 posts every other point, 3 posts every third, etc. This is often used to reduce the number of displayed data points to avoid symbols overwriting each other.

Exclusion Filter

The *Exclusion Filter* allows a Boolean expression to specify how to exclude data. The *Exclusion Filter* can be used with any column in the worksheet that contains numbers. Columns in the worksheet that contain text or columns that are empty will not be excluded by the *Exclusion Filter*.

To use one of the X, Y, or Z columns, use X, Y, or Z in the *Exclusion Filter* . To use another column from the worksheet, use _A, _B, _C, etc. The underscore is required when specifying a worksheet column.

For example:

X=-999 or Y=-999 or Z=-999	Excludes any data with a -999 value in either the X, Y, or Z columns.
X<10 or X>20 or Y<10 or Y>20	Excludes all data except for points in the range 10 to 20 for both the X and Y directions.
Z < 0.0	Excludes any triplet with Z value less than 0.0.
_A > 10	Excludes any row in the worksheet that contains a value greater than 10 in column A.
Z < 0 AND _D = -999	Excludes any triplet with Z value less than 0.0 and whose row in the worksheet contains a value in column D equal to -999.

Boolean expressions, used by Grids | New Grid | Function, Grids | Calculate | Math, Grid | Data, and Grid | Variogram, include:

- logical operators (AND, OR, XOR, NOT)
- comparison operators (=, <>, <, >, <=, >=)
- the IF function - for example IF(condition,

The words AND, OR, XOR, NOT, and IF are reserved keywords and may not be used as variable names.

To use a stored function, click the  next to the current function. This will display the ten most recent functions used. The functions are stored in the registry, so the equations are stored between **Surfer** sessions. You can also start typing the function in the function box. If the function is in the ten function history, the entire function will auto-complete.

For example, consider the case of ignoring data outside of a grid. The original grid *X Maximum* is 50, but the grid *X Maximum* is reset to 40. To limit the search to data with X values less than 40, use the *Exclusion Filter* by entering X > 40 into the *Exclusion Filter* text box. This tells **Surfer** to exclude all data with X values greater than 40.

Consider a second case where data contains a numerical identifier in column D. When the value in this column is equal to -999, the data point is considered inaccurate and should not be used when gridding. To grid only those data where column D is not equal to -999, exclude column D with the *Exclusion Filter* by entering _D = -999 into the *Exclusion Filter* text box. This excludes all rows of data where column D contains the value -999.

Connection Line

The *Connection Line* section controls the line that connects points in the post map.

Connect Points

Check the box next to *Connect points* to connect all points in the post map layer with a line. Uncheck the box to remove the line from the post map layer. When the *Connect points* box is checked, points are connected in the order the points appear in the worksheet.

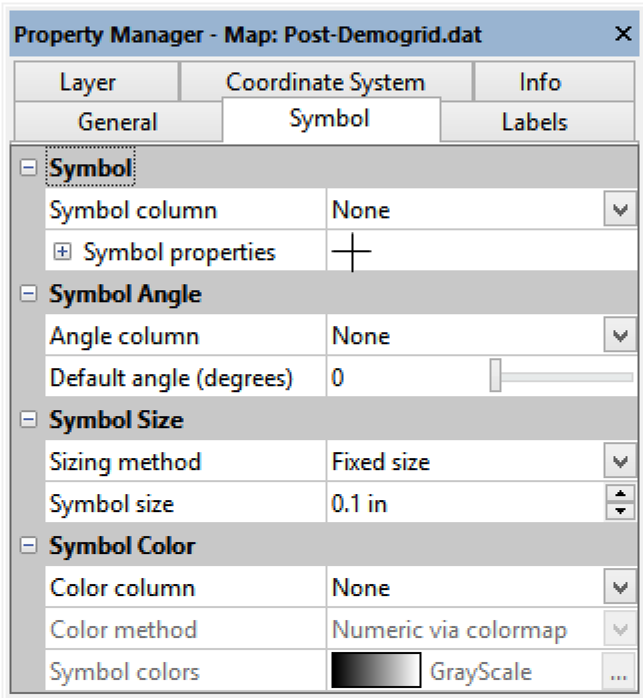
The line is drawn below the points. With open symbols, this can cause the line to appear inside the symbol. If the line should not be drawn through the symbol, use a closed symbol with the *Fill color* set to white instead of the open symbol.

Line Properties

Click on the ⊞ next to Line Properties to set the connecting line properties. The *Style, Color, Opacity* , and *Width* can be set.

Post Layer Symbol Properties

To edit a post map, click once on the post map to select it. In the Properties window, click on the **Symbol** tab. The post map properties **Symbol** page contains the following options:



Change post map symbol properties in the ***Properties*** *window on the* ***Symbol*** *page.*

Symbol

The *Symbol* section contains the *Symbol column* and default symbol properties.

Symbol Column

The *Symbol column* command specifies an optional column containing symbol information. See Symbol Specifications in the Data File for more information. To change the symbol column, click on the existing column name (or *None*) and select the column that contains the symbol information.

Symbol Properties

The *Symbol Properties* section is used to specify the symbol set and symbol (unless a *Symbol column* or *Color column* is used). Click the ⊞ next to *Symbol Properties* to open the section. From here, you can change the *Symbol* index, *Symbol Set*, *Fill color, Fill opacity*, *Line color,* and *Line opacity*.

If no symbol should be displayed, set the *Symbol size* to 0 inches or set the *Fill opacity* and *Line opacity* for the symbol to 0%.

Symbol Angle

The *Symbol Angle* section contains options for setting the *Angle column* and the default angle for all symbols.

Angle Column

The *Angle column* command specifies an optional column containing an angle to rotate each symbol. To change the angle column, click on the existing column name (or *None*) and select the column that contains the angle information. Positive values rotate the symbol counterclockwise the specified number of degrees up to +360. Negative values rotate the symbol clockwise the specified number of degrees down to -360. If an empty cell is encountered, the *Default angle (degrees)* is used for that point.

Default Angle (degrees)

The *Default angle (degrees)* specifies the angle in degrees to apply to all the symbols on the map (unless an *Angle* column is used). Positive angles of up to +360 degrees rotate the symbol in a counterclockwise fashion. To change the *Default angle (degrees)*, highlight the existing angle value and type a new value or click and drag the slider to the desired value.

Symbol Size

The *Symbol Size* section controls the symbol size in two ways: fixed or proportional.

Fixed Size

Change the *Sizing method* to *Fixed size* to use the same size symbols throughout the map. To set a new symbol size, click on the existing number next to *Symbol size*. Type in a new value, or use the spin buttons to change the *Symbol size* value. Select a value between 0 and 4 inches (0 and 10.16 cm). The fixed size is either *in* or *cm*, depending on whether *Inches* or *Centimeters* is selected for the *Page units* on the General page of the **Options** dialog.

If no symbol should be displayed, set the *Symbol size* to 0 inches or set the *Fill opacity* and *Line opacity* for the symbol to 0%.

Proportional

Change the *Sizing method* to *Proportional* to use different sized symbols on the map. Click the *Scaling* button next to *Proportional* to open the Proportional Scaling dialog. Use the scaling information in the **Proportional Scaling** dialog to size all symbols proportionally based on a data column.

Symbol Color

The *Symbol Color* section allows the symbol color to be determined from a column in two ways: using a color name or mapping a worksheet value to a color using a colormap.

Color Column

The *Color column* command specifies an optional column containing either a value or color name used to change the color of each symbol. To change the color column, click on the existing column name (or *None*) and select the column that contains the color information. To not use a color column, select *None* from the list.

Color Method

The *Color method* command controls the type of data that specifies the symbol color. Available options are *Explicit color names* and *Numeric via colormap*. To change the method, click on the existing option and select the desired option from the list.

When the *Color method* option is set to *Explicit color names*, the selected *Color column* should contain color names, RGB (red green blue) or RGBA (red green blue alpha) values. If using color names, refer to the names in any color box for a list of names. If using RGB or RGBA color values, the worksheet should contain a single column with data in the format Rxxx Gxxx Bxxx or Rxxx Gxxx Bxxx Axxx where the xxx is any value between 0 and 255. If a color name or value is missing for any row, the default post map symbol property is used.

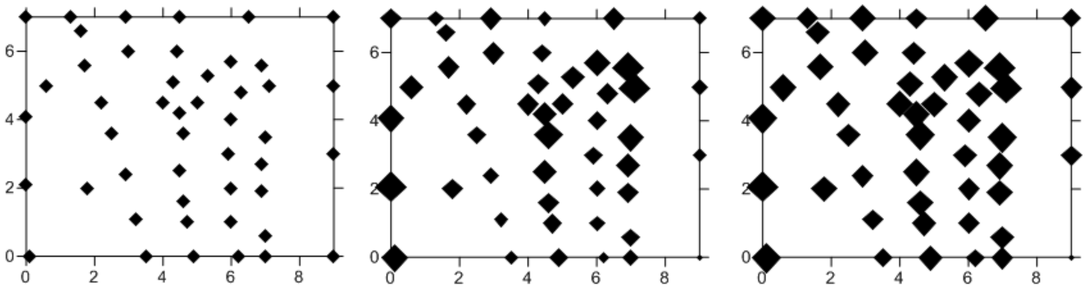
When the *Color method* is set to *Numeric via colormap*, the *Color column* should contain numeric values. The values are mapped to a colormap using the *Symbol colors* option.

Symbol Colors

The *Symbol colors* option is only available when the *Color method* is set to *Numeric via colormap*. The *Symbol colors* option defines the colormap used to map the values in the *Color column* to a color if the *Color method* is set to *Numeric via colormap*. To change the colormap used, click on the existing color bar next to *Symbol colors*. Select the new colormap from the list. If the desired color map is not listed, click the ... button to the right of the color. The Colormap Editor appears. Make any changes and click *OK* to see the change on the map.

Proportional Scaling

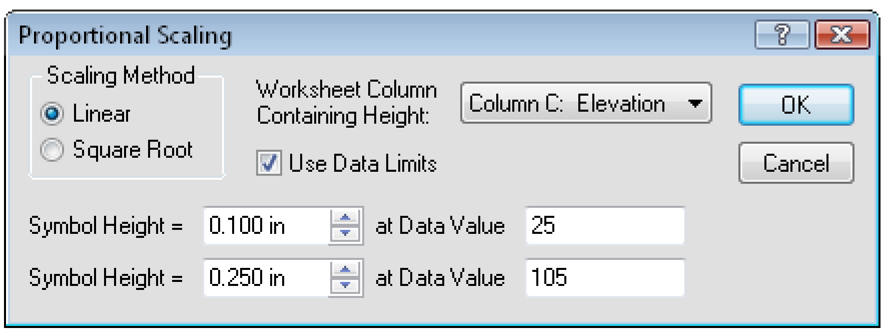
The *Symbol Size* group on the Symbol page of the post map properties allows you to specify the symbol *Sizing method* in two ways: fixed or proportional. *Fixed size* uses the same size symbols throughout the map. Set the *Sizing method* to *Fixed size* and set the *Symbol size* in page units. Change the *Sizing method* to *Proportional* to use symbols that are scaled relative to a data column.



These post maps display the same data. The map on the left shows a fixed symbol size. The map in the middle shows proportional symbols with the Scaling Method set to Linear. The map on the right shows proportional symbols with the Scaling Method set to Square Root.

Proportional Scaling Dialog

To open the **Proportional Scaling** dialog, click on the post map to select it. In the Properties window, click on the **Symbol** tab. Change the *Sizing method* to *Proportional* to use proportional scaling. Click the *Scaling* button next to *Proportional* to open the **Proportional Scaling** dialog. The *Proportional* option sizes all symbols proportionally, based on scaling information in the **Proportional Scaling** dialog. By default, the scaling is based on the data in the third worksheet column. This column usually contains the Z values. If you want to scale your symbols based on data in a different column, click the *Scaling* button to display the **Proportional Scaling** dialog. Select the desired column in the *Worksheet Column Containing Height* list.



The ***Proportional Scaling*** *dialog is opened by clicking on the* Scaling *button in the* Symbol Size *section of the* ***Symbol*** *page in the* ***Properties*** *window.*

Symbol Height

For proportional scaling, **Surfer** uses two *Symbol Height* to *Data Value* pairs to define the scaling equations. These pairs define the symbol size at two different data values, usually the minimum and maximum values in the height column. All other values are scaled proportionally. Assume the first symbol height is defined as H_1 and is associated with the minimum Z value in the data file (Z_1). Similarly, the maximum symbol height is defined as H_2 and is associated with the maximum Z value in the data file (Z_2). Then Hnis the symbol height for a symbol with the proportional value

Linear Scaling Method

If the *Scaling Method* is set to *Linear*, a standard linear interpolation is used:

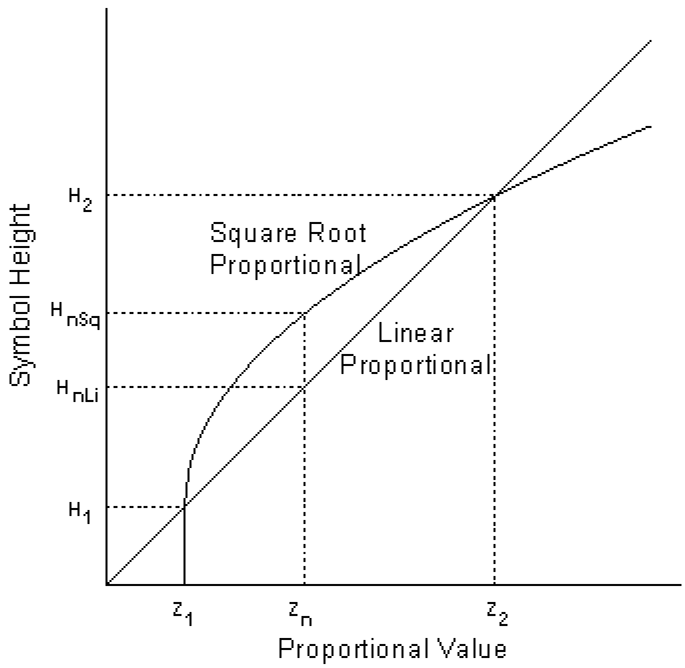
$$H_n = [((Z_n - Z_1)/(Z_2 - Z_1))*(H_2 - H_1)] + H_1$$

Square Root Scaling Method

If the *Scaling Method* is set to *Square Root*, a standard square root interpolation equation is used:

$$H_n = \left[\sqrt{(Z_n - Z_1)/(Z_2 - Z_1)} * (H_2 - H_1)\right] + H_1$$

Square root scaling is commonly used with solid symbols to offset the fact that the area increases as a function of the symbol height squared. Square root scaling is essentially making the area of the symbol proportional to the Z value, rather than making the size of the symbol proportional to the Z value.



This graph shows the height versus Z value relationship for linear proportional symbols and square root proportional symbols.

Enter the values you wish to use for H_1, H_2, Z_1, and Z_2 into the *Symbol Height* and *Data Value* fields, respectively.

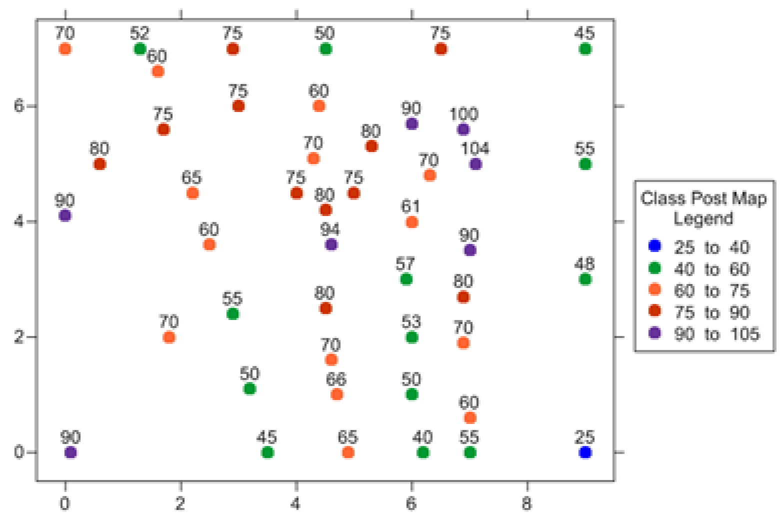
Use Data Limits

Check the *Use Data Limits* option to use the minimum and maximum data values in the specified height column for Z_1, and Z_2.

Classed Post Map

Classed post maps indicate XY locations with symbols and labels and require Z values in addition to the XY coordinates. Classed post maps group the data into discrete classes (bins). The data points are displayed using the symbol assigned to the class. Classed post maps can also include a legend.

The limits of a classed post map are based on the limits of the X and Y coordinates contained within the associated data file. These limits can be changed if necessary.



This classed post map displays colored symbols and labels. The legend to the right adds information about how the symbols are classed.

Data Files

Classed post maps are created from data files containing X and Y coordinates. The X and Y coordinates are used to determine the locations for symbols on the post map. A third worksheet column of numbers is used to separate the points into different classes. The data files can contain additional information to post text associated with the point location. To create a classed post map you must first create a data file in the format described in the Data Files Used for Posting topic.

Creating a New Classed Post Map

To create a classed post map:

1. Click the **Home | New Map | Classed Post** command or the  button.
2. Select a data file in the **Open Data** dialog and click *Open*.

The map is automatically created with reasonable defaults.

If the classed post map does not have any symbols in it, the map may not have the correct worksheet columns specified or may have text instead of numerical values in the worksheet columns. To determine if this is the case, click on the classed post map to select it. In the Properties window, click on the General tab. Open the *Worksheet Columns* section, if necessary. Adjust the *X coordinates* and *Y coordinates* columns. If the columns are specified correctly, check the data file to verify that the columns contain numeric data. If the data columns are correct, the class post map limits may be incorrect, preventing the data from being displayed. Click on the *Map* in the Contents window to select it. Check the values on the **Limits** tab.

Editing an Existing Classed Post Map

To change the features of the classed post map, click once on the classed post map in either the plot window or the **Contents** window. The properties are displayed in the **Properties** window.

Loading and Saving Classed Post Map Class .CLS Files

Surfer has the ability to load or save a classed post map's color and symbol class definitions. To load an existing class file, click on the post map to select it. In the **Properties** window, click on the **Classes** tab. Click the *Edit Classes* button to open the dialog where you can set the class properties. Click the *Load* button to import the information from an existing class file. To save the information to a new .CLS class file, click the *Save* button.

Adding a Map Layer

When classed post maps are created, they are independent of other maps in the plot window. For example, creating a classed post map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the classed post map data points on the contour map, select both maps by clicking the Select All command. Overlay the maps using the **Map Tools | Map Tools | Overlay Maps** command.

Alternatively, you can add the classed post map directly to the existing contour map by creating the classed post map using the **Home | Add to Map | Layer | Classed Post** or **Map Tools | Add to Map | Layer | Classed Post** command. This automatically adds the classed post map to the contour map axes.

Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the classed post map using the **Home | New Map | Post |**

Classed Post command. This creates two separate maps. Click on the classed post map, hold down the left mouse button, and drag the classed post map into the contour map. A single map with two map layers, using one set of axes and scaling parameters is created.

You can combine any number of classed post maps with any other map.

Classed Post Layer Properties

The classed post layer properties contains the following pages:

General
Labels
Classes
Layer
Coordinate System
Info

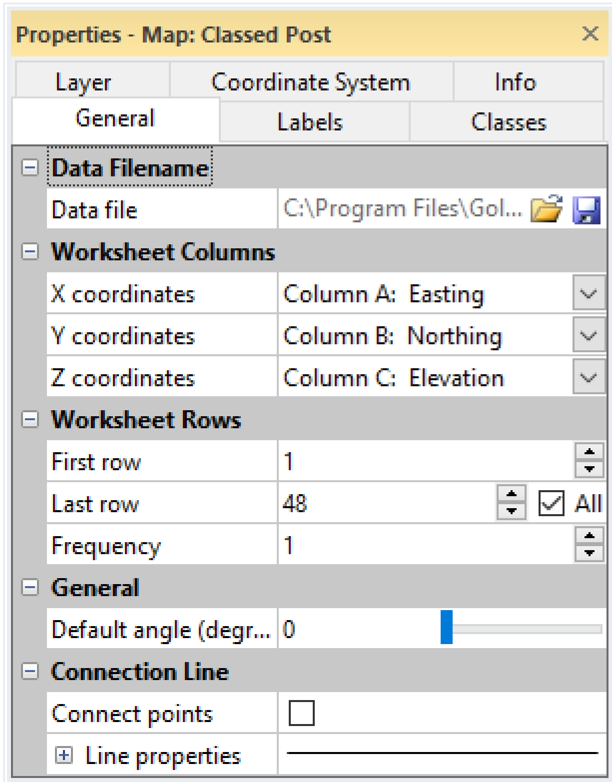
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info

Classed Post Layer General Properties

To edit a classed post map, click once on the classed post map to select it. In the Properties window, click on the **General** tab. The classed post map properties **General** page contains the following options:



*Change classed post map properties in the **Properties** window on the **General** page.*

Data Filename

The *Data Filename* section specifies the data file to use and allows the file to be saved or changed. Click the ⊞ next to *Data Filename* to open the *Data Filename* section.

Data File

The *Data file* displays the current file used in the classed post map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Change File

Click the button to display the **Open Data** dialog. This allows a new or updated data file to be specified for the classed post map. Select the new data file and click *Open* to reference the new file.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

Click the button to display the **Save As** dialog. This allows the data file used for the classed post map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file. If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, the Export Options dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Worksheet Columns

The *Worksheet Columns* section specifies the columns to be used from the data file. Click the ⊞ next to *Worksheet Columns* to open the *Worksheet Columns* section and set the columns for the classed post map layer.

X and Y Coordinates

Set *X coordinates* and *Y coordinates* to the columns containing the X and Y coordinates respectively. The X and Y columns are used to place the points on the map. If an empty cell is encountered in either of these columns, the corresponding point is excluded from the map. To change a column, click on the existing column name and select the desired column from the list.

Z Coordinates

Set the *Z coordinates column* to the column containing the values that will be used to determine the classes. If an empty cell is encountered in the Z column, the corresponding point is excluded from the map. To change a column, click on the existing column name and select the desired column from the list.

Worksheet Rows

The *Worksheet Rows* section specifies the rows to be used from the data file. Click the ⊞ next to *Worksheet Rows* to open the *Worksheet Rows* section and set the rows and frequency for the classed post map layer.

First Row

Set the *First row* to the first row in the worksheet for which you want a posted symbol to be displayed. The default value is 1, which will start at the first numeric row of data in the worksheet. To change the *First row*, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click the button to increase or decrease the *First row*.

Last Row

Set the *Last row* to the last row in the worksheet for which you want a posted symbol to be displayed. The default is the last row of data in the worksheet. To change the last row, highlight the existing value and type a new value. To return to showing all of the rows in the worksheet, check the *All* box. If the *First row* is set to 1 and the *Last row* is set to *All* , all of the rows of data in the worksheet will be used. To change the *Last row* , highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click the button to increase or decrease the *Last row*.

Frequency

Use the *Frequency* setting to control how often data points are posted. A frequency of 1 posts every point. A frequency of 2 posts every other point, 3 posts every third, etc. This is often used to reduce the number of displayed data points to avoid symbols overwriting each other. To change the number of points, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click the button to increase or decrease the *Frequency*.

Exclusion Filter

The *Exclusion Filter* allows a Boolean expression to specify how to exclude data. The *Exclusion Filter* can be used with any column in the worksheet that contains numbers. Columns in the worksheet that contain text or columns that are empty will not be excluded by the *Exclusion Filter*.

To use one of the X, Y, or Z columns, use X, Y, or Z in the *Exclusion Filter* . To use another column from the worksheet, use _A, _B, _C, etc. The underscore is required when specifying a worksheet column.

For example:

X=-999 or Y=-999 or Z=-999	Excludes any data with a -999 value in either the X, Y, or Z columns.

X<10 or X>20 or Y<10 or Y>20	Excludes all data except for points in the range 10 to 20 for both the X and Y directions.
Z < 0.0	Excludes any triplet with Z value less than 0.0.
_A > 10	Excludes any row in the worksheet that contains a value greater than 10 in column A.
Z < 0 AND _D = -999	Excludes any triplet with Z value less than 0.0 and whose row in the worksheet contains a value in column D equal to -999.

Boolean expressions, used by Grids | New Grid | Function, Grids | Calculate | Math, Grid | Data, and Grid | Variogram, include:

- logical operators (AND, OR, XOR, NOT)
- comparison operators (=, <>, <, >, <=, >=)
- the IF function - for example IF(condition,

The words AND, OR, XOR, NOT, and IF are reserved keywords and may not be used as variable names.

To use a stored function, click the next to the current function. This will display the ten most recent functions used. The functions are stored in the registry, so the equations are stored between **Surfer** sessions. You can also start typing the function in the function box. If the function is in the ten function history, the entire function will auto-complete.

For example, consider the case of ignoring data outside of a grid. The original grid *X Maximum* is 50, but the grid *X Maximum* is reset to 40. To limit the search to data with X values less than 40, use the *Exclusion Filter* by entering X > 40 into the *Exclusion Filter* text box. This tells **Surfer** to exclude all data with X values greater than 40.

Consider a second case where data contains a numerical identifier in column D. When the value in this column is equal to -999, the data point is considered inaccurate and should not be used when gridding. To grid only those data where column D is not equal to -999, exclude column D with the *Exclusion Filter* by entering _D = -999 into the *Exclusion Filter* text box. This excludes all rows of data where column D contains the value -999.

General

The *General* section controls the default angle and the legend display. Click the ⊞ next to *General* to open the *General* section and set the default angle and legend display for the classed post map layer.

Symbol Angle

The *Default angle (degrees)* specifies the angle to apply to all the symbols on the map. Positive angles of up to +360 degrees rotate the symbol in a counterclockwise fashion. To change the symbol rotation, highlight the existing angle value and type a new value or click and drag the slider until the symbols are rotated to the desired amount.

Connection Line

The *Connection Line* section controls the line that connects points in the post map. Click the ⊞ next to *Connection Line* to open the *Connection Line* section and line properties for the connecting line for the classed post map layer.

Connect Points

Check the box next to *Connect points* to connect all points in the post map layer with a line. Uncheck the box to remove the line from the post map layer. When the *Connect points* box is checked, points are connected in the order the points appear in the worksheet.

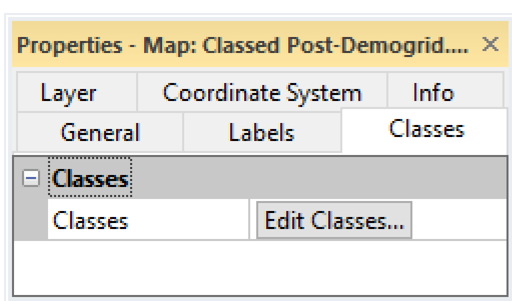
The line is drawn below the points. With open symbols, this can cause the line to appear inside the symbol. If the line should not be drawn through the symbol, use a closed symbol with the *Fill color* set to white instead of the open symbol. Symbols are set on the Classes tab.

Line Properties

Click the ⊞ next to Line Properties to set the connecting line properties. The *Style, Color, Opacity* , and *Width* can be set.

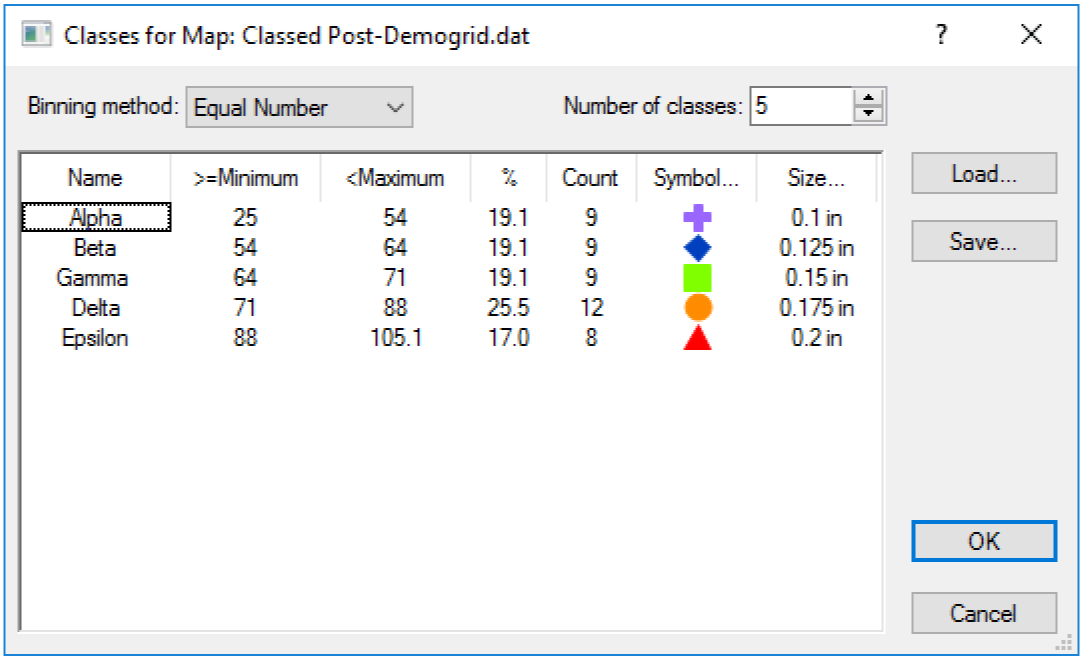
Classed Post Layer Classes Properties

The classed post map **Classes** page allows you to open the dialog to set classed post map classes. To edit classed post map classes, click once on the classed post map to select it. In the **Properties** window, click on the **Classes** tab. Click the *Edit Classes* button.



*The **Classes** page provides access to the **Classes for Map** dialog.*

In the **Classes for Map** dialog, define the classes used to group the data values. Each class is represented by a unique symbol in the classed post map.



*The classed post **Classes for Map** dialog allows you to define classes that group data values. Hover over any portion of the dialog for additional information.*

Binning Method

The *Binning method* specifies the method used to calculate the limits of the classes. There are three options available: *Equal Number, Equal Intervals*, and *User Defined*. To change the *Binning method*, click on the existing option and select the desired option from the list.

- *Equal Number* assigns the class ranges such that approximately equal numbers of points are included in each class. Normally in this case, the interval of each class is different.
- *Equal Intervals* assigns the class ranges so the interval between the >= *Minimum* value and the *<Maximum* value is equal for each class. Normally in this case, a different number of points is assigned to each class.
- *User Defined* allows you to set the *>=Minimum* value and the *<Maximum* value for each class individually. This allows you to specify your own ranges for the classes. To change the *>=Minimum* or *<Maximum* value, double-click the values in the class list.

Number of Classes

The *Number of classes* value is used to indicate how many classes or groupings there will be on the map. When you change the *Number of classes* value, the class list box is automatically updated to reflect the change. **Surfer** allows the creation of up to 200 classes. To change the number of classes, click the button to increase or decrease the number of classes or highlight the existing number and type in a new value from 1 to 200 classes.

Insert Class

When the *Binning method* is set to *User Defined*, the *Insert Class* option is available. To add a new class, click anywhere in the class list on the row where the new bin should be created. Click the *Insert Class* button or right-click on the selected row and select *Insert Class*. The new class is added. The new class automatically has the *>=Minimum* and *<Maximum* values set to the selected row's minimum value. When a new bin is created, it automatically uses the symbol properties and size for the previously selected bin.

For example: Suppose you have three bins 0-1, 1-2, and 2-3 and you want to create a new bin with values between 0.5 and 1, you would click anywhere on the second bin's (1-2) row. The new class will be set with 1 for the *>=Minimum* and *<Maximum* values. Double-click on the *>=Minimum* value. In the **Class Limit** dialog, type 0.5 and click *OK*. The new bin will be updated to show 0.5 to 1. The new class will use the same symbol as the 1-2 class.

Delete Class

When the *Binning method* is set to *User Defined*, the *Delete Class* option is available. Click on the class that should be deleted and click the *Delete Class* button or right-click on the selected row and select *Delete Class* . The class is removed from the bin list.

Changing the Class Name

Specify a class name in the **Classes for Map** dialog to display class names in a legend. By default, classes are named *Class 1*, *Class 2*, etc. Double-click a class name in the name column to change the name. Type the desired class name in the pop-up dialog.

Changing the Bin Limits for Each Class

The class list box displays summary statistics and allows you to specify the properties for each class. Available columns are: *>=Minimum, <Maximum, %, Count, Symbol,* and *Size*. The *>=Minimum* and *<Maximum* control the limits of the class.

The >= *Minimum* list specifies the lower limit for each class of data. The minimum value is included in the bin. The <*Maximum* list specifies the upper limit for each class of data. The maximum value is not included in the bin. Double-click on any number in the >=*Minimum* or <*Maximum* column, or right-click on a row and select *Minimum* or *Maximum* to open the **Class Limit** dialog. Change the value for the selected class and click *OK* in the dialog to make the change. Bin limits are adjacent. The <*Maximum* value for the previous bin is automatically set to the >=*Minimum* value of the following bin. When one value changes, the other automatically updates.

Note, when using date/time formats for the bin minimum and maximum, the >=*Minimum* and the <*Maximum* must always be in numbers, not in date/time format.

Statistical Information about the Points in Each Class

The class list box displays summary statistics and allows you to specify the properties for each class. Available columns are: *>=Minimum, <Maximum, %, Count, Symbol,* and *Size*. The *%* and *Count* columns display statistical information about the class. The *%* column indicates the percentage of data points in the particular class. The *Count* column indicates the number of points included in each class. These values cannot be directly edited and are for informational purposes only.

Changing an Individual Class Symbol

The *Symbol* column displays the symbol used for each class. To change a symbol or symbol property used for a particular class, double-click the class symbol or right-click and select *Symbol Properties* . Make changes in the Symbol Properties dialog. To change the symbols for all classes, click the *Symbol* button at the top of the column.

Changing an Individual Class Size

The *Size* column specifies the size of the symbol. To change the size of a symbol for a particular class, double-click the *Size* value in the class row or right-click and select *Symbol Properties*. Change the value in the **Symbol Properties** dialog. To change the symbol size for all classes, click the *Size* button at the top of the column.

Changing All Class Symbols at the Same Time

Click the *Symbol* button to set the properties for all symbols at the same time. The Class Symbol Properties dialog opens. Change the properties in the dialog to set all symbols to the same (uniform) properties or to assign different properties according to a gradational colormap. The symbol set, symbol, fill color, and line color can be changed for all classes at the same time.

Changing All Class Symbol Sizes at the Same Time

Click the *Size* button to set the symbol size for all symbols at the same time. The Class Symbol Size Properties dialog opens. Set the *Minimum Size* and *Maximum size* . When the sizes are the same, all classes use the same sized symbol. When the sizes are different, each class is sized so that the first class has the *Minimum Size* and the last class has the *Maximum Size* . Consecutive classes have the same difference in sizes, so the symbols increase proportionally.

Loading a Classed Post Map Class .CLS File

Click the *Load* button to load an existing classed post map .CLS class file.

Saving a Classed Post Map Class .CLS File

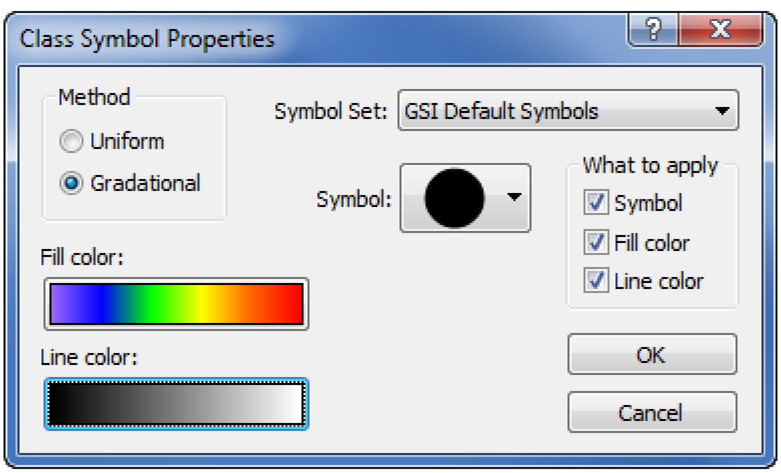
Click the *Save* button to save a classed post map .CLS class file. The range minimum, range maximum, symbol set, symbol index number, color, and symbol size are saved in the .CLS file. The color is in RGBA format.

OK or Cancel

Click *OK* to make the changes to the classed post map. Click *Cancel* to close the dialog without making changes to the classed post map.

Class Symbol Properties Dialog

The **Class Symbol Properties** dialog is opened by clicking the *Symbol* button in the Classes for Map dialog.



Set the symbol set, symbol, symbol line, and symbol fill color for all classes in the ***Class Symbol Properties*** *dialog.*

Method

The *Method* is the method used to set the symbol line and fill colors. Available options are *Uniform* and *Gradational*. Setting the *Method* to *Uniform* makes all classes use the same line color or fill color for the symbols. Setting the *Method* to *Gradational* allows each class to have a different line or fill color for the symbols based on a colormap.

Symbol Set

Select the *Symbol Set* from the list. The *Symbol Set* can be any TrueType font installed on your system. To change the *Symbol Set*, click on the existing font name. Select the new font from the list. The same *Symbol Set* is used for all classes.

Symbol

Choose the *Symbol* by clicking the existing symbol and selecting a new symbol from the symbol palette. The same *Symbol* is used for all classes.

Fill Color and Opacity

When the *Method* is set to *Uniform*, the *Fill color and opacity* section is available. The *Fill color* is the inside color of a solid symbol. When changed, all classes will use the same fill color for the symbols. Change the *Fill color* of the symbol by selecting a new color from the color palette. Create new colors by clicking the *Custom* button at the bottom of the color palette. Change the *Fill opacity* of the symbol by highlighting the existing value and typing a new value, by clicking the to increase or decrease the opacity, or by clicking and dragging the to change the opacity percentage. Percentages range from 0% (completely transparent) to 100% (completely opaque).

When the *Method* is set to *Gradational,* the *Fill color* section is available. The *Fill color* is the inside color of a solid symbol. When changed, symbol fill color for each class will be based on a colormap. Click the color scale bar to change the colormap.

Line Color and Opacity

When the *Method* is set to *Uniform*, the *Fill color and opacity* section is available. The *Line color* is the outside edge color of the symbol. When changed, all classes will use the same line color for the symbols. Change the *Line color* of the symbol by selecting a new color from the color palette. Create new colors by clicking the *Custom* button at the bottom of the color palette. Change the *Line opacity* of the symbol by highlighting the existing value and typing a new value, by clicking the to increase or decrease the opacity, or by clicking and dragging the to change the opacity percentage. Percentages range from 0% (completely transparent) to 100% (completely opaque).

When the *Method* is set to *Gradational,* the *Line color* section is available. The *Line color* is the outside edge color of the symbol. When changed, symbol line color for each class will be based on a colormap. Click the color scale bar to change the colormap.

What to Apply

After setting the *Symbol Set, Symbol, Method, Fill color*, and *Line color*, check the boxes in the *What to apply* section to apply the selected changes. Available options are *Symbol, Fill color*, and *Line color*. When the box is checked, the properties from the **Class Symbol Properties** dialog for the checked box are applied to all classes in the classed post map. When the box is not checked, the properties from the **Class Symbol Properties** dialog are not applied.

Setting the Same Symbol for All Classes

To set the same symbol for all classes, click the *Symbol* button in the Classes for Map dialog. In the **Class Symbol Properties** dialog, click on the *Symbol Set* and select the desired font. Click on the *Symbol* and select the desired symbol. Check the box next to *Symbol* in the *What to apply* section and click *OK*. All classes are changed to use the selected symbol.

Setting the Same Symbol Line or Fill Properties for All Classes

To set the same symbol fill properties for all classes, click the *Symbol* button in the Classes for Map dialog. In the **Class Symbol Properties** dialog, set the *Method* to *Uniform*. Click on the *Fill color and opacity* color box and select the desired color from the list. Change the *Fill opacity*, as desired. Check the box next to *Fill color* in the *What to apply* section and click *OK*. All classes are changed to use the selected symbol fill color.

To set the same symbol line properties for all classes, click the *Symbol* button in the Classes for Map dialog. In the **Class Symbol Properties** dialog, set the *Method* to *Uniform*. Click on the *Line color and opacity* color box and select the desired color from the list. Change the *Line opacity*, as desired. Check the box next to *Line color* in the *What to apply* section and click *OK*. All classes are changed to use the selected symbol line color.

Setting Gradational Symbol Line or Fill Properties for All Classes

To set gradational symbol fill properties for all classes, click the *Symbol* button in the Classes for Map dialog. In the **Class Symbol Properties** dialog, set the *Method* to *Gradational*. Click on the *Fill color* button. In the **Colormap Editor**, set the colormap, scaling, and opacity. Click *OK*. Check the box next to *Fill color* in the *What to apply* section and click *OK*. All classes are changed to use the colormap to determine the symbol fill color.

To set gradational symbol line properties for all classes, click the *Symbol* button in the Classes for Map dialog. In the **Class Symbol Properties** dialog, set the *Method* to *Gradational*. Click on the *Line color* button. In the **Colormap Editor**, set the colormap, scaling, and opacity. Click *OK*. Check the box next to *Line color* in the *What to apply* section and click *OK*. All classes are changed to use the colormap to determine the symbol line color.

OK or Cancel

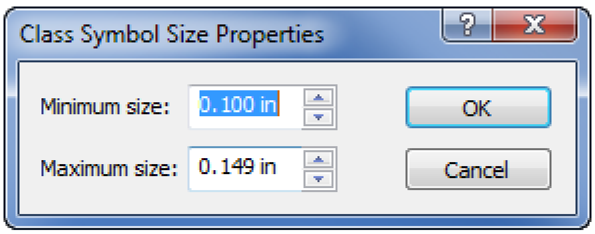
Click *OK* to make the changes to the class symbol properties. Click *Cancel* to close the dialog without making changes.

Dialog Persistence

Due to the nature of the settings allowed in the **Class Symbol Properties** dialog, only the last set of dialog settings are recalled after closing and reopening the dialog. The settings in the **Class Symbol Properties** dialog are intended to be additive, adding new line, fill, or symbol properties to the existing class properties. Therefore, not all previous settings are displayed when opening the dialog.

Class Symbol Size Properties Dialog

The **Class Symbol Size Properties** dialog is opened by clicking the *Size* button in the Classes for Map dialog.



*Set the minimum and maximum class symbol size in the **Class Symbol Size Properties** dialog.*

Minimum Size

The *Minimum size* is the smallest size to use for all classes. The *Minimum size* is used for the first class. To change the value, highlight the existing value and type the new value. Alternatively, click the to increase or decrease the value. Sizes range from zero to 4 inches (0 to 10.16 centimeters).

Maximum Size

The *Maximum size* is the smallest size to use for all classes. The *Maximum size* is used for the last class. To change the value, highlight the existing value and type the new value. Alternatively, click the to increase or decrease the value. Sizes range from zero to 4 inches (0 to 10.16 centimeters).

Same Symbol Size for All Classes

Setting the *Minimum size* and the *Maximum size* to the same value results in all classes having symbols that are the same size.

Incrementing Symbol Size for All Classes

Setting the *Minimum size* and *Maximum size* to different values results in all classes having symbols that increment symbol sizes by a fixed amount. To determine the incrementing amount, subtract the *Minimum size* from the *Maximum size* and divide by the *Number of classes*.

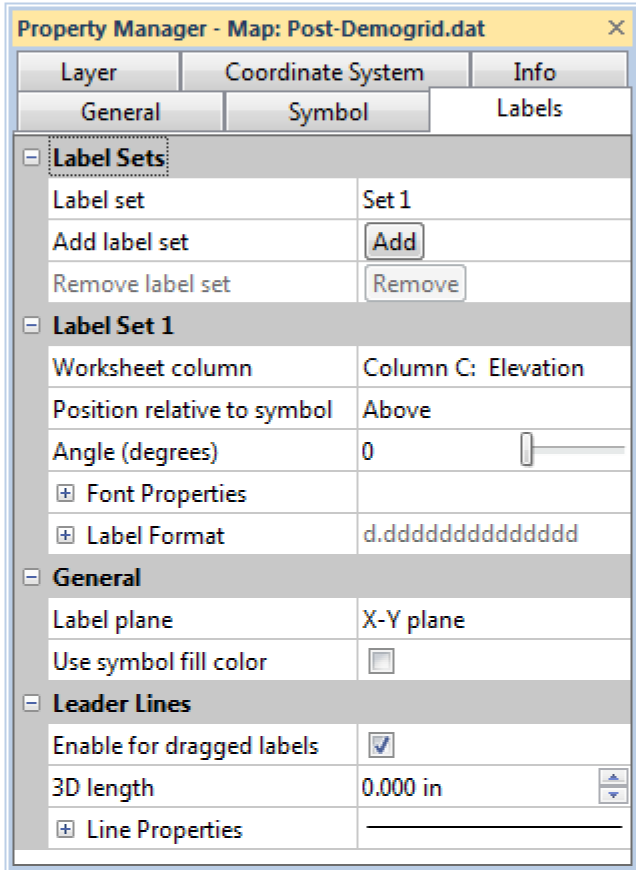
OK or Cancel

Click *OK* to make the changes to the symbol sizes for all classes. Click *Cancel* to close the dialog without making changes.

Labels Properties

When posting data points on a post map or classed post map, you can associate text from the worksheet with each posted symbol. The position, font, and numeric format can also be specified on the **Labels** page in the post properties and classed post properties.

To add labels to a post map or classed post map, click once on the post map to select it. In the Properties window, click on the **Labels** tab.



*Change post map or classed post map label properties on the **Labels** page in the **Properties** window.*

Label Sets

The *Label Sets* section controls which label is being controlled by the *Label Set #* section. Multiple labels can be displayed by changing the *Label Sets* settings. Click the ⊞ next to *Label Sets* to open the *Label Sets* section.

Label Set

The *Label set* displays the name of the label that is currently being changed. Click on the set name (*Set 1*, for instance) to select a different set name from the list. If only *Set 1* is displayed in the list, only a single label set is currently created. All of the *Label Set #* properties apply to only the selected *Label set*.

Add Label Set

The *Add label set* option allows new labels to be added. Click the *Add* button to create a new *Label set*. The *Label set* option automatically changes to the new set. If this is the second label to be added, a *Set 2* is added to the *Label set* list. If this is the third label to be added, a *Set 3* is added to the *Label set* list, and so on.

Remove Label Set

The *Remove label set* option deletes a label from the *Label set* list. To delete a label, first set the *Label set* to the label that should be deleted. Then, click the *Remove* button next to *Remove label set* to delete the label. All properties of the label set are removed.

Label Set

The *Label Set #* section controls the worksheet column, position, angle, font, and label properties for the selected *Label set* . Before making any changes in this section, change the *Label set* option to the desired label. The *Label Set #* changes to display the name of the *Label set* . For instance, if *Label set* is set to *Set 3* , the *Label Set #* section name changes to *Label Set 3*.

Worksheet Column

Select the worksheet column containing the values or text from which you wish the *Label set* to display near the posted points in the *Worksheet column* list. Click on the existing worksheet column or *None* to change to a new worksheet column. Labels may be the original data values for the data points, or may be other identifying text such as well names or sample numbers. Labels can use math text instructions to define custom character formatting. Different worksheet columns can be selected for each *Label set*.

Position Relative to Symbol

The *Position relative to symbol* controls the offset of the selected label from the symbol center. To change the position, first change the *Label set* to the desired label. Then, click on the existing position in the *Position relative to symbol* option and select the new position from the list. Available options are *Center*, *Left*, *Right*, *Above*, *Below*, and *User defined*. *User defined* allows you to specify the exact offset (in page units) in the *X offset* and *Y offset* boxes. The posted labels are all placed in the same position relative to the associated symbol.

Positive offset values move the labels in the positive axis direction. For reversed axes, this means that the label is moved in the opposite direction of the *Position relative to symbol* specified. For instance, if the bottom axis is reversed, setting the *Position relative to symbol* to *Left* will move the label to the right of the symbol (in the negative direction). Setting the *Position relative to symbol* to *User defined* and setting the *X offset* to 0.25 inches will move the label 0.25 inches in the negative axis direction.

X Offset and Y Offset

When the *Position relative to symbol* option is set to *User defined*, the *X offset* and *Y offset* options are available. The *X offset* controls the location of the label in the left-right direction. Positive values in the *X offset* shift the label position toward the positive direction (to the right of the symbol for normal axes, to the left for reversed axes). Negative values in the *X offset* shift the label position toward the negative direction (to the left of the symbol for normal axes, to the right for reversed axes). A value of zero places the label directly in the center of the symbol.

The *Y offset* controls the placement of the label in the up-down direction. Positive values in the *Y offset* shift the label position toward the positive direction (upward for normal axes, downward for reversed axes). Negative values in the *Y offset* shift the label position toward the negative direction (downward for normal axes, upward for reversed axes). A value of zero places the label directly in the center of the symbol.

The *X offset* and *Y offset* values are numbers between -4 and +4 inches (-10.16 and +10.16 centimeters). To change the value, highlight the existing value and type a new value. Alternatively, click the to increase or decrease the value.

Angle (degrees)

The *Angle (degrees)* box specifies the angle in degrees to rotate the selected label set. Positive angles rotate the labels counterclockwise. To change the *Angle (degrees),* highlight the existing angle value. Type a new value or click and drag the to the desired angle position.

Font

Click the ⊞ next to *Font Properties* to display the Font Properties section. The font properties are applied to all labels in this label set. Different label sets can contain different font properties.

Format

Click the ⊞ next to *Label Format* to display the Label Format section. The numeric format is applied to all numbers read from the specified label column in this label set. Different label sets can contain different label formatting.

When the *Type* is set to *Date/time*, if *Invalid date* text is displayed instead of the actual date/time format, the value for the label is outside the defined date/time range.

General

The *General* section controls the label plane for all label sets. In addition, all label sets can be set to use the symbol fill color for the label font color.

Label Plane

The *Label plane* list specifies the coordinate plane that contains the label. To change the *Label plane*, click on the existing plane and choose the new plane from the list. Available options are *X-Y plane* and *Screen*. If *X-Y plane* is selected, labels are oriented parallel to the X-Y plane of the map. As the map is tilted in 3D, the labels tilt as well. This can make it hard to read the labels at shallow tilt angles since the labels are viewed on edge. In this case, it may be better to orient the labels in the *Screen* plane. *Screen* oriented labels are always displayed perpendicular to the viewer's line of sight, no matter how the map is tilted or rotated. All labels from all label sets use the same *Label plane*.

Use Symbol Fill Color

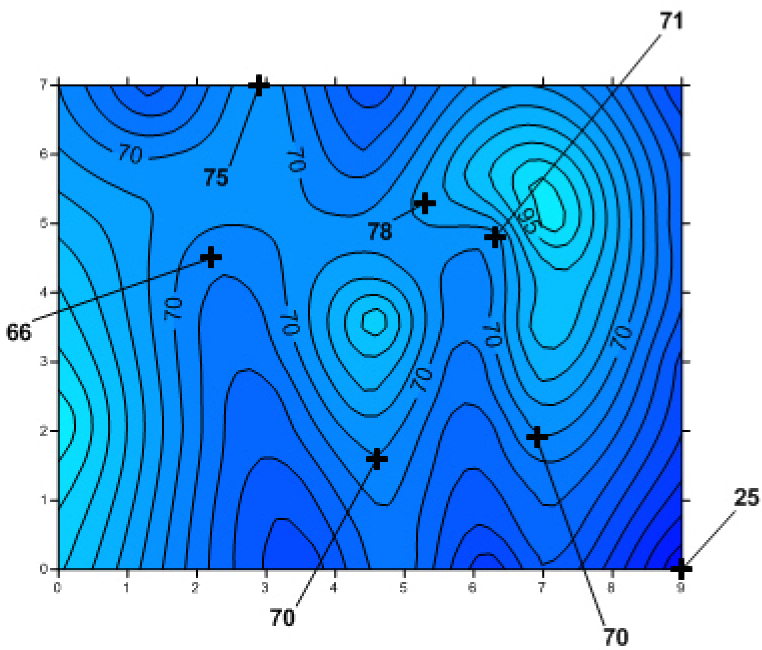
The *Use symbol fill color* option controls whether the color of the symbol is used for all label sets. When the *Use symbol fill color* option is checked, all label sets for a point use the symbol color for the text color instead of the *Font Properties* section *Foreground color* option. When the symbol color changes, the label color automatically changes, as well. When the *Use symbol fill color* option is unchecked, the *Font Properties* section *Foreground color* option controls the label color. Each label set can then be a separate color.

Leader Lines

The *Leader Lines* section is used when the post map or classed post map has labels that are offset in the Z direction or when a label is moved to a new location. The leader line is drawn from the symbol center to the new label location.

Enable for Dragged Labels

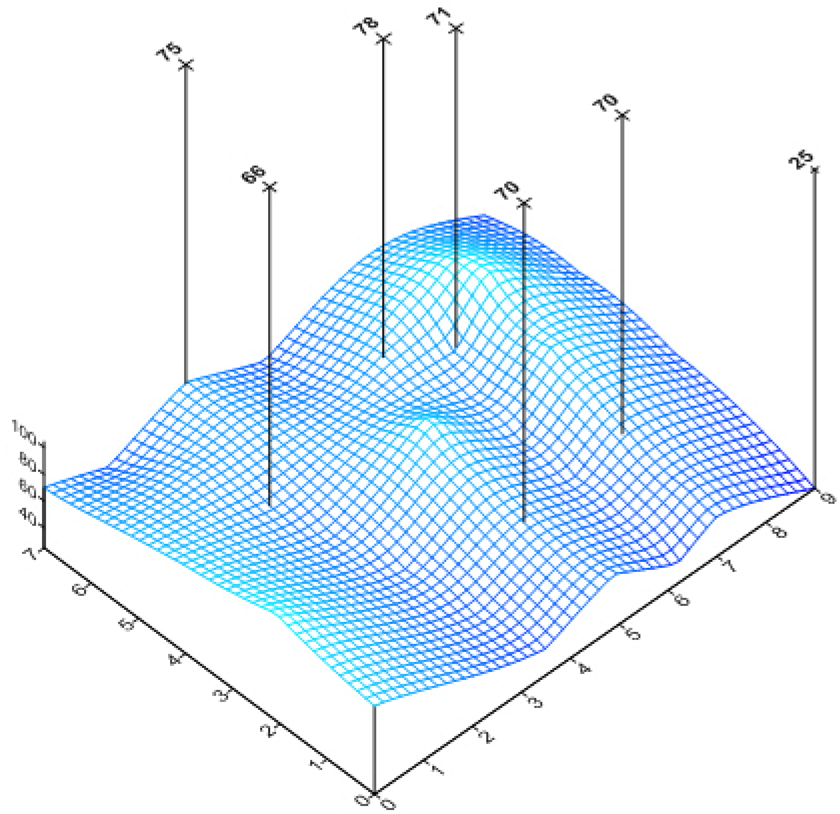
Check the *Enable for dragged labels* option to display leader lines when labels are manually dragged to a new location with the Edit Post Labels command. The leader line will display a line from the original label location to the dragged location.



This map has a contour map layer and a post map layer. The post points are labeled. The labels have been moved with the ***Edit Post Labels*** *command. Leader lines go from the data point to the posted label.*

3D Length

The *3D length* defines the distance above the map that the points are posted when the post map is tilted at a value other than 90 degrees. The *3D length* box specifies how long the label lines are and how far above the wireframe or tilted post map the labels are drawn. *3D Leader Lines* are not drawn on post maps overlaid on a 3D surface map. The *3D length* is a value between 0 and 10 inches (0 and 25.4 centimeters). To change the length, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click the buttons to increase or decrease the line length.



This map has a 3D wireframe layer and a post map layer. The post points are labeled and have 3D leader lines from the data point to the post label.

Leader Line Properties

Click the ⊞ next to *Line Properties* displays the Line Properties options, allowing you to specify the line *Style, Color, Opacity,* and *Width* for leader lines.

Creating a Post Map with Multiple Labels

Post maps and classed post maps can have multiple labels from different columns displayed on the map. To create a single map with multiple labels:

1. Click the **Home | New Map | Post** command to create a new post map.
2. Select the data file in the **Open Data** dialog and click *Open*.
3. Click on the *Post* layer to select it in the **Contents** window.
4. In the **Properties** window, click on the **Labels** tab.
5. With the *Label set* as *Set 1*,
   a. In the *Label Set 1* section, set the *Worksheet column* for the first label.
   b. Change the *Position relative to symbol* to the desired position.
   c. Set any *Font Properties* or *Label Format* options.
6. Click the *Add* button next to the *Add label set* option.
7. With the *Label set* as *Set 2*,

a. In the *Label Set 2* section, set the *Worksheet column* for the second label.
b. Change the *Position relative to symbol* to the desired position.
c. Set any *Font Properties* or *Label Format* options.

Edit Post Labels

Post map and classed post map labels can be moved interactively so that labels do not overlap or so that labels appear in a more desirable location. To move labels, a post map or classed post map must have the *Worksheet column* on the Labels page set to a column other than *None* to use the **Map Tools | Edit Layer | Layer Labels** command.

Enter Edit Post Labels Mode

To enter the post map label edit mode, click on a post map or classed post map in the plot window or Contents window. Once the post map is selected, click the **Map Tools | Edit Layer | Layer Labels** command or the button. Alternatively, right-click on the selected map and click **Edit Labels** or the button.

The cursor will change to a to indicate you are now in post label editing mode. The post map labels can now be individually moved.

Move Individual Post Labels

To move a label, enter the **Edit Post Labels** mode. Click once on the label you wish to move. Once the label is selected, hold down the left mouse button and drag the label to a new location. When the label is in the desired location, release the mouse button.

Labels can also be moved using the keyboard. To click on a label using the keyboard, hold down the ARROW keys until the cursor is above the label. Press and hold the SPACEBAR to select the label. Press the ARROW keys to move the label to a new location. When the label is in the desired location, release the SPACEBAR.

Exit Edit Post Labels Mode

To end post map label edit mode, press the ESC key or click the **Map Tools | Edit Layer | Layer Labels** command again.

Move Around the Plot Window in Edit Mode

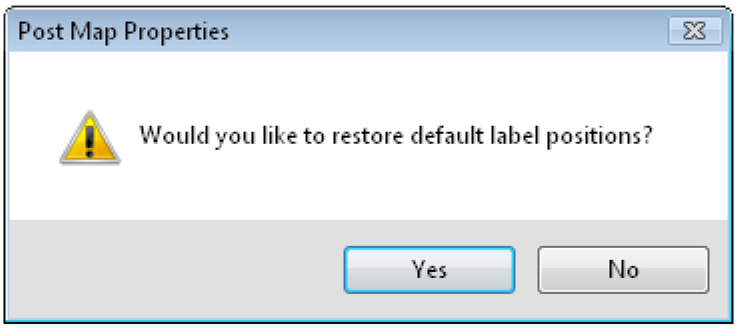
If you are zoomed in on the post map, use the scroll bars to move to locations that appear off the screen. Alternatively, click and hold the scroll button of a mouse wheel to pan the plot window.

Custom Label Location and Changed Coordinate System

When either the source coordinate system or the target coordinate system changes, all post map label customizations are returned to the default location.

Reset Labels to Default

All labels in the post map can be reset to the default position. To reset the labels, click on the post map to select it. In the Properties window, click on the Labels tab. Change the *Position relative to symbol* to any position other than the currently selected position to reset the labels to the default locations. The **Post Map Properties** dialog message appears. Click *Yes* to restore all of the labels to the default positions.



*The **Post Map Properties** warning message is displayed when you change the label* Position relative to symbol *after moving post map labels.*

Data Files Used for Posting

Data files used to produce post maps contain the *X and Y coordinates* used to locate points on the map. They can also provide additional information used to place labels on the map, define the symbols to use, the symbol size, the symbol color, and the angle at which each symbol is drawn. The first row of the data file may optionally contain the title for each column. These titles are used by the dialogs to associate a descriptive name with each column letter. Following the titles are the actual data, organized as one point (record) per row. Columns within the data file contain various properties for the point such as its X coordinate, Y coordinate, Z value, etc. The data must contain, at a minimum, the X and Y coordinates. Incomplete or blank records in the X and Y coordinate columns are not plotted on the map.

Data files used for post maps and classed post maps can contain slightly different information. Both types of files must contain X, Y coordinates used to locate the points on the map. In addition, data files for classed post maps should contain a column with data values used to determine the class for each point. If a third data column is not available for the *Z coordinates* for a classed post map, the first data column is used.

XY Coordinates in the Data File

The X and Y coordinates define the symbol locations and the extents of the post map. These coordinates can be in any columns in the data file, but **Surfer** assumes X is in column A and Y is in column B by default.

Z Values in the Data File

Z values in the data values, such as elevation or concentration, are associated with each X,Y location. This is typically the same column that was used when creating a grid file. Z values can be used to proportionally scale the posted symbols. You can define the minimum and maximum symbol size, and all points are scaled in proportion to these sizes. See Specifying Scaled Symbols for more information. Z values can also be used to assign colors to symbols. A colormap is defined based on the Z values and the symbols change color according to the colormap.

For classed post maps, the Z value is used to determine which class or bin will contain the data. Classes are based on specified data ranges, and each class is assigned a unique symbol. Classes can be saved and loaded for future use.

A common application for post maps and classed post maps is as overlays on contour maps. While the Z values used to scale the posted symbols in a post map, or to define the classes in a class post map, may be the same Z values used to generate the contour map, this is not required. For example, the posted symbols could be used to indicate rock type, while the contours show ore grade.

Data Labels in the Data File

Data labels are text strings or numbers associated with each point on a post map. Labels may be the original data values for the data points, or may be other identifying text such as well names or sample numbers. Labels can use math text instructions to define custom character formatting.

Updating Post Map and Classed Post Map Data Files

A copy of the data file is created and embedded within the post map at the time the post map is created. This means any subsequent changes made to the data file are not reflected in the post map. To incorporate changes made to a data file into its associated post map, the data file must be reassigned to the map.

To automatically updated a post or classed post map:

1. Select the map to be updated.
2. Use the File | Reload Map Data command. All map layers on the selected map are reloaded. This can change your map properties (i.e. base map containing an image).

To manually update a post or classed post map:

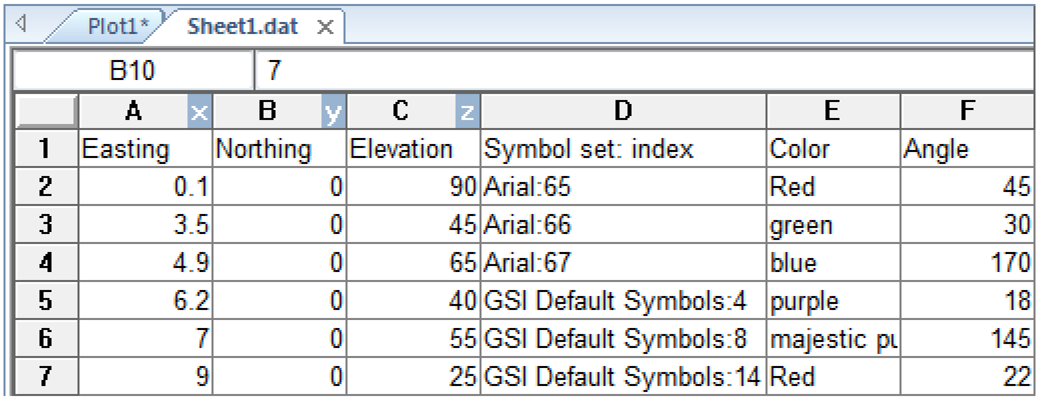
1. Select the map to be updated.
2. Click once on the post map or classed post map in the Contents window to select it.
3. In the Properties window, click on the **General** tab.
4. Click on the  button next to *Data file*.
5. Select the updated data file and click *Open*.

The map automatically redraws with the updated data.

Symbol Specifications in the Data File

Post map data files can contain a column defining which symbol set and symbol index to use for each posted point. This information can be specified in one of three ways:

Form	**Description**
SymbolSet:Index	This form allows both the symbol set and the symbol index to be specified. *SymbolSet* specifies the face name of the desired symbol set. The colon character must appear between the symbol set and the index. If the specified face name is invalid, the default symbol set specified in the post map properties is used instead.
Index	If a single integer is specified, it is interpreted as a symbol index for the current symbol set. The current symbol set is the last specified symbol set or the default symbol set specified in the post map properties on the General tab.
<Empty>	If the cell is empty, the last specified symbol set and default symbol index is used.



	A	B	C	D	E	F
1	Easting	Northing	Elevation	Symbol set: index	Color	Angle
2	0.1	0	90	Arial:65	Red	45
3	3.5	0	45	Arial:66	green	30
4	4.9	0	65	Arial:67	blue	170
5	6.2	0	40	GSI Default Symbols:4	purple	18
6	7	0	55	GSI Default Symbols:8	majestic pu	145
7	9	0	25	GSI Default Symbols:14	Red	22

This is a sample data file containing the symbol set and index, symbol color, and symbol angle for symbols on a post map.

Symbol Index

The symbol index is the symbol or font number as it appears in the Symbol Properties. This is the 0-based offset of the symbol within the symbol set. To use the symbol index in a data column, use the value displayed in the **Symbol Properties**. This value is the ASCII code minus 32. For example, the ASCII code for the Arial font lowercase "a" is 97. The index value displayed in the symbol properties and used in the symbol column is 65.

When saving your plot to a **Surfer13** or older .SRF file, i.e. *Surfer 13 Document (*.srf)*, *Surfer 12 Document (*.srf)*, or *Surfer 11 Document (*.srf)*, the symbol index is automatically updated in the internal data to display the correct symbol in the previous **Surfer** version, i.e. 32 is added to the index value from the current **Surfer** version. Before version 14, **Surfer** required the use of the ASCII code (index + 32) in the data file. After opening the .SRF file in a previous version of **Surfer**, this automatically updated data file can be saved by clicking the button in the *Data file* field in the post layer General properties.

When opening a **Surfer** .SRF file from a previous **Surfer** version, the symbol index is automatically updated in the internal data to display the correct symbol in the post layer, i.e. 32 is subtracted from the worksheet value from the previous version. Previous **Surfer** versions required the use of the ASCII code (index + 32) in the data file. This automatically updated data file can be saved in the current **Surfer** version by clicking the button in the *Data file* field in the post layer General properties.

You can use the Window's character map utility to determine the ASCII code for font symbols and subtract 32 to determine the index value. Though it is generally faster to obtain the index value from the **Symbol Properties**. Note that the character map utility displays ASCII codes in hexadecimal.

If anything about the symbol specified in the data file is incorrect or missing, then the default symbol is used.

Symbol Angle Values from the Data File

The angle (in degrees) for the posted symbol can be specified in the data file for non-classed post maps. Positive angles rotate the symbols in a counterclockwise direction. On classed post maps, all symbols are drawn at the same orientation, which is specified within the classed post map properties dialog.

Symbol Color from the Data File

The posted symbol color can be specified in the data file for non-classed post maps. Color names, as shown above, RGB or RGBA color values (red, green, blue, alpha values) can be used or a value in the *Color column* can be mapped to a colormap.

Chapter 11 - 3D Surface Maps

3D Surface

The **Home | New Map | 3D Surface** command can be used to create a three-dimensional shaded rendering from a grid file. You can also click the **Home | New Map | 3D Surface | 3D Surface** command to create a 3D surface. The height of the surface corresponds to the Z value of the associated grid node. Denser grids show greater detail on the surface.

Color can be used to show Z values on surfaces. The colors are blended to form a smooth gradation. Once the color is selected, the lighting can be adjusted, changing the appearance of the map.

This 3D surface map is displayed with custom colors and no axes. Customize your map to display exactly the map you want.

Creating a 3D Surface

To create a 3D surface:

1. Click on the **Home | New Map | 3D Surface** command or the button.
2. Select a grid file in the **Open Grid** dialog and click *Open*.

The map is automatically created with reasonable defaults.

Editing an Existing 3D Surface

To change the features of the 3D surface map, click once on the surface map in the plot window or in the Contents window to select it. The properties are displayed in the Properties window.

Adding a Map Layer

When surface maps are created, they are independent of other maps in the plot window. For example, creating a surface map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the contour map on the surface map, select both maps by clicking the Home | Selection | Select All command. Overlay the maps using the Overlay Maps command.

Alternatively, you can add the contour map directly to the existing surface map by creating the contour map using the **Home | Add to Map | Layer | Contour** or **Map Tools | Add to Map | Layer | Contour** command. This automatically adds the contour layer to the existing surface map axes.

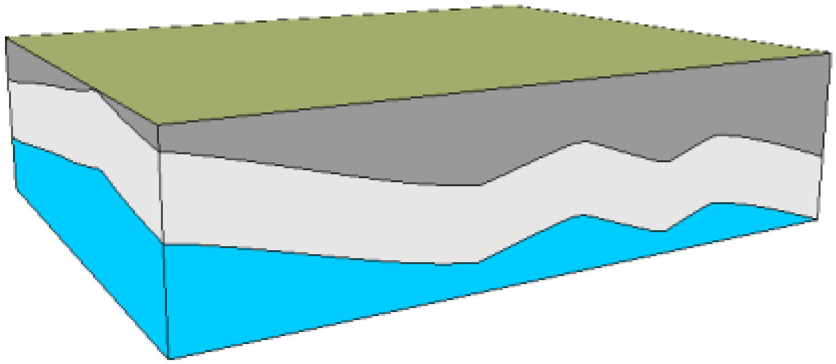
Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the surface map using the **Home | New Map | 3D Surface** command. This creates two separate maps. Click on the contour map, hold down the left mouse button, and drag the contour map into the surface map. A single map with two map layers, using one set of axes and scaling parameters is created.

Surface maps can be layered with images, vector files, and other 3D surfaces. You can overlay other map types, with the exception of 3D wireframes, on a surface. 3D surface maps can be created independently of other maps, or can be combined with other maps in map layers. Surfaces can be scaled, limited (clipped), or moved in the same way as other types of maps.

If there are map layers on a 3D surface, and the surface is made invisible through the **Contents** window, the layers will not be visible.

Overlaying Coincident 3D Surfaces

Multiple coincident 3D surfaces can be overlaid. The bases can be displayed by checking *Show base* on the **Properties** window General page to create a block diagram. Control which base color is displayed by arranging the 3D surface layers in the Contents window. The 3D surface layer that is to be drawn first should be placed at the bottom of the *Map* object in the **Contents** window. The 3D surface that should be drawn last should be placed highest in the *Map* object. Vary the base fill colors to represent different layers in the block diagram.



This is an example of multiple overlaid 3D surface layers with different color bases.

3D Surface Tips

- Image base maps, i.e. base (raster) layers, can be added to surface maps if both maps contain the same coordinate ranges. You may need to change the image coordinates so the maps use the same coordinates.
- Two or more 3D surface maps can be overlaid with one another. This is useful if you want to join two adjacent surface maps. Alternatively, you can use Grids | Resize | Mosaic to create one grid.
- Surface maps do not have a source coordinate system. All maps that are overlaid onto a surface map must have the same coordinates.

3D Surface Layer Properties

The surface properties contains the following pages:

General
Mesh
Lighting
Overlays
Info

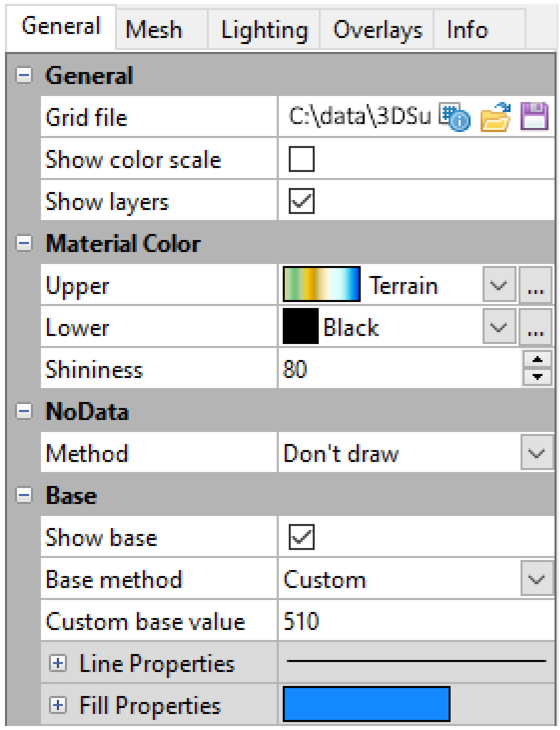
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Info

3D Surface Group General Properties

To edit a 3D surface map, click once on the surface map to select it. The properties for the surface map are displayed in the Properties window.



*Change 3D surface map properties in the **Properties** window on the **General** page.*

General

Input Grid File

The *Grid file* displays the path and file name for the grid file used for the map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Grid Information

The  button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the surface map. Select the new grid file and click *Open* to reference the new file. When the grid file is changed, the map limits are reset but the data limits are not recalculated.

If the Z range for the new grid is outside the old Z range, the full color spectrum may not be used. Click the ... button next to the *Upper* option in the *Material Color* section. In the Colormap Editor, check the *Use data limits* box to use the full color spectrum. Click *OK* and the map is updated.

Save File

Click the button to display the **Save Grid As** dialog. This allows the grid file used for the surface map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file.

Color Scale

You can check the *Show color scale* option to display a color scale bar adjacent to the map. The color scale bar indicates the colors assigned to the Z levels on the map and the associated Z level values are displayed as labels on the color scale bar. If the colormap is set to a logarithmic scale, the color scale bar will automatically display logarithmic scaling.

Layers

Check the *Show layers* box to show the overlay layers on the surface. For example, add a post map layer to a surface. Check the *Show layers* box to make the post map visible. Multiple layers can be combined into a single composite map, and this allows the overlays to be displayed on an individual basis. The appearance of the overlay layers is controlled on the Overlays page.

Material Color

The *Material Color* section shows the spectrum of colors used to color the upper and lower sides of the surface. The light angle and colors also affect the appearance of the surfaces.

Upper

The *Upper* color controls the main color spectrum of the surface map. To change the upper surface colors, click on the color next to *Upper*. Select any new colormap from the list. If the desired colormap is not displayed, click the ... button next to the selected colormap. The Colormap Editor opens, allowing you to specify additional color options. To set the colormap to use a logarithmic scale, click the ... button to the right of the selected colormap. Check the *Logarithmic* *scaling* option and click OK. The color map is automatically updated to show logarithmic scaling.

When overlaying map layers on a surface, especially image base layers, the layers will be brighter if you change the Upper material color to a completely white colormap:

1. Click the ... button next to *Upper*.
2. In the Colormap Editor, change the *Presets* to *Grayscale*.
3. Click the lower limit (left) anchor node.
4. Change the *Color* to white.
5. Click *OK* in the **Colormap Editor**.

Lower

The *Lower* button controls the single color of the bottom of the surface. Unless the bottom of the surface is at a sufficient angle to the light source, there will be insufficient reflection to see it. Also, as you increase the ambient light (colors closer to white) the lower surface color becomes more visible because ambient light is everywhere and directionless. To change the *Lower* surface color, click on the current color next to *Lower*. Click on the new color in the list. To use a custom color or set additional color options, click the ... button next to the selected color. The Colors dialog opens, allowing you to set additional options.

Shininess

Shininess controls the size of the specular reflections. As the shininess percentage increases, the reflections become more focused. Select a value between 0 and 100. To change the value, highlight the existing *Shininess* percentage value and type a new value or use the up and down arrow buttons to scroll to a new value.

NoData Nodes

The *NoData* section defines how nodes assigned the NoData value are handled in the surface map. Set the *Method* to *Don't draw* or *Remap to*.

- If *Don't draw* is selected, the NoData areas are not drawn on the map and appear as small transparent gaps in the surface.
- If *Remap to* is selected, type the new Z value into the *Remap value* option below. The *Remap value* is in Z coordinate units. Make sure the new value is within the range of Z values in the current map. The default *Remap value* is the grid Z minimum.

Base

Check the *Show base* box to fill the base with the look of a filled solid. The *Show base* option is useful when creating block diagrams. Control which base color is visible in the block diagram by arranging the 3D surface layers in the Contents window. The bottom-most layer in the map should be the bottom layer in the **Contents** window, and the top-most layer in the map should be the top layer in the **Contents** window.

Base Method

Select an option from the *Base method* property to fill the base down to a specific Z value.

- *Map minimum* fills the base down to the minimum Z value of all grids in the map.
- *Grid minimum* fills the base down to the minimum Z value of the grid used to create the selected layer.
- *Custom* fills the base down to the Z value that you specify in the *Custom base value* property. Click the **Info** tab to view minimum and maximum Z values.

Base Line Properties

Click the ⊞ button to view the *Line Properties* for the base line. The line properties for the 3D surface map base line are different than line properties for other objects.

- Click the current line *Style* and select *None* or *Solid* to hide or display the base line.
- To change the base line color , click the current selection next to *Color* and select a new color from the color palette. Alternatively, click the ... button and select or create a color in the Colors dialog.
- Change the base line width by typing a value in page units into the *Width* field.

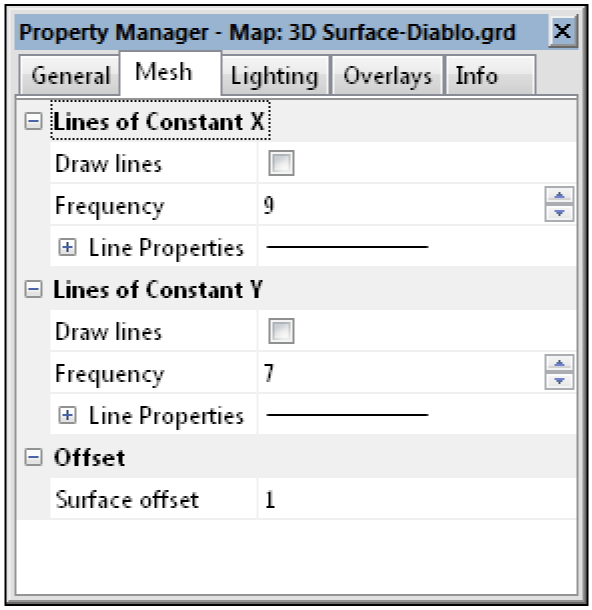
Base Fill Properties

Click the ⊞ button to view the *Fill Properties* for the base line. The fill properties for the 3D surface map base fill are different than fill properties for other objects.

- Click the current fill *Style* and select *None*, *Solid*, or *Lit* to change the fill style for the 3D surface base. *Solid* shows the base with a uniform fill, and the fill is not affected by the 3D surface lighting. *Lit* applies the 3D surface lighting to the base.
- To change the base fill color , click the current selection next to *Color* and select a new color from the color palette. Alternatively, click the ... button and select or create a color in the Colors dialog.

3D Surface Layer Mesh Properties

To edit a 3D surface map, click once on the surface map to select it. The properties for the surface map are displayed in the Properties window. The 3D surface **Mesh** page is used to add mesh lines to the surface of the map. Mesh lines can be used to simulate a wireframe.



*Change mesh properties in the **Properties** window on the **Mesh** page.*

Lines of Constant X

Check the *Draw lines* box under the *Lines of Constant X* to draw lines along the surface at constant X values.

Frequency

You can change the frequency of the X lines by entering a new number into the *Frequency* box. If this value is one, every grid node in the X direction will have a mesh line. If this value is two, every other grid node in the X direction will have a mesh line. If this value is three, every third grid node in the X direction will have a mesh line, and so on.

Line Properties

Click the ⊞ next to the *Line Properties* to open the line properties section for the X direction mesh lines. Line style, color, opacity, and width can be altered. 3D surface map mesh lines do not support complex line styles.

Lines of Constant Y

Check the *Draw lines* box under the *Lines of Constant Y* to draw lines along the surface at constant Y values.

Frequency

You can change the frequency of the Y lines by entering a new number into the *Frequency* box. If this value is one, every grid node in the Y direction will have a mesh line. If this value is two, every other grid node in the Y direction will have a mesh line. If this value is three, every third grid node in the Y direction will have a mesh line, and so on.

Line Properties

Click the ⊞ next to the *Line Properties* to open the line properties section for the Y direction mesh lines. Line style, color, opacity, and width can be altered. 3D surface map mesh lines do not support complex line styles.

Surface Offset

If you overlay two or more 3D surface maps, the mesh lines may appear dashed or broken. Experiment with the *Surface offset* values to change the distance between where the surface and mesh lines are drawn. The value is not associated with map units, instead it is offset by a factor. It is recommended that small changes be made to the *Surface offset* when trying to make the lines more clear.

Mesh Tips

- The default line frequency is one line in each direction.
- The mesh offset value o is calculated by:

  o = (m)(factor)+r

  where,

  m = maximum depth slope of the polygon

  factor = user-specified value in *Surface offset*, values typically range from 0.0 to 5.0

  r = the smallest value guaranteed to produce a resolvable difference in window coordinate depth values (a constant)

To Draw Mesh Lines

1. Click on the surface map to select it.
2. In the **Properties** window, click on the **Mesh** tab.

3. Check the *Draw lines* boxes in the *Lines of Constant X* and *Lines of Constant Y* sections.
4. Change the Line Properties for the X and Y lines.
5. You can change the frequency of the lines by entering a new number into the *Frequency* boxes.
6. If you overlay two or more surface maps, the mesh lines may appear dashed or broken. Experiment with the *Surface offset* values to change the distance above the surface that the mesh is drawn.

Lighting Properties

To edit a 3D surface map, click once on the surface map to select it. The properties for the surface map are displayed in the Properties window. To edit the **Lighting** in the 3D View, click *Environment* in the Contents window. Changes made to the environment lighting will be automatically applied to the lighting of point vector data in the 3D view. The 3D view window lighting properties are displayed in the **Properties** window.

The **Lighting** page controls the lighting for the entire multi-layer map. This includes the surface, any overlays that may have been combined with the surface, and point vector data in 3D. The light source is fixed relative to the surface, so if the surface is rotated, the light rotates with it.

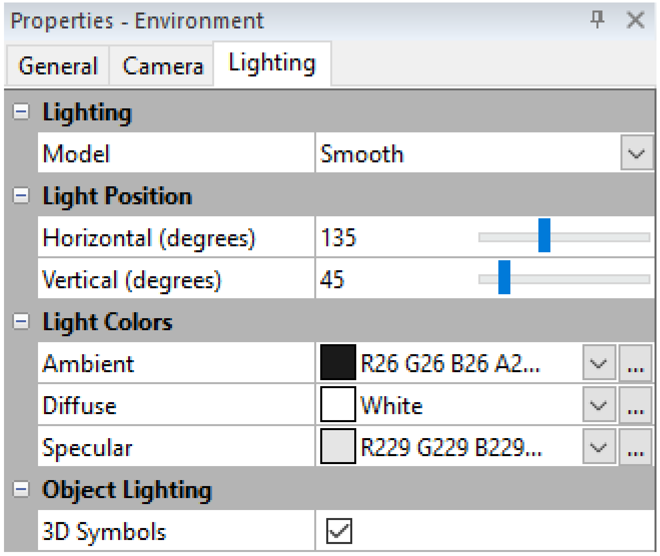
Hill and Reflectance Shading in the 3D View

The 3D view lighting is turned off when the map in the 3D view contains a hill shaded or reflectance shaded color relief layer. This is because the map is already shaded by hill shading or reflectance. The cumulative effect of both lighting systems would result in very dark regions of the map in the 3D view.

To use the 3D view **Lighting** properties instead of the color relief hill or reflectance shading, the *Terrain representation* option on the color relief layer General page must be set to *Color only*.

Lighting Properties

The following properties control the lighting appearance of the 3D surface map in the plot window or the lighting of the map in the 3D view.



*Change lighting properties in the **Properties** window on the **Lighting** page.*

Lighting

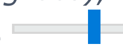
There are three lighting options. Click on the current lighting option next to *Model*. In the list, select the desired lighting option.

- *None* disables all lighting effects. The color shown is from the surface material color only.
- *Smooth* splits each grid cell is into two triangular polygons. *Gouraud* shading is used to interpolate colors within the triangles from the three vertices of each triangle. This results in smooth transitions across the triangles and the entire grid, but it is slightly slower than flat shading.
- *Flat* uses a single vertex (grid node) to define the shaded color for the entire polygon. Note that each grid cell is divided into two triangular polygons. This results in a faceted look since each triangle is only filled with a single color.

Light Position

The *Light Position* section specifies the orientation of the light source. The light source can be thought of as the sun shining on a topographic surface.

The *Horizontal (degrees)* box defines the direction for the light source in the horizontal plane. Zero degrees corresponds to the light source shining from due east toward the west. Positive angles rotate the light source counterclockwise. For example, a specified horizontal angle of 90 degrees places the light source north of the unrotated surface. 180 degrees places the light source west of the unrotated surface and shining east. 270 degrees places the light source south of the unrotated surface and shining north. The default horizontal angle is set at 135 degrees, or NW. To change the *Horizontal (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position.

The *Vertical (degrees)* box rotates the light source in the vertical plane. A vertical angle of zero degrees places the light source at the horizon and shining horizontally. An angle of 90 degrees places the light source directly overhead and shining down onto the map. 180 degrees places the light source at the opposite horizon and shining horizontally. 270 degrees places the light source directly below the map and shining up. The default vertical angle is 45 degrees. As the vertical angle approaches zero, shadows lengthen and the overall display shifts to the colors at the left end of the color spectrum. To change the *Vertical (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position.

Light Colors

There are three different types of light color, *Ambient*, *Diffuse*, and *Specular*. Note that these colors are used to represent reflectivity. White is 100% reflective and reflects the material color unaltered to the viewer. Black is 0% reflective, and causes all material color to be absorbed. Our perception of color is based on reflected and absorbed light. For example, a leaf appears green because it absorbs all colors in the light spectrum EXCEPT green. Since only green is reflected to your eye, the leaf appears green.

Surfer uses a pure white light source. The light "strikes" the surface and some of the light is absorbed based on the color of the surface material at the point the light ray struck it. Some light is reflected to the viewer according to the type of light (*Ambient*, *Diffuse*, and *Specular*), and the reflectivity color associated with each type of light specified in the *Light Colors* section.

- *Ambient* refers to light that has been scattered so evenly by the environment that its direction is impossible to determine. Increasing the ambient light component brightens the scene without casting shadows. The default *Ambient* color is 90% black which means that the ambient light contribution is fairly small.
- *Diffuse* refers to light coming from a particular direction and is brighter if aimed directly down on a surface than barely glancing off the surface. When diffuse light hits the surface, it is scattered uniformly in all directions so that it appears equally bright no matter where the eye is located. Increasing diffuse light intensifies shadow effects. The default *Diffuse* color is white, which is the maximum amount of reflectivity.
- *Specular* refers to light coming from a particular direction, and tends to bounce off the surface in a preferred direction. A shiny surface such as metal has a high specular component, while a surface like carpet has almost no specular component. Increasing the percentage of specular light results in strong shadow effects and more pronounced "shiny" or glare spots. The default *Specular* color is 90% black.

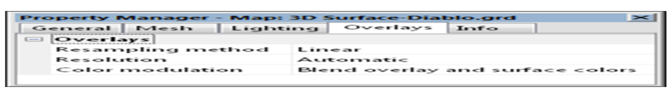
In general, these reflectivity colors should be specified as shades of gray in order to evenly reflect the surface material color components. However, special effects are possible by specifying non-gray colors for the reflectivity. For example, assume the *Ambient* reflectivity is set to pure red, and the *Diffuse* and *Specular* components are set to pure black. The *Diffuse* and *Specular* components are essentially disabled by setting their reflectivity color to black. The only light that is reflected to the viewer is red ambient light. Portions of the surface that lack a red component in the material color will appear black, since only red light is reflected to the viewer.

Object Lighting

The *3D Symbols* property is selected by default to enable the lighting to illuminate and cast shadows on point vector data as it does on the surface displayed in the 3D View window. When this property is cleared, shadows on 3D symbols remain static. The *Symbol method* for 3D vector point data must be set to *Symbols* and the symbol style either *Shaded Circle*, *Sphere* or *Cube*. The lighting *Model* must be either *Smooth* or *Flat*.

3D Surface Layer Overlays Properties

To edit a 3D surface map, click once on the surface map to select it. The properties for the surface map are displayed in the Properties window. The 3D surface **Overlays** page contains options to control how overlay layers are combined with the surface plot. Overlay layers are converted into an image known as a texture map. This texture map is applied to the surface by stretching it and shrinking it as necessary. Note that this stretching applies to all graphics in the overlays, including text. If the text is positioned over a steep portion of the surface, it can be stretched quite a bit, resulting in significant distortion. The usual solution to this is to view the surface from a higher angle to minimize stretching along the Z axis.



*Change overlay properties in the **Properties** window on the **Overlays** page.*

Resampling Method

When the texture map is stretched, the colors in the original overlay must be resampled to a new size and position. The *Resampling method* specifies how the texture map is is resampled. To change the *Resampling method*, click on the current method. In the list, select the desired method. The options are *Linear* and *Nearest*. *Linear* uses bilinear interpolation to combine the four surrounding pixels. Bilinear interpolation results in higher quality, but is usually slower. *Nearest* uses the nearest pixel in the source image.

Resolution

The *Resolution* refers to the resolution (in pixels) of the overlaid texture map on the surface map. High-resolution texture maps result in more detail, but line and text features become thinner and may eventually fade or break up. If lines appear too thin, you should change the *Resolution* to a smaller value. The default value of *Automatic* allows **Surfer** to automatically determine what the best resolution should be. Options are 64, 128. 256, 512, 1024, 2048, or 4096.

The maximum allowed value is determined by the video driver. This means that surface maps with overlays may appear differently on computers that do not have the same video driver resolution.

Color Modulation

Color modulation refers to the method used to combine the texture map and surface material colors. You can *Use surface color only*, *Use overlay color only*, or you can *Blend overlay and surface colors*. When set to *Use surface color only*, any other overlays will not be shown in the map. The surface map color is defined by the *Upper* colormap on the General page. When *Use overlay color only* is selected, the surface map color does not appear. The color is defined by any maps that are overlaid onto the surface. When *Blend overlay and surface colors* is selected, the surface map color and the overlaid maps are combined to form a new color. To change the *Color modulation* , click on the current option. In the list that appears, select the new option. The map will automatically update.

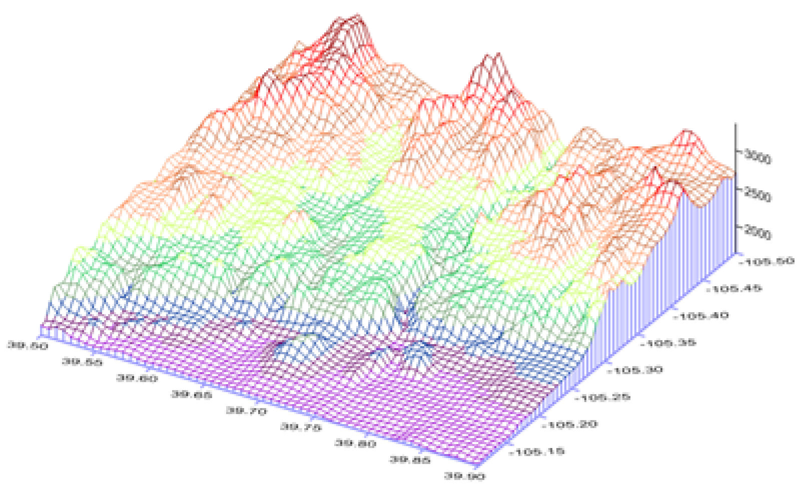
Map Layer Visibility

If there are map layers on a 3D surface, and the surface is made invisible through the **Contents** window, the layers will not be visible.

Chapter 12 - 3D Wireframe Maps

3D Wireframe

Wireframes are three-dimensional representations of a grid file. Wireframes are created by connecting Z values along lines of constant X and Y. Each XY intersection occurs at a grid node and the height of the wireframe is proportional to the Z value assigned to that node. The number of columns and rows in the grid file determines the number of X and Y lines drawn on the wireframe.



This wireframe displays gradationally colored lines that show elevation. The intersections of the lines occur at grid nodes.

Wireframes can display any combination of X lines, Y lines, or Z contour lines. On the wireframe, X lines correspond to the columns in the grid file and Y lines correspond to rows in the grid file.

The grid limits define the extent of the wireframe. When creating a grid file, set the grid limits to the desired limits for the wireframe. The wireframe cannot be larger or smaller than the extent of the grid file. Use the Grids | Resize | Extract or Grids | Resize | Mosaic commands to produce a subset or to thin a grid file. Use Grid | Spline Smooth to increase the density of a grid file in order to add additional X and Y lines to the wireframe. Use the Grid | Data command to create a new grid file with different limits.

Creating a 3D Wireframe

1. Click on the **Home | New Map | 3D Surface | 3D Wireframe** command or the button.
2. Select a grid file in the **Open Grid** dialog and click *Open*.

The map is automatically created with reasonable defaults.

Editing an Existing 3D Wireframe

To change the features of the 3D wireframe map, click once on the wireframe map in the plot window or in the Contents window to select it. The properties are displayed in the Properties window.

Adding a Map Layer

When wireframe maps are created, they are independent of other maps in the plot window. For example, creating a wireframe map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the contour map on the wireframe map, select both maps by clicking the Home | Selection | Select All command. Overlay the maps using the Overlay Maps command.

Alternatively, you can add the contour map directly to the existing wireframe map by creating the contour map using the **Home | Add to Map | Layer | Contour** command. This automatically adds the contour map to the existing wireframe map axes.

Another alternative is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the wireframe map using the **Home | New Map | 3D Surface | Wireframe** command. This creates two separate maps. Click on the contour map, hold down the left mouse button, and drag the contour map into the wireframe map. A single map with two map layers, using one set of axes and scaling parameters is created.

Wireframes cannot be overlaid with other wireframes, point cloud layers, or on raster-based maps such as 3D surfaces, color relief maps, and base (raster) layers. Wireframes can be overlaid with contour maps and other map types.

Wireframe Layer Properties

The wireframe properties contains the following pages:

General
Z Levels
Color Zones
Layer
Info (Grids)

Map Properties

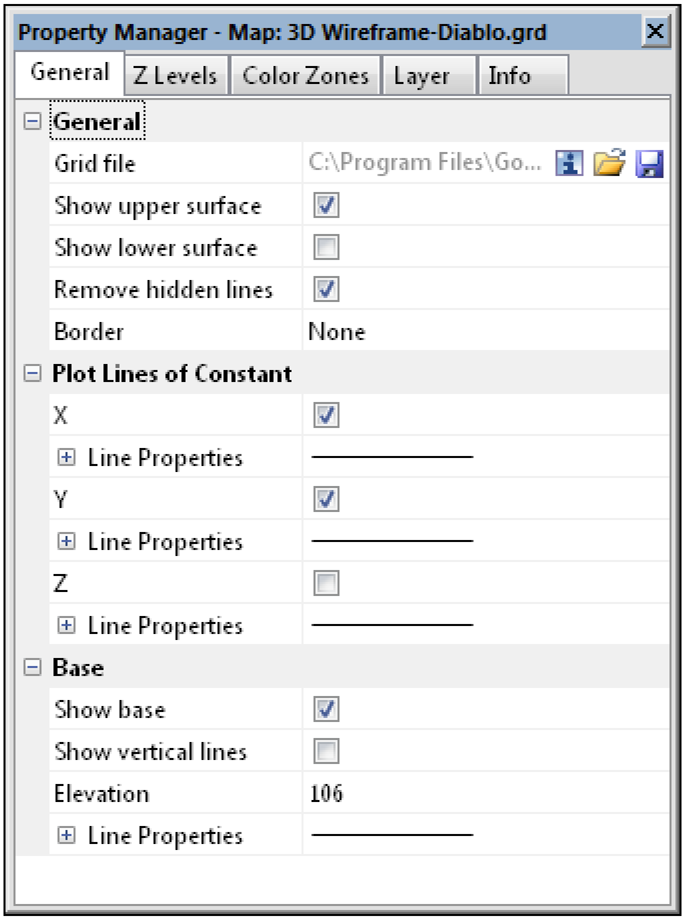
The map properties dialog contains the following pages:

View

Scale
Frame
Info (Objects)

3D Wireframe Layer General Properties

To edit a 3D wireframe, click once on the wireframe map to select it. The properties for the wireframe map are displayed in the Properties window. The 3D wireframe properties **General** page contains the following options:



*Change 3D wireframe map properties in the **Properties** window on the **General** page.*

Input Grid File

The *Grid file* displays the path and file name for the grid file used for the map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Grid Information

The button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the wireframe map. Select the new grid file and click *Open* to reference the new file. When the grid file is changed, the map limits are reset but the data limits are not recalculated.

Save File

Click the button to display the **Save Grid As** dialog. This allows the grid file used for the wireframe map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file.

Show Upper Surface

Check the *Show upper surface* box to display only the top portion of the surface. If the *Remove hidden lines* box is checked, only those portions of the upper surface not obscured by other portions of the surface are drawn.

Show Lower Surface

Check the *Show lower surface* box to display only the bottom of the surface. If the *Remove hidden lines* box is checked, only those portions where the bottom of the wireframe is not obscured by other portions of the surface are drawn. A base cannot be displayed when viewing the lower surface.

Remove Hidden Lines

Check the *Remove hidden lines* box to remove the display of X and Y lines behind other lines on the surface. When hidden lines are not removed, the wireframe has a transparent appearance.

Border

The *Border* option controls the display of the border line around the outside of the wireframe, at the intersection of the wireframe with the sides of the base. To change the border display, click on the existing option next to *Border* . Select the new option in the list. Options are to display the border on the *Front Only*, on *All Sides*, or on *None* of the sides. The border line is not visible on wireframes displaying both X and Y lines. The border line color is set in the *Plot Lines of Constant* section.

Plot Lines of Constant

The *Plot Lines of Constant* section allows you to specify the lines to be used when displaying the wireframe. Any combination of X, Y, and Z lines may be used. The properties on the Z Levels tab and Color Zones tab are only applied if the *Plot Lines of Constant* boxes are checked for the appropriate X, Y, or Z lines.

X Lines

Check the box next to *X* to display lines of constant X value on the wireframe.

Line Properties

Click the ⊞ next to the *Line Properties* to open the line properties section for the X direction lines. Line *Style*, *Color*, *Opacity*, and *Width* can be altered. 3D wireframes do not support complex line styles.

Y Lines

Check the box next to *Y* to display lines of constant Y value on the wireframe.

Line Properties

Click the ⊞ next to the *Line Properties* to open the line properties section for the Y direction lines. Line *Style*, *Color*, *Opacity*, and *Width* can be altered. 3D wireframes do not support complex line styles.

Z Lines

Check the box next to *Z* to display lines of constant Z value on the wireframe.

Line Properties

Click the ⊞ next to the *Line Properties* to open the line properties section for the Z direction lines. Line *Style*, *Color*, *Opacity*, and *Width* can be altered. 3D wireframes do not support complex line styles.

Base

The Base section controls the display of a base on the wireframe. The base is the area "underneath" the wireframe. The bottom of the base can be drawn at any Z level relative to the surface.

Show Base

Check the *Show base* box to display the base. The base on a wireframe is the lines connecting the wireframe to the corners of the axes.

Show Vertical Lines

Add X and Y lines to the base by checking the *Show vertical lines* box.

Elevation

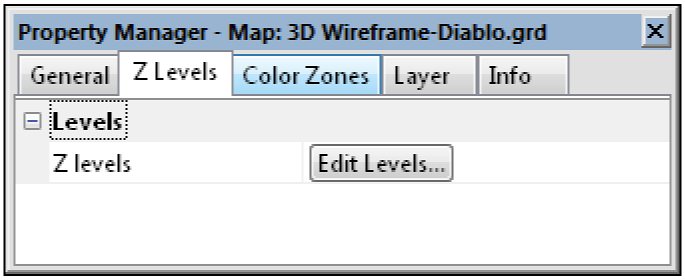
You can adjust the elevation of the base in the *Elevation* box. To change the elevation, highlight the existing value and type a new one.

Line Properties

Click the ⊞ next to the *Line Properties* to open the line properties section for the base lines. Line *Style*, *Color*, *Opacity*, and *Width* can be altered. 3D wireframes do not support complex line styles.

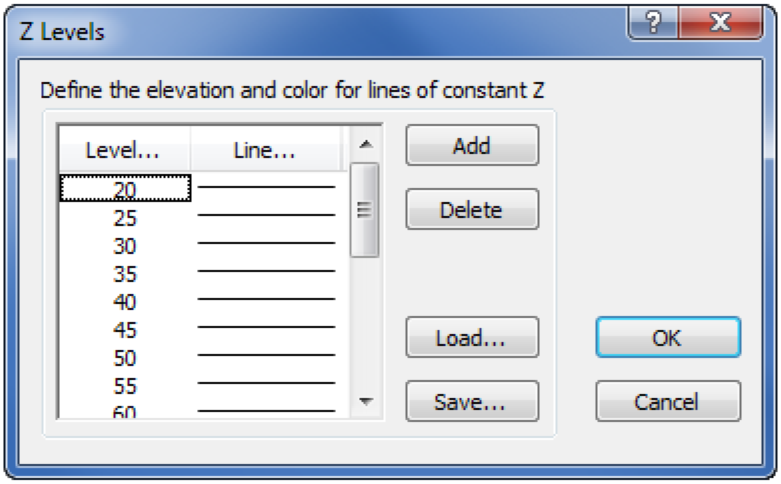
3D Wireframe Layer Z Levels Properties

To edit a 3D wireframe, click once on the wireframe map to select it. The properties for the wireframe map are displayed in the Properties window.



In the ***Properties*** *window, click the* Edit Levels *button to open the dialog and set theZ Levelsoptions for a wireframe.*

In the **Properties** window, click on the **Z Levels** tab. Then, click the *Edit Levels* button to open the **Z Levels dialog**. The 3D wireframe **Z Levels** dialog controls the display of lines of constant Z. When displaying the Z lines on a 3D wireframe, use the options on the **Z Levels** dialog to control which levels of Z lines to display. You can specify the Z contour levels and the line properties for the individual Z lines on this page. For the **Z Levels** options to appear, the box next to *Z* in the *Plot Lines of Constant* section on the General tab must be checked.



*Change the Z level properties in the **Z Levels** dialog.*

Level

Evenly spaced Z levels are assigned by clicking the *Level* button. This displays the Contour Levels dialog. Set the *Minimum, Maximum,* and *Interval* and click *OK.*

Level Value For One Level

To set a specific contour level, double-click the level value in the list that you wish to change. The Z Value dialog opens. Enter a new value, and click *OK*. This method creates unequal intervals between wireframe Z level lines.

Line

Click the *Line* button to open the Line Spectrum dialog and create gradational line color.

Level Line Property For One Level

To set a specific contour level line property, double-click the sample line next to the level value you wish to change. The Line Properties dialog opens. Make changes to the line properties and click *OK*. Line *Style*, *Color*, *Opacity*, and *Width* can be altered. 3D wireframes do not support complex line styles.

Add

Click the *Add* button to add a level. The added level is placed between the currently selected level and the level above the currently selected level with a value that is halfway between the two level values.

Delete

Highlight a level and then click the *Delete* button to remove a level.

Load

Click the *Load* button to load a level file. Level files can be created manually, or with wireframe or contour maps. When the level file contains additional options, like labels and hachures, the selected level line properties affect the Z contours on the 3D wireframe and do not display contour labels or hachures.

Save

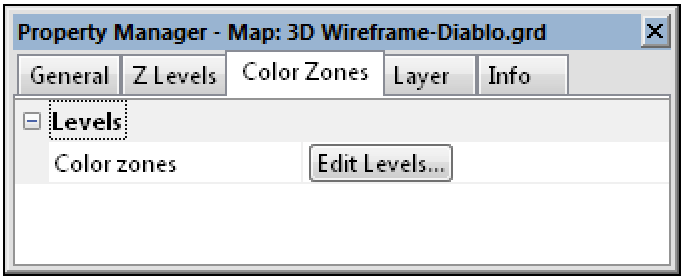
Click the *Save* button to save a level file.

Editing Z Levels Zones

1. Click on the wireframe map to select it.
2. In the **Properties** window, click on the **General** tab.
3. In the *Plot Lines of Constant* section, check the box next to *Z*.
4. Click on the **Z Levels** tab.
5. Click the *Edit Levels* button next to *Z levels*.
6. In the **Z Levels** dialog, change the desired level and line properties.
7. Click *OK* and the 3D wireframe is updated.

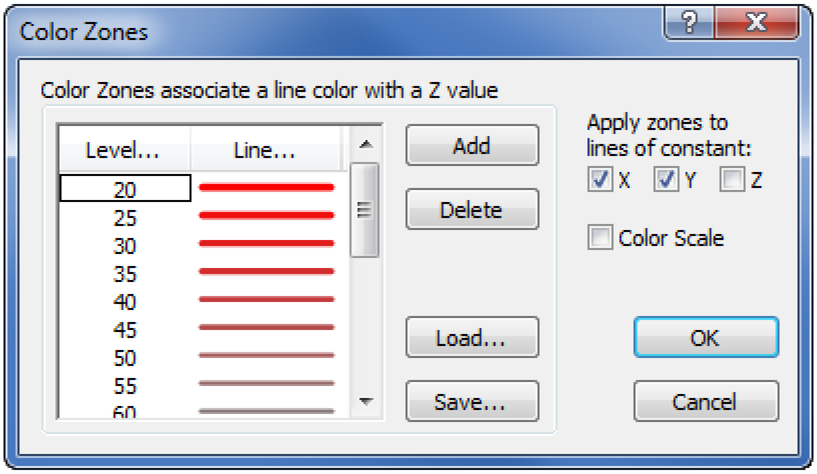
3D Wireframe Layer Color Zones Properties

To edit a 3D wireframe, click once on the wireframe map to select it. The properties for the wireframe map are displayed in the Properties window.



*Click on the **Color Zones** tab to edit the properties.*

In the **Properties** window, click on the **Color Zones** tab. Then, click on the *Edit Levels* button to open the 3D wireframe **Color Zones** dialog. The **Color Zones** dialog controls the color to apply to lines of constant X, Y, and Z based on Z values in the grid file. For the **Color Zones** options to appear, the same box next to *X, Y,* or *Z* in the *Plot Lines of Constant* section on the General tab must also be checked.



Change the color zone properties in the ***Color Zones*** *dialog.*

Level

Evenly spaced Z levels are assigned by clicking the *Level* button. This displays the Contour Levels dialog. Set the *Minimum, Maximum,* and *Interval* and click *OK.*

Level Value For One Level

To set a specific contour level, double-click the level value in the list that you wish to change. The Z Value dialog opens. Enter a new value, and click *OK*. This method creates unequal intervals between wireframe Z level lines.

Line

Click the *Line* button to open the Line Spectrum dialog and create gradational line color.

Level Line Property For One Level

To set a specific contour level line property, double-click the sample line next to the level value you wish to change. The Line Properties dialog opens. Make changes to the line properties and click *OK*. Line *Style*, *Color*, *Opacity*, and *Width* can be altered. 3D wireframes do not support complex line styles.

Add

Click the *Add* button to add a level. The added level is placed between the currently selected level and the level above the currently selected level with a value that is halfway between the two level values.

Delete

Highlight a level and then click the *Delete* button to remove a level.

Load

Click the *Load* button to load a level file. Level files can be created manually, or with wireframe or contour maps. When the level file contains additional options, like labels and hachures, the selected level line properties affect the Z contours on the 3D wireframe and do not display contour labels or hachures.

Save

Click the *Save* button to save a level file.

Apply Zones to Lines of Constant X, Y, and Z

The *Apply zones to lines of constant* allows the selection of which type of lines you want the color zones applied. Check the boxes next to *X, Y*, and *Z* to apply the colors to lines of constant X, Y, or Z.

Note that the same boxes must be checked on the General tab in the *Plot Lines of Constant* section for the color zone changes to appear on the map.

Color Scale Bar

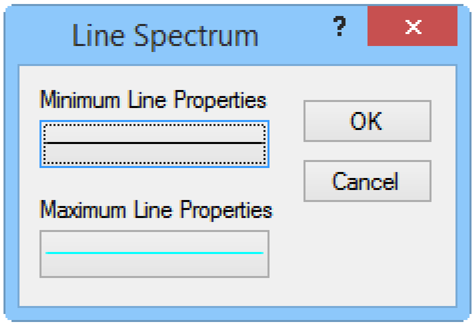
A color scale bar can be displayed on the map by checking the box next to the *Color Scale* option on this page.

Editing Color Zones

1. Click once on the wireframe to select it.
2. In the **Properties** window, click on the **General** tab.
3. In the *Plot Lines of Constant* section, check the box next to *X, Y,* or *Z* . The box must be checked on this tab for the color zone line properties to have an affect.
4. In the **Properties** window, click on the **Color Zones** tab.
5. Click the *Edit Levels* button next to *Color zones*.
6. In the **Color Zones** dialog, change the desired line and level properties.
7. Check the *X, Y,* or *Z* boxes to apply the color zone to that type of constant line.
8. Click *OK* and the wireframe is updated.

Line Spectrum Dialog

Click the *Line* button in the 3D wireframe dialog Color Zones or Z Levels page, or click the *Line* button in the grid node editor Contour Levels dialog to open the **Line Spectrum** dialog and create gradational line color.



*Specify the line properties in the **Line Spectrum** dialog.*

Minimum and Maximum Line Properties

Click the *Minimum Line Properties* and *Maximum Line Properties* buttons to specify the line style, color, and width for the minimum and maximum levels. When you click the *Minimum Line Properties* or *Maximum Line Properties* button, the Line Properties dialog is displayed. The minimum line property is assigned to the lowest contour level and the maximum line property is assigned to the highest contour level. **Surfer** automatically assigns gradational line colors or widths between the minimum and maximum levels.

If you choose different line styles for the minimum and maximum contours, there is no gradational change in line styles. The lower half of the contour lines use the style assigned for the minimum and the upper half of the contour lines use the style assigned for the maximum.

Invisible Line

If you do not want to display contour lines for a plot you can assign an invisible line style for both the minimum and maximum lines from the **Line Spectrum** dialog.

Wireframe Level Files

For 3D wireframes, .LVL level files contain information for the Z level line properties and for the color zone line properties. After defining custom level and line properties on the Z Levels page or the Color Zones page, save the level and color information in a level file. This level file can be recalled for any other 3D wireframe or contour maps.

Level files created from the wireframe properties dialog contain only information for the Z level line properties. Color fill, contour label, or hachure information is not written to the level file if the level file is created from the wireframe. However, a level file created from the advanced contour map level properties dialog may contain such additional information. A level file created from the wireframe properties dialog can be used for a contour map and vice versa. Any additional information not used is ignored.

Creating Level Files

To create a .LVL level file containing level and line property information from the wireframe properties:

1. Click once on a wireframe map to select it. The wireframe map properties will be shown in the **Properties** window.
2. Click on the **Z Levels** tab or the **Color Zones** tab.
3. Click the *Edit Levels* button. The **Properties** dialog opens.
4. Change any of the parameters, including the contour levels or line properties.
5. After changing the settings, click *Save.*
6. In the **Save As** dialog, type the name of the level file and click the *Save* button. The level file is saved with a .LVL extension.
7. Click *OK* to close the dialog. The wireframe map updates.

Using Level Files

To use an existing .LVL file with any wireframe map:

1. Click once on a wireframe map to select it. The wireframe map properties will be shown in the **Properties** window.
2. Click on the **Z Levels** tab or the **Color Zones** tab.
3. Click the *Edit Levels* button. The **Properties** dialog opens.
4. Click *Load*.
5. Select the .LVL file and click *Open.*
6. Click *OK* to close the dialog. The wireframe map updates.

Note, all wireframe maps using the level file must have comparable Z data ranges, otherwise lines will not appear on the map.

Color Filled Wireframe

Wireframes do not allow you to create filled contours, but you can create a 3D wireframe and add a filled contour map layer. This allows you to create a filled contour wireframe. Usually, 3D surface maps are better for creating this type of map.

To create a filled contour map and overlay on a 3D wireframe map:

1. Create a 3D wireframe map with the **Home | New Map | 3D Surface | Wireframe** command.
2. Select the grid file and click *Open*.
3. Click once on the wireframe map to select it.
4. Click the **Home | Add to Map | Layer | Contour** command.
5. Select the grid file and click *Open*.
6. Click once on the contour map in the Contents window to select it.
7. In the Properties window, click on the **Levels** page.
8. Check the box next to the *Fill contours* option.
9. Set any additional parameters for the contour map. The map will automatically update to display the changes.
10. The result is a filled contour map draped over the 3D wireframe map. Since no hidden line removal is performed on the contour map, it may be necessary to adjust the tilt so all areas are visible. Select the Map Tools | View | Trackball command or right-click on the map and select **Trackball**. Set the desired orientation parameters for the 3D wireframe. Press the ESC key on your keyboard and the wireframe is redrawn at the orientation you specify.
11. Alternatively, you can adjust the *tilt*, *field of view*, *rotation*, and *projection* values on the View page of the map properties.

Specifying the Lines to Draw on a Wireframe

Any combination of X, Y, and Z lines can be displayed on wireframe maps. When using X and Y lines to represent the surface, a mesh appears on top of the surface. When using the Z lines, an elevated contour map is drawn.

To specify which lines to draw on the 3D wireframe:

1. Click once on the wireframe map to select it.
2. In the Properties window, click on the General tab.
3. In the *Plot Lines of Constant* section, check the appropriate boxes to display any combination of X, Y, and Z lines on the wireframe.
4. When displaying the Z lines on a wireframe, use the options on the Z Levels page to control which levels of Z lines to display. Specify the Z contour levels and the line properties for the individual Z lines on this page. The selected levels affect the Z contours on the wireframe and do not display contour labels or hachures.

5. Note: To display contour labels or hachures on a wireframe, add a contour map layer. Select the wireframe map and use the **Home | Add to Map | Layer | Contour**.
6. Set any other desired options on the General page.
7. The wireframe is displayed with the line types you have chosen.

Adding Color Zones to a 3D Wireframe

You can change the line colors of any 3D wireframe by applying color zones. In this example, we will change color zones gradationally and individually.

To add a color zone to a 3D wireframe:

1. Click on the wireframe layer to select it.
2. In the Properties window, click on the **General** tab.
3. In the *Plot Lines of Constant* section, check the box next to *X* and *Y*. Whichever line you want to have the color zone applied to must be checked in this section.
4. Click the **Color Zones** tab.
5. Click the *Edit Levels* button next to *Color zones* .
6. Click the *Line* button to display the Line Spectrum dialog.
7. Click the *Minimum Line Properties* button to open the Line Properties dialog. From here, you can select the line *Color*, *Style*, *Width,* or *Opacity*.
8. Click the *Color* button and select *Blue* from the list.
9. Click *OK* to return to the **Line Spectrum** dialog.
10. Click the *Maximum Line Properties* button and change the line color to *Red* using the steps above.
11. Click *OK* in the **Line Spectrum** dialog to return to the **Color Zones** page.
12. Check the *Apply Zones to Lines of Constant X and Y* boxes.
13. Click *OK* and the wireframe is displayed with gradational X and Y lines varying colors by the Z variable.

To change the properties of an individual Z value:

1. Click on the wireframe, and the 3D wireframe properties are displayed in the Properties window.

2. Click the **Color Zones** tab.

3. Click the *Edit Levels* button next to *Color zones*.

4. Double-click the line sample for the contour level at Z = 70.

5. You can select the line *color*, *style*, *width,* or *opacity* for the selected line in the **Line Properties** dialog. In the *Width* box, click the up arrow and change the width value to 0.030 in.

6. Click *OK* in the **Line Properties** dialog and the **Color Zones** page is updated to reflect the change.

7. Click *OK* in the 3D wireframe **Properties** dialog and the map is redrawn. The color zone at Z = 70 is drawn with a thicker line, and is emphasized on the map.

Line Property Precedence

Wireframe line properties can be assigned on three different pages in the Properties window depending on the effect you are trying to achieve.

1. Color Zones page
2. Z Levels page
3. General page

The highest precedence is applied to line properties assigned on the **Color Zones** page. When the *Apply zones to lines of constant* check boxes on the **Color Zones** page are enabled, those properties take precedence over any other assigned line properties.

Line properties assigned in the *Plot Lines of Constant* section on the **General** page have the same precedence as the line properties assigned on the **Z Levels** page. If the options on the **Color Zones** page are not used, the next highest precedence are selections on the **General** page or **Z Levels** page, whichever is changed last. The **Z Levels** page colors are only used on Z lines.

Line Properties

Line properties for wireframes are controlled from the wireframe properties in the **Properties** window. The X and Y line properties specified in the *Plot Lines of Constant* section on the **General** page are applied over the entire surface without regard to the relative level of the surface.

To assign X and Y line properties so that they change over the vertical range of the surface, set the line properties on the **Color Zones** page. Color zone line properties vary based on Z values in the grid file.

Use the *Line Properties* in the *Plot Lines of Constant* section on the **General** page to apply the same line properties to all lines of constant Z without regard to level. To apply Z line properties that vary according to elevation, use the options in the **Z Levels** or **Color Zones** pages.

Wireframe Base

A 3D wireframe base is the area below the wireframe. The bottom of the base can extend to any Z level. The sides of the base can be highlighted with vertical lines that are extensions of the X and Y lines.

To change the base parameters:

1. Click on the wireframe map to select it.
2. In the Properties window, click on the General tab.
3. Check the *Show base* box to display the base lines.
4. To display vertical base lines, check the *Show vertical lines* box. Vertical lines are drawn on the face between the base and the surface border. The vertical lines represent the X and Y grid lines extended from the border of the surface to the base.
5. To specify base height, enter the value into the *Elevation* box. The value is in Z data units. The smaller the number entered in the *Elevation* box, the farther the base begins from the bottom of the wireframe surface, creating a thicker base.
6. Specify the line properties for the base in the *Line Properties* section.
7. The wireframe is drawn with the specified base parameters.

Smoothing a Wireframe

The smoothness of a 3D wireframe map is a function of the density and topography of the grid. For example, a 10 by 10 grid file (ten rows and ten columns) appears much more angular than a 50 by 50 grid file created from the same data. To smooth the wireframe map, you need to increase the number of rows and columns in the grid file.

You can increase the number of grid rows or columns in one of two ways. One method is to create a new grid file with more grid nodes in the X and Y directions in the Grid Data dialog.

Alternatively, use the Grids | Edit | Spline Smooth command to insert additional rows and columns into an existing grid file. Also, the Grids | Edit | Filter command can be used to increase the smoothness of the map.

Wireframe NoData Regions

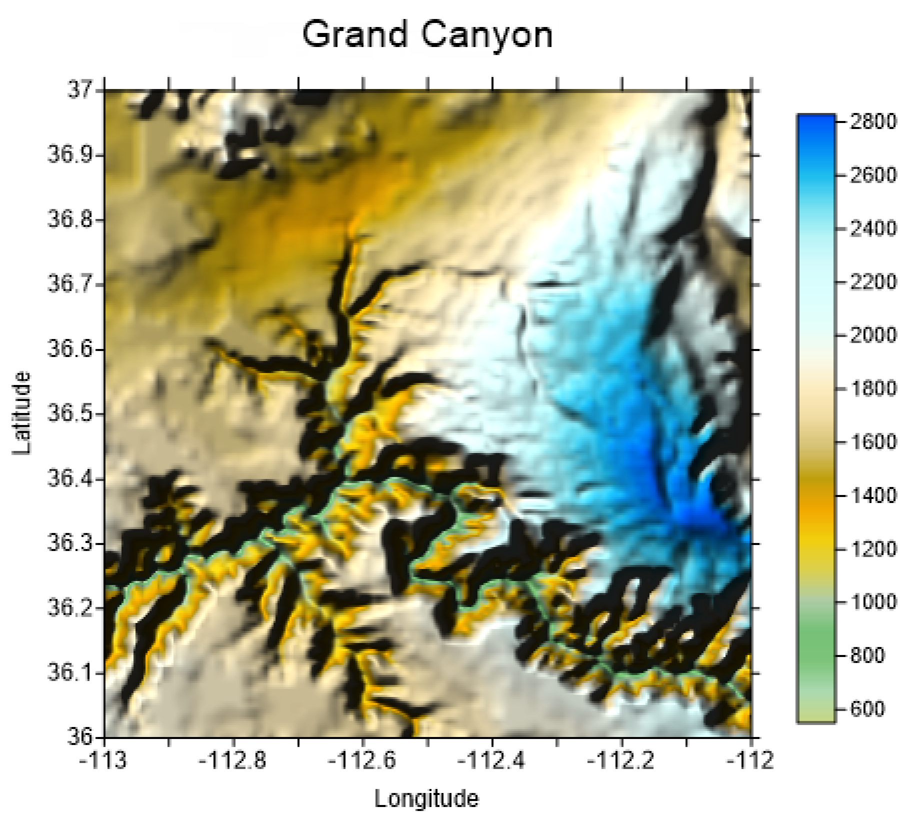
NoData regions of a grid file are represented by low flat areas on a 3D wireframe map. The level of the NoData region is set to the minimum Z value for the grid file or the *Elevation* value on the General page, whichever is greater. For example, consider a grid file that ranges from 0 to 100 in the Z dimension. When a wireframe is created, any NoData region appears as a large flat area at the Z= 0 value, and the *Elevation* property value also equals 0. If the *Elevation* property is changed to -15, the NoData region remains and Z=0. If the *Elevation* value is increased to 10, the NoData region is a low flat area at Z=10.

Digital Elevation Models (DEM) may have some NoData regions along the edges of the surface. The DEM polygon contained within a 7.5-minute quadrangle consists of profiles that do not always have the same number of data points because of the variable angle between the UTM coordinate system and true north. The result is a regular, stair-stepped arrangement of the data points along the edges of the map that can translate to grid nodes assigned the NoData value along the edges of grid files produced in **Surfer**.

Chapter 13 - Color Relief Maps

Color Relief Map

Color relief maps are raster maps based on grid files. These maps represent Z values (e.g. elevations) with user specified colors. NoData regions on color relief maps are shown as a separate color. Hill shading or reflectance can be enabled to enhance contrast and visualization.



This hill shaded color relief map shows the Grand Canyon. Colors show elevation in the area.

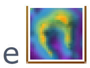
A colormap is used to map Z values to specific colors. **Surfer** automatically blends colors to produce a smooth color gradation over the map. 16 million colors are available (true color definition), creating smooth color variations.

The color schemes for color relief maps can be saved in Color Spectrum .CLR files. With such a file, the colors defined for one color relief map can be used with any other color relief map. Since the colors are stored as a percentage of the grid data range, a single color spectrum file can be used for multiple maps, even if the associated grid files cover significantly different data ranges.

In previous versions of **Surfer**, shaded relief maps and color relief maps were different layer types. Now the two layer types have been combined under the color relief layer reflectance shading (shaded relief) and hill shading options.

Creating a Color Relief Map

To create a color relief map:

1. Click the **Home | New Map | Color Relief** command or use the  button.
2. Select a grid file in the **Open Grid** dialog and click *Open*.

The map is automatically created.

Editing an Existing Color Relief Map

To change the features of the color relief map, click once on the color relief layer in the plot window or in the Contents window to select it. The properties are displayed in the Properties window.

Adding a Map Layer

When color relief maps are created, they are independent of other maps in the plot window. For example, creating a color relief map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the color relief layer on the contour layer, select both maps by clicking the Select All command. Overlay the maps using the Overlay Maps command.

Alternatively, you can add the color relief map directly to the existing contour map by creating the color relief map using the **Home | Add to Map | Layer | Color Relief** or **Map Tools | Add to Map | Layer | Color Relief** command. This automatically adds the color relief layer to the contour map axes.

Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the color relief map using the **Home | New Map | Color Relief** command. This creates two separate maps. Click on the color relief map, hold down the left mouse button, and drag the color relief map into the contour map. A single map with two map layers, using one set of axes and scaling parameters is created.

Color relief maps cannot be added as a map layer to 3D wireframe maps, but a color relief map can be added to any other map layer, including 3D surface maps.

Color Relief Layer and the 3D View

When the map in the 3D view contains a reflectance or hill shaded color relief layer, the 3D view Lighting is turned off. This is because the map is already shaded by the color relief layer hill shading. The cumulative effect of both lighting systems would result in very dark regions of the map in the 3D view.

To use the 3D view **Lighting** properties instead of the color relief hill shading, the *Enable hill shading* option on the color relief layer General page must be cleared.

Color Relief Layer Properties

The color relief layer properties contains the following pages:

General
Layer
Coordinate System
Info

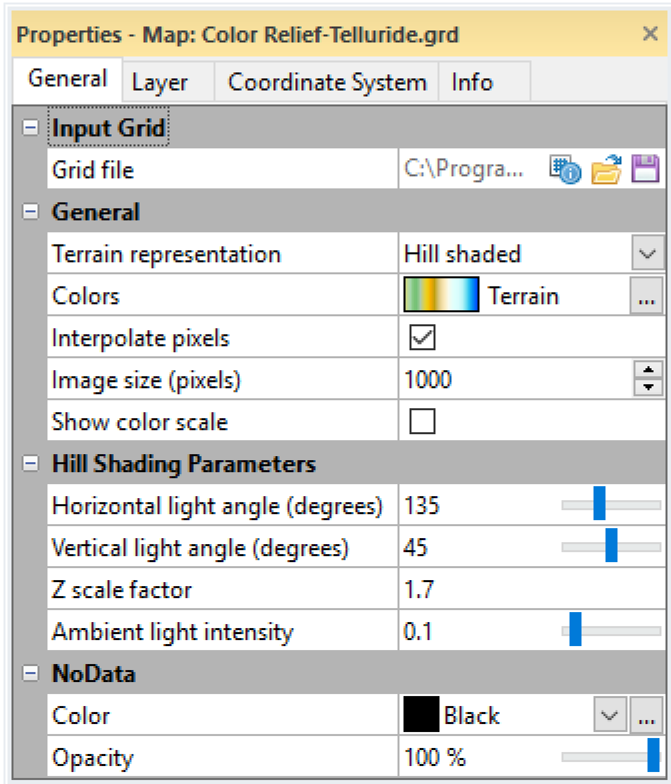
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info

Color Relief Layer General Properties

To edit a color relief layer, click once on the color relief layer to select it. The properties for the color relief layer are displayed in the Properties window. The color relief properties **General** page contains the following options:



*The color relief layer properties **General** page controls the layer color, color scale bar, hill shading, pixel interpolation, and missing data.*

Input Grid File

The *Grid file* displays the path and file name for the grid file used for the map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Grid Information

The button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the color relief map. Select the new grid file and click *Open* to reference the new file.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

Click the button to display the **Save Grid As** dialog. This allows the grid file used for the color relief map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file. If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, an **Export Options** dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Terrain Representation

The *Terrain Representation* property specifies if the color relief layer visualization is enhanced by hill shading or reflectance or if only colors are displayed. Select *Color only* to color the layer by the Z value and colormap only. Select *Hill shaded* to apply hill shading to the map. Select *Reflectance* to apply reflectance shading to the map.

Hill shading is a method of representing relief by depicting shadows cast by a point light source. Select the *Hill shaded* to apply hill shading to the color relief map. For hill shading a color relief map with a *GrayScale* colormap, it is recommended you change the lower color of the colormap to a slightly lighter black (e.g. *80% Black*). The *Hill Shading Parameters* are displayed when the *Hill shaded* option is selected.

Reflectance uses colors to indicate the local orientation of the surface relative to a user-defined light source direction. **Surfer** determines the orientation of each grid cell and calculates reflectance of a point light source on the grid surface. The light source can be thought of as the sun shining on a topographic surface. Portions of the surface that face away from the light source reflects less light toward the viewer, and thus appear darker. The colors on a reflectance shaded color relief map are based on the reflectance from the grid surface. Reflectance values range from zero to one. A reflectance value of zero means that no light is reflected toward the viewer. A reflectance value of one means that all incident light is reflected toward the viewer. The *Reflectance Parameters* are displayed when the *Reflectance* option is selected.

Hill shading and *Reflectance* replaces the 3D View Lighting properties. To use the 3D view lighting properties when viewing the map in the 3D view, set the *Terrain representation* to *Color only*.

Colors

The *Colors* option defines the colormap used to fill the color relief map. The colormap can use a linear or logarithmic scale. Change the color by clicking the existing color bar next to *Colors*. Select the new colormap from the list. If the desired color map is not listed, click the ... button to the right of the selected colormap. The Colormap Editor appears. Make any changes and click *OK* to see the change on the map. To set the colormap to use a logarithmic scale, click the ... button to the right of the selected colormap. Check the *Logarithmic scaling* option and click *OK*. The color map is automatically updated to show logarithmic scaling.

Interpolate Pixels

The *Interpolate pixels* check box activates color smoothing on the map. When checked, *Interpolate pixels* uses bilinear interpolation to calculate colors on the map. Bilinear interpolation makes the color gradations smoother, but it can slightly slow the on-screen drawing of the color relief map.

When a dense grid (a grid with relatively large numbers of rows and columns) is used, little difference is seen between the final color relief maps whether the *Interpolate pixels* option is checked or not. For dense grids, on-screen drawing time can be reduced when the *Interpolate pixels* option is unchecked.

When a coarse grid (a grid with relatively few rows and columns) is used and the *Interpolate pixels* option is not checked, all pixels within a single grid square are assigned the same color. This creates a grid-square map, and can result in a color relief map with a very blocky appearance. For coarse grids, therefore, a smoother appearance results when the *Interpolate pixels* check box is checked.

The non-interpolated color relief map maps each grid node to a pixel, i.e. a grid cell, which is expanded to a block of pixels to make the map the correct size. The interpolated color relief map assigns colors to the pixel or pixels at a grid node and then interpolates the colors for the pixels between the grid nodes.

Image Size (Pixels)

The *Image size (pixels)* property determines the number of pixels along the longest side of the color relief map. The number of pixels along the shorter side of the map is calculated from the *Image size (pixels)* and aspect ratio of the input grid file. The default *Image size (pixels)* value is 1000. Click the buttons to increase or decrease the *Image size (pixels)* by 100.

Increase the *Image size (pixels)* value to smooth the appearance of the color relief layer. Decrease the *Image size (pixels)* value to speed up the color relief layer rendering time. The *Image size (pixels)* property can be any integer value between 2 and 32767. Use caution when increasing the *Image size (pixels)* value to very large numbers. Large values can significantly increase the time to render the color relief layer.

Show Color Scale

The *Show color scale* option permits the display of a color scale bar adjacent to the map. The color scale bar indicates the colors assigned to the Z levels on the map, and the associated Z level values are displayed as labels on the color scale bar. When the *Show color scale* box is checked, the color scale bar is displayed. If the colormap is set to a logarithmic scale, the color scale bar will automatically display logarithmic scaling.

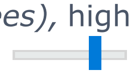
Hill Shading Parameters

The *Hill Shading Parameters* are displayed when the *Hill shaded* option is selected.

Horizontal Light Angle

The *Horizontal light angle (degrees)* box defines the direction for the light source in the horizontal plane. Zero degrees corresponds to the light source shining from due east toward the west. Positive angles rotate the light source counterclockwise. For example, a specified horizontal angle of 90 degrees places the light source north of the unrotated surface. 180 degrees places the light source west of the unrotated surface and shining east. 270 degrees places the light source south of the unrotated surface and shining north. The default horizontal angle is set at 135 degrees, or NW. To change the *Horizontal light angle (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position. The *Horizontal light angle (degrees)* option is disabled when the *Enable hill shading* box is not checked.

Vertical Light Angle

The *Vertical light angle (degrees)* box rotates the light source in the vertical plane. A vertical angle of zero degrees places the light source at the horizon and shining horizontally. An angle of 90 degrees places the light source directly overhead and shining down onto the map. The default vertical angle is 45 degrees. As the vertical angle approaches zero, shadows lengthen. To change the *Vertical light angle (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position. The *Vertical light angle (degrees)* option is disabled when the *Enable hill shading* box is not checked.

Z Scale Factor

The *Z scale factor* box allows you to specify the Z scale to use for the surface. As you increase the *Z scale factor*, the surface becomes more exaggerated vertically. Increasing the factor enhances the shading effect, and can be useful for bringing out more detail, especially on relatively flat surfaces. The default *Z scale factor* scales the Z coordinates to 1/4th the XY diagonal of the input grid. The *Z scale factor* option is disabled when the *Enable hill shading* box is not checked. The *Z scale factor* must be greater than 0. If a value less than or equal to 0 is entered into the *Z scale factor* box, the value will be automatically set to approximately 1.192×10^{-7}.

Ambient Light Intensity

The *Ambient light intensity* property controls the overall brightness of the surface. The *Ambient light intensity* is a value between 0 and 1. To change the *Ambient light intensity*, highlight the existing angle value. Type a new value or click and drag the  to the desired angle position. The *Ambient light intensity* option is disabled when the *Enable hill shading* box is not checked.

Reflectance Parameters

The *Reflectance Parameters* are displayed when the *Reflectance* option is selected.

Reflectance Method

The *Reflectance method* specifies the algorithm used to compute the reflected light at each grid cell. The *Reflectance method* determines how the colors are distributed in relation to the slopes and slope directions (aspect) over the extent of the map.

Horizontal Light Angle

The *Horizontal (degrees)* box defines the direction for the light source in the horizontal plane. Zero degrees corresponds to the light source shining from due east toward the west. Positive angles rotate the light source counterclockwise. For example, a specified horizontal angle of 90 degrees places the light source north of the unrotated surface. 180 degrees places the light source west of the unrotated surface and shining east. 270 degrees places the light source south of the unrotated surface and shining north. The default horizontal angle is set at 135 degrees, or NW. To change the *Horizontal (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position.

Vertical Light Angle

The *Vertical (degrees)* box rotates the light source in the vertical plane. A vertical angle of zero degrees places the light source at the horizon and shining horizontally. An angle of 90 degrees places the light source directly overhead and shining down onto the map. 180 degrees places the light source at the opposite horizon and shining horizontally. 270 degrees places the light source directly below the map and shining up. The default vertical angle is 45 degrees. As the vertical angle approaches zero, shadows lengthen and the overall display shifts to the colors at the left end of the color spectrum. To change the *Vertical (degrees),* highlight the existing angle value. Type a new value or click and drag the to the desired angle position.

Z Scale Factor

The *Z scale factor* box allows you to specify the Z scale to use for the surface. As you increase the *Z scale factor*, the surface becomes more exaggerated vertically. Increasing the factor enhances the shading effect, and can be useful for bringing out more detail, especially on relatively flat surfaces. The default *Z scale factor* scales the Z coordinates to 1/10th the XY diagonal of the input grid. The *Z scale factor* must be greater than 0. If a value less than or equal to 0 is entered into the *Z scale factor* box, the value will be automatically set to approximately 1.192×10^{-7}.

NoData

The *NoData* section sets the color for grid nodes that have been assigned the NoData value.

NoData Color

To select a different NoData region color, click on the existing color to the right *Color* and click on the desired color from the palette. Click the ... button to the right of the selected color to define new colors in the Colors dialog.

NoData Opacity

To change the *Opacity* of the NoData regions, enter a value from 0% (completely transparent) to 100% (completely opaque) by highlighting the existing value and typing a new value or by dragging the to change the opacity percentage.

Reflectance Shading Methods

There are four reflectance methods available for color relief layers using *Reflectance* for the *Terrain representation*. To set the *Reflectance method*, click on the color relief layer to select it. In the Properties window, click on the General tab. Change the *Terrain representation* to *Reflectance* and change the *Reflectance method* to the desired method, described below.

Simple

Simple is the fastest of the shading methods, but produces a rather crude image. With *Simple*, the *Horizontal* (azimuth) and *Vertical* (zenith) values are fixed at 135 and 45 degrees, respectively, and cannot be changed.

Peucker’s Approximation

Peucker’s approximation uses a piecewise linear approximation. This method gives somewhat better results than the *Simple* method, but redrawing the map takes slightly longer. With *Peucker’s approximation*, the *Horizontal* (azimuth) and *Vertical* (zenith) values are fixed at 135 and 45 degrees, respectively, and cannot be changed.

Lambertian Reflection

Lambertian reflection assumes an ideal surface that reflects all the light that strikes it and the surface appears equally bright from all viewing directions. The *Light Position Angles* section allows the *Horizontal (angle)* and *Vertical (angle)* values to be specified with this option. This is the default *Reflectance method.*

Lommel-Seeliger Law

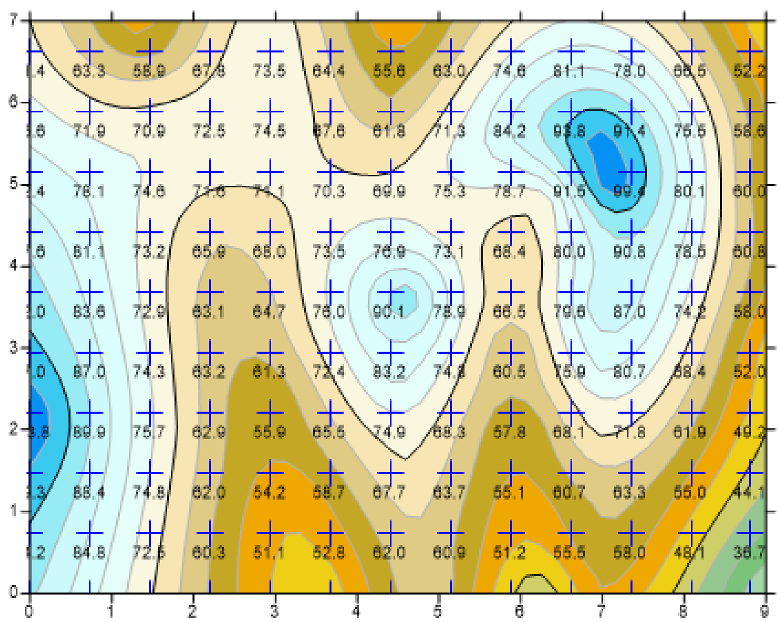
The *Lommel-Seeliger law* method is based on an analysis of light scattering from a surface. This method is a compromise between an ideal diffuser and a real surface. With some surfaces, this may actually give better results than the *Lambertian reflection* method. The *Light Position Angles* section allows the *Horizontal (angle)* and *Vertical (angle)* values to be specified with this option.

Chapter 14 - Grid Values Maps

Grid Values Map

The **Home | New Map | Specialty | Grid Values** command creates a grid values map. A grid values map indicates the location and values of the grid nodes with symbols and labels. Posting grid values on a map can be useful for visualizing the density of the grid, as well as for verifying the plausibility of your gridded data. The frequency of the plotted grid nodes can be controlled in the X and Y directions independently. The grid values can also be displayed for only a specific range of values. You can specify the symbol type, size, color, and angle for the grid nodes. Associated data values may be placed next to the posted point. The size, angle, color, and typeface for the label can also be specified. Grid lines can also be plotted to help you visualize the grid density.

The **Home | Add to Map | Layer | Grid Values** or **Map Tools | Add to Map | Layer | Grid Values** command adds a grid values map layer to the selected map.



The grid values map displays grid node location and values for all or a subset of the grid nodes. In this example every fourth grid node value and location is displayed in the X and Y directions. The grid values layer is overlaid with a filled contour layer.

Creating a Grid Values Map

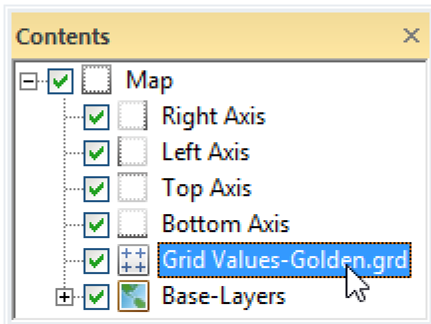
To create a new grid values map:

1. Click **Home | New Map | Specialty | Grid Values**.
2. Select a grid file in the **Open Grid** dialog. Click *Open*.

The map is automatically created with symbols and labels displayed across the map. The X and Y frequencies are automatically set to display approximately ten rows and columns of grid nodes.

Editing an Existing Grid Values Map

To change the features of the grid values map, click once on the grid values layer in the Contents window or plot window. When the grid values layer is selected, the grid values properties are displayed in the Properties window.



Click on the Grid Values *map layer to select it.*

Grid Values Layer Properties

The **Properties** window for a grid values layer contains the following pages:

General
Symbols
Labels
Layer
Coordinate System
Info (Grids)

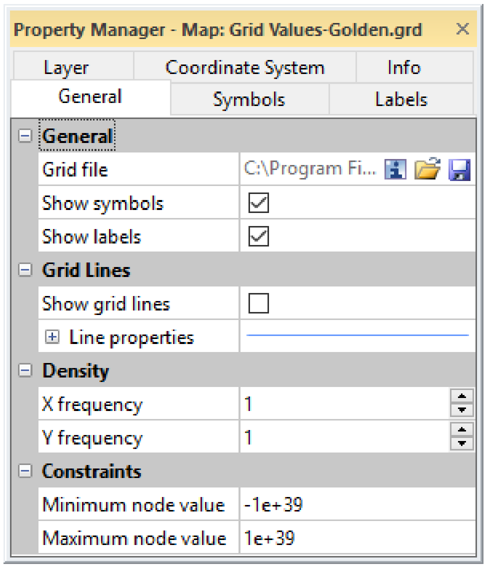
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info (Objects)

Grid Values Layer General Properties

The **General** page for a grid values layer controls the display of symbols, labels, and lines, as well as the grid values density and constraints.



*Specify general display options on the **General** page of the **Properties** window.*

General

The *General* section of the **General** page includes options for the layer grid file and display of symbols and labels.

Input Grid File

The *Grid file* lists the current grid file used in the contour map. The path and file name are the location of the grid file when the map was created or the grid file was most recently changed.

Grid Information

The  button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

The button displays the **Open Grid** dialog. This allows you to select a new grid file, or an updated version of the grid file used to create the grid values map. Select a grid file and click *Open*.

If the new grid exceeds the current map limits, another warning will appear asking you to adjust the map limits. If you click *Yes*, the limits are automatically adjusted to fit the new grid. If you click *No*, the limits are not automatically adjusted. The map may not be displayed. To change the map limits, click on the Map object in the **Contents** window and the **Limits** tab in the **Properties** window.

You may also see a warning message that the current map scale may result in an un-viewable map. Clicking *OK* allows the map scale to automatically be adjusted.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

The button displays the **Save Grid As** dialog. Type a *File name* and change the *Save as type* to the desired grid file format. Click *Save.* If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, an **Export Options** dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Show Symbols

Symbols are displayed at grid node locations when the *Show symbols* check box is checked. Symbols are displayed by default. Clear the *Show symbols* check box to hide the symbols. The frequency of the symbols is controlled by the properties in the *Density* section.

Show Labels

Grid node value labels are displayed relative to the grid node locations when the *Show labels* check box is checked. Labels are displayed below the symbols by default. When symbols are not displayed, labels are positioned relative to the grid node location. Clear the *Show labels* check box to hide the labels. The frequency of the labels is controlled by the properties in the *Density* section.

Grid Lines

The *Grid Lines* section of the **General** page contains options for the display of grid lines and their properties.

Show Grid Lines

Check the *Show grid lines* check box to display grid lines for grid rows and columns. Clear the *Show grid lines* check box to hide the grid lines. The frequency of the grid lines is controlled by the properties in the *Density* section.

Line Properties

The *Line properties* control the display of the grid lines. See the Line Properties topic for more information on editing line properties.

Density

The *Density* section of the **General** page contains options for the density of the symbol, label, and/or line display.

X Frequency

The *X frequency* value specifies the frequency of drawn symbols, labels, and/or lines in the X direction. An *X frequency* value of 1 displays every grid node column. A frequency of 2 displays every other grid node column, 3 displays every third column, etc. The default value is the value in which approximately ten grid columns are visible across the map. Type a value in the *X frequency* field or click the buttons to change the density of the symbols, labels, and/or lines in the X direction.

Y Frequency

The *Y frequency* value specifies the frequency of drawn symbols, labels, and/or lines in the Y direction. A *Y frequency* value of 1 displays every grid node row. A frequency of 2 displays every other grid node row, 3 displays every third row, etc. The default value is the value in which approximately ten grid rows are visible across the map. Type a value in the *Y frequency* field or click the buttons to change the density of the symbols, labels, and/or lines in the Y direction.

Constraints

The *Constraints* section of the **General** page controls the minimum and maximum values for grid nodes displayed on the map.

Minimum Node Value

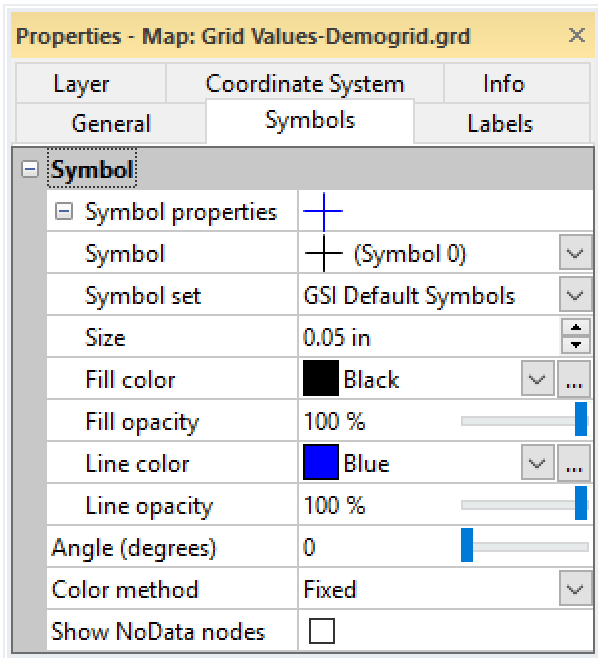
The *Minimum node value* sets the smallest grid node value that will have a symbol and/or label displayed on the grid values map. By default the *Minimum node* *value* is -1e+39, i.e. -1 x 10^{39}. Only grid nodes with a value greater than or equal to the *Minimum node value* will be displayed.

Maximum Node Value

The *Maximum node value* sets the largest grid node value that will have a symbol and/or label displayed on the grid values map. By default the *Maximum node value* is 1e+39, i.e. 1 x 10^{39}. Only grid nodes with a value less than or equal to the *Maximum node value* will be displayed.

Grid Values Layer Symbols Properties

The **Symbols** page controls the appearance of symbols in a grid values layer. Symbols are displayed by checking the *Show symbols* check box on the General page.



Specify symbol display options on the ***Symbols*** *page of the* ***Properties*** *window.*

Symbol Properties

The *Symbol properties* section controls the appearance of grid node symbols in the map. See the *Symbol Properties* topic for more information on changing symbol properties.

Angle

The *Angle (degrees)* property sets the rotation of the symbols in the grid values layer. Type a value in degrees in the *Angle (degrees)* field between 0 and 360 or click and drag the slider to select a value. Symbols rotate counterclockwise as the *Angle (degrees)* increases.

Color Method

The symbols in a grid values layer can all be the same color or be varied by grid node value. When the *Color method* is set to *Fixed*, the symbols are all the same color and all of the properties in the *Symbol properties* section applies to the symbols.

Set the *Color method* to *Colormap* to apply the colors from a colormap to the symbols based on grid node value. When the *Color method* is set to *Colormap*, the *Fill color*, *Fill opacity*, *Line color*, and *Line opacity* properties in the *Symbol properties* section are ignored. Symbols use the same line and fill color and opacity, which is determined by the grid node value and colormap.

Colors

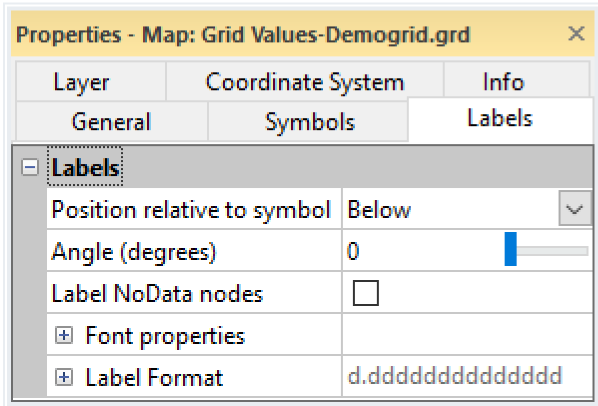
The *Colors* property is enabled when the *Color method* is set to *Colormap*. Click the current colormap and select a predefined colormap from the list or create a custom colormap in the Colormap Editor by clicking the ... button in the *Colors* field.

Show NoData Nodes

Check the *Show NoData nodes* check box to display symbols at grid nodes assigned the NoData value. NoData nodes will not have a symbol by default. Clear the *Show NoData nodes* check box to hide symbols at NoData grid nodes.

Grid Values Layer Labels Properties

The **Labels** page controls the appearance of labels in a grid values layer. Labels are displayed by checking the *Show labels* check box on the General page.



*Specify label display options on the **Labels** page of the **Properties** window.*

Position Relative to Symbol

Select the relative position for labels in the *Position relative to symbol* field. Click the current selection and select *Center*, *Left*, *Right*, *Above*, *Below*, or *Custom* from the *Position relative to symbol* list. If you select *Custom*, the *X offset* and *Y offset* properties are displayed.

X Offset

The *X offset* positions the labels relative to the symbol position in the X direction (left and right). Positive *X offset* values move the labels to the right. Negative *X offset* values move the labels to the left. Type a value in page units between -4.0 and 4.0 inches (-10.16 to 10.16 centimeters) to set the *X offset*, or click the button to change the *X offset* .

Y Offset

The *Y offset* positions the labels relative to the symbol position in the Y direction (up and down). Positive *Y offset* values move the labels up. Negative *Y offset* values move the labels down. Type a value in page units between -4.0 and 4.0 inches (-10.16 to 10.16 centimeters) to set the *Y offset*, or click the button to change the *Y offset*.

Angle

The *Angle (degrees)* property sets the rotation of the labels in the grid values layer. Type a value in degrees in the *Angle (degrees)* field between 0 and 360 or click and drag the slider to select a value. Labels rotate counterclockwise as the *Angle (degrees)* increases.

Label NoData Nodes

Check the *Label NoData nodes* check box to display labels at grid nodes with the NoData value. NoData nodes will not have a label by default. Clear the *Label NoData nodes* check box to hide labels at NoData nodes. "N/A" is displayed for labels at NoData grid nodes.

Font Properties

The *Font properties* section controls the font for grid node labels in the map. See the Text and Font Properties topic for more information on changing font properties.

Label Format

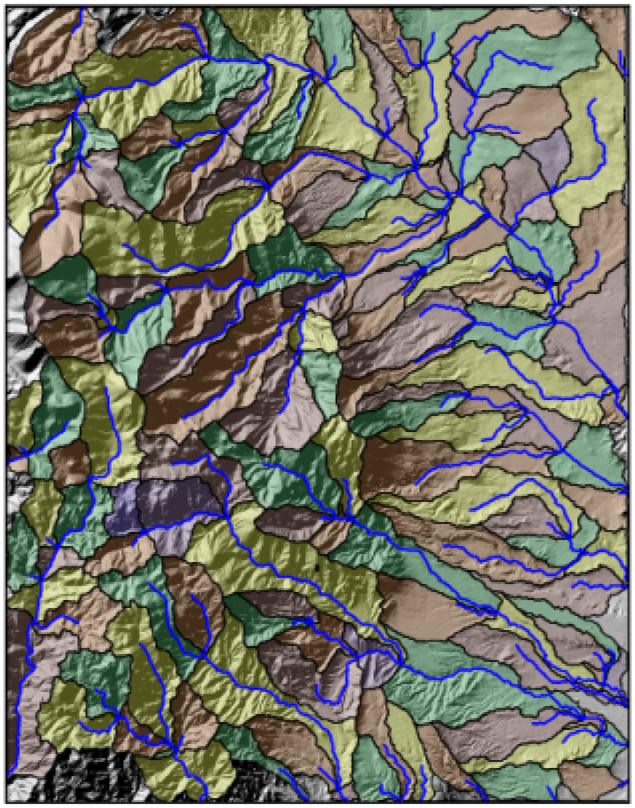
The *Label format* section controls the format of the grid node labels in the map. See the *Label Formats* topic for more information on changing label format properties

Chapter 15 - Watershed Maps

Watershed

The **Home | New Map | Specialty | Watershed** command or the button can be used to create a watershed map from a grid file. The **Home | Add to Map | Layer | Watershed** or **Map Tools | Add to Map | Layer | Watershed** command adds a watershed map layer to the selected map.

A watershed is defined as the region draining into a stream, stream system, or body of water. A watershed map reads the data from a grid file and splits the grid up into basin, or catchment, areas. Basin areas are areas that drain water to the stream. Stream paths are calculated based on the amount of flow into the grid node from all surrounding grid nodes. This shows the path water will take across the grid. Stream lines mark the low points on the map. Depressions are areas of internal drainage, such as a sink. Depressions can be forced to overfill so that water continues to move across the surface. Depression areas can be exported. Pour points are the outlets of the stream systems and are calculated at all drainage intersections and where the drainage leaves the edge of the grid. The pour points can be generated automatically or selected from a file. **Surfer** uses the eight-direction pour point algorithm to calculate the flow direction at each grid node. Custom methods are used to calculate the flow direction through flat areas and for filling depressions.



This example map shows a watershed with several basins and streams. A relief layer is overlaid.

Creating a Watershed

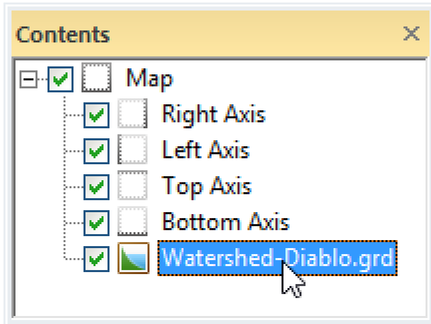
To create a watershed:

1. Click the **Home | New Map | Specialty | Watershed** command.
2. Select a grid file in the **Open Grid** dialog and click *Open*.

The map is automatically created with reasonable defaults.

Editing an Existing Watershed

To change the features of the watershed map layer, click once on the watershed map layer in the plot window or in the Contents window to select it. The properties are displayed in the Properties window.



Click on the Watershed *layer to select it.*

Adding a Map Layer

When watershed maps are created, they are independent of other maps in the plot window. For example, creating a watershed map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the contour map on the watershed map, select both maps by clicking the Home | Selection | Select All command. Overlay the maps using the Overlay Maps command.

Alternatively, you can add the contour map directly to the existing watershed map by creating the contour map using the **Home | Add to Map | Layer | Contour** command. This automatically adds the contour map to the existing watershed map axes.

Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the watershed map using the **Home | New Map | Specialty | Watershed** command. This creates two separate maps. Click on the contour layer, hold down the left mouse button, and drag the contour map into the watershed map. A single map with two map layers, using one set of axes and scaling parameters is created.

Any number of peaks and depressions maps can be combined with any other map layers.

Watershed Properties

The watershed properties contains the following pages:

General
Layer
Coordinate System
Info (Grids)

Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info (Objects)

Watershed References

Many of the parameters for the watershed map are technical in origin, and a detailed development is beyond the scope of this document. For additional information, see the following resources:

O'Callaghan JF, Mark DM. (1984) "The Extraction of Drainage Networks from Digital Elevation Data." *Computer vision, Graphics and Image Processing*, Vol. 28, 323-344.

Wang, L. & H. Liu (2006) "An Efficient Method for Identifying and Filling Surface Depressions in Digital Elevation Models for Hydrologic Analysis and Modelling." *International Journal of Geographical Information Science*, Vol. 20, No. 2: 193-213.

Garbrecht, J. and Martz, L.W. (1997) "The assignment of drainage direction over flat surfaces in raster digital elevation models." *Journal of Hydrology,* Vol. 193, 204-213.

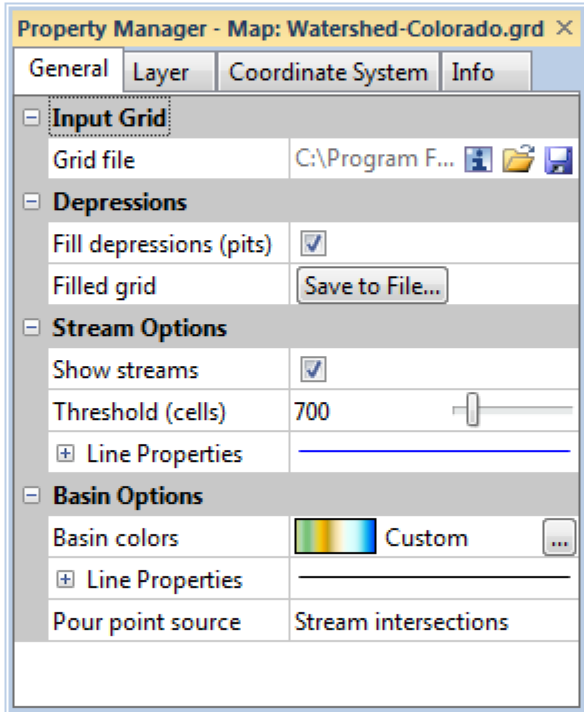
Welsh, D. J. A.; Powell, M. B. (1967), "An upper bound for the chromatic number of a graph and its application to timetabling problems." *The Computer Journal*, Vol. 10 (1): 85–86.

Jenson S. K. and J. O. Domingue. 1988. "Extracting Topographic Structure from Digital Elevation Data for Geographic Information System Analysis." *Photogrammetric Engineering and Remote Sensing,* Vol. 54 (11): 1593-1600.

Pierre Soille, Jurgen Vogt, and Roberto Colombo, 2003, "Carving and adaptive drainage enforcement of grid digital elevation models." *Water Resources Research*, Vol. 39, NO. 12, 1366.

Watershed Layer General Properties

To edit a watershed map, click once on the watershed map layer to select it. The properties for the watershed layer are displayed in the Properties window. The watershed properties **General** page contains the following options:



Change watershed layer properties in the ***Properties*** *window on the* ***General*** *page.*

Input Grid File

The *Grid file* displays the path and file name for the grid file used for the map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Grid Information

The  button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the  button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the watershed map. Select the new grid file and click *Open* to reference the new file.

If the new grid exceeds the current map limits, a warning will appear asking you to adjust the map limits. If you click *Yes*, the limits are automatically adjusted to fit the new grid. If you click *No*, the limits are not automatically adjusted. The map may not be displayed. To change the map limits, click on the Map object in the **Contents** window and the **Limits** tab in the **Properties** window.

You may also see a warning message that the current map scale may result in an un-viewable map. Clicking *OK* allows the map scale to automatically be adjusted.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

Click the button to display the **Save Grid As** dialog. This allows the grid file used for the watershed map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file. If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, the Export Options dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Depressions

The *Depressions* section contains the option to fill the depressions and save the data created by the filled depressions to a grid file. *Depressions* are areas of internal drainage. The only way water can leave a depression is by checking the *Fill depressions (pits)* option. Click the ⊞ next to *Depressions* to set the fill options and export a grid file that contains the depressions.

Fill Depressions

Check the box next to the *Fill depressions (pits)* option to fill all depressions in the grid. Depressions cause the flow of water to terminate. Checking this option causes the flow from all grid cells to exit the edge of the grid, allowing no trapped water areas. Unchecking this option allows areas to become internal sinks and allows the flow to terminate in these sinks. This may or may not be appropriate, depending on the topography of the grid. Changing this option does affect the resulting basin boundaries.

When the *Fill depressions (pits)* box is checked or unchecked, the *Threshold (cells)* value is re-evaluated. If the *Threshold (cells)* value is greater than the minimum flow and smaller than the maximum flow, the *Threshold (cells)* does not change. If the new value is outside the range, the *Threshold (cells)* value changes to be within the new range of flow. This can create a different number of basins by simply unchecking and checking the *Fill depressions (pits)* box.

Filled Grid

Click the *Save to File* button next to *Filled grid* to save the depressions to a grid file. The resulting grid file has the Z value in depressions changed to match the lowest Z value of surrounding grid nodes. This makes the surface flat in these depression areas. The *Save to File* button is disabled when *Fill depressions (pits)* is unchecked.

Stream Options

Streams show the path water will take across the grid. Stream paths are calculated based on the amount of flow into the grid node from all surrounding grid nodes. Streams start when the number of upstream grid cells draining into a grid cell exceeds the specified *Threshold (cells)* value. Click the ⊞ next to *Stream Options* to set the threshold and line properties for streams.

Show Streams

Check the box next to *Show streams* to show the drainages with flow values greater than or equal to the *Threshold (cells)* value. Set the line properties for the streams in the *Line Properties* section.

Threshold (cells)

The *Threshold (cells)* value is the number of upstream cells that are flowing into the grid cell that are required to create a stream line. Higher values create less streams and basins. Lower values create more streams and basins.

Line Properties

Click the ⊞ next to *Line Properties* to set the stream line properties, including style, color, opacity, and width.

Basin Options

Basin areas are areas that drain water to the stream. You can set the basin colors and the pour point source in the *Basin Options* section by clicking the ⊞ next to *Basin Options* to open the section.

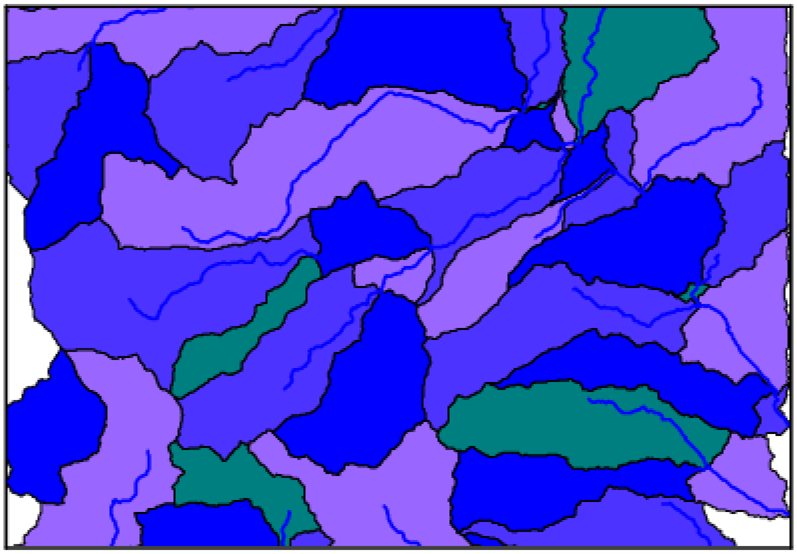
Basin Colors

The *Basin colors* option controls the colors used for creating the watershed basin areas. To change the colors, click on the color bar next to *Basin colors*. Select one of the predefined colormaps from the list. To create custom colors, click on the ... button to the right of the selected colormap to open the Colormap Editor.

Surferapplies a custom algorithm to determine how many colors are needed so that no adjacent basins have the same color. This algorithm usually determines that between three and six colors are necessary. Once the number of colors is determined, the selected colormap is sampled to choose the colors. The colormap ranges in value from 0 to 1. The first color is picked at a value of 0 on the left side of the colormap. The second color is picked 1/10th to the right on the colormap at a value of 0.1. The third color is picked at a value of 0.2, the fourth at 0.3, and so on. With most colormaps, this provides a smooth transition of colors for the basins in the watershed map.

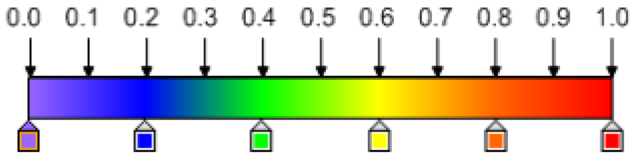
Example

Consider this watershed map and the *Rainbow*colormap.



The watershed is filled with four colors, ranging from a purple to green.

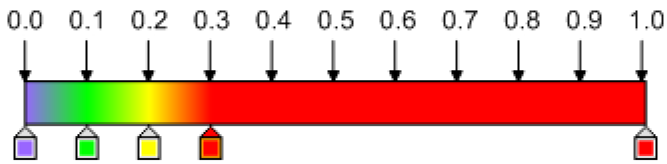
Because the watershed requires four colors, the colors at 1/10th increments (at values 0.0, 0.1, 0.2, and 0.3) are selected to fill the basins. These correspond to the light purple, blue-purple, blue, and blue-green colors.



The colors at values of 0, 0.1, 0.2, and 0.3 are used for the watershed map above.

If additional colors were needed, the green color at value 0.4, the yellow-green color at 0.5, the yellow color at 0.6, and the yellow-orange color at 0.7 would be used.

Because it is rare that more than six colors will ever be needed, the colors on the right side of the colormap are rarely used. If you want dramatically different colors for the watershed basins, you should vary the colors on the left side of the colormap. For instance, if the four colors for the map above should be purple, green, yellow, and red, the blue and orange nodes should be removed. The green and yellow nodes should be moved to the left, aligning with the 0.1 and 0.2 values. A new node should be added at a value of 0.3 with the color red. To determine the value a node is at, click on the node in the **Colormap Editor**. The *Value* is displayed at the top of the dialog.



Setting a variety of colors on the left side of the colormap will result in more differentiation in the watershed basin colors.

Line Properties

Click the ⊞ next to *Line Properties* to set the basin boundary line properties, including style, color, opacity, and width.

Pour Point Source

Pour points are the outlets of the stream systems and are calculated at all drainage intersections and where the drainage leaves the edge of the grid. The pour points can be generated automatically or selected from a file. **Surfer** uses the eight-direction pour point algorithm to calculate the flow direction at each grid node.

To change the *Pour point source*, click on the current option and select the desired option from the list. Available options are *None, Stream intersections,* and *File*. When set to *None*, the basins are not drawn or filled on the map. Only the streams appear. When set to *Stream intersections*, pour points are computed at the intersection of all tributary streams. When set to *File*, a .BLN file is imported that defines the locations where pour points should be located. The easiest way to create this file is to click on the map and click the Digitize command. Click on the map at the desired locations. Save the points to a .BLN file and load the file in the *Pour point file* option.

Too Many Pour Points

The maximum number of pour points is 65535. If you exceed the maximum number of pour points a "Too many pour points" or "Too many control points" error will be displayed. If the error occurs while *Pour point source* is set to *Stream intersections* , increase the *Threshold (cells)* value. If *Pour point source* is set to *File* , do not load a file with more than 65535 points.

Pour Point File

The *Pour point file* is used to define the pour points when the *Pour point source* is set to *File.* This .BLN file should contain point locations that are within the watershed map grid limits. To select the file, click on the button to the right of *Pour point source*. In the Open dialog, select the .BLN file and click *Open*.

Snap Pour Points

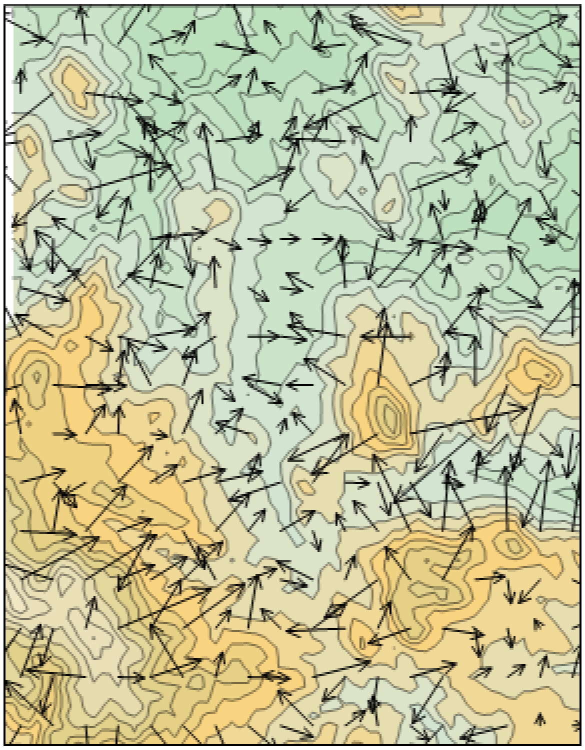
The *Snap pour points* option is used when the *Pour point source* is set to *File*. A common problem with using digitized points to define the pour points is locating the point slightly off the desired location, such as on the side of a hill. Checking this option moves the points in the *Pour point file* to if any of the eight neighboring grid node cells have a higher flow value. This allows more flow and better delineation of the basins.

Chapter 16 - Vector Maps

1-Grid Vector Map

Vector map information, direction and magnitude, can be derived from one grid. The arrow symbol points in the "downhill" direction and the length of the arrow depends on the magnitude, or steepness, of the slope. A vector is drawn at each grid node unless some nodes are skipped by changing the *Frequency* setting on the Symbol page in the vector map properties.

For example, consider a grid containing elevation information. If water were poured over the surface, the direction arrows would point in the direction water flows - from high elevation to low elevation. Magnitude is indicated by arrow length. In the water flow example, the steeper slopes would have longer arrows.



This vector map is overlaid on a filled contour map.
Longer arrows indicate areas of steeper slope.

Creating a 1-Grid Vector Map

To create a 1-grid vector map:

1. Click the **Home | New Map | Specialty | 1-Grid Vector** command or the  button.
2. Select a grid file in the **Open Grid** dialog and click *Open*.

The map is automatically created with reasonable defaults.

Editing a 1-Grid Vector Map

To change the features of the vector map, click once on the vector map in the plot window or in the Contents window to select it. The properties are displayed in the Properties window.

Adding a Map Layer

When vector maps are created, they are independent of other maps in the plot window. For example, creating a vector map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the vector map on the contour map, select both maps by clicking the Select All command. Overlay the maps using the Map Tools | Map Tools | Overlay Maps command.

Alternatively, you can add the vector map directly to the existing contour map by creating the vector map using the **Home | Add to Map | Layer | 1-Grid Vector** command. This automatically adds the vector map to the contour map axes.

Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the vector map using the **Home | New Map | Specialty | 1-Grid Vector** command. This creates two separate maps. Click on the vector map, hold down the left mouse button, and drag the vector map into the contour map. A single map with two map layers, using one set of axes and scaling parameters is created.

Any number of vector maps can be combined with any other map layers.

1-Grid Vector Map Properties

1-grid vector map layers contain the following property pages:

Data
Symbol
Scaling
Layer
Coordinate System
Info (Grids)

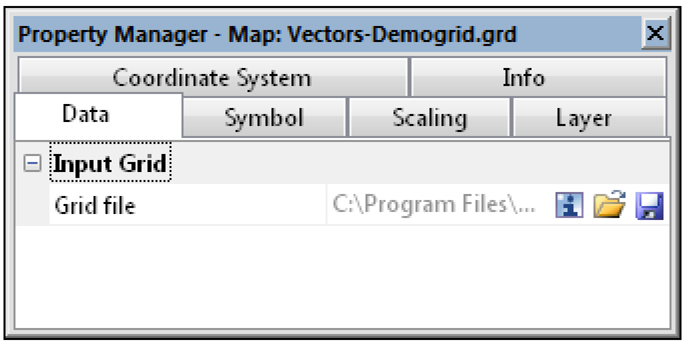
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info (Objects)

1-Grid Vector Map Data Properties

To edit a 1-grid vector map, click once on the vector map to select it. The properties for the vector map are displayed in the Properties window. The vector map **Data** page contains the following options:



*Specify the input grid file on the **Data** page for a 1-Grid vector map.*

Input Grid File

The *Grid file* displays the path and file name for the grid file used for the map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Grid Information

The button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the  button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the 1-grid vector map. Select the new grid file and click *Open* to reference the new file.

Coordinate System Note

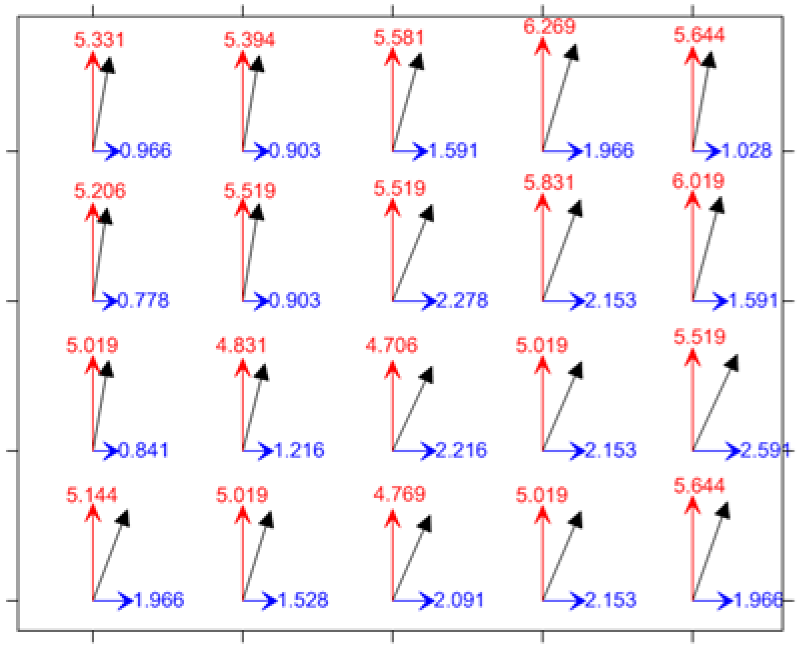
Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the  button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

Click the  button to display the **Save Grid As** dialog. This allows the grid file used for the 1-grid vector map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file. If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, the Export Options dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

2-Grid Vector Map

2-grid vector maps use two separate grid files to determine the vector direction and magnitude. The grids can contain Cartesian or polar data. With Cartesian data, one grid consists of X component data and the other grid consists of Y component data. With polar data, one grid consists of angle information and the other grid contains length information.



This map shows two Cartesian grid files. The blue vectors are X component grid values. The red vectors are Y component grid values. The black is the vector that Surfer plots when the two grids are specified when creating a 2-grid Cartesian vector map.

Creating a 2-Grid Vector Map

To create a 2-grid vector map:

1. Use the **Home | New Map | Specialty | 2-Grid Vector** command or the button.
2. Select the first grid file in the **Open Grid** dialog. This is the *X component* or *Angle component* grid file. Click *Open*.
3. Select the second grid file in the **Open Grid** dialog. This is the *Y component* or the *Length component* grid file. Click *Open*.

The map is automatically created with reasonable defaults.

Editing an Existing 2-Grid Vector Map

To change the features of the 2-grid vector map, click once on the 2-grid vector map in the plot window or the Contents window. The properties are displayed in the Properties window.

Adding a Map Layer

When vector maps are created, they are independent of other maps in the plot window. For example, creating a vector map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the vector map on the contour map, select both maps by clicking the Select All command. Overlay the maps using the Map Tools | Map Tools | Overlay Maps command.

Alternatively, you can add the vector map directly to the existing contour map by creating the vector map using the **Home | Add to Map | Layer | 2-Grid Vector** command. This automatically adds the vector map to the contour map axes.

Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the vector map using the **Home | New Map | Specialty | 2-Grid Vector** command. This creates two separate maps. Click on the vector map, hold down the left mouse button, and drag the vector map into the contour map. A single map with two map layers, using one set of axes and scaling parameters is created.

Any number of vector maps can be combined with any other map layers.

2-Grid Vector Map Properties

The 2-grid vector map layer properties contains the following pages:

Data
Symbol
Scaling
Layer
Coordinate System
Info (Grids)

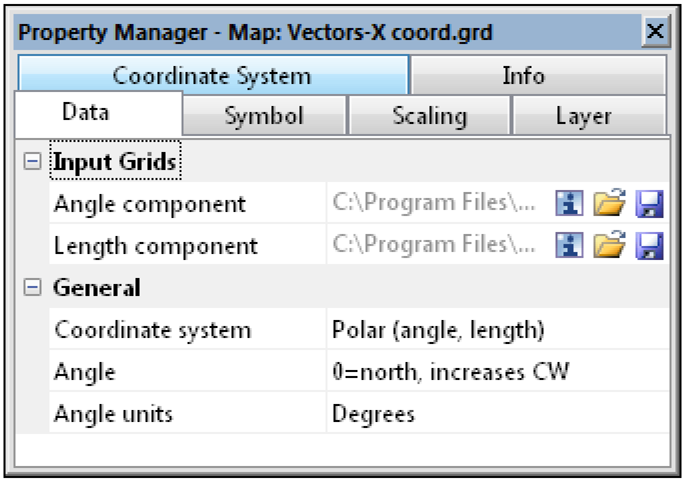
Map Properties

The map properties contains the following pages:

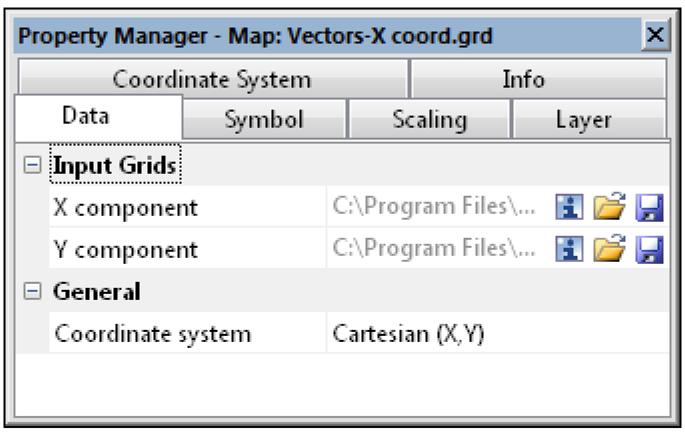
View
Scale
Limits
Frame
Coordinate System
Info (Objects)

2-Grid Vector Map Data Properties

To edit a 2-grid vector map, click once on the vector map to select it. The properties for the vector map are displayed in the Properties window. The vector map **Data** page contains the following options:



*Specify the Coordinate System, Angle, Units, and two grid files on the **Data** page for a 2-grid polar vector map.*



*Specify the X component and Y component on the **Data** page for a 2-grid Cartesian vector map.*

Input Grid Files

The *Input Grid* section displays the path and file name for the grid files used for the map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

For a *Polar (angle, length)* 2-grid vector map, the grids contain *Angle component* and *Length component* grids. The *Angle component* grid contains direction information. The vector at each grid node will be pointed in the direction of the

Angle component. The *Length component* grid contains the length of the vector. The vector at each grid node will be as long as the value in the *Length component*.

For a *Cartesian (X, Y)* 2-grid vector map, the grids contain *X component* and *Y component* grids. The *X component* grid contains the length of the vector in the X direction. The *Y component* grid contains the length of the vector in the Y direction. The direction of each vector will be found by adding the two grid files. The magnitude is found using the Pythagorean theorem.

Grid Information

The button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the 2-grid vector map. Select the new grid file and click *Open* to reference the new file.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

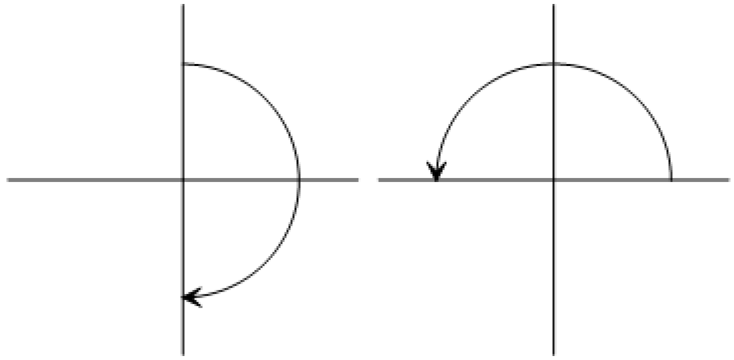
Click the button to display the **Save Grid As** dialog. This allows the grid file used for the 2-grid vector map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file. If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, the Export Options dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Coordinate System

Cartesian (X, Y) or Polar (Angle, Length) can be used in a two-grid vector map. To change the type of 2-grid vector map, click on the existing *Coordinate System* and select the desired coordinate system from the list. The *Cartesian (X, Y)* coordinate system requires one grid file with X magnitude components and the second grid file with Y magnitude components. The *Polar (Angle, Length)* coordinate option requires one grid with angle (direction) information and the second grid with length (magnitude) direction for the vectors. The *Length Component Grid File* cannot contain negative values.

Angle

When working with polar coordinates, set the location of the zero value to the north or east in the *Angle* option. Setting *Angle* to *0=north, increases CW,* creates a polar system with increasing angles as shown in the left image below. Setting *Angle to 0=eash, increases CCW*, creates a polar system with increasing angles as shown in the right image below.



This image shows the two coordinate Angle orientations. In the image on the left, zero = North. Numbers increase clockwise. In the image on the right, zero = East. Numbers increase counter-clockwise.

Angle Units

When working with polar coordinates,you can set the type of angle units used in the grid file. Set the *Angle units* to either *Degrees* or *Radians* . This is the unit of the data in the *Angle component* grid.

Cartesian Data

2-grid vector maps created with Cartesian data require that one grid contains X component data and a Y component data grid. The data from the two grids are combined to produce vector direction and magnitude. The components can be in either negative or positive space. The direction and magnitude of the resulting vectors are determined from the two grids as shown in the following example.

Example

The lower right corner's X component is 3.5 and the Y component is 3.5. To simplify the math, the origin of both are zero. The direction of the resulting vector is found by adding the two vectors together:

The magnitude of the vector is:

$$C = (C_1, C_2) = (3.5, 3.5)$$
$$|C| = \sqrt{C_1^2 + C_2^2}$$
$$|C| = \sqrt{3.5^2 + 3.5^2}$$
$$|C| = \sqrt{24.5}$$
$$|C| = 4.95$$

Polar Data

2-grid vector maps created with polar data require that one grid contains angle (direction) data and the other contains length (magnitude) data.

Negative Data and Polar Grids

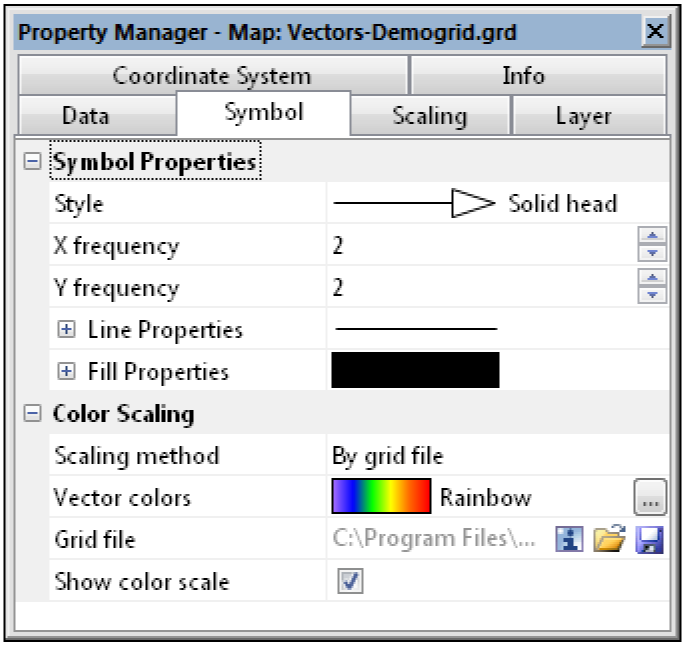
Magnitude values cannot be negative. If you have chosen a gridding technique that displays trends in the data, such as Kriging, you may end up with negative values in your grid even if the original data are all positive.

When creating a grid, you can use the *Grid Z Limits* section of the Grid Data dialog to limit the grid Z values to a specific range. Set the *Minimum* option to *Custom* and the value to 0 to ensure negative values are not created in the gird. The *Grid Z Limits* options can be used with any *Gridding Method*.

Alternatively, regridding the magnitude data and decreasing the grid spacing is one way to make the grid honor the data better. If you regrid the magnitude data you must also regrid the angle data using the same output grid geometry. If you do not want to change the output grid geometry of the grids or if decreasing the grid nodes is not completely eliminating the negative data, then you can use Grid | Math to set all negative values equal to zero. Another alternative is to use a gridding method that does not trend the data as much (Triangulation with Linear Interpolation or Natural Neighbor).

Vector Map Symbol Properties

To edit a 1-grid vector map or 2-grid vector map, click once on the vector map to select it. The properties for the vector map are displayed in the Properties window. The vector map **Symbol** page contains the following options for the vector symbol drawn at each grid node:



*Change vector symbol properties in the **Properties** window on the **Symbol** page.*

Style

The *Style* list contains several symbols to use to display the vectors. To change the symbol *Style* , click on the existing style. A list appears. Click on a new symbol from the list and the map automatically updates to show the new symbol.

Frequency

The *X frequency* and *Y frequency* settings control the number of symbols displayed on the vector map. If the frequency is set to one, every symbol is displayed on the vector map. If the frequency is set to two, every other symbol is displayed. If the frequency is set to three, every third symbol is displayed on the vector map, and so on. The symbols are located at grid nodes. So, skipping symbols skips vectors at the specified number of grid nodes in the X or Y direction. If there are too many symbols on the vector map, increase the *X frequency* and *Y frequency* numbers until the map becomes legible. To change the frequency, highlight the existing value and type a new number. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click the buttons to increase or decrease the frequency.

Line Properties

Click on the ⊞ next to Line Properties to open the line properties section. The *Line Properties* control the line on the outside of a filled symbol. If the *Scaling method* is set to *By magnitude* or *By grid file* , the *Line Properties* indicated in this section are not used.

Fill Properties

Click on the ⊞ next to Fill Properties to open the fill properties section. The *Fill Properties* control the color and patter of the area inside a filled symbol. If the *Scaling method* is set to *By magnitude* or *By grid file*, the *Fill Properties* indicated in this section are not used.

Color Scaling

The *Color Scaling* section contains options for gradationally coloring the symbols.

Scaling Method

The *Scaling method* determines how to gradationally color the vector map symbols. To change the *Scaling method*, click on the current method. A list appears. Select the desired method and the map is updated.

- When the *Scaling method* is set to *Disabled*, the *Line Properties* and *Fill Properties* specified in the *Symbol Properties* section are used to color the vector symbols.
- When the *Scaling method* is set to *By magnitude*, the vectors are gradationally colored, by the length of the vector.
- When the *Scaling method* is set to *By grid file*, the vectors are gradationally colored by the value in another grid file. The grid file must contain the same output grid geometry as the vector map grid files. This means the number of rows and columns and the grid range must be the same for all grids. The vectors are colored according to the corresponding Z value in the color grid file.

Vector Colors

The *Vector colors* option defines the colormap used to fill the vector map symbol when *Scaling method* is set to either *By magnitude* or *By grid file*. Change the color by clicking the existing color bar next to *Vector colors*. Select the new colormap from the list. If the desired color map is not listed, click the ... button to the right of the selected colormap. The Colormap Editor appears. Make any changes and click *OK* to see the change on the map. The range for the colors is also set from the **Colormap Editor**. To set the colormap to use a logarithmic scale, click the ... button to the right of the selected colormap. Check the *Logarithmic scaling* option and click *OK*. The color map is automatically updated to show logarithmic scaling.

Grid File

The *Grid file* displays the path and file name for the grid file used to color the vector symbols when the *Scaling method* is set to *By grid file*. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Grid Information

The button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the vector symbol colors. Select the new grid file and click *Open* to reference the new file. If a new unreferenced file is selected, select the appropriate coordinate system in the **Assign Coordinate System** dialog and click *OK*.

Save File

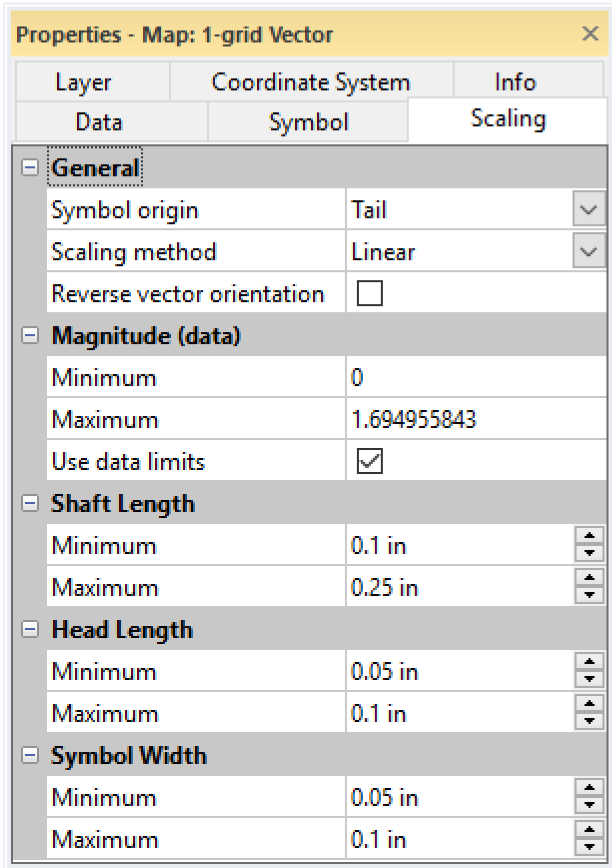
Click the button to display the **Save Grid As** dialog. This allows the grid file used to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file.

Show Color Scale

The *Show color scale* option permits the display of a color scale bar adjacent to the map. The color scale bar indicates the colors assigned to the vector symbols on the map, and the associated Z level values are displayed as labels on the color scale bar. When the *Show color scale* box is checked, the color scale bar is displayed. If the colormap is set to a logarithmic scale, the color scale bar will automatically display logarithmic scaling.

Vector Map Scaling Properties

To edit a 1-grid vector map or 2-grid vector map, click once on the vector map to select it. The properties for the vector map are displayed in the Properties window. The **Scaling** page contains placement and sizing options for vector symbols. Each symbol is located at a grid node.



Change vector scaling properties in the ***Properties*** *window on the* ***Scaling*** *page.*

Symbol Origin

The symbol is located on the grid node with the *Symbol origin* options. To change the *Symbol origin*, click on the existing option and select a new option from the list. The symbol can be placed at the *Tail*, at the *Center*, or at the *Head*. At the *Tail*, places the end of the vector symbol on the grid node. At the *Head* , places the tip of the arrow symbol at the grid node. At the *Center* places the center of the vector at the grid node.

Scaling Method

There are three ways to scale vectors listed in the *Scaling method* option. You can scale the symbols between the minimum and maximum data values linearly, logarithmically, or by square root. *Linear* scaling provides a better visual representation of symbols that, for the most part, are scaled in one dimension (such as arrows with varying shaft length). When scaling the arrows in two dimensions (symbol width and shaft length), *Square root* or *Logarithmic* scaling displays the arrows more effectively. To change the *Scaling method* , click the current option. A list is displayed. Click on the new option and the map automatically updates.

Reverse Vector Orientation

The direction the arrowhead points can be reversed by checking the *Reverse vector orientation* check box. By default, vectors point in the "downhill" direction. Checking the *Reverse vector orientation* points the vectors in the "uphill" direction.

Vector Magnitude (data)

Set the *Minimum* and *Maximum* data values for the vectors in the *Magnitude (data)* section. Check the *Use data limits* box to set the vector minimum and maximum to the grid minimum and maximum values. Setting a new *Minimum* and *Maximum* is useful in displaying a series of maps in which you would like all of the vectors to be scaled the same even though the data minimum and maximum may differ.

To set the *Minimum* or *Maximum* value, highlight the existing value and type a new value. Values are in Z magnitude units.

Shaft Length

Set the range of the arrow shaft length in the *Shaft Length* section. The *Shaft Length* is the length of the arrow symbol from the tip of the tail to the tip of the arrow head. The *Minimum* value is the smallest shaft length displayed in the map at the *Minimum* value specified by the *Magnitude (data)* section. The *Maximum* value is the longest shaft length displayed in the map at the *Maximum* value specified by the *Magnitude (data)* section. Enter a new value into the boxes to change the length. Values are in page units. Values can be between 0 and 10 inches (0 and 25.4 centimeters).

Head Length

Set the range of the head length in the *Head Length* section. The *Head Length* is the length of the arrow head portion of the arrow symbol. The *Minimum* value is the smallest arrow head displayed in the map at the *Minimum* value specified by the *Magnitude (data)* section. The *Maximum* value is the longest arrow head displayed in the map at the *Maximum* value specified by the *Magnitude (data)* section. Enter a new value into the boxes to change the length. Values are in page units. Values can be between 0 and 10 inches (0 and 25.4 centimeters). If the *Head Length* values match the *Shaft Length* values, the arrow heads will be the entire length of the vector map symbol.

Symbol Width

Set the range of the symbol width in the *Symbol Width* section. The *Symbol Width* is the width of the arrow head portion of the arrow symbol at the widest point. The *Minimum* value is the smallest arrow head displayed in the map at the *Minimum* value specified by the *Magnitude (data)* section. The *Maximum* value is the longest arrow head displayed in the map at the *Maximum* value specified by the *Magnitude (data)* section. Enter a new value into the boxes to change the length. Values are in page units. Values can be between 0 and 5 inches (0 and 12.7 centimeters).

Clipping Symbols on Vector Maps

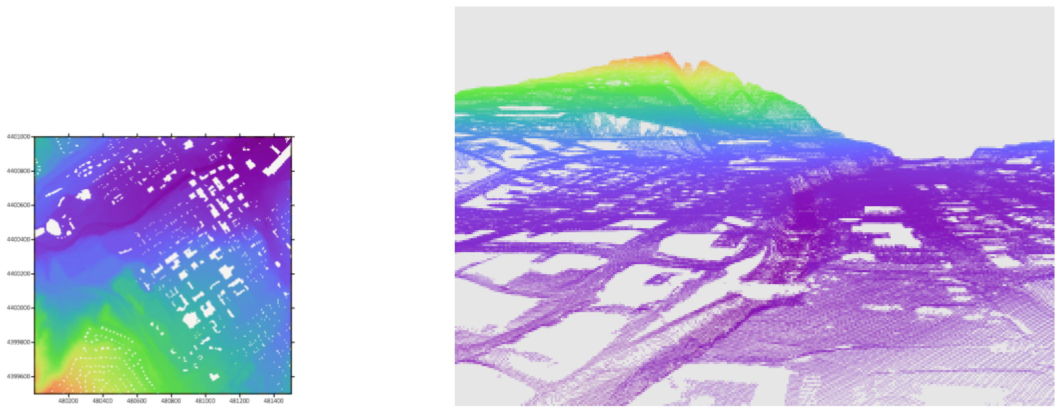
In a 1-grid vector map or a 2-grid vector map, any symbols at the edge of the map are truncated at the map limits. This results in a partial vector being drawn when the vector extends beyond the map limits. If a partial vector is drawn, but a full vector is desired, a change in the map limits is required. Follow these steps to change the map limits:

1. Create a vector map using the **Home | New Map | Specialty | 1-Grid Vector Map** or **Home | New Map | Specialty | 2-Grid Vector Map** command.
2. Click on the map in the Contents window or plot window to select it.
3. Click on the Limits page in the Properties window.
4. If a partial vector is displayed at the bottom or left of the map, decrease the values in the *xMin* and *yMin* boxes to display the entire vector.
5. If a partial vector is displayed at the top or right of the map, increase the value in the *xMax* and *yMax* boxes to display the entire vector.

Chapter 17 - Point Cloud Maps

Point Cloud Map

Point cloud maps display LiDAR data as points at XY locations. Color is assigned to the points by elevation, intensity, return number, or classification. The point cloud layer includes commands for modifying, classifying, and exporting points. Point cloud layers are displayed in the 3D View as three-dimensional points. LiDAR data can be combined from multiple files and filtered with various criteria when creating a point cloud map. The point cloud layer requires one or more LAS/LAZ LiDAR data files to be selected for the input files.



This example shows a point cloud layer in the plot window (left) and 3D view (right). The colormap is applied by elevation.

Creating a Point Cloud Map

To create a point cloud map:

1. Click the **Home | New Map | Specialty | Point Cloud** command or the button.
2. Select one or more LiDAR files in the **Open** dialog and click *Open*.
3. Add or remove files, set the coordinate system, and apply filters to the points in the Import Points dialog, if desired, and click *OK*.

The map is automatically created with reasonable defaults. The point cloud layer name includes the input file name in the **Contents** window, e.g. *Point Cloud - myData.las* . If the point cloud layer consists of multiple input files, a "+" is appended to the layer name, e.g. *Point Cloud - myData.las+* .

Editing an Existing Point Cloud Map

To change the features of the point cloud map, open the point cloud map properties by clicking on the point cloud map in the plot window or clicking on the point cloud layer name in the **Contents** window. The properties are displayed in the **Properties** window.

To classify, modify, or export points or grids, open the Point Cloud ribbon tab by clicking the point cloud map in the plot window or clicking the point cloud layer name in the **Contents** window. The **Point Cloud** tab is added to the ribbon and automatically selected.

Adding a Map Layer

When point cloud maps are created, they are independent of other maps in the plot window. For example, creating a point cloud map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the point cloud map data points on the contour map, select both maps by clicking the Home | Selection | Select All command. Overlay the maps using the Overlay Maps command.

Alternatively, you can add the point cloud map directly to the existing contour map by creating the point cloud map using the **Home | Add to Map | Layer | Point Cloud** command. This automatically adds the point cloud map to the contour map axes.

Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the point cloud map using the **Home | New Map | Specialty | Point Cloud** command. This creates two separate maps. Click on the point cloud layer in the Contents window, hold down the left mouse button, and drag the point cloud layer into the contour map. A single map with two map layers, using one set of axes and scaling parameters is created.

Point cloud maps cannot be added as a map layer to 3D wireframe maps, but a point cloud map can be added to any other map layer, including 3D surface maps.

Saving a Point Cloud Map

Surfer embeds most grid and vector data in the Surfer SRF file. However, **Surfer** does not embed the point cloud data in the SRF file. **Surfer** saves a path to the source data, the Import Points options, and point cloud layer settings. When sharing an SRF file with a point cloud layer with another user, provide the user with the LAS source data files. Consider storing LAS source data in a network location if you often share SRF files with point cloud layers.

When opening an SRF file that includes a point cloud layer, **Surfer** first checks the saved file path to locate the data. If the data is not found in the same path with the same name, **Surfer** then checks paths relative to the SRF file location for the LAS source data. If the source data is still not found, the **Import Points** dialog is displayed with missing data files highlighted. Click *Update File Path* in the **Import Points** dialog to locate the missing source data files.

When opening an SRF file with a point cloud layer, the point cloud data loading progress is displayed in place of the layer until the source data has finished loading. You can begin working on your map while the point cloud data loads.

Point Cloud Layer Properties

The point cloud layer properties contains the following pages:

General
Layer
Coordinate System
Info

Map Properties

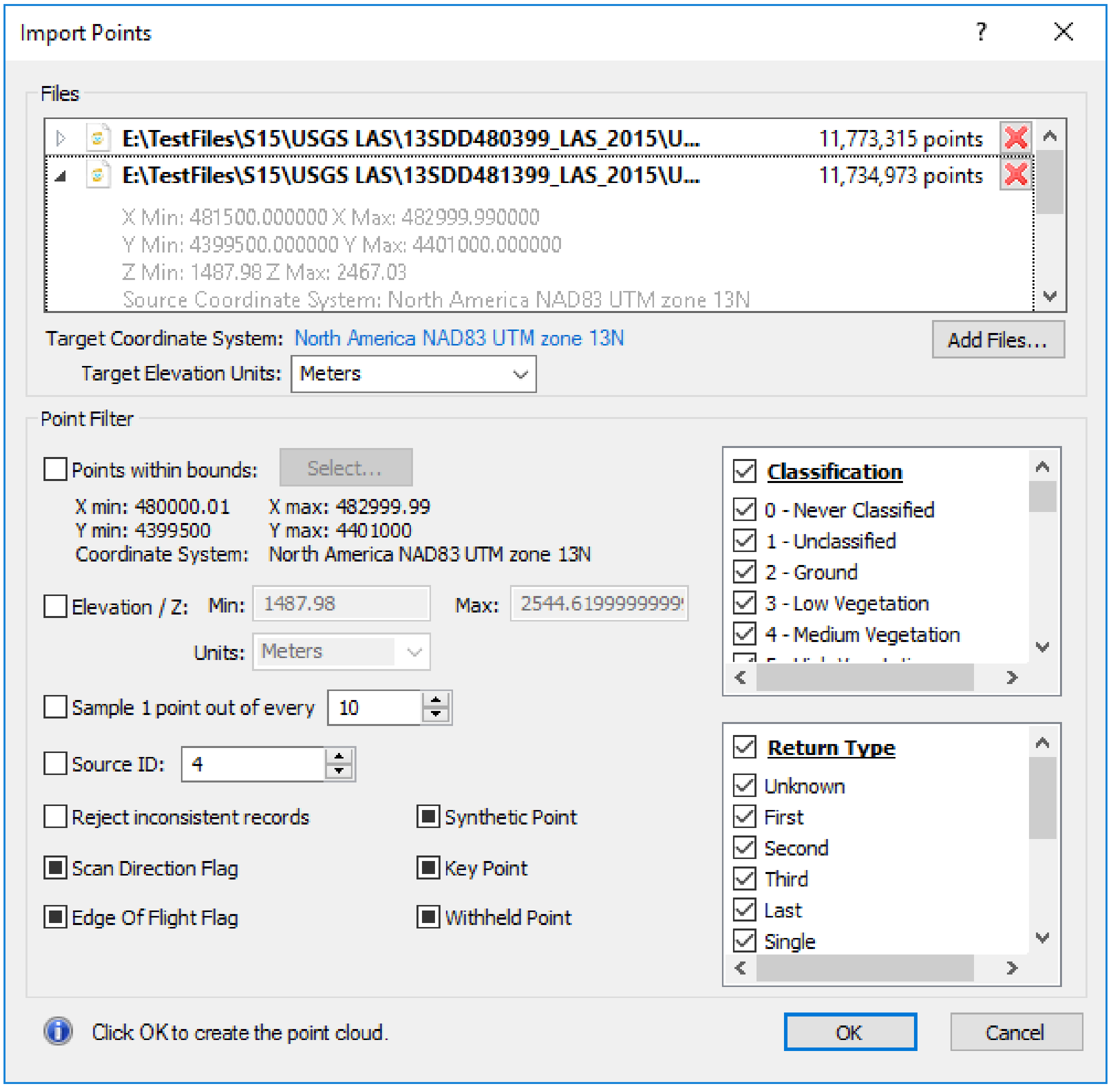
The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info

Import Points Dialog

The **Import Points** dialog specifies the input LAS/LAZ files, the source file elevation units, the layer source coordinate system, and various data filters for the input data. The **Import Points** dialog is displayed when creating a point cloud layer or when changing the *Sources* for the layer in the General page.

The *Import Points* dialog is also displayed when opening an SRF file and one or more LAS source data files were not found. Missing files are highlighted in red and include the error message, "Unable to open file <filename>." Click *Update File Path* to update the path to the missing data files, or click  to remove the missing files from the *Files* list.



*Select the source data and point filtering for the point cloud layer in the **Import Points** dialog.*

Files

The *Files* section lists the source data files. The *Files* section includes commands for adding and removing source files, specifying the data elevation units, and setting the output layer coordinate system.

File List

The *File list* displays the source LAS/LAZ files, the number of points in the file, the file XYZ extents, and the file coordinate system. Click the ▹ to expand the file details in the file list. Click the ◢ to collapse the file details in the file list. When a source file is collapsed, only the file name and number of points are displayed.

Source files with errors will be highlighted in red. The error description is displayed below the file name when there is a source file error. Errors must be resolved before the *OK* button is enabled and the point cloud layer can be created.

File Extents

The spatial extents of the source data is displayed below the file name. The extents include the *X Min*, *X Max*, *Y Min*, *Y Max*, *Z Min*, and *Z Max*.

Source Coordinate System

The *Source Coordinate System* is displayed below the file spatial extents. The source data files can have different coordinate systems. All coordinates will be converted to the *Target Coordinate System*.

The *Source Coordinate System* will be set automatically if the coordinate system information is available in the data file. If the *Source Coordinate System* is incorrect, click *Set* to specify the source data coordinate system.

When the *Source Coordinate System* is unknown for one or more files, the *Set all unreferenced* option is available. Click *Set all unreferenced* to specify the same coordinate system for all source files that are currently unreferenced.

Source Elevation Units

The *Source Elevation Units* are displayed below the *Source Coordinate System*. The source data files can have different elevation units. All elevation units will be converted to the *Target Elevation Units*.

The *Source Elevation Units* may be determined automatically if the information is available in the source file. Click *Set* to change the elevation units to the appropriate units if the *Source Elevation Units* is incorrect.

When one or more source file *Elevation Units* are *Unknown*, the *Set all unknown* option is available. Click *Set all unknown* to specify the same *Elevation Units* for all source files that currently have their *Elevation Units* set to *Unknown*.

Remove Files

Click the  to the right of the points count to remove a file from the *File list*.

Add Files

Click *Add files* to add new files to the *File List* with the Open dialog. More than one file from the same directory can be selected. Only LiDAR data files can be selected for creating a point cloud layer.

Target Coordinate System

The *Target Coordinate System* is the coordinate system to which all the source file coordinates are transformed. This becomes the point cloud layer source coordinate system when the point cloud layer is created.

Target Elevation Units

Select the *Target Elevation Units* from the *Target Elevation Units* list. The source elevation values will be converted to the corresponding value in the *Target Elevation Units* in the point cloud layer.

Point Filter

The points in the input files can be filtered by various criteria in the *Point Filter* section.

Boundary Filter

The *Points within bounds* filter limits the imported points by XY extents. Select the *Points within bounds* check box to apply the *Points within bounds* filter. Click *Select* to set the bounds in the Select Bounds dialog. The minimum and maximum XY values and their coordinate system are displayed below *Points within bounds*.

Elevation/Z Filter

The *Elevation / Z* filter limits the imported points to a specific elevation/Z value range. Select the *Elevation / Z* check box to apply the *Elevation / Z* filter. Specify the minimum Z value to be imported in the *Min* field. Specify the maximum Z value to be imported in the *Max* field.

Select the units for the *Min* and *Max* values from the *Units* list. The *Source Elevation Units* must be specified in the *Files* list to use the *Elevation / Z Units* . If any input file has *Source Elevation Units* set to *Unknown* , then the *Elevation / Z Units* must also be set to *Unknown* . When set to *Unknown* , the Z values are compared directly to the *Min* and *Max* values without any unit conversions.

Sample Filter

The *Sample 1 point out of every* filter limits the imported points to every nth point. Select the *Sample 1 point out of every* check box to apply the sample filter. For example to import every other point, type 2 in the *Sample 1 point out of every* field. Type 10 to import every tenth point. Click the buttons to increase or decrease the value, or type a value directly in the *Sample 1 point out of every* field.

Source ID Filter

The point source ID value indicates the file from which the point originated. The point source ID is a value from 0 to 16384 inclusive. Select the *Source ID* check box to apply a source ID filter to the data. Type the desired source ID number in the *Source ID* field to limit the import to points with the specified point source ID.

Reject Inconsistent Records

The *Reject inconsistent records* filter removes all invalid or inconsistent data from import. Select the *Reject inconsistent records* check box to reject invalid or inconsistent data.

Scan Direction Flag

The scan direction flag indicates whether the point was located on a forward scan sweep or backward scan sweep. The *Scan Direction Flag* filter limits the imported points to forward scan sweep data, backwards scan sweep data, or both. Click the *Scan Direction Flag* check box to change the filter state.

A check mark ☑ indicates only forward scan sweep points are imported. A cleared check box ☐ indicates only backwards scan sweep points are imported. A black square ▣ indicates no scan direction filtering is applied. No *Scan Direction Flag* filter is applied by default.

Edge of Flight Flag

The edge of flight flag indicates whether the point is located at the farthest reach of the scanner during the sweep. The *Edge of Flight Flag* filter limits the imported points to points with the edge of flight flag, points without the edge of flight flag, or both. Click the *Edge of Flight Flag* check box to change the filter state.

A check mark ☑ indicates only points with an edge of flight flag are imported. A cleared check box ☐ indicates only points without an edge of flight flag are imported. A black square ▣ indicates no edge of flight filtering is applied. No *Edge of Flight Flag* filter is applied by default.

Synthetic Point

The synthetic point flag indicates the point was created by a means other than LiDAR collection, such as a point created via interpolation in post-processing software or digitized from a photogrammetric stereo model. The *Synthetic Point* filter limits the imported points to points with the synthetic point flag, points without the synthetic point flag, or both. Click the *Synthetic Point* check box to change the filter state.

A check mark ☑ indicates only points with a synthetic point flag are imported. A cleared check box ☐ indicates only points without a synthetic point flag are imported. A black square ▣ indicates no synthetic point filtering is applied. No *Synthetic Point* filter is applied by default.

Key Point

The key point flag indicates the point is considered a model key point and should not be removed. The *Key Point* filter limits the imported points to points with the key point flag, points without the key point flag, or both. Click the *Key Point* check box to change the filter state.

A check mark ☑ indicates only points with a key point flag are imported. A cleared check box ☐ indicates only points without a key point flag are imported. A black square ▣ indicates no key point filtering is applied. No *Key Point* filter is applied by default.

Withheld Point

The withheld point flag indicates the point is considered suspect or redundant and should not be included in processing. The *Withheld Point* filter limits the imported points to points with the withheld point flag, points without the withheld point flag, or both. Click the *Withheld Point* check box to change the filter state.

A check mark ☑ indicates only points with a withheld point flag are imported. A cleared check box ☐ indicates only points without a withheld point flag are imported. A black square ▣ indicates no withheld point filtering is applied. No *Withheld Point* filter is applied by default.

Classification Filter

The *Classification* filter limits the imported points to only the selected classes. Select the *Classification* check box to import all classes. Clear the *Classification* check box to clear all the classes. Select individual classes for import by selecting the check boxes next to the desired classes.

Return Type Filter

The *Return type* filter limits the imported points to only the specified return types. The return types are defined by the return number and number of returns from the pulse. For example, *Single* selects all points with a return number of 1 and a number of returns of 1. The following table defines the *Return type* options.

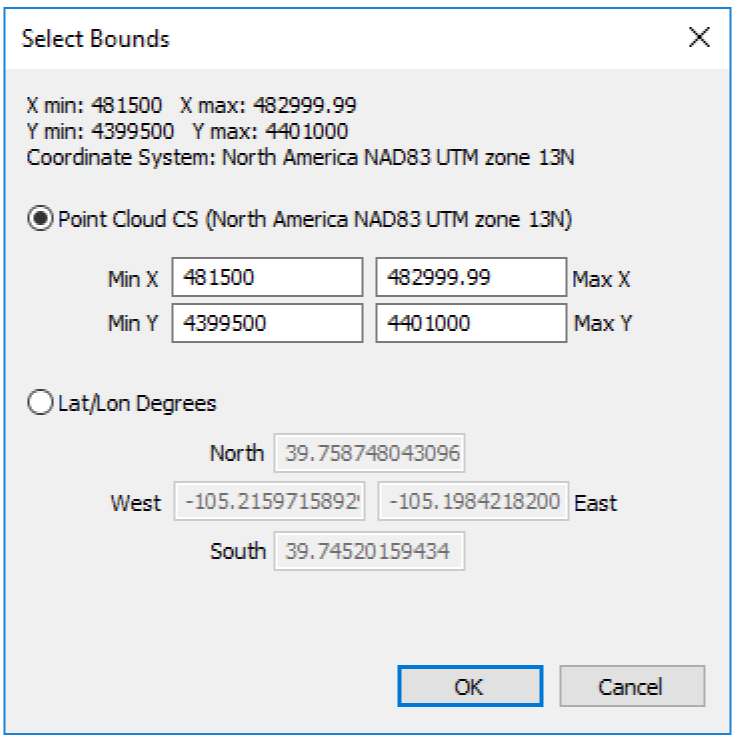
Option	**Definition**
Unknown	The return number is unknown
First	Return Number = 1
Second	Return Number = 2
Third	Return Number = 3
Last	Return Number = Number of Returns
Single	Return Number = Number of Returns = 1 (e.g. 1st of 1)
First of Many	Return Number < Number of Returns AND Return Number = 1 (e.g. 1st of 3)

Second of Many	Return Number < Number of Returns AND Return Number = 2 (e.g. 2nd of 4)
Third of Many	Return Number < Number of Returns AND Return Number = 3 (e.g. 3rd of 4)
Last of Many	Return Number = Number of Returns AND Return Number > 1 (e.g. 2nd of 2, 3rd of 3, etc.)

Select the *Return type* check box to import all return types. Clear the *Return type* check box to clear all the return types. Select individual return types for import by selecting the check boxes next to the desired return type.

Select Bounds Dialog

The **Select Bounds** dialog is displayed when defining the boundary filter in the Import Points dialog or the boundary criteria in the Select by Criteria dialog. The data filtering or selecting bounds can be specified in the **Import Points** *Target Coordinate System* or in latitude and longitude degrees.



*Filter the data by XY extents with the **Select Bounds** dialog.*

Input Bounds Information

The complete extents of the input data is displayed at the top of the **Select Bounds** dialog: *X min*, *X max*, *Y min*, and *Y max*. The *Coordinate System* indicates the target coordinate system of the input LAS data in the **Import Points** dialog.

Point Cloud Coordinate System

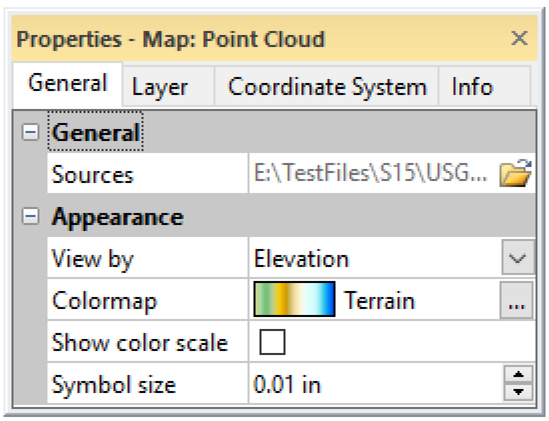
Select *Point Cloud CS* to limit the imported points to, or select the points within, bounds defined in the same coordinate system as the point cloud layer. Type the desired extents in the *Min X*, *Max X*, *Min Y*, and *Max Y* fields.

Lat/Lon Degrees

Select *Lat/Lon Degrees* to limit the imported points to, or select the points within, bounds defined in World Geodetic System 1984 latitude and longitude coordinates. Type the desired coordinates in the *North*, *West*, *East*, and *South* fields.

Point Cloud Layer General Properties

The **General** page of the Properties window contains the properties for defining the source data and modifying the point cloud layer appearance.



*Set the point cloud layer appearance properties in the **General** page.*

Sources

The *Sources* property lists the source LiDAR data files. Click  to open the Import Points dialog to add or remove source files or change the point filters for the source data.

Appearance

The *Appearance* section includes properties for modifying the point cloud layer appearance in the plot window and 3D view.

View By

The *View by* property specifies whether the *Colormap* is applied to the points by *Elevation*, *Intensity*, *Return number*, or *Classification*.

- *Elevation* applies point color by elevation value. The default colormap is *Terrain*.
- *Intensity* applies point color by intensity value. The default colormap is *GrayScale*.
- *Return number* applies point color by return number. The default colormap is *GrayScale*.
- *Classification* applies point color by classification value. A custom colormap is used by default. See the LAS LiDAR Binary File Description for a list of classes and colors.

Click the current selection and select the desired option from the list to change the *View by* property.

Colormap

The *Colormap* property specifies the colormap for the four *View by* options. Each *View by* option, *Elevation*, *Intensity*, *Return number*, and *Classification* uses a separate colormap. The previous *Colormap* settings are maintained when switching to a new *View by* option. To change the colormap for the current *View by* selection, click the current colormap and select a predefined colormap from the list. To modify the colormap, create a custom colormap, or change the data to color mapping, click ... in the *Colormap* field to open the Colormap Editor.

Show Color Scale

The *Show color scale* option displays a color scale bar adjacent to the map. The color scale bar indicates the colors assigned to the levels on the map, and the associated values are displayed as labels on the color scale bar. When the *Show color scale* box is checked, the color scale bar is displayed. If the colormap is set to a logarithmic scale, the color scale bar will automatically display logarithmic scaling.

The *Show color scale* option is not available when *View by* is set to *Classification*. When *Show color scale* is selected and the *View by* property is changed to *Classification* , the color scale is removed. Change the *View by* property to *Elevation* , *Intensity* , or *Return number* to enable the *Show color scale* property.

Symbol Size

The *Symbol size* property specifies the diameter of the points in the layer in page units. Type a value between 0 and 0.25 in (0 and 0.635 cm) in the *Symbol size* field to change the size of the point cloud points. Alternatively, click the to increase or decrease the point *Symbol size* . When the *Symbol size* is set to 0, the point symbols will be one pixel in size.

Point Cloud Tab Commands

When a point cloud layer is selected, the contextual **Point Cloud** ribbon tab is available and automatically selected. The **Point Cloud** tab contains commands for selecting, classifying, removing, and exporting points in the point cloud.

Criteria	Select points by various criteria: XY bounds, Z value, classification, return type, every n^{th}, etc.
Rectangle	Select points by drawing a rectangle
Polygon	Select points by drawing a polygon
Deselect All	Deselect all points
Reclassify	Change the classification for selected points
Remove Classifications	Remove the classification for select points
Crop to Selection	Remove all non-selected points from the layer
Remove Selected Points	Remove all selected points from the layer
Create Grid	Create a grid file from the point cloud layer
Export LAS/LAZ	Export the point cloud layer to an LAS or LAZ file

Point Cloud Tab Commands

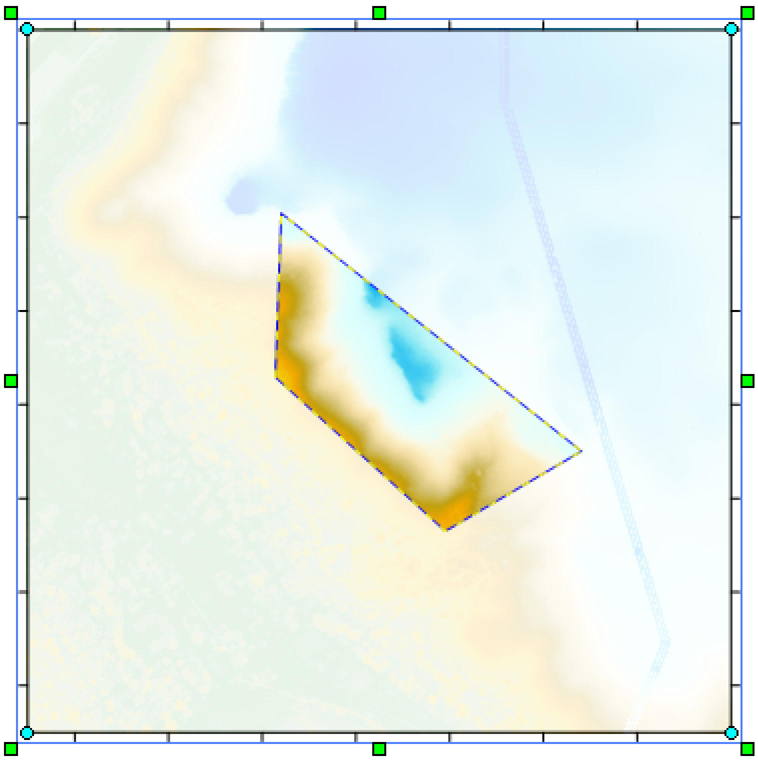
When a point cloud layer is selected, the contextual **Point Cloud** ribbon tab is available and automatically selected. The **Point Cloud** tab contains commands for selecting, classifying, removing, and exporting points in the point cloud.

Criteria	Select points by various criteria: XY bounds, Z value, classification, return type, every n^{th}, etc.
Rectangle	Select points by drawing a rectangle
Polygon	Select points by drawing a polygon
Deselect All	Deselect all points
Reclassify	Change the classification for selected points
Remove Classifications	Remove the classification for select points
Crop to Selection	Remove all non-selected points from the layer

Remove Selected Points	Remove all selected points from the layer
Create Grid	Create a grid file from the point cloud layer
Export LAS/LAZ	Export the point cloud layer to an LAS or LAZ file

Selecting Points

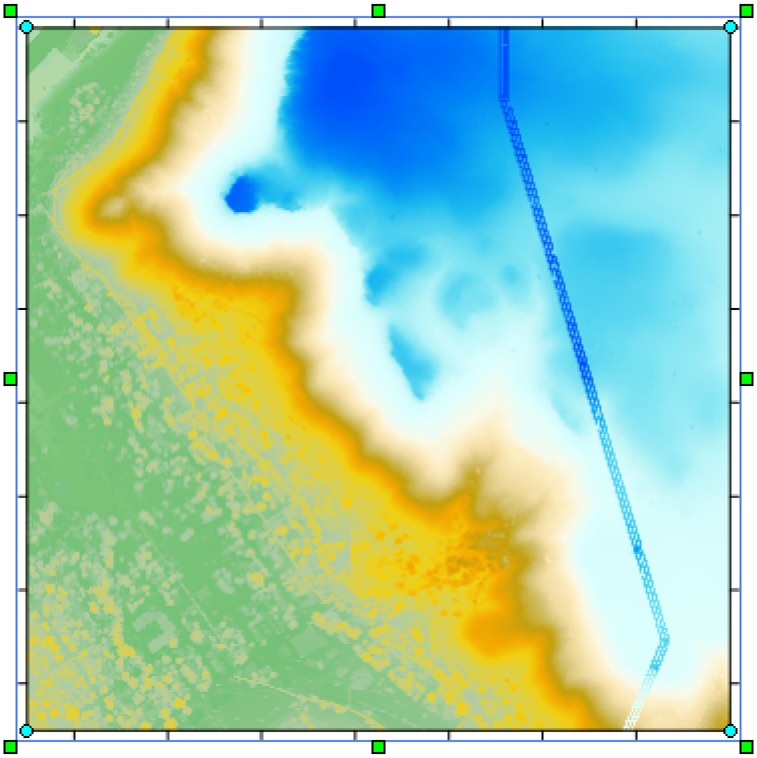
Points in the point cloud layer can be reclassified, removed, and/or exported. First the points must be selected. The Point Cloud ribbon tab includes three commands for selecting points in the point cloud layer: Criteria, Rectangle, and Polygon. When one or more points is selected the selected point(s) are surrounded by a blue dashed line. Regions of the map with no selected points are faded, and the region that includes the selected points is highlighted.



*The **Polygon** command was used to select points in the trapezoidal region in the center of the map.*

Once you have selected the desired points, use the Reclassify, Remove Classifications, Crop to Selection, or Remove Selected Points commands to modify the layer. To export the selected points, use the Export LAS/LAZ command and select the *Selected points only* option in the **Export Point Cloud** dialog.

Click the Deselect All command to deselect all points. The dashed blue line is removed, and the layer returns to its normal appearance.



The map returns to its normal appearance when no points are selected.

Criteria Select

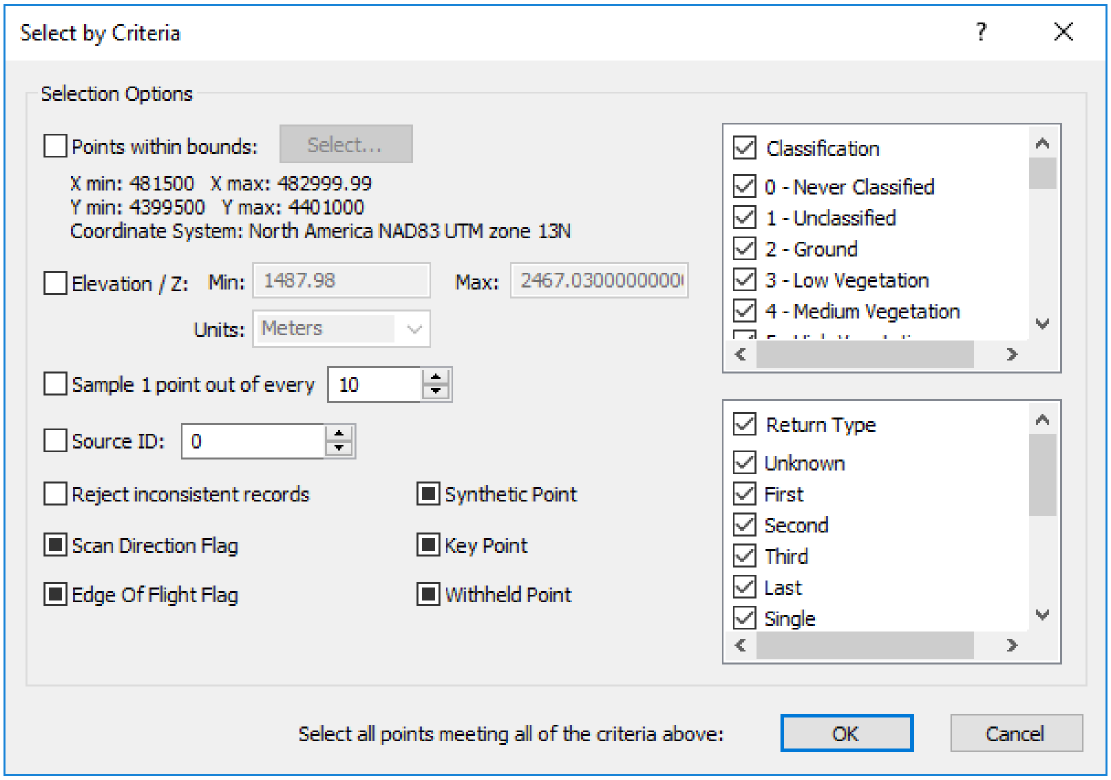
The **Point Cloud | Point Selection | Criteria** command selects points in the point cloud layer based on spatial, random sampling, flag, classification, or return type criteria. Click the **Point Cloud | Point Selection | Criteria** command or the  button to select points based on specific criteria. Specify the desired criteria in the **Select by Criteria** dialog, and then click *OK* to select the points in the point cloud layer that meet the criteria.

There are two indications of the point selection in the **Surfer** window.

- The number of selected points is displayed in the status bar.
- If the selected points fall within a specific region of the map, the selected points are highlighted, and the non-selected points are faded out. Note that with criteria select, the highlighted region may include points that are not selected due to the various selection criteria. If the selected points cover the extent of the map, no region will appear faded out.

Select by Criteria Dialog

The **Select by Criteria** dialog contains spatial, classification, return type, and flag criteria for selecting points in the point cloud layer.



Select points by criteria with the ***Select by Criteria*** *dialog.*

Boundary Criteria

The *Points within bounds* criteria selects the points by XY extents. Select the *Points within bounds* check box to apply the *Points within bounds* criteria. Click *Select* to set the bounds in the Select Bounds dialog. The minimum and maximum XY values and their coordinate system are displayed below *Points within bounds*.

Elevation/Z Criteria

The *Elevation / Z* criteria selects the points within a specific elevation/Z value range. Select the *Elevation / Z* check box to apply the *Elevation / Z* criteria. Specify the minimum Z value to be selected in the *Min* field. Specify the maximum Z value to be selected in the *Max* field. Select the units for the *Min* and *Max* values from the *Units* list.

Sample Criteria

The *Sample 1 point out of every* criteria selects the every nth point. Select the *Sample 1 point out of every* check box to apply the sample criteria. For example to select every other point, type 2 in the *Sample 1 point out of every* field. Type 10 to select every tenth point. Click the buttons to increase or decrease the value, or type a value directly in the *Sample 1 point out of every* field.

Source ID Criteria

The point source ID value indicates the file from which the point originated. The point source ID is a value from 0 to 16384 inclusive. Select the *Source ID* check box to apply a source ID criteria to the data. Type the desired source ID number in the *Source ID* field to select the points with the specified point source ID.

Reject Inconsistent Records

The *Reject inconsistent records* filter removes all invalid or inconsistent data from the selection. Select the *Reject inconsistent records* check box to reject invalid or inconsistent data.

Scan Direction Flag

The scan direction flag indicates whether the point was located on a forward scan sweep or backward scan sweep. The *Scan Direction Flag* criteria selects forward scan sweep data, backwards scan sweep data, or both. Click the *Scan Direction Flag* check box to change the criteria state.

A check mark ☑ indicates only forward scan sweep points are selected. A cleared check box ☐ indicates only backwards scan sweep points are selected. A black square ▣ indicates no scan direction criteria is applied. No *Scan Direction Flag* criteria is applied by default.

Edge of Flight Flag

The edge of flight flag indicates whether the point is located at the farthest reach of the scanner during the sweep. The *Edge of Flight Flag* criteria selects the points with the edge of flight flag, points without the edge of flight flag, or both. Click the *Edge of Flight Flag* check box to change the criteria state.

A check mark ☑ indicates only points with an edge of flight flag are selected. A cleared check box ☐ indicates only points without an edge of flight flag are selected. A black square ▣ indicates no edge of flight criteria is applied. No *Edge of Flight Flag* criteria is applied by default.

Synthetic Point

The synthetic point flag indicates the point was created by a means other than LiDAR collection, such as a point created via interpolation in post-processing software or digitized from a photogrammetric stereo model. The *Synthetic Point* criteria selects points with the synthetic point flag, points without the synthetic point flag, or both. Click the *Synthetic Point* check box to change the criteria state.

A check mark ☑ indicates only points with a synthetic point flag are selected. A cleared check box ☐ indicates only points without a synthetic point flag are selected. A black square ▣ indicates no synthetic point criteria is applied. No *Synthetic Point* criteria is applied by default.

Key Point

The key point flag indicates the point is considered a model key point and should not be removed. The *Key Point* criteria selects points with the key point flag, points without the key point flag, or both. Click the *Key Point* check box to change the criteria state.

A check mark ☑ indicates only points with a key point flag are selected. A cleared check box ☐ indicates only points without a key point flag are selected. A black square ▣ indicates no key point criteria is applied. No *Key Point* criteria is applied by default.

Withheld Point

The withheld point flag indicates the point is considered suspect or redundant and should not be included in processing. The *Withheld Point* criteria selects points with the withheld point flag, points without the withheld point flag, or both. Click the *Withheld Point* check box to change the criteria state.

A check mark ☑ indicates only points with a withheld point flag are selected. A cleared check box ☐ indicates only points without a withheld point flag are selected. A black square ▣ indicates no withheld point criteria is applied. No *Withheld Point* criteria is applied by default.

Classification Filter

The *Classification* filter selects points within the selected classes. Select the *Classification* check box to select all classes. Clear the *Classification* check box to clear all the classes. Select individual classes for selection by selecting the check boxes next to the desired classes.

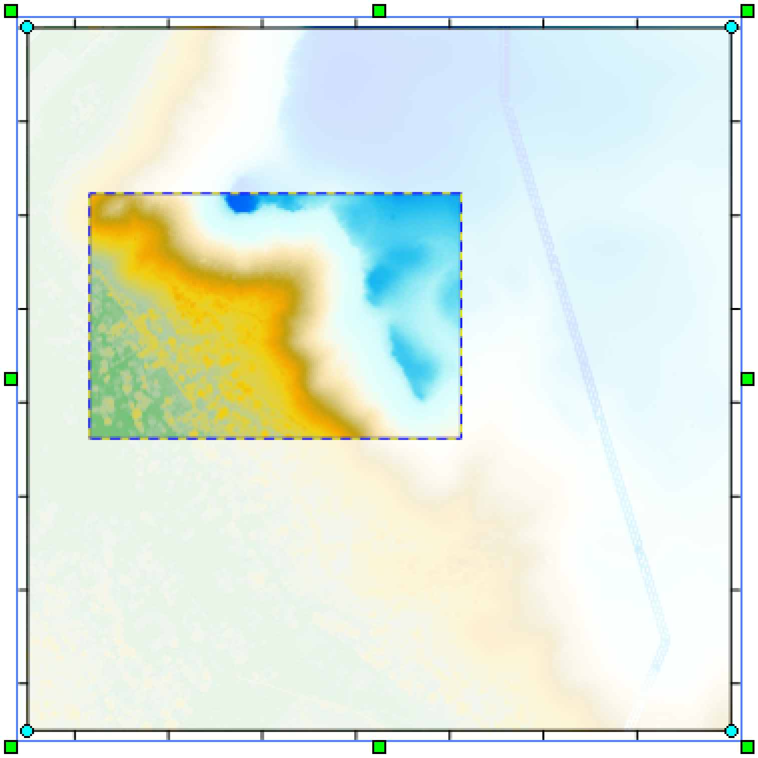
Return Type Filter

The *Return type* filter selects points within the specified return types. Select the *Return type* check box to select all return types. Clear the *Return type* check box to clear all the return types. Select individual return types for selection by selecting the check boxes next to the desired return type.

Rectangle Select

Click the **Point Cloud | Point Selection | Rectangle** command or the button to select points in the point cloud layer by drawing a rectangle. The cursor changes to a cross hair when the **Rectangle** command is selected.

1. Click the **Point Cloud | Point Selection | Rectangle** command.
2. Click and drag in the map to draw a rectangle around the points you wish to select.
3. Release the left mouse button to finish drawing the rectangle. The points within the rectangle are selected.



The selected points are indicated by the highlighted area and blue dashed line.

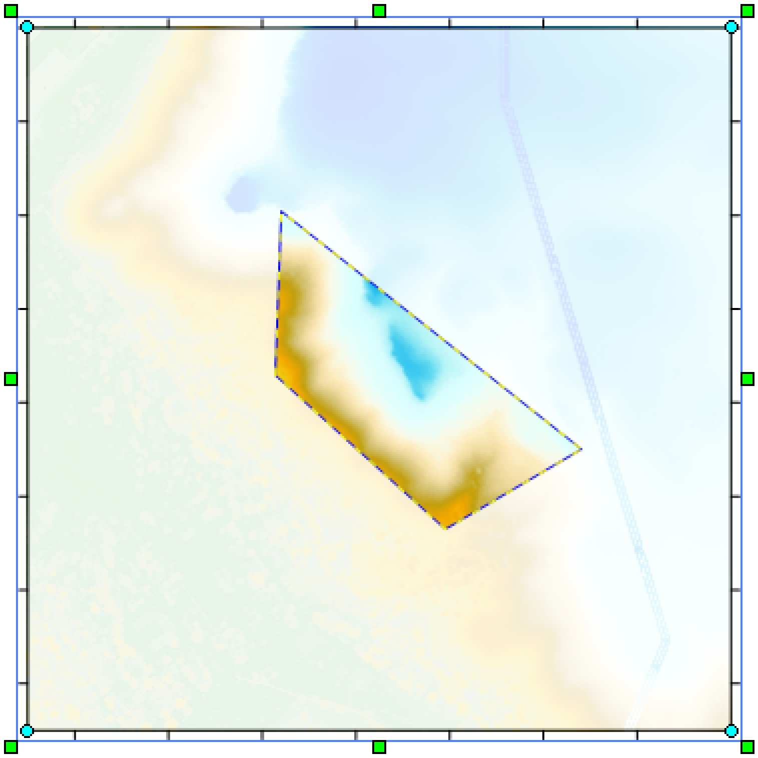
Drawing Tips

- Hold down the CTRL key while dragging the mouse to draw a square rather than a rectangle.
- Hold down the SHIFT key while dragging the mouse to draw a rectangle from the center point rather than from corner to corner.
- Hold down the SHIFT and CTRL keys while dragging the mouse to draw the square from the center point rather than from corner to corner.
- The number of selected points is indicated in the status bar.

Polygon Select

Click the **Point Cloud | Point Selection | Polygon** command or the button to select points in the point cloud layer by drawing a polygon. The cursor changes to a cross hair when the **Polygon** command is selected.

1. Click the **Point Cloud | Point Selection | Polygon** command.
2. Click in the map to add a vertex to the polygon.
3. Continue adding vertices around the points you wish to select.
4. Double-click the last vertex or press the ENTER key to finish drawing the polygon. The points within the polygon are selected.



The selected points are indicated by the highlighted area and blue dashed line.

Drawing Tips

- Click points on the page to draw a polygon, or click and hold the left mouse button and drag the cursor to draw a continuous stream of points.
- Click the right mouse button to remove the last drawn point. This can be done repeatedly.
- If the CTRL key is pressed while clicking points, the points are constrained to 45-degree angles.
- Double-click the left mouse button or press the ENTER key to close the polygon.
- To cancel drawing a polygon, press the ESC key before closing the polygon.
- The number of selected points is indicated in the status bar.

Deselect All Points

Click the **Point Cloud | Point Selection | Deselect All** command to deselect all points in the point cloud layer. When the points in the point cloud layer are deselected, the blue dashed line indicating the selected points bounding box is removed.

Reclassify

The **Point Cloud | Classify | Reclassify** field displays the classification of the selected points. When the selected points include more than one classification value, no classification is displayed in the **Reclassify** field.

To reclassify the selected points, select the desired classification from the **Reclassify** list. The reclassify list includes the following classifications:

- Never Classified
- Unclassified
- Ground
- Low Vegetation
- Medium Vegetation
- High Vegetation
- Building
- Low Point Noise
- Mass Point
- Water
- Railroad
- Road Surface

Remove Classifications

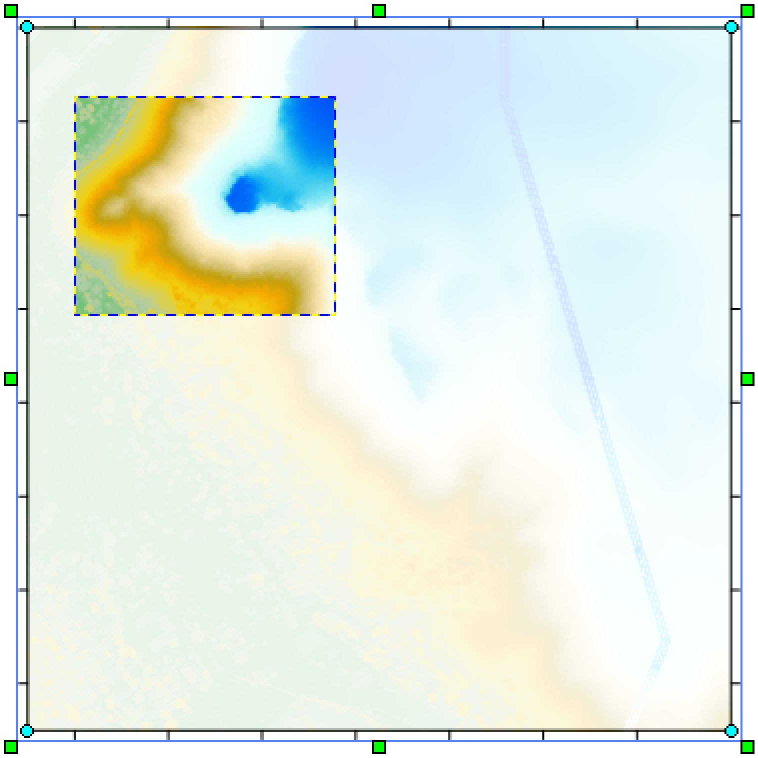
Click the **Point Cloud | Classify | Remove Classifications** command or the button to remove the classifications from the selected points.

Crop to Selection

Click the **Point Cloud | Modify | Crop to Selection** command or the button to crop the point cloud layer to the selected points. The **Crop to Selection** command removes points from the layer that are outside the rectangular extents of the selected points. For example if you create a selection with the Polygon command, the crop area will be the smallest rectangle that completely surrounds the selection polygon.

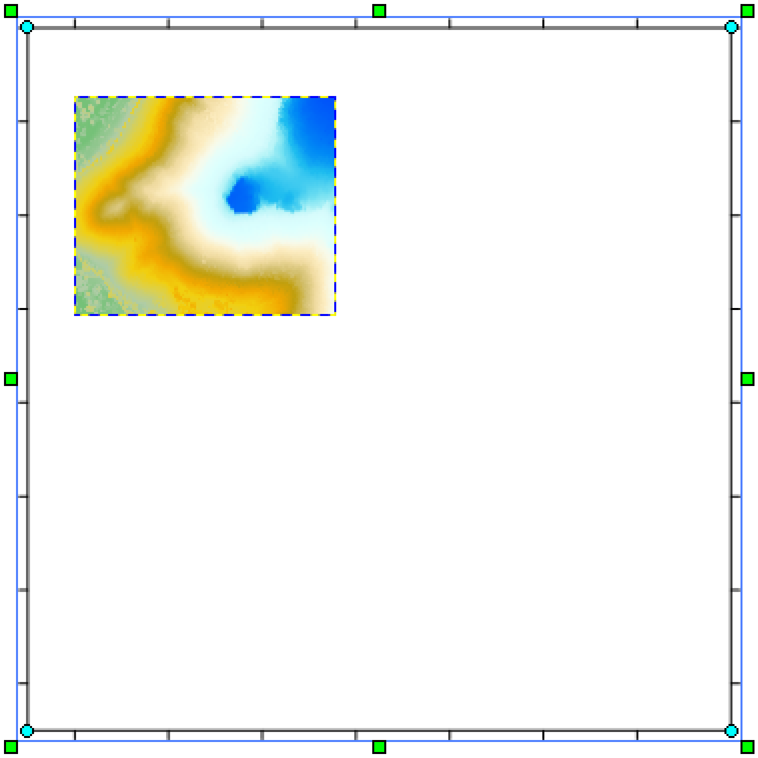
To crop the layer to the selected points:

1. Select points using the Criteria, Rectangle, or **Polygon** commands.



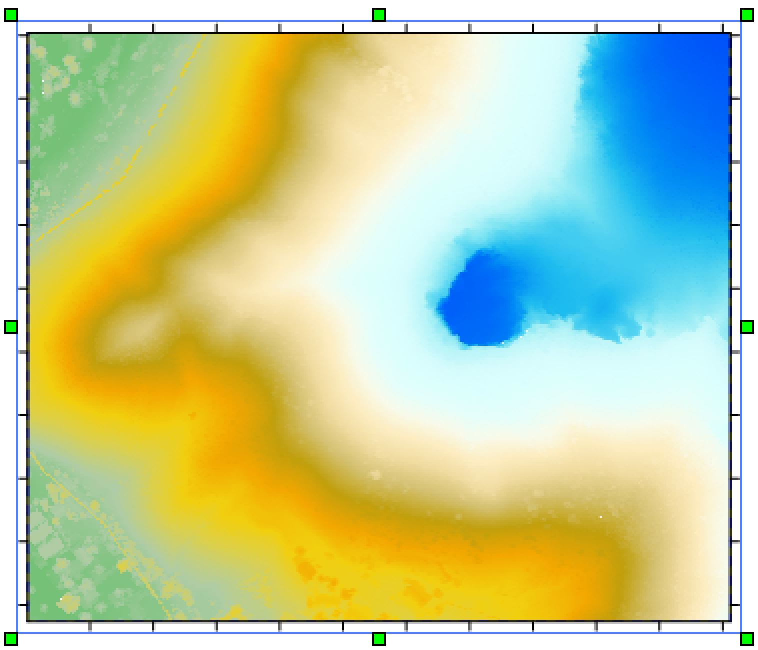
Select the points you wish to keep when using the ***Crop to Selection*** *command.*

2. Click the **Point Cloud | Modify | Crop to Selection** command. All points that are not selected are removed from the map. The map extents are not changed.



Crop to Selection *removes all points outside the extents of the selection.*

3. If desired, change the map limits and scale in the **Properties** window or with the Set Limits command.



You may wish to change the map limits and scale after using the ***Crop to Selection*** *command.*

Remove Selected Points

Click the **Point Cloud | Modify | Remove Selected Points** command or the button to remove the selected points from the layer.

Grid Point Cloud

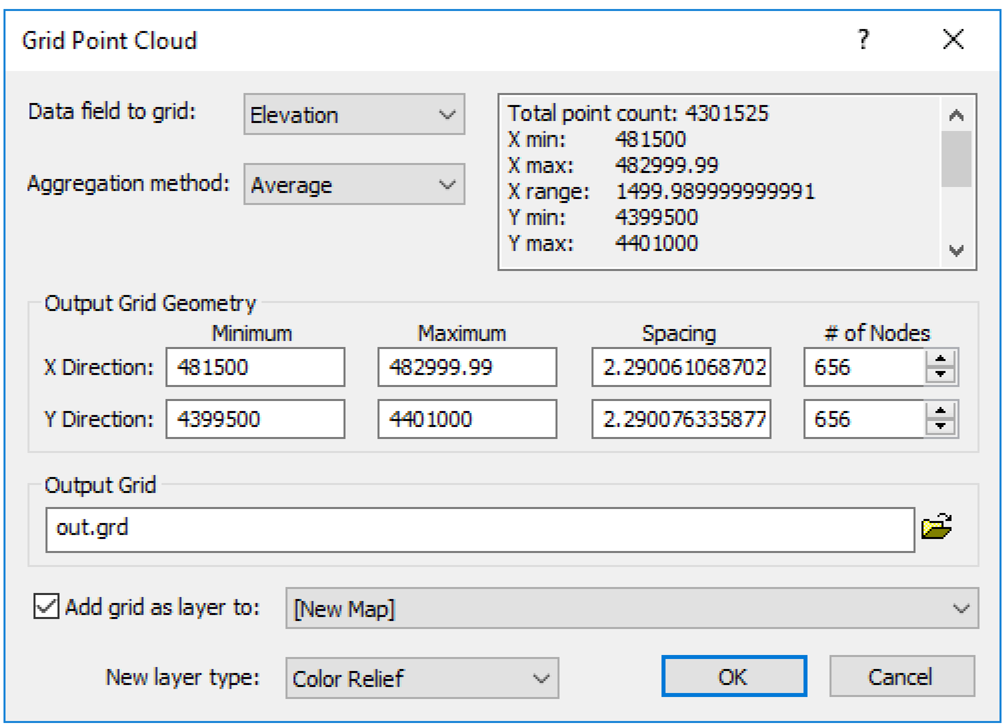
Click **Point Cloud | Features | Create Grid** or the button to create a grid file from the point cloud layer. The **Grid Point Cloud** dialog is displayed when the **Create Grid** command is clicked.

The **Create Grid** command uses a binning algorithm to create a grid file from the point cloud data. Each grid node is at the center of a grid cell, and all points within the cell are aggregated by the selected *Aggregation method* to compute the grid node value. The binning algorithm does not extrapolate beyond the convex hull of the data, and any grid nodes with no points within their cell are assigned the NoData value.

If you wish to create a grid with one of the gridding methods, export the point cloud layer to an LAS/LAZ file and use the newly created file with the Grid Data command.

Grid Point Cloud Dialog

The **Grid Point Cloud** dialog is displayed when the **Point Cloud | Features | Create Grid** command is clicked. Specify the data field, aggregation method, output grid geometry and file name in the **Grid Point Cloud** dialog.



Create a grid file from a point cloud layer with ***Grid Point Cloud*** .

Data Field to Grid

A grid can be created from the elevation, intensity, return number, or classification value. Select the field *Elevation*, *Intensity, Return number*, or *Classification* in the *Data field to grid* list.

Aggregation Method

The *Aggregation method* specifies how the grid node values are calculated from the points in the grid cell.

- *Average* assigns the average value of the points within the cell.
- *Minimum* assigns the minimum value from the points within the cell.
- *Maximum* assigns the maximum value from the points within the cell.
- *Nearest* assigns the value from the point nearest the grid node.
- *IDW* assigns the inverse distance squared weighted average value of the points within the cell.

Select the desired method from the *Aggregation method* list.

Data Statistics

The data statistics section shows the number of points and the spatial extents of the point cloud layer.

Output Grid Geometry

The *Output Grid Geometry* section defines the grid limits and grid density. By default, the *Output Grid Geometry* settings are spaced to include an average of ten points per grid node cell.

Minimum and Maximum X and Y Coordinate (Grid Limits)

Grid limits are the minimum and maximum X and Y coordinates for the grid. **Surfer** computes the minimum and maximum X and Y values from the point cloud layer. These values are used as the default minimum and maximum coordinates for the grid.

Grid limits define the X and Y extent of contour maps, color relief maps, 3D surfaces, etc. created from grid files. When creating a grid file, you can set the grid limits to the X and Y extents you want to use for your map. Once a grid file is created, you cannot produce a grid-based map larger than the extent of the grid file. If you find you need larger grid limits, you must regrid the data. You can, however, read in a subset of the grid file to produce a map smaller than the extent of the grid file.

Spacing and # of Nodes (Grid Density)

Grid density is usually defined by the number of columns and rows in the grid, and is a measure of the number of grid nodes in the grid. The *# of Nodes* in the *X Direction* is the number of grid columns, and the *# of Nodes* in the *Y Direction* is the number of grid rows. By defining the grid limits and the number of rows and columns, the *Spacing* values are automatically determined as the distance in data units between adjacent rows and adjacent columns.

Output Grid

Choose a path and file name for the grid file in the *Output Grid* section. You can type a path and file name, or click the button to browse to a new path and enter a file name in the **Save Grid As** dialog. By default the grid file name is based on the first file in the point cloud metadata.

Add Grid as Layer

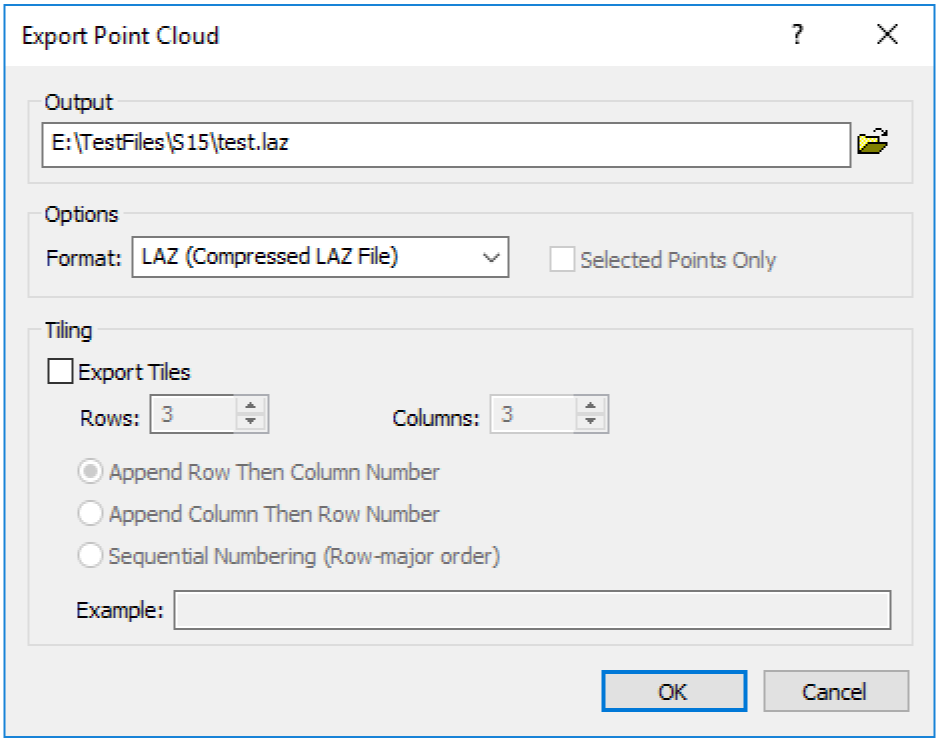
Check the *Add grid as layer* to check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking the current selection in the *New layer type* field and selecting the desired layer type from the list.

Export LAS/LAZ

Click **Point Cloud | Features | Export LAS/LAZ** or the button to export the points in the point cloud layer to an uncompressed LAS or compressed LAZ LiDAR Binary file.

Export Point Cloud Dialog

The **Export Point Cloud** dialog is displayed when the **Point Cloud | Features | Export LAS/LAZ** command is clicked. Specify the output file path and name, format, and tiling options in the **Export Point Cloud** dialog.



Set the export options in the ***Export Point Cloud*** *dialog.*

Output

Choose a path and file name for the LiDAR file in the *Output* section. You can type a path and file name, or click the button to browse to a new path and enter a file name in the **Export Points to File** dialog. By default the file name is based on the first file in the point cloud metadata.

Options

Select the *Format* and whether to export only selected points in the *Options* section.

Format

Select *LAS (Uncompressed LAS File)* or *LAZ (Compressed LAZ File)* in the *Format* list to export an LAS or LAZ file.

Selected Points Only

Select the *Selected points only* option to export only selected points in the LAS/LAZ file. Clear the *Selected points only* check box to export all points in the point cloud layer to the LAS/LAZ file.

Tiling

The *Tiling* section controls the data tiling options. Tiling is used to limit the size of the exported files. For example, an 11.7 million point layer may be exported to a single 313MB LAS file, or split into nine tiles with file sizes varying between 28 and 45MB.

Export Tiles

Select the *Export tiles* option to export the point cloud layer as multiple indexed files to reduce file size. The number of tiles is the product of the *Rows* and *Columns* values. For example, selecting three (3) rows by three (3) columns leads to nine (9) tiles.

Rows

Specify the number of *Rows* for tiling the point cloud layer. Type the desired number in the *Rows* field or click the to change the value. More *Rows* means a larger number of smaller files. Fewer *Rows* leads to a smaller number of larger files.

Columns

Specify the number of *Columns* for tiling the point cloud layer. Type the desired number in the *Columns* field or click the to change the value. More *Columns* leads to a larger number of smaller files. Fewer *Rows* leads to a smaller number of larger files.

Tile Indexing

The tile file names can be indexed in one of three ways:

- *Append row then column number* - the row number is appended first and then the column number is appended to the end of the file name in the *Output* field.
- *Append column then row number* - the column number is appended first and then the row number is appended to the end of the file name in the *Output* field.
- *Sequential numbering (Row-major order)* - the tile in the first row and first column is numbered 0, and then the tiles are numbered in row-major order.

	Col 0	Col 1	Col 2
Row 0	Append row then column number: _0_0 Append column then row number: _0_0 Sequential numbering: _0	Append row then column number: _0_1 Append column then row number: _1_0 Sequential numbering: _1	Append row then column number: _0_2 Append column then row number: _2_0 Sequential numbering: _2
Row 1	Append row then column number: _1_0 Append column then row number: _0_1 Sequential numbering: _3	Append row then column number: _1_1 Append column then row number: _1_1 Sequential numbering: _4	Append row then column number: _1_2 Append column then row number: _2_1 Sequential numbering: _5

The table above demonstrates the indexing for the tile file names.

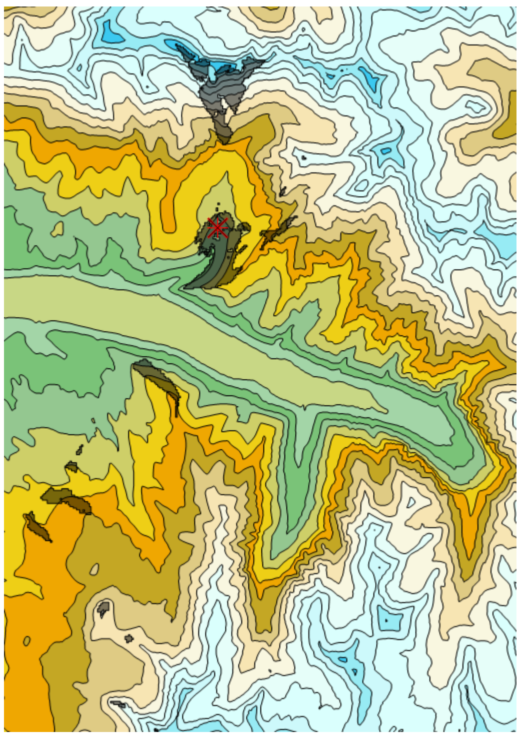
Example

The *Example* shows an example of the tile file names for the first three tiles in row 0.

Chapter 18 - Viewshed Layers

Viewshed

The **Home | Add to Map | Layer | Viewshed**, the button, or **Map Tools | Add to Map | Layer | Viewshed** command adds a viewshed analysis layer to a map. The viewshed analysis indicates which grid cells are visible from an observer's, or transmitter's, location. Since the viewshed requires a grid and a transmitter location, the viewshed analysis must be added to a map object that already contains a 2D grid layer, i.e. contour, color relief, 1-grid vector, 2-grid vector, or watershed map. The current map layer provides a visual reference for placing the transmitter location.



In this viewshed example, areas that are visible from the transmitter location are indicated with the partially transparent black fill.

Add a Viewshed Layer

1. Click the map object or map layer to which the viewshed will be added in the Contents.
2. Click the **Home | Add to Map | Layer | Viewshed** or **Map Tools | Add to Map | Layer | Viewshed** command. The cursor changes to a cross-hair .

3. Click the desired transmitter location on the current map. Use the current map layer and the cursor location coordinates in the status bar to aid in transmitter location placement. The transmitter location can be refined or changed in the Properties window once the viewshed is created.

Viewsheds and 3D Surface Maps

Viewsheds can only be added to maps that are not tilted and in the orthographic projection. To temporarily change a 3D Surface map to this orientation,

1. Click on the *Map* object in the **Contents** window.
2. In the **Properties** window, click on the **View** tab.
3. Change the *Projection* to *Orthographic*.
4. Click and drag the *Tilt (degrees)* to 90.
5. Then, click the **Home | Add to Map | Layer | Viewshed** command.

Once the viewshed is located on the map, the *Projection* and *Tilt (degrees)* can be changed back to their previous values. Viewsheds cannot be added to 3D Wireframe maps.

Viewshed Properties

Once created, a viewshed can be modified with the properties in the Properties window. Click on the viewshed layer in the **Contents** window to view its properties in the **Properties** window. The viewshed properties contain the following pages:

General
Layer
Coordinate System
Info (Grids)

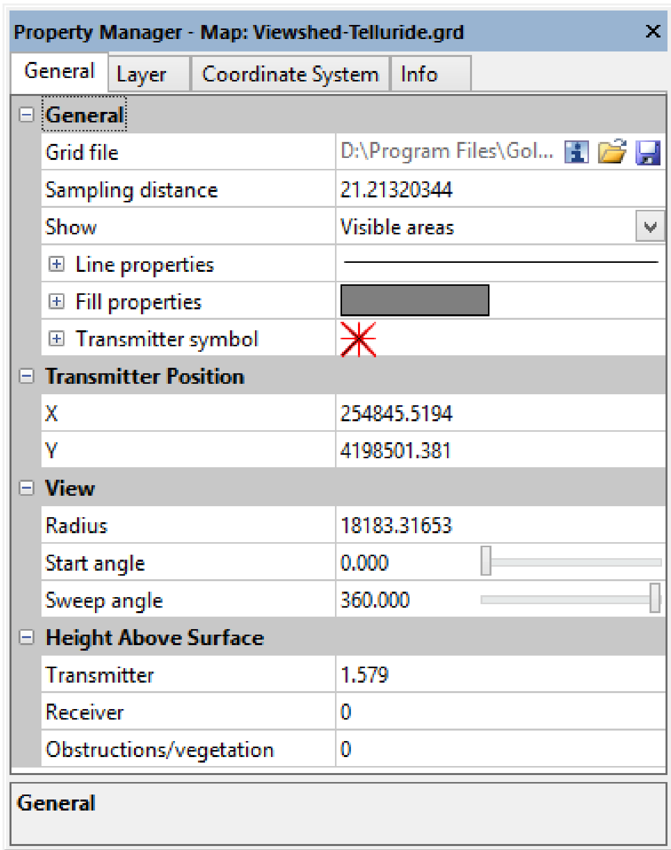
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info (Objects)

Viewshed Layer General Properties

To edit a viewshed layer, click once on the viewshed layer to select it. The properties for the viewshed layer are displayed in the Properties window. The viewshed layer properties **General** page contains the following options:



*Edit the viewshed properties on the **General** page of the **Properties** window.*

Input Grid File

The *Grid file* lists the current grid file used in the viewshed layer. The path and file name are the location of the grid file when the viewshed layer was created or the grid file was most recently changed.

Change File

The  button displays the **Open Grid** dialog. This allows you to select a new grid file, or an updated version of the grid file used to create the map. Select a grid file and click *Open*.

If the new grid exceeds the current map limits, another warning will appear asking you to adjust the map limits. If you click *Yes*, the limits are automatically adjusted to fit the new grid. If you click *No*, the limits are not automatically adjusted. The map may not be displayed. To change the map limits, click on the Map object in the **Contents** window and the **Limits** tab in the **Properties** window.

You may also see a warning message that the current map scale may result in an un-viewable map. Clicking *OK* allows the map scale to automatically be adjusted.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the  button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

The  button displays the **Save Grid As** dialog. Type a *File name* and change the *Save as type* to the desired grid file format. Click *Save.* If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, an **Export Options** dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Sampling Distance

The viewshed is calculated by casting a ray from the transmitter position to each grid node. Starting at the transmitter position, a point is sampled at each *Sampling distance* along the ray and visibility is determined. Small *Sampling distance* values increase the viewshed accuracy, but increase the time to create the viewshed. Generally the viewshed calculation is performed very quickly, but if necessary the *Sampling distance* can be increased to reduce the time to create the viewshed. The default *Sampling distance* value is one-half the length of a grid node diagonal.

Click the current value in the *Sampling distance* field, and type a new value into the field to change the *Sampling distance*. The sampling distance is in the linear distance units specified by the source coordinate system.

Show

The *Show* property specifies whether the visible or invisible regions of the layer are highlighted. When *Visible areas* is selected, regions that are visible from the transmitter location are displayed with the viewshed line and fill properties. When *Invisible areas* is selected, regions that are not visible from the transmitter are displayed with the viewshed line and fill properties. Click the current selection in the *Show* field, and select *Visible areas* or *Invisible areas* from the list to change the *Show* property.

Line Properties

The line properties for the polygons created by the viewshed analysis are edited in the *Line properties* section of the **General** page. See the Line Properties help topic for more information on changing line properties.

Fill Properties

The fill properties for the polygons created by the viewshed analysis are edited in the *Fill properties* section of the **General** page. See the Fill Properties help topic for more information on changing fill properties.

Transmitter Symbol

The transmitter location is represented on the viewshed layer by a symbol. Change the Transmitter symbol properties in the *Transmitter symbol* section of the **General** page. See the Symbol Properties help topic for more information on changing symbol properties.

Transmitter Position

The transmitter position is first specified when creating the viewshed. The transmitter position can be changed by changing the values in the *X* and *Y* coordinates fields. The *X* and *Y* coordinates are specified in reference to the source coordinate system. Change the *X* or *Y* coordinate by clicking on the current value and typing a new value into the field.

View Radius and Angle

The viewshed view *Radius* property limits the viewshed to a specified distance from the transmitter location. The viewshed is limited radially by the *Start angle* and *Sweep angle* properties.

Radius

By default the *Radius* is equal to the diagonal of the grid extents, so the entire grid is included in the viewshed analysis. Change the viewshed *Radius* by clicking the current value in the *Radius* field and typing a new value. The viewshed radius is in the linear distance units specified by the source coordinate system.

Start Angle

The *Start angle* specifies the starting direction for the viewshed analysis. The *Start angle* can be from 0 to 360 degrees, where 0 or 360 degrees is East, 90 degrees is North, 180 degrees is West, and 270 degrees is South. Change the *Start angle* by typing a value into the field or clicking and dragging the bar.

Sweep Angle

The *Sweep angle* controls which portion of the map is included in the viewshed analysis. The viewshed analysis is begins at the *Start angle* and continues radially for the number of degrees specified by the *Sweep angle*. The *Sweep angle* can vary from -360 to 360 degrees. Positive values sweep in a counter-clockwise direction, and negative values sweep in a clockwise direction. For example, with a *Start angle* of 0 degrees and a *Sweep angle* of 90 degrees, the viewshed analysis is performed for grid nodes to the east and continues counter-clockwise until it reaches the grid nodes north of the transmitter location.

Change the *Sweep angle* value by typing a new value into the field or clicking and dragging the bar.

Height Above Surface

The height of the transmitter, receiver, and obstructions can be adjusted in the *Height Above Surface* section of the **General** page. The viewshed is calculated by casting a ray from the transmitter position to each grid node. If any of the sample points along the ray occlude the reciever, the node is determined to be invisible. The relative heights of the transmitter and receiver affect the slope of the ray. The obstruction height affects the height of the sample points. In this way, each height has an effect on what is and is not included in the viewshed analysis results. The heights are specified in the same units as the grid Z level.

Transmitter

The *Transmitter* height is the height of the observation point above the surface at the transmitter position. Often, but not always, a taller observation point generates a larger viewshed. The default value for the *Transmitter* height is 0.3% of the Z range of the grid ((Zmax-Zmin) * 0.003). Viewshed calculations are very sensitive to minute undulations in the surface, so it is not recommended that the *Transmitter* height is set to 0. Change the *Transmitter* height by clicking on the current value and typing a new value into the field.

Receiver

The *Receiver* height is the height above the surface that is evaluated at each grid node. The default value is 0. This means grid cells are visible when the surface is visible from the transmitter. This value can be increased to represent structures, for example a 150-foot tall power line or a ten-meter tall building. Change the

Receiver height by clicking on the current value and typing a new value into the field.

Obstructions/vegetation

The *Obstructions/vegetation* height is added to the surface between the transmitter and receiver during viewshed calculations. The transmitter and receiver heights change the slope of the line of sight. The *Obstructions/vegetation* height affects the likelihood of a sample point obstructing the line of sight, rendering the receiver grid node invisible in the viewshed analysis. As implied, the *Obstructions/vegetation* height is used to simulate vegetation, buildings, or any other obstructions between the transmitter and the receivers. The default value is 0. Change the *Obstructions/vegetation* height by clicking on the current value and typing a new value into the field.

Chapter 19 - Peaks and Depressions Maps

Peaks and Depressions Map

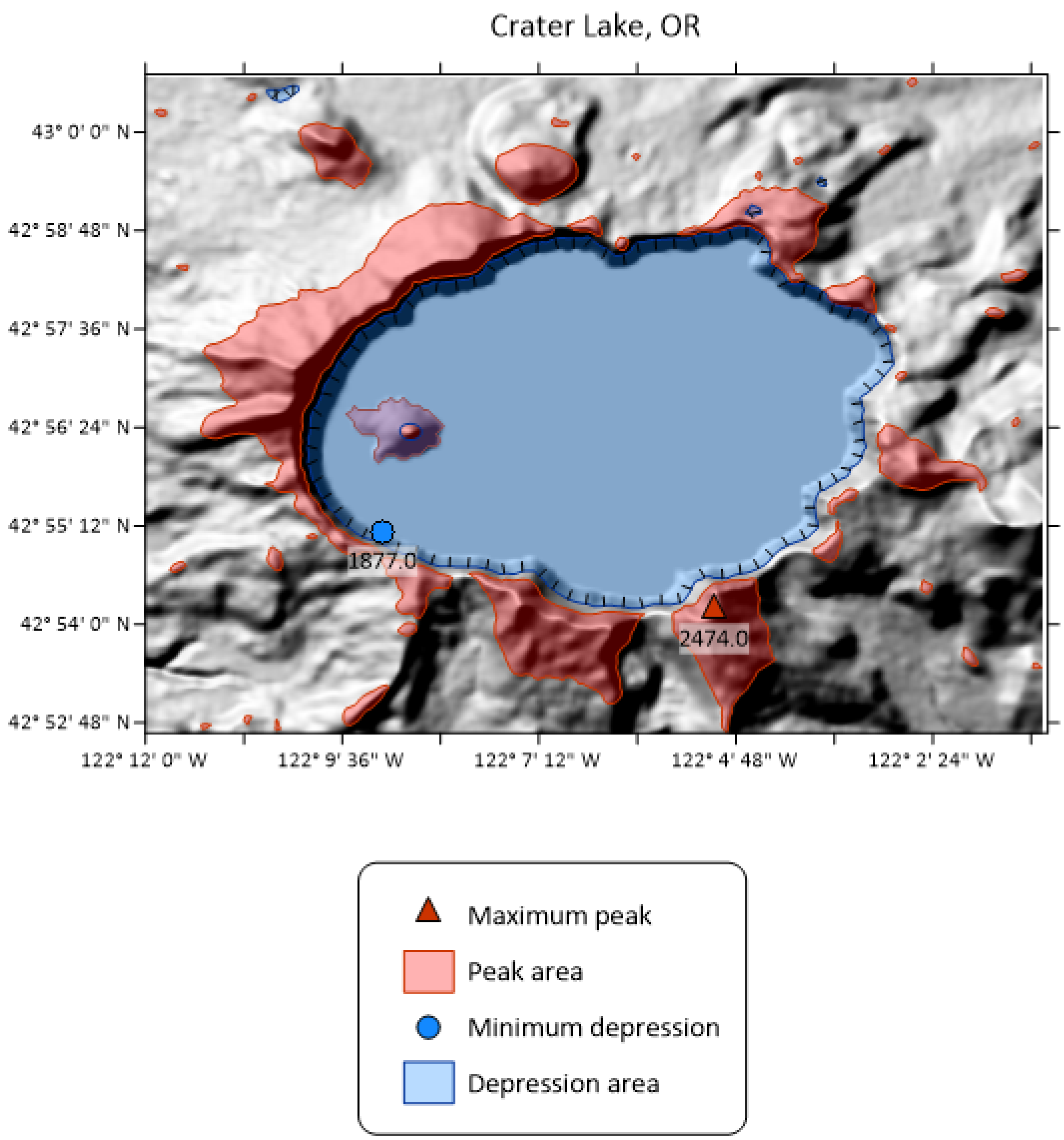
The **Home | New Map | Specialty | Peaks and Depressions** command can be used to create a peaks and depressions map from a grid file. The **Home | Add to Map | Layer | Peaks and Depressions** or **Map Tools | Add to Map | Layer | Peaks and Depressions** command adds a peaks and depressions map layer to the selected map.

A peaks and depressions map creates polygons around peaks and depressions in a grid file. The polygons represent the first or last closed polygon around an area, where drainage flows away from it (peak) or into it (depression). Peaks and depression areas can be labeled, hachured, the minimum and maximum points can be located with a symbol, and a report can be generated with statistical information about each peak and depression (e.g. length, width, height/depth, volume, orientation, etc).

Peaks and depressions maps are useful for modeling water moving over landmasses and landscapes characterized by caves, sinkholes, fissures, and underground streams.

For example, the availability of surface water and groundwater changes over time depending on precipitation, water runoff, evaporation, seepage, and discharge. Determining the current and future water availability also depends on the geology and geography of the catchment area or the building site and water demand. Volumes of surface water and ground water are informed by topographic data, Karst topography, and geographic data, which are captured in Surfer grid files and mapped in peaks and depression maps. Boundaries can be drawn around peaks where water flows from and depressions which capture water to create unique areas for statistical analysis. Sinkholes and hills can be specifically selected and analyzed. The boundary areas can be saved for later analysis when geographic or topographic data changes.

The following example peaks and depressions map is from PeaksAndDepressions.srf, which you can find in the Surfer sample files (C:\Program Files\Golden Software\Surfer\Samples).



Peak areas of Crater Lake are in the default red color and the depression area is in the default blue. Hachures indicate the direction of the slope from the contour line in the depression area. The highest point is labeled with a triangle and its elevation in feet, while the lowest point is marked with a blue circle and labeled with its elevation.

Creating Peaks and Depressions

To create a peaks and depressions map:

1. Click the **Home | New Map | Specialty | Peaks and Depressions** command.
2. Select a grid file in the **Open Grid** dialog and click *Open*.

The map is automatically created with reasonable defaults.

Editing a Peaks and Depressions Map

To change the features of the peaks and depressions map, click once on the peaks and depressions map layer in the plot window or in the Contents window to select it. The properties are displayed in the Properties window.

Adding a Map Layer

When peaks and depressions maps are created, they are independent of other maps in the plot window. For example, creating a peaks and depressions map while a contour map is present in the plot window yields two separate maps, each with its own set of axes and scaling parameters. To create a single map with the contour map on the peaks and depressions map, select both maps by clicking the Home | Selection | Select All command. Overlay the maps using the Map Tools | Map Tools | Overlay Maps command.

Alternatively, you can add the contour map directly to the existing peaks and depressions map by creating the contour map using the **Home | Add to Map | Layer | Contour** command. This automatically adds the contour map to the existing peaks and depressions map axes.

Another alternative is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. Create the peaks and depressions map using the **Home | New Map | Specialty | Peaks and Depressions** command. This creates two separate maps. Click on the contour layer, hold down the left mouse button, and drag the contour map into the peaks and depressions map. A single map with two map layers, using one set of axes and scaling parameters is created.

Any number of peaks and depressions maps can be combined with any other map layers.

Peaks and Depressions Layer Properties

The peaks and depressions map properties contains the following pages:

General
Peaks
Depressions
Layer
Coordinate System
Info (Grids)

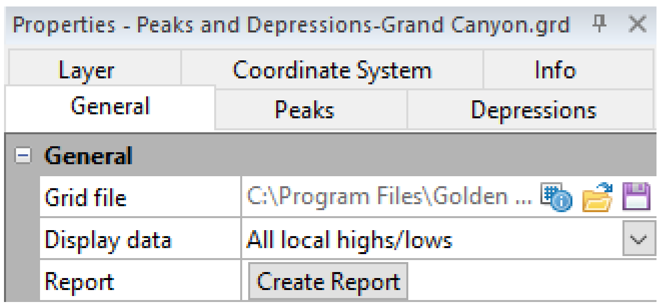
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info (Objects)

Peaks and Depressions Layer General Properties

To edit a peaks and depressions map, click once on map layer to select it. The properties for the peaks and depressions map are displayed in the Properties window. The peaks and depressions properties **General** page contains the following options:



Change peaks and depressions layer properties in the ***Properties*** *window on the* ***General*** *page.*

Grid File

The *Grid file* displays the path and file name for the grid file used for the map. If the entire file name is not shown, place the mouse over the file name. A small window will appear with the full path and name displayed.

Grid Information

The  button displays information about the grid file used to produce the map layer. The information includes the grid size, the minimum and maximum X, Y, Z values contained in the grid file, and statistics. If the grid file contains more than 40 million nodes, you are asked if you wish to create a detailed report or a quick report. Click *Yes* in the message to create a detailed grid report, or click *No* to create a shorter quick grid report.

Change File

Click the button to display the **Open Grid** dialog. This allows a new or updated grid file to be specified for the peaks and depressions map. Select the new grid file and click *Open* to reference the new file.

If the new grid exceeds the current map limits, a warning will appear asking you to adjust the map limits. If you click *Yes*, the limits are automatically adjusted to fit the new grid. If you click *No*, the limits are not automatically adjusted. The map may not be displayed. To change the map limits, click on the Map object in the **Contents** window and the **Limits** tab in the **Properties** window.

You may also see a warning message that the current map scale may result in an un-viewable map. Clicking *OK* allows the map scale to automatically be adjusted.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Save File

Click the button to display the **Save Grid As** dialog. This allows the grid file used for the peaks and depressions map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file. If a coordinate system has been defined on the Coordinate System tab in the **Properties** window, the Export Options dialog appears. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Click *OK* and the file is saved.

Display Data

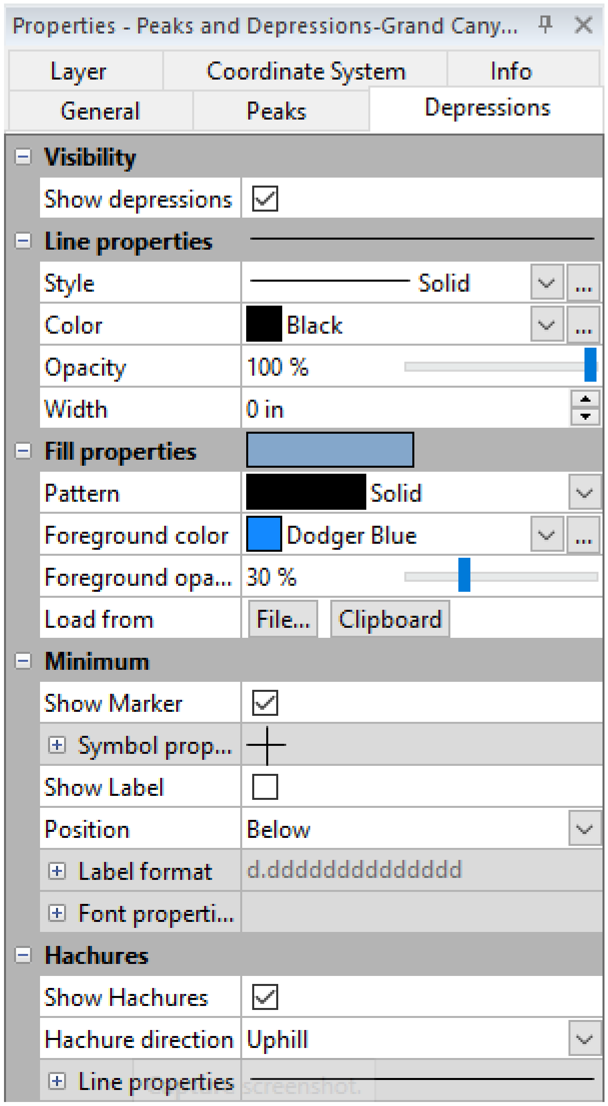
Display either *All local highs/lows* or the *Maximum high/Minimum low*.

Report

Click *Create Report* to run a report of statistics of the peak and depression areas. Statistics are only included for peaks and or depressions displayed on the map. In the **Peaks and Depressions Report Options** dialog enter the appropriate number if the Z dimension is in different units than the X and Y dimensions. The default is 1. The statistics can be saved or printed when opened in the report window.

Peaks and Depressions Layer Depressions Properties

To edit the depressions in a peaks and depressions map click on the grid file in the Contents window to access the **Depressions** page in the Properties window. The **Depressions** page contains the following options:



*Change the properties of depressions in the **Properties** window on the **Depressions** page.*

Visibility

Check the *Show depressions* box to show depressions on the map.

Line Properties

Use the **Line Properties** sections in the Properties window to change line properties for selected lines in the document. You can set default line properties in the **Options** dialog Defaults page by clicking on *Line*.

Style

Click the line next *Style* to open the line style palette. Click on a style to use it for the selected line. The line style sample updates to show the new selection. Click on the ... button at the right of the line style to open the **Custom Line** dialog, where you can specify a custom line style.

Complex line styles can be selected for most object types. Wireframe map layers and surface mesh lines do not support complex line styles. When a complex line style is selected, the line *Width* automatically increased to 0.03125 inches (0.079 cm). Most complex line styles require lines with the *Width* set to something larger than 0.03125 inches to fully distinguish the line style.

Color

Click the color next to *Color* to open the color palette. Click on a color to use it for the selected line. The color box and the sample line update to show the new selection. Click on the ... button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Opacity

The *Opacity* is the amount of transparency of the line. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click and drag the slider to change the opacity percentage.

Width

The *Width* controls the thickness of the line in page units. The value can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide. To change the *Width*, highlight the existing number and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the width.

Fill Properties

Most fill properties are edited in the Properties window in a *Fill Properties* section. When changing fill properties for a selected object, the *Fill Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Fill Pattern

Change the *Pattern* by selecting a pattern from the fill pattern palette. Open the pattern palette by clicking the pattern button. Click on a new pattern in the list to select it.

Foreground Color

Foreground color is the color of the pattern lines or pixels. Select a new color by clicking on the color in the color palette. Only the foreground color can be applied to solid colors. The foreground colors can be applied to any stock hatch pattern or grayscale image pattern. They cannot be applied to the *None* pattern or non-grayscale image patterns. Click the ... button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Foreground Opacity

The *Foreground opacity* is the amount of transparency of the fill. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change.

Alternatively, click and drag the slider to change the opacity percentage. Opacity is disabled if it does not apply to the current pattern. Note that for true color image fill patterns the *Foreground opacity* applies to the image in its entirety.

Load a Fill Pattern File

To load a fill pattern from a raster image file, click the *File* or *Clipboard* button in the *Load from* field. The *Clipboard* button is only active when there is an image on the clipboard suitable to use for a fill pattern. If you select *File,* an **Import** dialog will appear. Click on the image file and click *Open* to load the image as the fill pattern.

Minimum

The Minimum section includes properties for displaying symbols at the depressions.

Show Marker

Check the *Show marker* box to mark the minimum point in the depressions or polygon(s).

Symbol Properties

Expand Symbol properties to change or set symbol properties.

Show Label

Check the *Show label* box to display labels.

Position

The *Position* controls the offset of the selected label. To change the position, select the new position from the list. Available options are *Center*, *Left*, *Right*, *Above*, *Below*, and *User defined*. *User defined* allows you to specify the exact offset (in page units) in the *X offset* and *Y offset* boxes.

Format

Click the ⊞ next to *Label format* to display the Label Format section. The numeric format is applied to all numbers read from the specified label column in this label set.

Font Properties

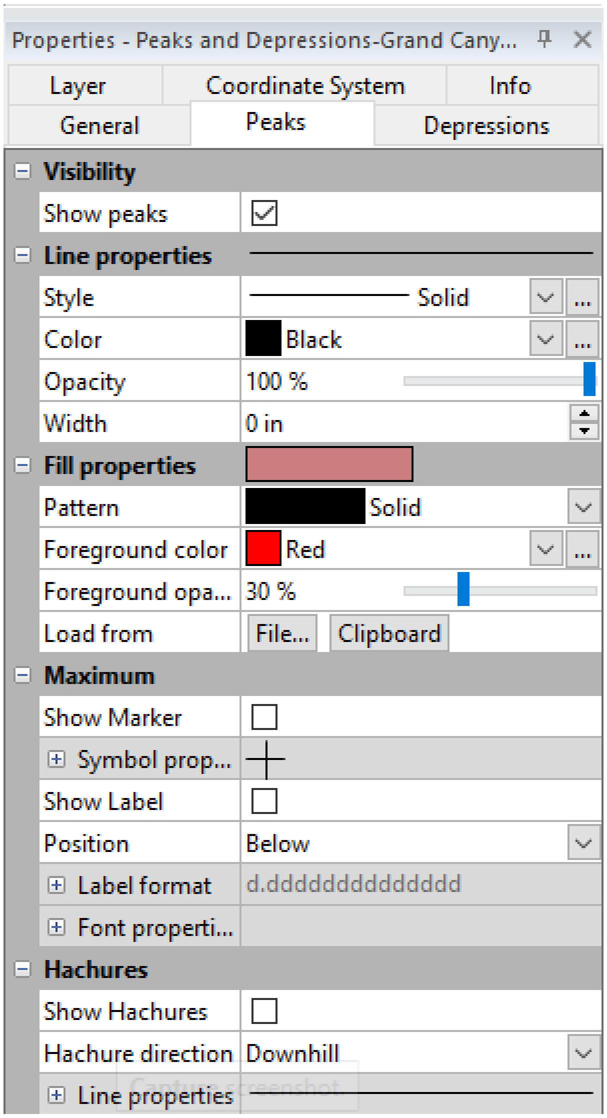
Click the ⊞ next to *Font properties* to display the Font Properties section. The font properties are applied to all labels in this label set.

Hachures

Check the *Show Hachures* box to display hachures in the areas of the depressions. Change the direction of all the hachures to either *Downhill* or *Uphill* to indicate the direction of slope.

Peaks and Depressions Layer Peaks Properties

To edit the peaks in a peaks and depressions map click on the grid file in the Contents window to access the **Peaks** page in the Properties window. The **Peaks** page contains the following options:



Change the properties of peaks in the ***Properties*** *window on the* ***Peaks*** *page.*

Visibility

Check the *Show peaks* box to show peaks on the map.

Line Properties

Use the **Line Properties** sections in the Properties window to change line properties for selected lines in the document. You can set default line properties in the **Options** dialog Defaults page by clicking on *Line*.

Color

Click the color next to *Color* to open the color palette. Click on a color to use it for the selected line. The color box and the sample line update to show the new selection. Click on the ... button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

The default color for peaks is red.

Opacity

The *Opacity* is the amount of transparency of the line. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click and drag the slider to change the opacity percentage.

Style

Click the line next *Style* to open the line style palette. Click on a style to use it for the selected line. The line style sample updates to show the new selection. Click on the ... button at the right of the line style to open the **Custom Line** dialog, where you can specify a custom line style.

Complex line styles can be selected for most object types. Wireframe map layers and surface mesh lines do not support complex line styles. When a complex line style is selected, the line *Width* automatically increased to 0.03125 inches (0.079 cm). Most complex line styles require lines with the *Width* set to something larger than 0.03125 inches to fully distinguish the line style.

Width

The *Width* controls the thickness of the line in page units. The value can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide. To change the *Width*, highlight the existing number and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the width.

Fill Properties

Most fill properties are edited in the Properties window in a *Fill Properties* section. When changing fill properties for a selected object, the *Fill Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Fill Pattern

Change the *Pattern* by selecting a pattern from the fill pattern palette. Open the pattern palette by clicking the pattern button. Click on a new pattern in the list to select it.

Foreground Color

Foreground color is the color of the pattern lines or pixels. Select a new color by clicking on the color in the color palette. Only the foreground color can be applied to solid colors. The foreground colors can be applied to any stock hatch pattern or grayscale image pattern. They cannot be applied to the *None* pattern or non-grayscale image patterns. Click the ... button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Foreground Opacity

The *Foreground opacity* is the amount of transparency of the fill. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click and drag the  to change the opacity percentage. Opacity is disabled if it does not apply to the current pattern. Note that for true color image fill patterns the *Foreground opacity* applies to the image in its entirety.

Load a Fill Pattern File

To load a fill pattern from a raster image file, click the *File* or *Clipboard* button in the *Load from* field. The *Clipboard* button is only active when there is an image on the clipboard suitable to use for a fill pattern. If you select *File,* an **Import** dialog will appear. Click on the image file and click *Open* to load the image as the fill pattern.

Maximum

The Maximum section includes properties for displaying symbols at the peaks.

Show Marker

Check the *Show marker* box to mark the maximum point in the peaks or polygon (s).

Symbol Properties

Expand Symbol properties to change or set symbol properties.

Show Label

Check the *Show label* box to display labels.

Position

The *Position* controls the offset of the selected label. To change the position, select the new position from the list. Available options are *Center*, *Left*, *Right*, *Above*, *Below*, and *User defined*. *User defined* allows you to specify the exact offset (in page units) in the *X offset* and *Y offset* boxes.

Format

Click the ⊞ next to *Label format* to display the Label Format section. The numeric format is applied to all numbers read from the specified label column in this label set.

Font Properties

Click the ⊞ next to *Font properties* to display the Font Properties section. The font properties are applied to all labels in this label set.

Hachures

Check the *Show Hachures* box to display hachures in the areas of the peaks. Change the direction of all the hachures to either *Downhill* or *Uphill* to indicate the direction of slope.

Peaks and Depressions Report

The Peaks and Depressions Report is created by clicking the *Generate Report* button on the **General** page of the Peaks and Depressions **Properties** page. It contains the *Time Stamp*, *Data Source*, and separate sections with the statistics for *Peaks*, *Depressions*, and *Individual Peaks*. Save or print the report from the **File** menu commands. See below for the information contained in each section of the report.

Time Stamp

The date and time the report was created.

Data Source

The Grid file name provides the path and file name of the data used in gridding.

Peaks and Depressions Sections

The peaks section and depressions section each provide statistics for the total of all the peaks and/or depressions as long as *Show peaks/depressions* is checked respectively on the Peaks page or Depressions page in the **Properties** window.

This list of statistics is calculated for *Height* (Peaks) and *Depth* (Depressions), *Length*, *Width*, *Planar Area*, and volume. Three methods are used to determine volumes. **Surfer** approximates the necessary one-dimensional integrals using three classical numerical integration algorithms: Extended Trapezoidal Rule, Extended Simpson's Rule, and Extended Simpson's 3/8 Rule; see Press et al., 1988, Section [4.1]. The difference in the volume calculations by the three different methods measures the accuracy of the volume calculations. If the three volume calculations are reasonably close together, the true volume is close to these values. If the three values differ somewhat, a new denser grid file should be used before performing the volume calculations again. The net volume can be reported as the average of the three values.

5%-tile	5 percent of the values are smaller than this number and 95 percent of the values are larger
95%-tile	95 percent of the values are smaller than this number and 5 percent of the values are larger
Minimum	minimum value
Maximum	maximum value
Mean	arithmetic average of the data $Mean = \frac{1}{N}\sum_{i=1}^{N} X_i$
Median	middle data value, 50 percent of the data values are larger than this number and 50 percent of the data are smaller than this number

Individual Peaks and Depressions Sections

A list of polygons within the peaks or depressions areas provides information about the *X* and *Y* coordinates, *Elevation* which is also the *Z* coordinate, *Height*, *Length*, *Width*, *Orientation*, *Planar Area*, *Extended Trapezoidal Rule*, *Extended Simpson's Rule*, and *Extended Simpson's 3/8 Rule:* are shown. Add labels to the peaks or depressions by clicking the Show label property in the Peaks and/or Depressions pages in the Properties window to cross reference them with the Elevation column in the report.

Chapter 20 - Drillhole Maps

Drillhole Map

Surfer allows you to create 2D maps and 3D models of the path and shape of drillholes, boreholes, shafts, wells, and tunnels to help you build models of those objects. All of these types of data can be displayed as lines and symbols on a 2D map or as shaped 3D cylinders in a 3D model. Using interval data, Surfer can also display a 3D segmented view with a color scale. For example, if your data contains contaminant or deviation data, you can model those aspects in 3D in different colors and shapes along the path of the drillhole, well, or borehole. You can create labels using values in your data files. To help visualize and share these maps and 3D models, export 2D and 3D images to other applications or use the Surfer plot window or the 3D view window to dynamically view the borehole, drillhole, or well paths.

This help topic uses sample drillhole data to walk you through the process of defining both 2D and 3D drillhole maps. You can add drillholes to a new map or to an existing map. Use the **Home | New Map | Specialty | Drillhole** command to create a new map with a drillhole layer or use the **Home | Add to Map | Layer | Drillhole** or **Map Tools | Add to Map | Layer | Drillhole** command to add a drillhole layer to an existing map.

After creating a drillhole layer, you can view it in 3D, Add a Drillhole Layer, define 2D properties, and define 3D properties.

This help topic contains the following main sections:
Drillhole Data Requirements
Create a Drillhole Map
Add a Drillhole Layer
Tips and Hints

Drillhole Data Requirements

The data to create a drillhole map include required collars data, and optional survey, interval and point data. In addition to drillhole depth and path data, other data considerations include having supporting values in the point and interval data to map different aspects of your drillhole, borehole, or well (for example, various mineral concentrations). Our sample data include information about TiO_2 and MnO, which are shown in the example images in the 3D drillhole help topic. The following sections define the types of data that Surfer needs to create a drillhole layer and the types of data that Surfer can use to enhance the 3D modeling of drillholes and boreholes.

Note: The data below are examples (from C:\Program Files\Golden Software\Surfer\Samples\SampleDrillholeData.xlsx), your data column names may vary. The drillhole or borehole identifier (listed as *ID* or *Hole Id* in the examples) is a required column for any imported data table so that Surfer can link all related data to the same drillhole or borehole.

Collars Data

Collars data should have information about the drillhole or borehole, such as Hole ID (name), East (X) coordinate, North (Y) coordinate, elevation, starting depth, ending depth, azimuth, and inclination. The Hole ID, East (X), and North (Y) coordinate fields are required for this table type. Inclination and azimuth data can be used to calculate true vertical depth for deviated paths. This table can have multiple rows, but Hole IDs must be unique. A collars table is required unless you import a LAS (log-ascii) file as interval and point data.

Example Collars Data

ID	Easting	Northing	Elevation
MW-1	595299	3968125	2145
MW-2	595203	3968210	2145
MW-3	595176	3968113	2145

Example Collars Data Definitions

Hole ID	Contains the borehole or drillhole identifier.	Required
Easting	Contains the X value, easting, longitude, or other horizontal location value of the borehole or drillhole.	Required
Northing	Contains the Y value, northing, latitude, or other vertical location value of the borehole or drillhole.	Required
Elevation	Contains the Z value or elevation of the borehole or drillhole.	Optional
Starting Depth	Contains the starting Z value for the borehole or drillhole in depth or elevation units. Keep in mind that starting depth is not the same as elevation.	Optional
Ending Depth	Contains the ending Z value for the borehole or drillhole in depth or elevation units.	Optional
Inclination	The angle the borehole or drillhole is oriented, in degrees. Inclination varies from 0 to 180. 0 indicates vertical pointing down, 90 indicates horizontal, and 180 indicates vertical pointing up. When recorded in a collars table, the azimuth and inclination apply to the entire borehole or drillhole length. Negative and positive inclination values are treated the same for depth calculations. A negative inclination changes the direction (azimuth) to the opposite of the similar positive inclination. For example, the azimuth value of 90 and inclination of 45 describes the same orientation as the azimuth value of 270 and inclination of -45 degrees. Both combinations describe an eastward direction at 45 degrees down from the horizontal plane.	Optional
Azimuth	The compass orientation of the borehole or drillhole. Azimuth is in	Optional

	degrees and varies from 0 (true vertical north) to 360. When recorded in a collars table, the azimuth and inclination apply to the entire borehole or drillhole length.	

Survey Data

Optional survey data should have information about the orientation of the drillhole, such as Hole ID, depth, azimuth, and inclination. This table can have multiple rows with the same Hole IDs.

Example Survey Data

ID	MD	Azimuth	Inclination
MW-1	0	370	44.62
MW-1	20	370	44.89
MW-2	0	315	45.44

Example Survey Data Definitions

Hole ID	Contains the borehole or drillhole ID.	Required
Depth	Contains the depth or elevation for the recorded deviation.	Required
Inclination	The angle the borehole or drillhole is oriented in degrees. Inclination varies from 0 to 180. 0 indicates vertical pointing down, 90 indicates horizontal, and 180 indicates vertical pointing up. When recorded in a survey table, the azimuth and inclination apply from the depth to the next recorded depth.	Required
Azimuth	The compass orientation of the borehole's or drillhole's deviation. Azimuth is in degrees and varies from 0 (true vertical north) to 360. When recorded in a survey table, the azimuth and inclination (or dip) apply from the depth to the next recorded depth.	Required

Intervals Data

Optional interval data should have actual down hole information in one or more data files (e.g., XLSX, DAT, and CSV) or LAS files. LAS files import both the downhole data and the collars information. Interval data can have multiple rows with the same Hole IDs.

Example Intervals Data

ID	From	To	TiO2	MnO
MW-1	10	30	2.903	0.302
MW-1	20	40	2.915	0.299
MW-1	40	60	2.83	0.282

Example Intervals Data Definitions

Hole ID	Contains the borehole or drillhole ID associated with the interval.	Required
From	Contains the top depth or elevation of the recorded parameter.	Required

To	Contains the bottom depth or elevation of the recorded parameter.	Required
Parameter (s)	One or more columns that contain information, such as contamination, chemical concentration, etc. The parameter value is recorded across at the interval.	Optional

Points Data

Optional point data should have actual down hole information in one or more data files (e.g., XLSX, DAT, and CSV) or LAS files. LAS files import both the down-hole data and the collars information. All variable information relates to that specific depth. Points data can have multiple rows with the same Hole IDs.

Example Points Data

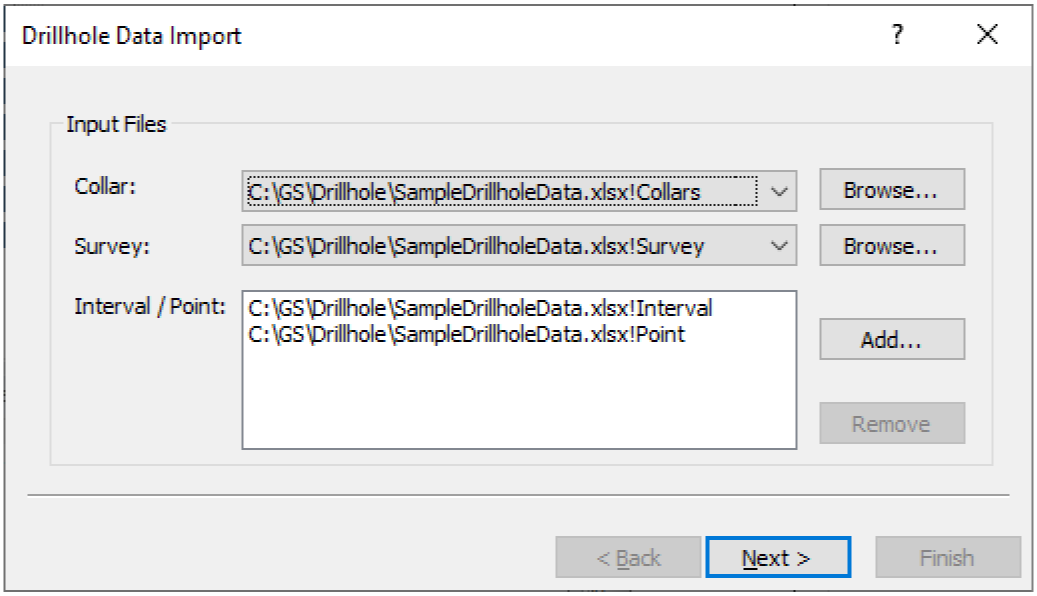
ID	Depth	TiO2	MnO
MW-1	10	2.903	0.302
MW-1	30	2.915	0.299
MW-1	50	2.83	0.282

Example Points Data Definitions

Hole ID	Contains the borehole or drillhole ID associated with the depth.	Required
Depth	Contains the depth or elevation of the recorded parameter.	Required
Parameter(s)	One or more columns that contain information, such as contamination, chemical concentration, etc. The parameter value is recorded at the depth.	Optional

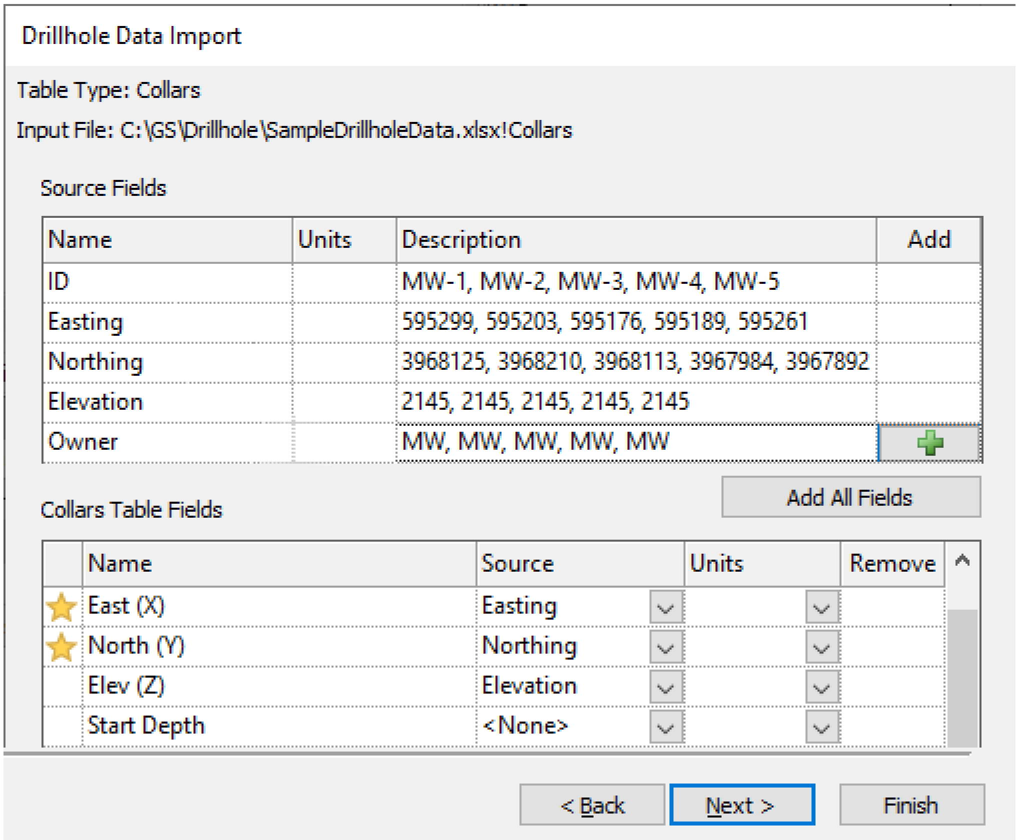
Create a Drillhole Map

1. In a new plot window click the **Home | New Map| Specialty | Drillhole** command. A **Drillhole Data Import** dialog appears.



Example ***Drillhole Data Import*** *data selection dialog using SampleDrillholeData.xls*

2. In the **Drillhole Data Import** dialog, click the *Browse* and *Add* buttons to select each of the drillhole data files (collar and optional survey, interval, and point data), and then click *Next* to open the next **Drillhole Data Import** dialog. A **Drillhole Data Import** dialog will appear for each data file selected to be imported. The top section contains a table of the selected data source (*Source Fields*) and the bottom section contains the mapping of that data source to the table structure (*Table Fields*) that Surfer will use to create the drillhole map.



Example **Drillhole Data Import** *dialog for collars data using SampleDrillholeData.xls*

3. To import collars or survey data from a **Drillhole Data Import** dialog:

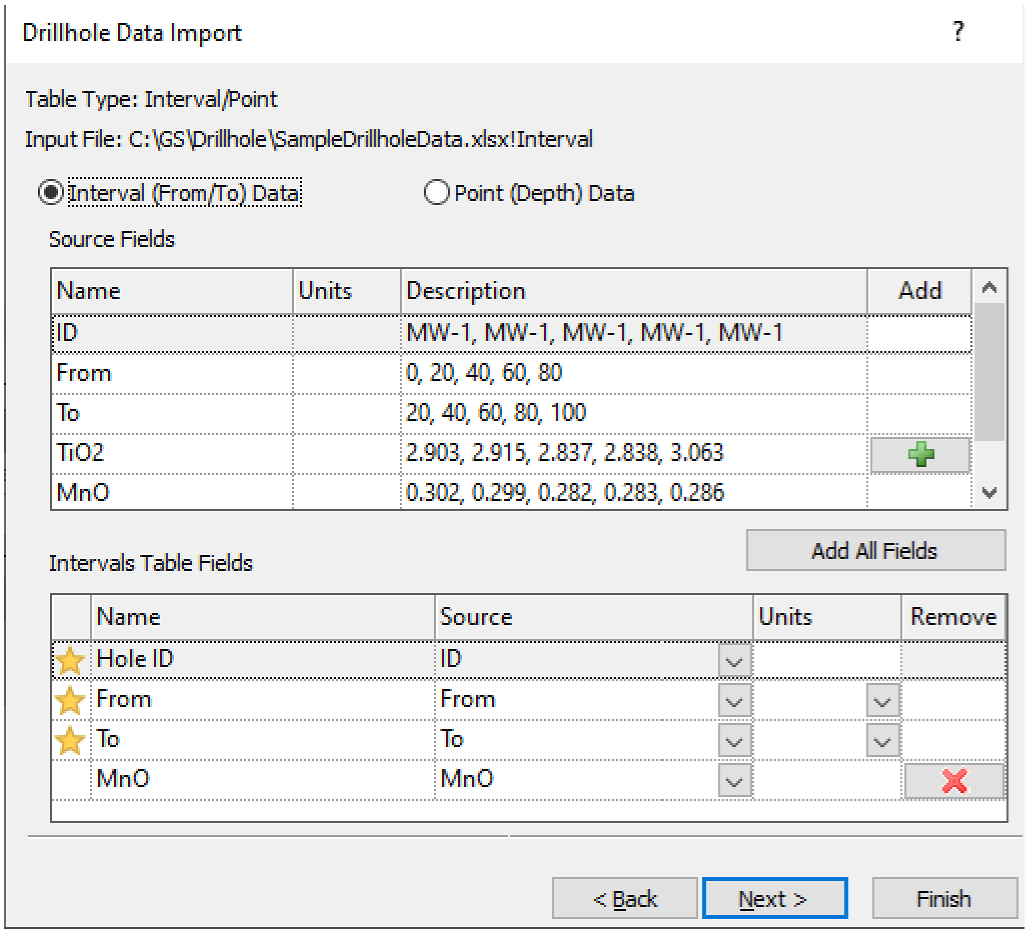
   Note: If both the collars and survey table have azimuth and inclination data, Surfer will use the survey table data. If neither the collars nor the survey data has azimuth or inclination data, then Surfer will assume that the hole is vertical.

   a. In the *Table Fields* section, assign a *Source* field to each *Name* field. Rows marked with a ⭐ are required. If a source field is not assigned to a predefined, non-required table field, the table field can be left empty (*<None>*) and have no values imported.
   b. Use the *Units* fields to define the units for source data. If *Units* are defined in the source field, the value will automatically be added to the selected table field. If needed, data conversion will occur for the selected unit of the table field. The select units will also show in the **Drillhole Manager**.
   c. Click the ➕ button next to any additional data in the *Source Fields* section to import that data or click the *Add All Fields* button to add all data. For any additional rows that were added to the *Table Fields* section,

click the button to not import that data. Any data not listed in the *Table Fields* will not be imported.

d. After associating the *Name* to the *Source* data and defining any *Units*, click *Next* to display the next data import dialog.

4. If interval or point data were selected to be imported, a **Drillhole Data Import** dialog appears for every data source. If the imported data are interval data, click *Interval (From/To) Data*. If the imported data are point data, click *Point (Depth) Data*. Interval data usually has rows with *Hole ID*, *From*, and *To*, and point data will have *Depth* data.



Example interval data ***Drillhole Data Import*** *dialog.*

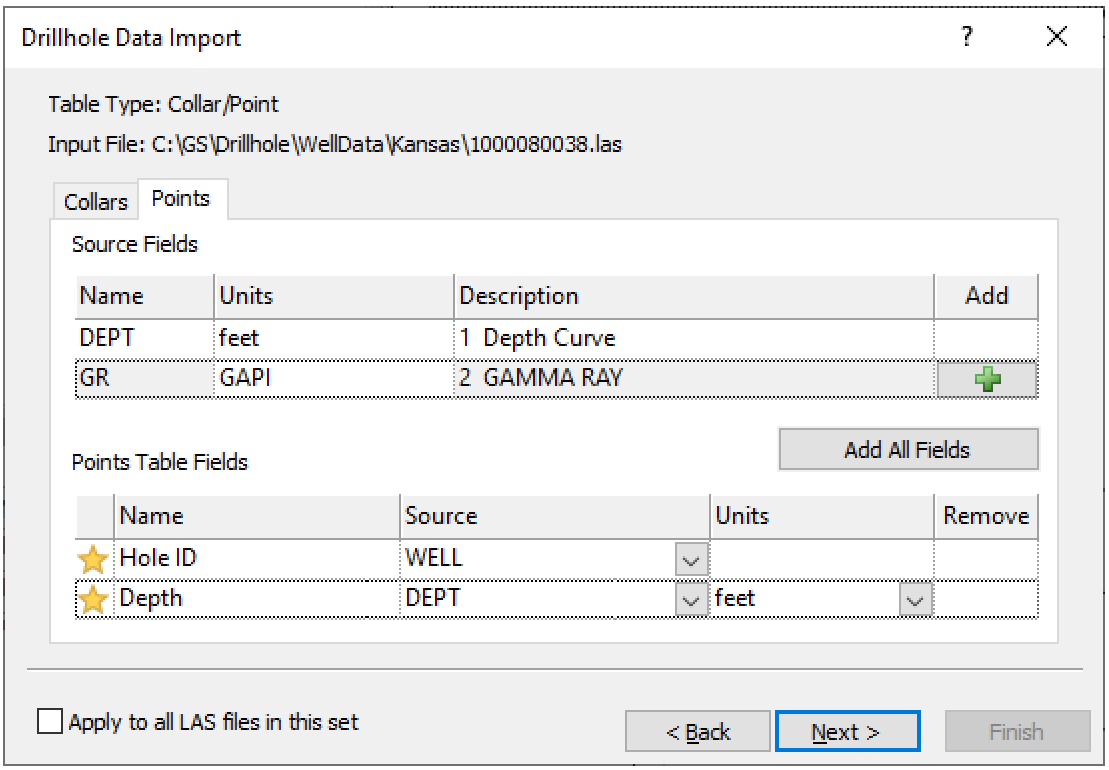
a. In the *Table Fields* section, assign a *Source* field to each *Name* field. Rows marked with a ☆ are required. If a source field is not assigned to a predefined, non-required table field, the table field can be left empty (<*None*>) and have no values imported.

b. Click the button next to any additional data in the *Source Fields* section to import the data or click the *Add All Fields* button to add all data.

For any additional rows that were added to the *Table Fields* section, click the button to not import that data.

c. After associating the *Name* to the *Source* data and defining any *Units*, click *Next*. If additional imported data need to be defined, another ***Drillhole Data Import*** will appear.

5. If one or more LAS files were selected to import, each LAS file will import data into the **Collars** and **Points** tabs in the **Drillhole Data Import** dialog for LAS files. The **Collars** tab is similar to the collars table **Drillhole Data Import** dialog described above and takes data from the well information block and the parameter information block in the LAS file. The **Points** tab takes data from the curve information block in the LAS file.



*Example **Points** tab for LAS data import in the*
***Drillhole Data Import** dialog*

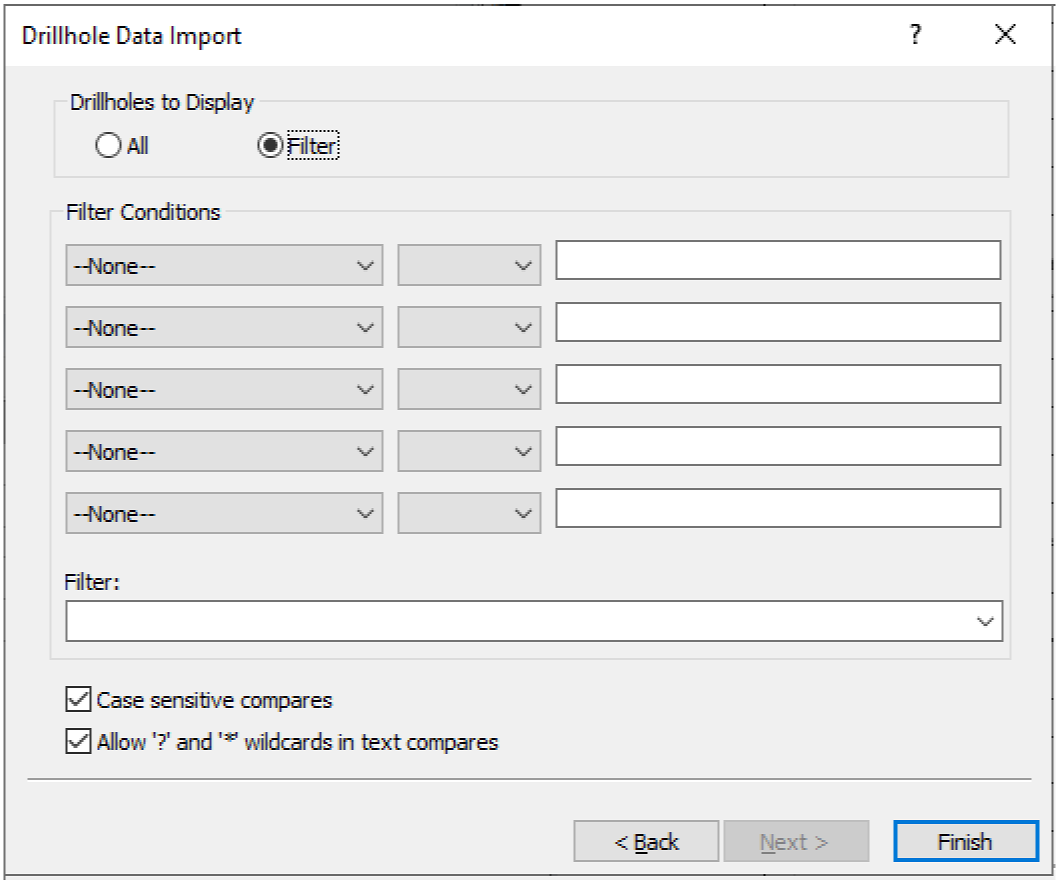
a. In the *Table Fields* section, assign a *Source* field to each *Name* field. Rows marked with a are required.

b. If all LAS files have the same data format, check the *Apply to all LAS files in this set* check box to apply this data import definition to all LAS files selected to be imported.

c. Click the button next to any additional data in the *Source Fields* section to import that data or click the *Add All Fields* button to add all data. For any additional rows that were added to the *Table Fields* section, click the button to not import that data.

d. After associating the *Name* to the *Source* data and defining any *Units*, click *Next* to display the next data import dialog.

6. After defining all imported data, the final **Drillhole Data Import** dialog appears.



*Example **Drillhole Data Import** filter dialog.*

a. Under *Drillholes to Display*, either click *All* to display all imported drillhole data on the map or click *Filter* to filter the data to only display a filtered group of drillholes. Use the five *Filter Conditions* rows to build a filter using fields in the collars table. Click a filter to view or select a previously used filter. If you choose to display all drillholes, you can still filter the display of drillholes using the *Drillholes to display* property on the **General** page of the drillhole layer properties.

2. The *Case sensitive compares* and *Allow ? and * wildcard in text compares* options control how text is compared in the query.
   - When the *Case sensitive compares* check box is checked, the text returned by the filter conditions must have the same case.
   - When the *Allow ? and * wildcard in text compares* check box is checked, a ? can be used to represent any single character and an * can be used to represent a group of characters.

c. After defining any needed filters, click the *Finish* button to create and display the drillhole map.

7. After creating a drillhole map, view a 3D Drillhole Properties, define drillhole properties, Add a Drillhole Layer, or use the Drillhole Manager to manage imported data.

Add a Drillhole Layer

To add a drillhole layer to an existing map:

1. Select the map and click **Home | Layer | Add to Map | Drillhole** or right click over the map layer and select **Add to Map | Drillhole**.
2. If drillhole data has already been imported into the plot document, then the **New Drillhole Layer** dialog appears.
    a. If you have new drillhole data you want to display in the new layer, click the *Add New Data* button to display the **Drillhole Data Import** dialog. Follow the steps above in the Create a Drillhole Map section, beginning with Step 2.
    b. If you want to use the same drillhole data to create the new layer, perhaps using different filter conditions, select the desired settings under **Drillholes to Display** and **Filter Conditions** and click *OK*.
3. If drillhole data has not already been imported into the plot document, then follow the steps above in the Create a Drillhole Map section, beginning with Step 2.

Note: When adding new drillhole data, you may receive a warning about adjusting map limits to include all data. If you select to adjust the map limits, and then decide to not include the new data, you can readjust the map limits to the original data (see the Limits Properties help topic).

Tips and Hints

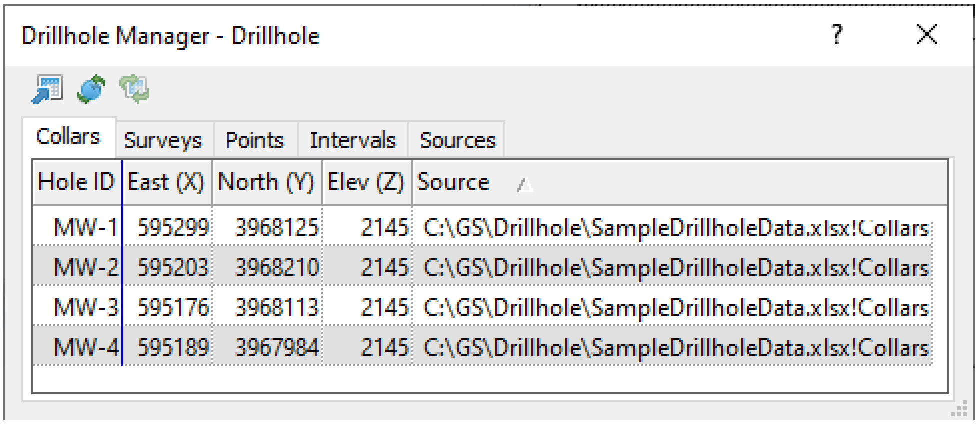
You can create a 3D model of vertical drillholes or boreholes using just collars data, if your collars data include ending depth values. The example below shows the first four columns of the collar data in the sample data file, with added end depth values (shown in blue). During the data import of the collars data, associate the collars table *End Depth* field with the end depth values in your collars data. You can also associate the collars *Start Depth* field to start depth values in your data. If you don't have or don't associate start depth data, Surfer will use a start depth of 0.

Example collars data with added EndDepth values:

ID	Easting	Northing	Elevation	EndDepth
MW-1	595299	3968125	2145	690
MW-2	595203	3968210	2145	670
MW-3	595176	3968113	2145	500

Drillhole Manager

The **Drillhole Manager** lets you view and manage imported drillhole or borehole data. To open the **Drillhole Manager**, select the Drillhole layer in the **Contents** window, and in the **Properties** window, click the *View* button on the **General** page. Alternatively, you can right click the Drillhole layer in the **Contents** window and click *Open Drillhole Manager* or click **Map Tools | Layer Tools| Open Drillhole Manager**.



Example ***Drillhole Manager*** *using SampleDrillholeData.xlsx*

View and Edit Data

Click the **Collars**, **Surveys**, **Points**, and **Intervals** tabs to view the imported data. Data can be edited in any of these tables, except in the *Hole ID* or *Source* fields. The changes will be saved in the **Drillhole Manager**, but the changes will not be saved to the source data files. The **Sources** table contains a list of data files imported into the **Drillhole Manager** and the date the data was last updated. This table cannot be edited.

Import and Reload Data

To import new data into the **Drillhole Manager**, click the  button. Select this option to add additional drillholes that have not yet been imported. Follow the same steps used to import the initial data.

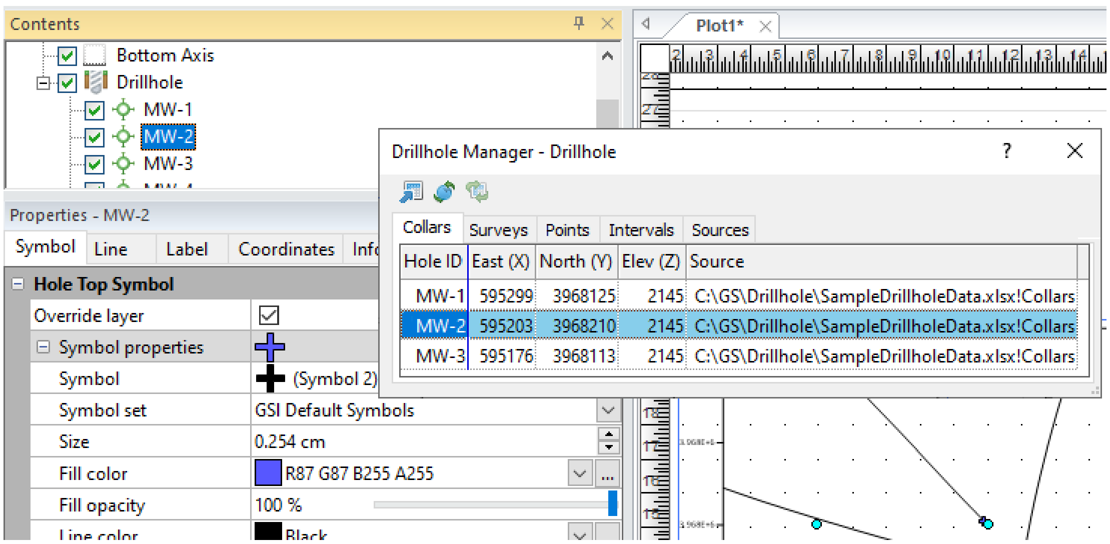
To reload all of the data files imported into the **Drillhole Manager**, click the  button.

To reload one or more existing data sources, select the **Sources** tab, select one or more data sources, and then select the  button to reload only the selected sources. This is a good option to select when only one or a few data files have updated data, so all files do not have to be reloaded.

Tips and Hints

The **Drillhole Manager** can also be used to help locate a specific drillhole. When a row is selected in the **Drillhole Manager**, the drillhole for that Hole ID is selected in the **Contents** window and in the plot window. This is useful when working in the **Drillhole Manager** and you want to change individual drillhole attributes, such as its symbol.

For example, when the **Drillhole Manager** is open, click one of the Hole IDs (Hole ID MW-2 is selected in our sample data example below). The selected Hole ID will be selected on the map and in the **Contents** window, and its properties will be displayed in the **Properties** window. In the **Properties** window, click on the **Symbol** page, if that page is not already active. If you check the *Override layer check* box in the **Hole Top Symbol** section, you can define a new symbol for the selected drillhole. When you select another Hole ID in the **Drillhole Manager**, that drillhole will be selected on the map and its properties displayed in the **Properties** window.



*Example **Drillhole Manager** showing a selected Hole ID and its symbol properties*

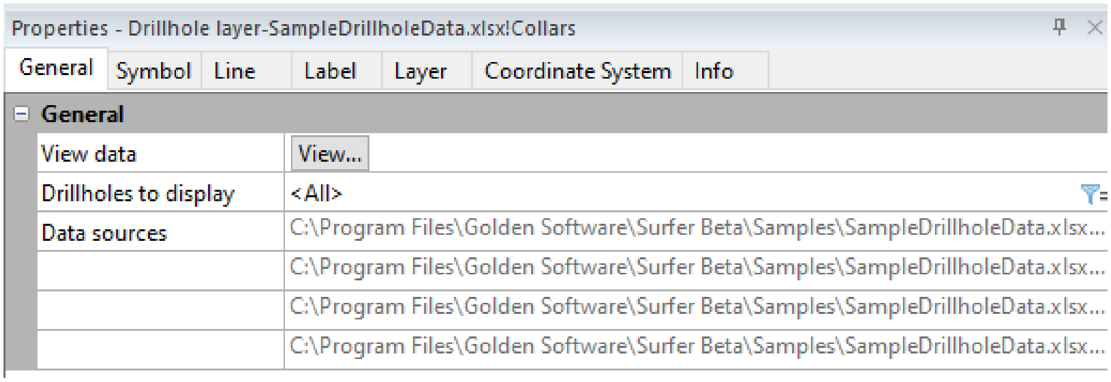
Drillhole Properties

A drillhole layer's properties can be modified in properties pages. Properties can also be modified for an individual drillhole (see Individual Drillhole Properties) and for the 3D view (see 3D Drillhole Properties). To define drillhole layer properties, click on the drillhole layer in the **Contents** window.

This help topic contains three main sections:
Drillhole Properties
Individual Drillhole Properties
Legend Properties

General Page

The **General** page of the **Properties** window contains the properties for viewing, reloading, and filtering drillhole data.



*Drillhole layer **General Properties***

View Data

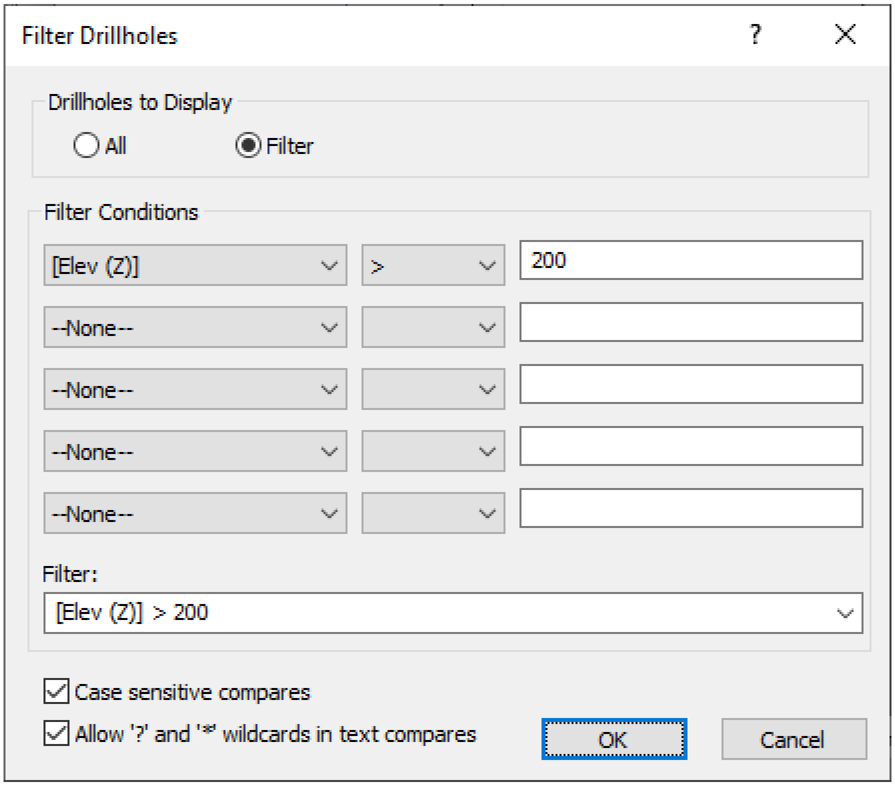
Select the *View* button to open the **Drillhole Manager** that allows you to manage imported drillhole data. See the Drillhole Manager help topic for more information.

Drillholes to Display

Use the *Drillholes to display* property to filter the drillhole data. If a filter was defined during the initial data import, that filter condition will be listed in the *Drillholes to display* property in the **General** page. If no filter was defined, *All* will be listed in the *Drillholes to display* property.

This filter is an inclusion filter to define the drillholes you want to display on the map. To create or edit a drillhole filter:

1. Click the button to the right of the *Drillholes to display* property. The **Filter Drillholes** dialog will appear.



*Example **Filter Drillholes** dialog*

2. Under *Drillholes to Display*, click the *Filter* option to filter the imported drill-hole data. Use the five *Filter Conditions* rows to build the *Filter* using fields in the collars table or type the filter directly into the *Filter* box. Click a filter row to select a previously used query.
3. The *Case sensitive compares* and *Allow ? and * wildcard in text compares* options control how text is compared in the filter query.
   - When the *Case sensitive compares* check box is checked, the text returned by the filter conditions must have the same case.
   - When the *Allow ? and * wildcard in text compares* check box is checked, a ? can be used to represent any single character and an * can be used to represent a group of characters.
4. After defining a filter, click the *OK* button to refresh the drillhole layer with only the drillholes that match the filter conditions.

Data Sources

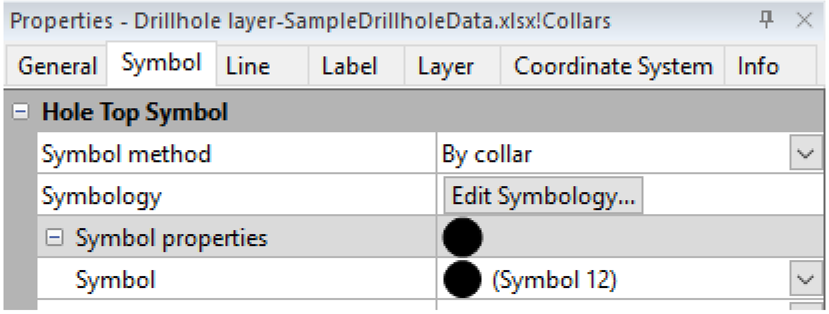
The *Data sources* property lists the data files that have been imported. You can also view the imported file list in the Drillhole Manager.

Symbol Page

The **Symbol** page for a drillhole layer has two sections: *Hole Top Symbol* and *Hole Bottom Symbol*. *Hole Top Symbol* defines the symbol properties for the location of the top of each drillhole, and *Hole Bottom Symbol* defines the symbol properties for the location at the bottom of each drillhole. If the drillhole is vertical, the location of the top of the hole and bottom of the hole will be at the same XY position, but they will have a different Z position. If inclination and azimuth data were imported, then the X, Y and Z positions are different.

Hole Top Symbol

The *Hole Top Symbol* defines the symbol properties for the top of the drillhole, using the East (X) and North (Y) location values imported into the collars table. The symbol properties can define a fixed symbol or different symbols using collars data.



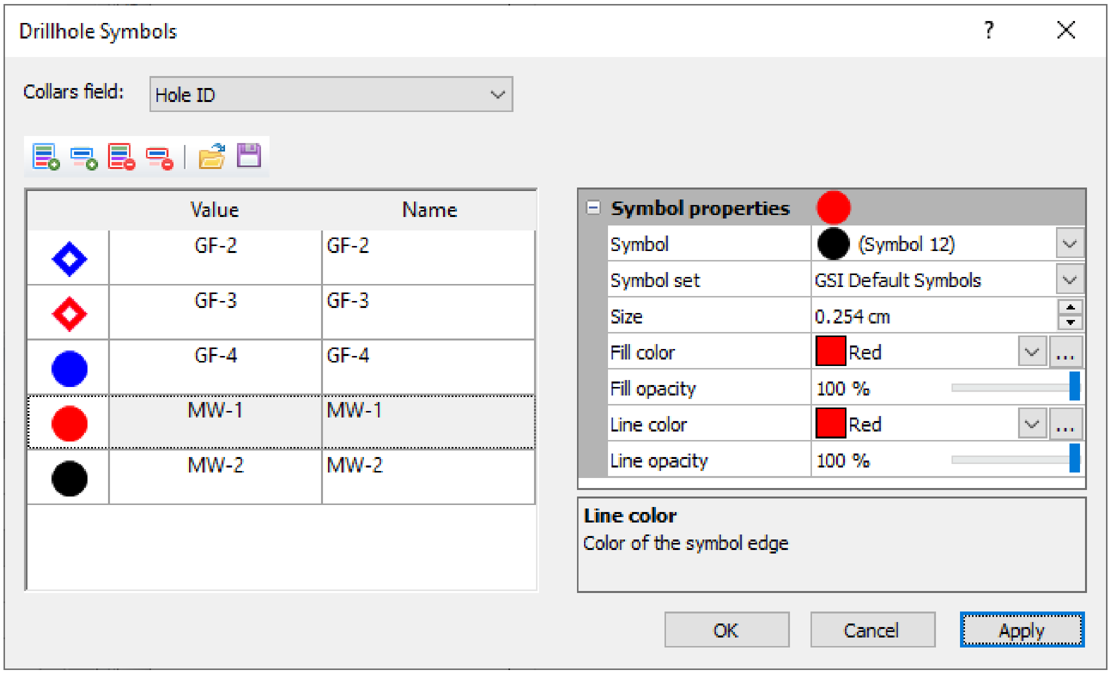
Example drillholes symbols properties

Symbol Method

To define fixed symbols so that all symbols are the same, select *Fixed* from the *Symbol method* list. See Symbol Properties for more information on setting common symbol properties. To define symbols that are specific to selected collars data, select by *By collar* from the *Symbol method* list, and then click the *Edit Symbology* button. The **Drillhole Symbols** dialog will appear.

Drillhole Symbology

The **Drillhole Symbols** dialog has two main sections: the section on the left displays the values of specific collars data to which the symbols will be applied, and the section on the right contains the properties that define those symbols.



*Example **Drillhole Symbols** dialog*

Follow these steps to create drillhole symbols.

1. To use an existing drillhole symbols definition, click the button, locate a saved drillhole symbols file, and click *Open*.
2. To define a new set of drillhole symbols:
   a. Select a value from the *Collars field* list to use as the hole top symbol and then use one of the buttons below to select specific collars values.
      - Click the button to add all values from the selected collars data.
      - Click the button to display a pick list of collars data.
      - Click the button to remove all collars data that were added to the collars values.
      - Click one or more values from the collars values and click the button to remove the selected rows.

   b. After the collars values are set, click one or more of the collar data values and use the *Symbols properties* section to define the symbols for those values. See Symbol Properties for more information on setting common symbol properties. After defining one or more symbols, you can click the *Apply* button to view the changes on the map.
   c. After defining symbols for all collar data values, click the button to save the symbol definition to a file.
   d. After saving changes to a file, click the *OK* button to close the dialog.

Hole Bottom Symbol

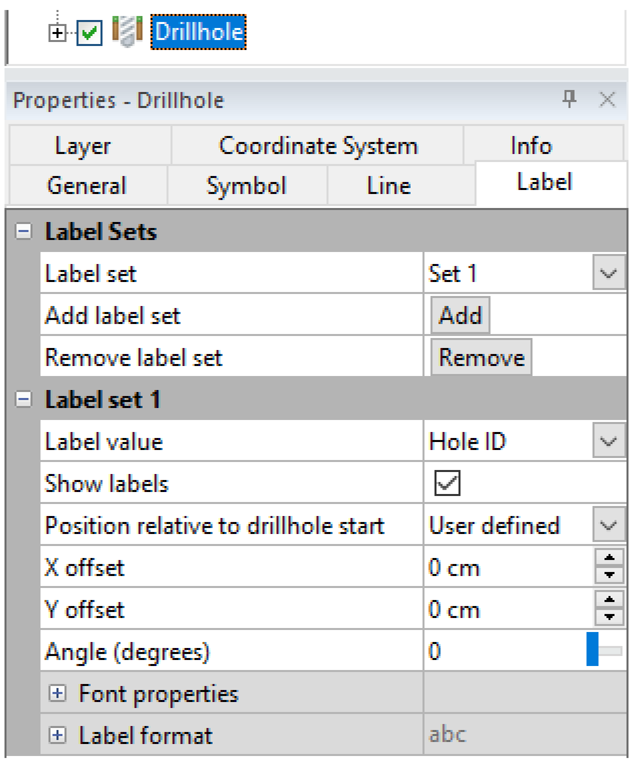
Check the *Show symbol* check box in the **Hole Bottom Symbol** section to display a symbol at the XY location of the bottom of each drillhole. See Symbol Properties for more information on setting common symbol properties.

Line Page

Check the *Show deviation path* check box to show the drillhole path defined in the survey data. If you did not import survey data, but you did import azimuth/inclination collars data, the path will be a straight line. See Line Properties for information on setting the other line properties.

Label Page

Use the *Label Sets* properties to label the drillholes on the map using any field in the collars data. You can label the drillholes using one or multiple collars fields by creating one or multiple label sets. For information on common label properties, such as *Font* and *Label Format*, see the Label Properties help topic.



*Example **Label** page properties*

Label Sets

The *Label Sets* section controls which label is being controlled by the *Label Set #* section. Multiple labels can be displayed by changing the *Label Sets* settings. Click the ⊞ next to *Label Sets* to open the *Label Sets* section.

Note: If only one label needs to be customized (e.g., moved to a different position), click on the specific drillhole in the **Contents** window. See the Individual Drillhole Label Page section below for more information.

Label Set

The *Label set* displays the name of the label that is currently being changed. Click on the set name (*Set 1*, for instance) to select a different set name from the list. If only *Set 1* is displayed in the list, only a single label set is currently created. All of the *Label Set #* properties apply to only the selected *Label set*.

Add Label Set

The *Add label set* option allows new labels to be added. Click the *Add* button to create a new *Label set*. The *Label set* option automatically changes to the new set. If this is the second label to be added, a *Set 2* is added to the *Label set* list. If this is the third label to be added, a *Set 3* is added to the *Label set* list, and so on.

Remove Label Set

The *Remove label set* option deletes a label from the *Label set* list. To delete a label, first set the *Label set* to the label that should be deleted. Then, click the *Remove* button next to *Remove label set* to delete the label. All properties of the label set are removed.

Label Set

The *Label Set #* section controls the worksheet column, position, angle, font, and label properties for the selected *Label set* . Before making any changes in this section, change the *Label set* option to the desired label. The *Label Set #* changes to display the name of the *Label set* . For instance, if *Label set* is set to *Set 3* , the *Label Set #* section name changes to *Label Set 3*.

Label Value

From the *Label Value* list, select the collars column containing the values or text you wish the *Label set* to display near the drillholes. Labels may be the original data values for the drillholes, or may be other identifying text such as well names or sample numbers.

Position Relative to Drillhole Start

The *Position relative to drillhole start* controls the offset of the selected label from the symbol center. To change the position, first change the *Label set* to the desired label. Then, click on the existing position in the *Position relative to drillhole start* option and select the new position from the list. Available options are *Center*, *Left*, *Right*, *Above*, *Below*, and *User defined*. *User defined* allows you to specify the exact offset (in page units) in the *X offset* and *Y offset* boxes. The labels are all placed in the same position relative to the associated symbol.

Positive offset values move the labels in the positive axis direction. For reversed axes, this means that the label is moved in the opposite direction of the *Position relative to symbol* specified. For instance, if the bottom axis is reversed, setting the *Position relative to symbol* to *Left* will move the label to the right of the symbol (in the negative direction). Setting the *Position relative to symbol* to *User defined* and setting the *X offset* to 0.25 inches will move the label 0.25 inches in the negative axis direction.

X Offset and Y Offset

When the *Position relative to symbol* option is set to *User defined*, the *X offset* and *Y offset* options are available. The *X offset* controls the location of the label in the left-right direction. Positive values in the *X offset* shift the label position toward the positive direction (to the right of the symbol for normal axes, to the left for reversed axes). Negative values in the *X offset* shift the label position toward the negative direction (to the left of the symbol for normal axes, to the right for reversed axes). A value of zero places the label directly in the center of the symbol.

The *Y offset* controls the placement of the label in the up-down direction. Positive values in the *Y offset* shift the label position toward the positive direction (upward for normal axes, downward for reversed axes). Negative values in the *Y offset* shift the label position toward the negative direction (downward for normal axes, upward for reversed axes). A value of zero places the label directly in the center of the symbol. The *X offset* and *Y offset* values are numbers between -4 and +4 inches (-10.16 and +10.16 centimeters).

Angle (degrees)

The *Angle (degrees)* box specifies the angle in degrees to rotate the selected label set. Positive angles rotate the labels counterclockwise. To change the *Angle (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position.

Font

Click the ⊞ next to *Font Properties* to display the Font Properties section. The font properties are applied to all labels in this label set. Different label sets can contain different font properties.

Format

Click the ⊞ next to *Label Format* to display the Label Format section. The numeric format is applied to all numbers read from the specified label column in this label set. Different label sets can contain different label formatting.

When the *Type* is set to *Date/time*, if *Invalid date* text is displayed instead of the actual date/time format, the value for the label is outside the defined date/time range.

Layer Page

Use the *Opacity* slider to define the opacity of the drillholes.

Coordinate System Page

See Assign Coordinate System for coordinate system information.

Info Page

See Info Properties for general information on **Info** pages and on how to define attributes.

Individual Drillhole Properties

In the **Contents** window, expand a drillhole layer to show each of its drillholes. Check or uncheck the check box next to one of the drillholes to show or not show that drillhole. Select one of the drillholes to define its properties. Individual drillholes have five **Properties** pages.

Note: Some of the check boxes in the individual drillhole property pages have three settings instead of just checked and unchecked.

- A filled check box means that the individual drillhole object property always matches the drillhole layer property. Changes made to the drillhole layer property are applied to each individual drillhole.
- Checked and unchecked check boxes mean that the property settings for an individual drillhole are applied and override any setting in the drillhole layer properties.

Individual Drillhole Symbol Page

The individual drillhole **Symbol** page for hole top and hole bottom symbols is similar to the drillhole layer symbol properties. The individual **Symbol** page has an *Override layer* check box for the hole top symbol and the hole bottom symbol. When checked, this box overrides the drillhole layer symbol properties. If an *Override layer* box is not checked, then only the drillhole layer symbol properties are displayed. The *Show symbol* check box in the **Hole Bottom Symbol** section has three settings: a filled check box uses the drillhole layer properties; checked shows the symbol and overrides the drillhole layer properties; and unchecked does not show the symbol and overrides the drillhole layer properties.

Individual Drillhole Line Page

The individual drillhole **Line** page is similar to the drillhole layer line properties. The individual **Line** page has an *Override layer* check box that overrides the drillhole layer line properties. If this box is not checked, then only the drillhole layer line properties are displayed. The *Show deviation path* check box has three settings: a filled check box uses the drillhole layer properties; checked shows the deviation path and overrides the drillhole layer properties; and unchecked does not show the path and overrides the drillhole layer properties.

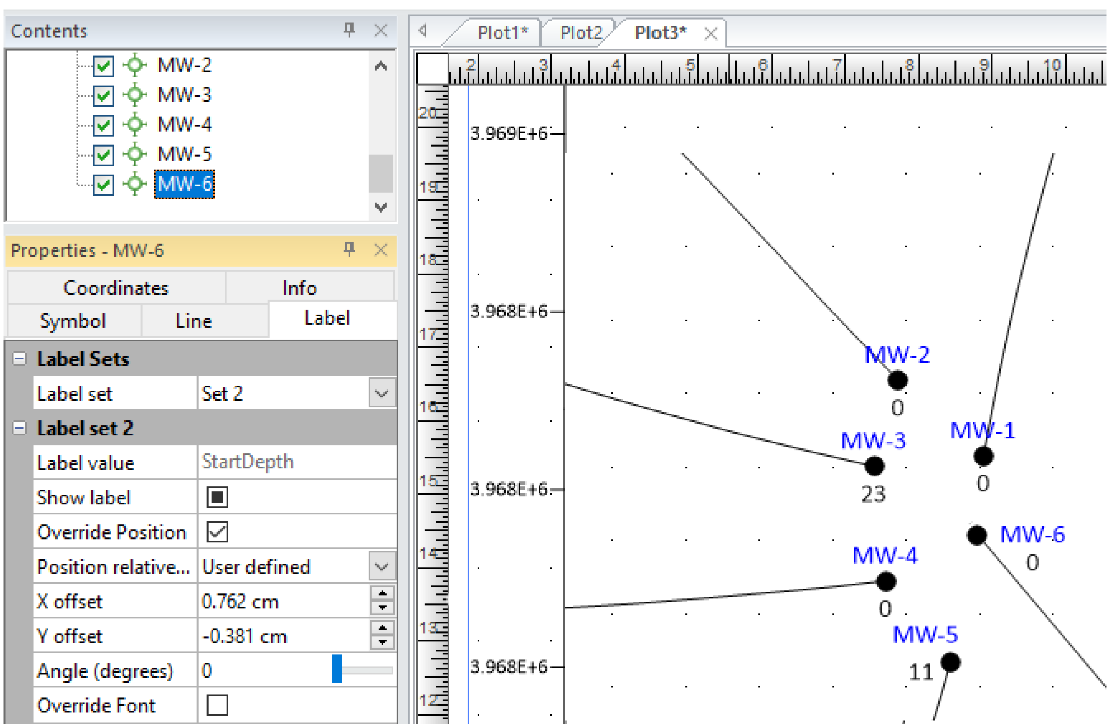
Individual Drillhole Label Page

Label sets are added in the drillhole layer properties and cannot be added or changed for individual drillholes. The label position, font and format for an individual drillhole can be changed. The individual drillholes **Label** page has three check boxes (*Override Position*, *Override Font*, and *Override Format*) that override the drillhole layer label properties. A filled check box uses the drillhole layer properties; checked applies the property to the individual drillhole and overrides the drillhole layer properties; and unchecked does not apply the property and overrides the drillhole layer properties.

See the Label Sets section above for more information on label sets and the shared drillhole layer and individual drillhole label properties. See the Font Properties and Label Format help topics for information on common label properties.

Example Individual Drillhole Labels

The following image is an example of two label sets. *Set 1* and *Set 2* create labels from drillhole collar data: Hole ID (in blue) and StartDepth (in black). The individual drillhole, "MW-6", is shown with its *Override Position* box checked, so a *User defined Position relative to drillhole start* property could be applied to change the *Set 2* StartDepth *X offset* value to move the label to a more readable position.



Example drillhole labeling with two collars data fields

Individual Drillhole Coordinates Page

This page shows the X, Y and TVD (True Vertical Depth) coordinates for the specific drillhole. The X and Y values are from the collars data. The TVD is calculated from the azimuth and inclination data (either in a collars or survey data), along with either the start and end depths (collars data) or the measured depth data (survey data). If there is no depth data, then Surfer uses the depth data available in the points or interval data.

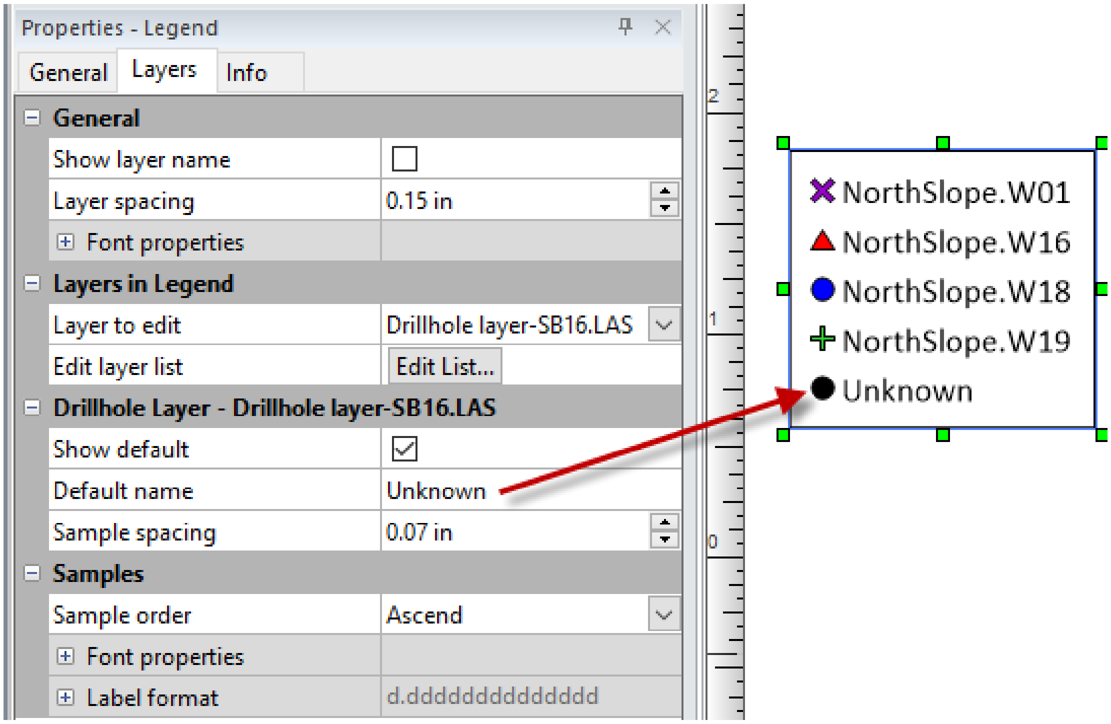
Note: When drillholes are deviated, the true vertical depth can be calculated by Surfer. Several calculation methods exist, and each method creates a mathematical approximation of the true X, Y, and Z value along the drillhole path. Surfer uses the tangential method for calculating true vertical depth from measured depth. This method is probably the simplest method for computing true vertical depth. It uses only the inclination and direction measured at the lower end of the borehole or drillhole. The drillhole path is assumed to be a straight line throughout the course. This method has historically been used more than any other, but can be the least accurate. Drillholes calculated with the tangential method can appear too shallow and the lateral displacement along the drillhole can be too large. The error is minimized if short intervals are used between points.

Individual Drillhole Info Page

This page contains information about the selected drillhole that can also be copied to the clipboard. All the fields from the collars data are displayed here with the values for the selected drillhole. If needed, an attribute can be created.

Legend Properties

If the drillhole layer **Symbol** property for *Symbol method* in the **Hole Top Symbol** section is set to *By collar*, you can add a legend to a drillhole layer. Click the **Map Tools | Add to Map | Legend** command to add a legend and then click the *Legend* object in the **Contents** window to display legend properties. Most of these properties, such as the **General** and **Info** properties, are similar to legends in other maps (see the Legend help topic).



*Example drillhole **Legend Layers Page** properties*

Drillhole Layers Legend Properties

The **Drillhole Layer** section of the **Layers** page is unique to drillholes.

- Check *Show default* to add the symbol displayed for drillholes that were not added to the symbology table in the drillhole properties.
- In the *Default name* field, enter a name to use to name those drillholes.
- Enter a value in the *Sample spacing* field to define the spacing between the lines in the legend.

New Drillhole Layer or Map

Chapter 21 - Downloading Layers from a Server

Base Map from Server

Click the **Home | New Map | Base | Base from Server** command or the button, the **Home | Add to Map | Layer | Base from Server**command, or the **Map Tools | Add to Map | Layer | Base from Server** command to download a base map from an online web server. Clicking one of the preceding commands opens the Download Online Maps dialog. Use the **Download Online Maps** dialog to select the server, map extents, and image resolution.

When using the **Download Online Maps** dialog, a server with appropriate data must be selected. Surfer currently supports four types of servers, web mapping service (WMS), Open Street Map (OSM), web coverage service (WCS), and web feature service (WFS). WMS, OSM, WCS, and WFS servers provide different types data.

- When selecting a WMS or OSM server, an image is downloaded to be used as a base (raster) layer.
- When selecting a WCS server, data values are downloaded to be used as a grid file. WCS servers cannot be used with the **Base from Server** command. Use the Grid from Server command or the *Download* button in the Open Grid dialog instead.
- When selecting a WFS server, vector data is downloaded with points, polylines, and/or polygons representing features in a base (vector) layer.

Grid from Server

Click the **Grids | New Grid | Grid from Server** command or the button to download data from an online web mapping server and create a new grid file. Clicking the **Grid from Server** command opens the Download Online Grids dialog.

The **Download Online Grids** dialog can also be accessed from the Open Grid dialog by clicking *Download*. The downloaded grid will be used to create a grid-based map layer or in the following grid command dialog, depending on which command was used to open the **Open Grid** dialog.

When using the **Download Online Grids** dialog, a server with appropriate data must be selected. Surfer currently supports four types of servers, web mapping server (WMS), Open Street Map server (OSM), web coverage server (WCS), and web feature server (WFS). WMS, OSM, WCS, and WFS servers provide different types data.

- When selecting a WMS or OSM server, an image is downloaded to be used as a grid file. There are no Z values associated with the image. When the image is used as a grid file, a value between 0 and 1 is assigned to each pixel in the image based on the color of that pixel. WMS and OSM servers are generally used to create base maps with the **Home | New Map | Base | Base from Server** command.
- When selecting a WCS server, data values are downloaded to be used as a grid file.
- When selecting a WFS server, vector data is downloaded with points, polylines, and/or polygons representing features. WFS servers cannot be used with the **Grid from Server** command.

Download Online Maps or Grids

The **Download Online Maps** and **Download Online Grids** dialogs are used in the same manner to download data from online servers when creating a map. The **Download Online Maps** dialog is displayed when creating a base map. The **Download Online Grids** dialog is displayed when creating a grid-based map. The difference between the two dialogs is very minor and is discussed in the **Select Image Resolution to Download** section below.

Click the **Home | New Map | Base from Server** command or **Map Tools | Add to Map | Layer | Base from Server** command to open the **Download Online Maps** dialog.

Click the **Grids | New Grid | Grid from Server** or the *Download* button in the Open Grid dialog to open the **Download Online Grids** dialog.

This dialog contains a server list and download options for downloading data from a web mapping service (WMS), Open Street Map (OSM), web coverage service (WCS), or web features service (WFS). Data can be downloaded from any of the predefined servers:

- Download images from the predefined WMS servers in the *US Imagery* or *Worldwide Imagery* categories. Use these images to create a base map. Images can be used to create a grid-based map, but the grid values will be an arbitrary color index on a scale from 0 to 1. In almost all cases WMS servers should not be used for grid-based maps.

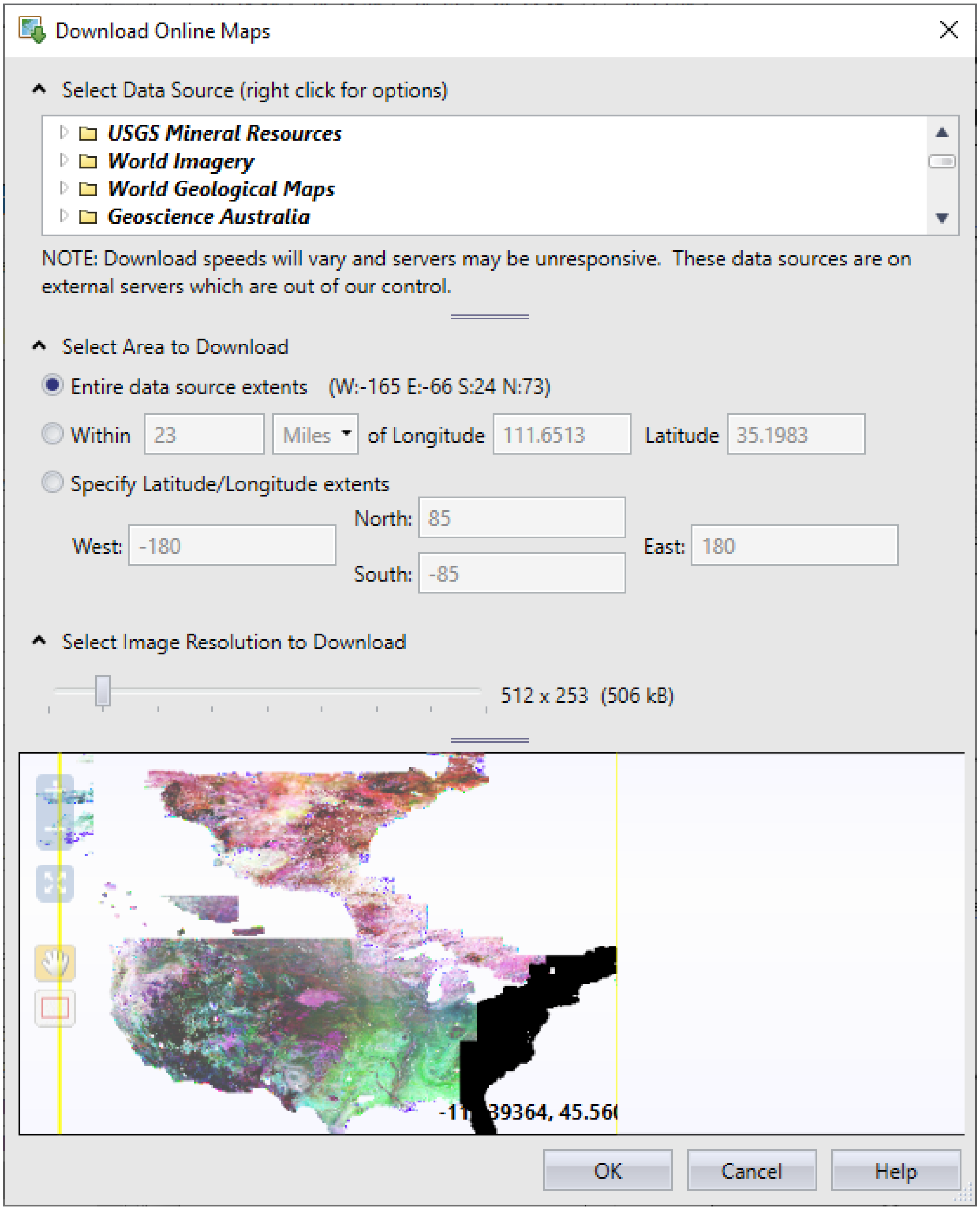
- Download images from the predefined OSM servers in the *OpenStreetMaps Imagery* category. Use these images to create a base map. Images can be used to create a grid-based map, but the grid values will be an arbitrary color index on a scale from 0 to 1. In almost all cases OSM servers should not be used for grid-based maps.

- Download vector data from the predefined WFS servers in the *US Vector Data* or *Worldwide Vector Data* categories. Use the vector data to create a base map. WFS servers cannot be used to create a grid-based map. **Surfer** can download data from WFS protocol 1.0.0, 1.1.0, and 2.0.0 servers.

- Download grids from the predefined WCS server. Use the grid data to create a grid-based map. WCS servers cannot be used to create a base map.

New categories and servers can be added to the list.

The **Download Online Maps** and **Download Online Grids** dialogs can access servers that require a proxy server connection. **Surfer** uses the proxy server settings set in the Windows **Internet Options**, accessed via the Windows Control Panel.



Select or add the server, specify extents and resolution, and download layers in the ***Download Online Maps*** *dialog.*

Data Source

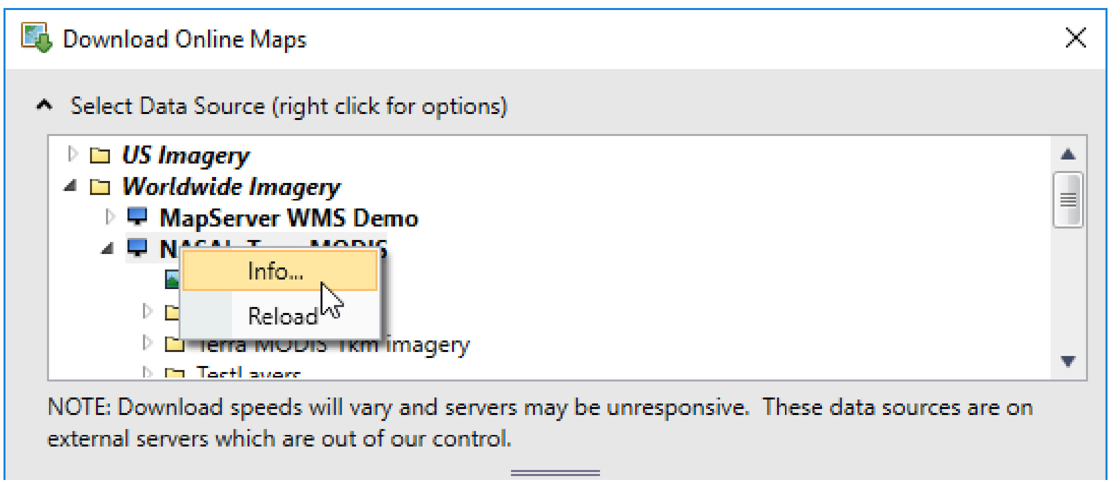
The *Select Data Source* list contains servers and layers, separated in categories, from which map data or images can be downloaded. The categories are *US Imagery*, *Worldwide Imagery*, *OpenStreetMaps Imagery*, *US Vector Data*, *Worldwide Vector Data*, and *Favorites*. These categories include sample servers. In the *Select Data Source* section, click the arrow ▷ to expand a category or server. The arrow turns black ◢ when a section is opened. Click the black arrow ◢ to collapse the category or server.

Adding New Categories

Right-click on an existing category or in the white space of the *Select Data Source* list and click **Add New Category** to add a user-defined category to the *Select Data Source* list. After clicking **Add New Category**, the a new category is created at the bottom of the list. The category can be renamed immediately after adding the category. If desired, type in a new name for the category. Then press ENTER or click elsewhere in the *Select Data Source* list to finish creating the category.

Server and Layer Information

To see additional options for a specific server, right-click on the server in the *Select Data Source* section, such as the **NASA Terra Modis** server.



Right-click on the server name to see the URL for the server and information about the server's layers.

Click **Info** to view data source and layer information for the selected server. The information is presented in the View Map Source dialog. The **View Map Source** dialog also displays layer information such as the minimum scale, height or width, layer name and title. None of the options are editable.

The layer list from the data source is cached on the local hard drive. This reduces the load time for the *Select Data Source* list. Update the layer information by right-clicking on the data source and selecting **Reload** in the context menu.

To copy the server and layer name, right-click on the layer name and select **Copy to Clipboard**. You can then paste the text to any text editor, word processing program, or to the window. The pasted information includes the name of the server, the server location, the layer title and layer name selected.

Arranging the Data Source List

Categories can be rearranged by clicking and dragging the category name above or below other categories. Layers and servers can be rearranged within a category in the same method. Layers can also be dragged and dropped into a different category.

Favorites Section

To add any layer to the *Favorites* section, right-click on the layer name and select **Add Favorite**. Only the selected layer is added to the favorites. Other layers from the same server are not added automatically.

To add a new server to the *Favorites* section, right-click on the *Favorites* category and select **Add Map Source**. Use the Add Map Source dialog to add the data source. To copy any existing server to the *Favorites* category, click and drag the server to the *Favorites* folder.

To remove a server or layer from the *Favorites* section, right-click on the server or layer name in the *Favorites* section and click **Delete**. Click *Yes* in the dialog to remove the server or layer from the *Favorites*. Click *No* in the dialog to keep the server or layer in the *Favorites*. Removing the server or layer from the *Favorites* section does not remove the server or layer from any other section.

Adding New Data Sources

To add new data sources to the list, right-click on the a category in the *Select Data Source* section. Click **Add Map Source** to open the Add Map Source dialog. Set the *Name, Type,* and *URL* to the new web service. Click *Validate* and the web server will be tested. If the test is successful, click *OK* and the new web service is added to the *Select Data Source* list. If the test is not successful, verify the *URL* and *Type* are correct.

All custom data sources are saved to an XML file, allowing multiple Golden Software programs to use the custom definitions. Servers that have been added in previous **Surfer** versions or in other **Golden Software** programs will be automatically added to the server list. The XML file is located at C:\Users\<username>\AppData\Roaming\Golden Software\Shared\Geode.xml. The file can be copied from one computer to another to share custom data sources between multiple computers and users.

When sharing the Geode.xml file between a previous version of **Surfer** and the current version, the *Data source* list may include servers with unsupported protocols for the previous **Surfer** version. If this happens, the unsupported server will be displayed in the *Data Source* list. However, the server name will be disabled in the list, and you will not be able to download data from the server.

Editing Data Sources

To edit the data source for any user-defined server, right-click on the server name and click **Edit**. The Edit Map Source dialog updates the web server *Name, Type,* and *URL*. This may be useful if the location for the server changes or if the name listed in the **Download Online Maps** dialog should be changed. The **Edit Map Source** dialog also displays information about the server and layers. Predefined servers cannot be edited.

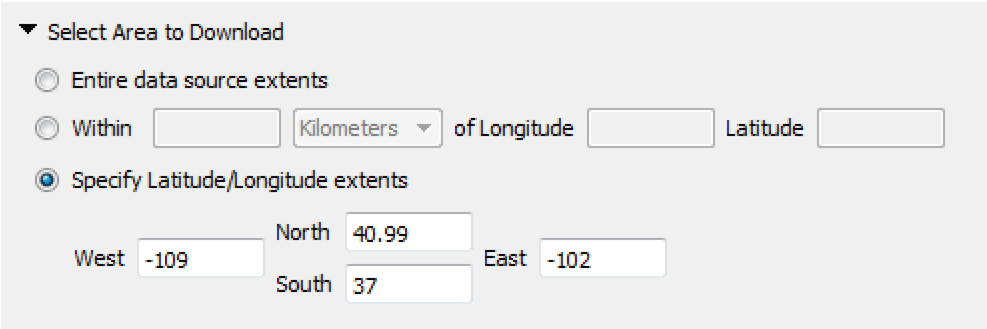
To view information about the server, right-click on the server name and click **Info**. The View Map Source dialog opens and displays the server and layer information. The *Name*, *Type*, and *URL* cannot be edited in the **View Map Source** dialog.

Deleting Data Sources

To delete a user-defined server from the *Select Data Source* list, right-click on the server name and click **Delete**. In the **Delete Data Source** dialog, click *Yes* to delete the server. Click *No* to keep the server in the *Select Data Source* list. Predefined servers cannot be deleted.

Select Area to Download

The *Select Area to Download* section controls the areal extent of the image being downloaded. Available options are *Entire data source extents, Within <distance> <units> of Longitude <Lon> Latitude <Lat>*, or *Specify Latitude/Longitude extents*. Click on the desired option to select it.

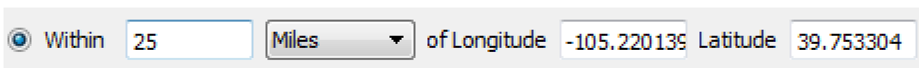


Select the extents of the map to download in the Select Area to Download section.

The *Entire data source extents* reads all of the data on the specified server. All of the data is downloaded in the single image. With servers that cover large areas, this option is not recommended because even at high resolutions, very little detail will appear on the downloaded map.

The *Within <distance> <units> of Longitude <Lon> Latitude <Lat>* option allows a single longitude and latitude value to be entered. A square area is downloaded based on the *distance* value and the *units* option. The square is centered on the value entered for the *Longitude* and *Latitude*. To use this option, enter valid values in each box.

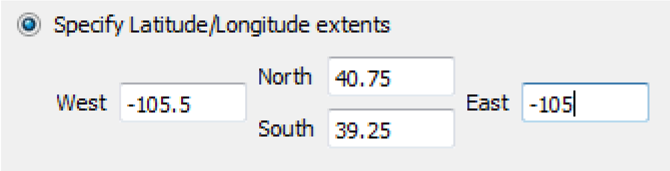
- The first box sets the width and height of the square area. For instance, 25 can be input.
- The second option (units) can be set to either *Kilometers* or *Miles*. Click on the existing option to select the desired option.
- The third option is the central *Longitude* value. In the example below, the center point of the downloaded image will be -105.220139. Longitude values must be between -180 and +180. Negative values are western hemisphere longitudes. Positive values are eastern hemisphere longitudes.
- The last option is the central *Latitude* value. In the example below, the center point of the downloaded image will be 39.753304. Latitude values must be between -90 and +90. Negative values are entered for the southern hemisphere. Positive values are entered for the northern hemisphere.



The downloaded area will be 25 miles wide and 25 miles tall. The image will be centered on the longitude and latitude location of -105.220139, 39.753304.

The *Specify Latitude/Longitude extents* option allows a rectangular area to be input in latitude and longitude coordinates. When adding a base map layer to an existing map with a defined coordinate system, this option is automatically selected. The boxes are filled in with the map limits of the current map. After selecting this option,

- Type in the western-most edge for the downloaded map in longitude degrees coordinates in the *West* box. Longitude values must be between -180 and +180. Negative values are western hemisphere longitudes. Positive values are eastern hemisphere longitudes.
- Type in the eastern-most edge for the downloaded map in longitude degrees coordinates in the *East* box. Longitude values must be between -180 and +180. Negative values are western hemisphere longitudes. Positive values are eastern hemisphere longitudes.
- Type in the northern-most edge for the downloaded map in latitude degrees coordinates in the *North* box. Latitude values must be between -90 and +90. Negative values are entered for the southern hemisphere. Positive values are entered for the northern hemisphere.
- Type in the southern-most edge for the downloaded map in latitude degrees coordinates in the *South* box. Latitude values must be between -90 and +90. Negative values are entered for the southern hemisphere. Positive values are entered for the northern hemisphere.



The downloaded area will cover from -105.5 to -105 degrees longitude and 39.25 to 40.75 degrees latitude.

Select Image/Grid Resolution to Download

The *Select Image Resolution to Download* section controls the quality of the image being downloaded. This section is called *Select Grid Resolution to Download* in the **Download Online Grids** dialog. Drag the slider to the right to increase the image resolution. Click a point on the slider bar and the slider will jump to that resolution. The farther to the right the slider is located, the better the resolution and the larger the image. Clicking on one of the lines on the left side of the slider will normally download a map of sufficient quality that is smaller in size. The larger the file size, the longer the image will take to process and the more memory the program will use to manipulate the image. If N/A is displayed or no size information is displayed, the image cannot be downloaded at the requested resolution. Change the resolution by moving the slider to select a different size image.

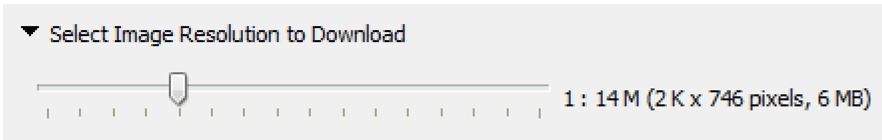
The selected WMS server may have an original data source resolution (for example, 1 meter/pixel) for its imagery. You may be able to select an image resolution higher or lower than this original data source resolution. Increasing the image resolution here cannot give you better image detail over the original data source. By requesting a higher resolution than the original data source, the image is simply larger in size and in the number of pixels. The server returns the requested image in the requested size. Some servers provide resolution information, and if so, this information and be found in the View Map Source dialog. If the server does not provide resolution information, consider checking the server provider's documentation.

OSM servers may have more than one layer available. When this is the case, increasing the *Image Resolution to Download* may download a more detailed image. For example with the *Carto Light* predefined server, using the default *Image Resolution to Download* will download the image as displayed in the preview, and increasing the resolution will download an image that includes more geographic names and more detailed streets. Just as with the WMS servers, eventually increasing the *Image Resolution to Download* will exceed the original data source resolution for the most detailed layer, and this will result in downloading larger images with no appreciable difference.

WFS servers provide vector data. When a WFS server is selected the *Select Image Resolution to Download* slider is disabled.

If a WCS server only provides only one resolution, such as the *SRTM Global 90m (GMU)* predefined server, the *Select Grid Resolution to Download* slider is disabled.

In some instances a ⊗ may be displayed next to the image size estimate. This occurs when the selected resolution is outside the recommended scale for the sever. Hover the cursor over the ⊗ to display information about the warning. Generally this warning is removed by increasing the *Image Resolution to Download* by moving the slider to the right or reducing the *Area to Download* selection.



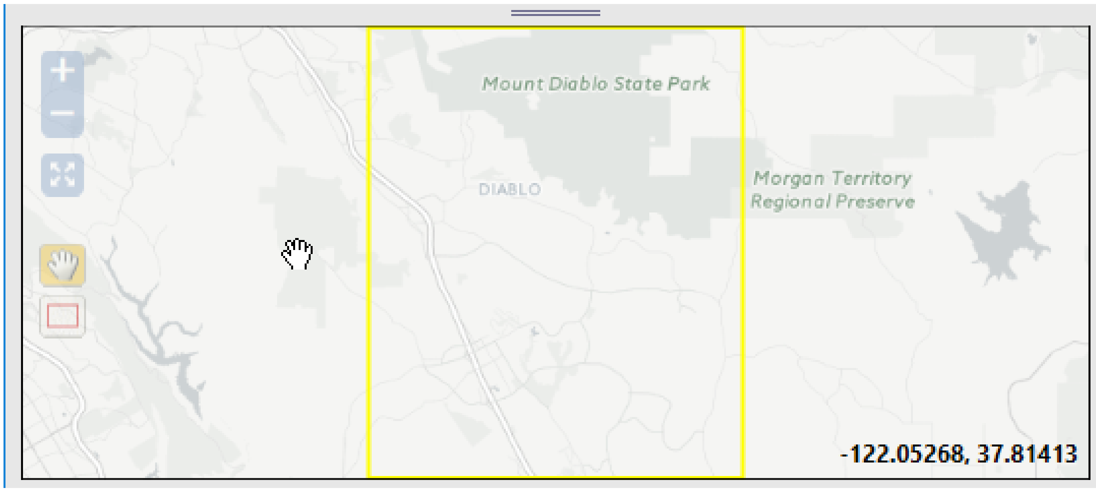
Set the image quality by dragging the slider to the right or left.

Image Preview

The *Preview* section displays a picture of the area to be downloaded. The *Preview* section uses low resolution previews. The final output image quality is controlled by the *Select Image Resolution to Download*. Some servers use scale-dependent rendering. If a preview is not displayed but no errors are visible in the *Data Source* or *Preview* sections, then try zooming in and/or out until you find the correct scale for the server.

- For OSM and WMS servers, the *Preview* section contains a low resolution preview of the image.
- For WCS servers, the *Preview* section shows a coverage footprint over a world map.
- For WFS servers, the *Preview* section shows a coverage footprint over a world map. The preview can also include vector data for the selected area to download. The preview is not displayed automatically for vector data from WFS servers, because all vector data must be downloaded first. The data download and rendering may take a few seconds or up to a few minutes depending on the size of the area to download and the WFS server. Set the *Select Area to Download* before enabling the vector data preview.

  The vector data preview must be enabled manually by clicking the button. If you preview the vector data and make no changes to the area to download extents, it will not be necessary to download the data again when clicking *OK*. If the preview is taking longer than you would like to wait, click the *Cancel* button in the lower right portion of the dialog, next to the preview status bar.



The Preview *window displays the area to download, buttons to zoom in/out, pan, or change the download area, a map scale, and the coordinates of map cursor.*

- Click the buttons on the left side of the image preview to zoom in or out on the area. Changing the zoom level does not affect the area to download. The area to download is indicated in the *Preview* section by a yellow outline.

- Click the button on the left side of the image preview to zoom to the selected area extents. Changing the zoom level does not affect the area to download. The area to download is indicated in the *Preview* section by a yellow outline.

- Click the button on the left side of the image preview to quickly move the image in the *Preview* section. The cursor changes to , indicating pan mode. Click and hold the left mouse button down and drag the image to change the view; the cursor changes to to indicate the left mouse button is held. Changing the view does not affect the area to download. The area to download is indicated in the *Preview* section by a yellow outline.

- Click the button on the left side of the image preview to change the extents of the image that is downloaded. The cursor changes to , indicating draw mode. Click and hold the left mouse button down and drag the mouse over the area to download. The zoom extents update and the yellow box coincides with the area drawn. The *Select Area to Download* also updates. Only the portion of the image highlighted by the yellow box will download.

- Click the button to download the vector preview for the selected area to download. Additionally, the vector data preview may be simplified in the **Download Online Maps** dialog. The icon is displayed if preview simplification has occurred. Even if the preview is simplified, all vector data will be added to the map layer when clicking *OK*.
- If the server or selected layer returns an warning, a and/or is displayed in the *Preview* section. Hover the cursor over the warning icon to view more information about the warning.

OK, Cancel and Help

Once all of the options are set, click *OK* to download the image, grid, or vector data. Click *Cancel* to cancel the download and return to the **Surfer** plot window. Click *Help* to open the help file. A progress bar is displayed showing download progress and number of bytes (kB, MB, or GB) downloaded.

Clicking *OK* in the **Download Online Maps** dialog downloads the map layer and creates a new base map or adds a base layer to the select map frame. Clicking *OK* in the **Download Online Grids** dialog opens the Save Grid As dialog or creates a new grid based layer.

If the *OK* button is not available, the layer is not selected. Go back to the *Select Data Source* section and select a specific layer, not a server name.

Server Responsiveness

Note: Download speeds will vary, depending on the server selected. Some servers may become unresponsive. These data sources are on external servers which are out of Golden Software's control. If one server is unresponsive or slow to download, you may wish to select a different server.

Naming Convention

Base maps from servers are named with the map server name, such as *Base-USGS_EROS_Ortho_NAIP*. In addition, the image downloaded with the base map from server adds attribute information, including the server name and layer title.

Grid-based maps created by clicking *Download* in the Open Grid dialog are named with the map type and server name, for example *Contours-NED Contiguous U.S. 1 3W arc second elevation data*.

Changing the Window and Section Sizes

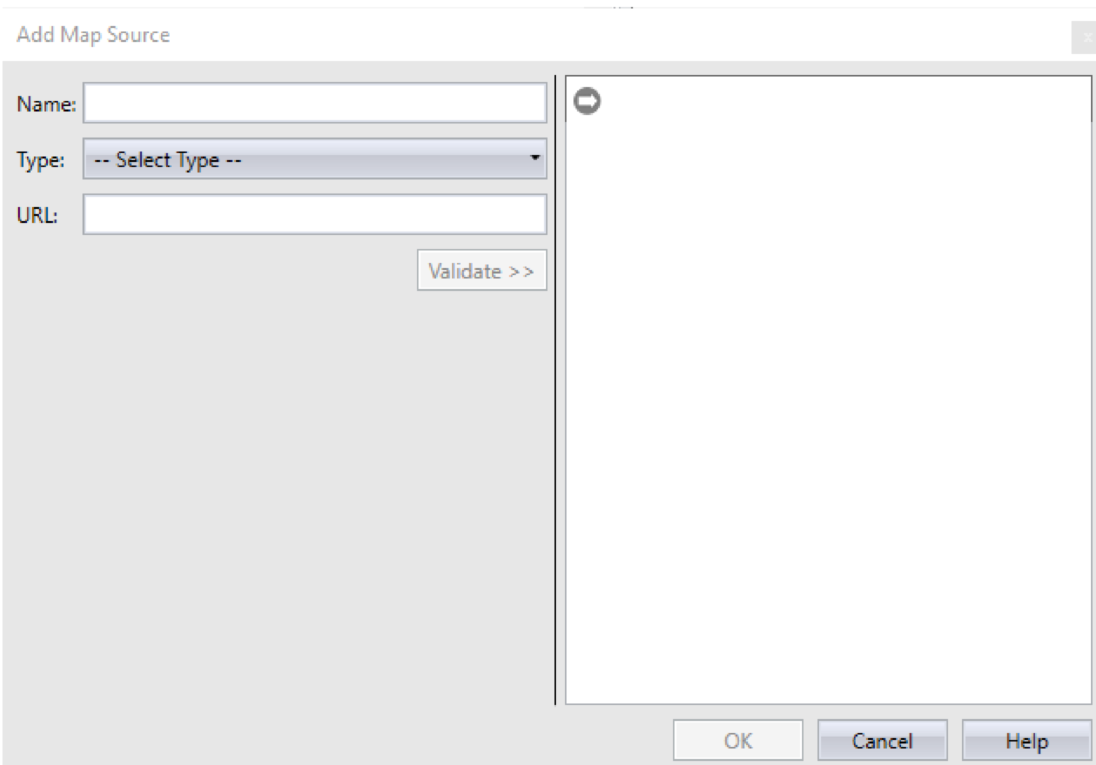
The **Download Online Maps** and **Download Online Grids** dialogs can be resized by clicking and dragging a side or corner, as with normal windows. The image preview and *Select Data Source* sections can also be enlarged or reduced by clicking and dragging the double horizontal bars between the sections of the dialog.

Map Source Dialogs

The following map source dialogs are accessed via the **Download Online Maps** or **Download Online Grids** dialog. The map source dialogs add, edit, and show information for source servers.

Add Map Source Dialog

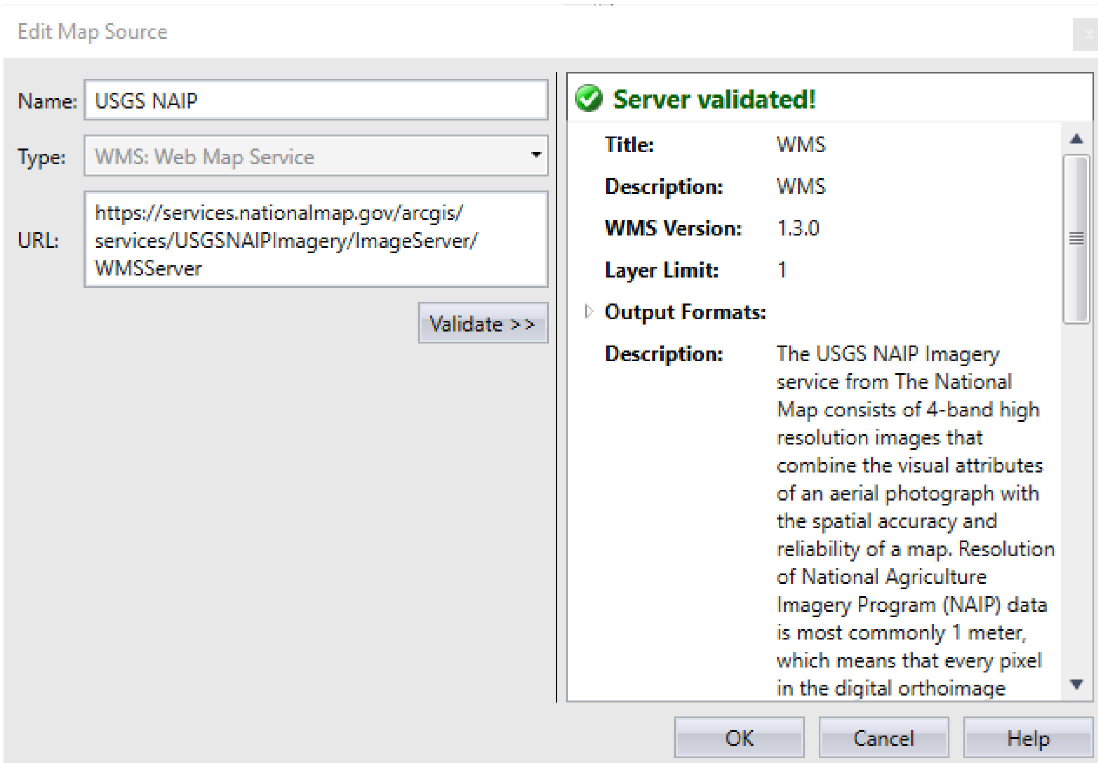
Right-click on an existing category in the Download Online Maps dialog, and then click *Add Map Source* to open the **Add Map Source** dialog. This dialog adds new web servers to the list in the **Download Online Maps** dialog.



*The **Add Map Source** dialog adds a new server to the selected category. Notice the* Validate *button is disabled. The* Validate *button is enabled once a* URL *is added to the **Add Map Source** dialog.*

Edit Map Source Dialog

Right-click on a user-added map source in the Download Online Maps dialog, and then click **Edit** to open the **Edit Map Source** dialog. This dialog is the same as the **Add Map Source** dialog, but includes the server information for the selected map source. Use the **Edit Map Source** dialog to edit the map source *Name* and *URL*.



*The **Edit Map Source** dialog allows you to change the* Name *and/or* URL *for the selected server. The **Edit Map Source** dialog validates the server immediately once the dialog is opened. The **Edit Map Source** dialog also displays server and layer information for the selected server.*

View Map Source Dialog

Right-click on an pre-existing server in the Download Online Maps dialog, and then click **Info** in the context menu to open the **View Map Source** dialog. This dialog shows server and layer information. Editing the server name and URL is disabled in the **View Map Source** dialog.

Dialog Controls

The following section describes the purpose and use of the components in the **View Map Source**, **Edit Map Source**, and **Add Map Source** dialogs.

Name

In the *Name* section, type the name of the server. This is the name that is listed in the **Download Online Maps** dialog, so the name can be as descriptive as desired.

Type

Select the *Type* from the list. Available types are *WCS: Web Coverage Service*, *WFS Web Feature Service*, *WMS: Web Map Service*, or *OSM: Open Street Maps*. **Surfer** can download vector data from WFS protocol 1.0.0, 1.1.0, and 2.0.0 servers.

URL

In the *URL* section, type the full internet location of the web service. The address must contain the http:// or https:// before the location. Sites with https:// or sites that redirect to SSL servers are supported.

Open Street Map URLs

OSM Open Street Map server URLs can be defined for multiple tiles by using variables in the URL:

- Use ${s[*i*-*j*]} where *i* and *j* are the tile letters or numbers you wish to include in the data source, for example http://${s[1-5]}.toolserver.org/tiles/hikebike/${z}/${x}/${y}.png specifies five tile servers for the Hike & Bike data source.
- Use ${s[*a*,*b*]} where values *a*, *b* are the comma-separated tile letters or numbers you wish to include in the data source, for example http://${s[a,b,c]}.tile.openstreetmap.org/${z}/${x}/${y}.png specifies the three tile servers for the OSM Mapnik data source.

Example OSM URLs are displayed in the **Add Map Source** dialog when *OSM: Open Street Maps* is selected in the *Type* field. Also, when copying and pasting an OSM URLs from a web browser, the %7B and %7D characters will be automatically converted to { and }.

Use Specific Protocol Version

When contacting a server, **Surfer** allows the server to pick which protocol is used. Occasionally, a server will select a protocol that does not work well or does not work at all. Try specifying the protocol in the *Use specific protocol version* list if you are having issues with a specific WCS or WFS server. Protocol 1.0.0 is often the most reliable.

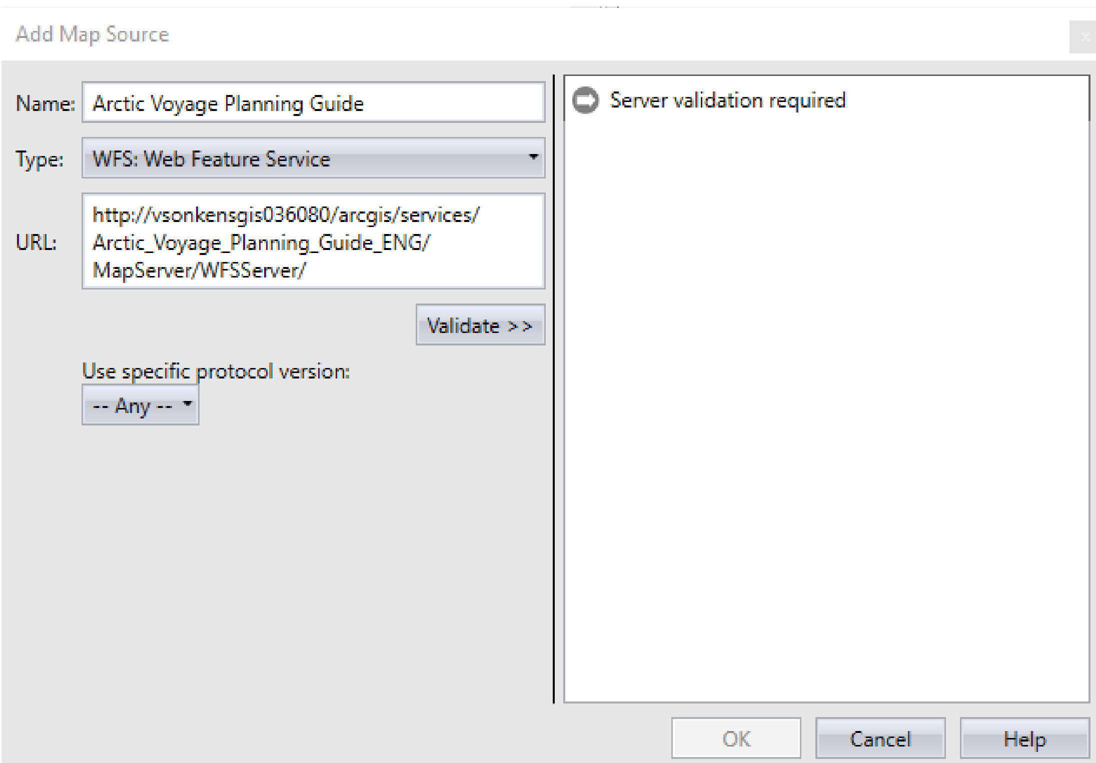
Surfer supports WCS protocols 2.0.1, 2.0.0, 1.1.3, 1.1.2, 1.1.1, 1.1.0, and 1.0.0. **Surfer** supports WFS protocols 1.0.0, 1.1.0, and 2.0.0.

Allow Server Redirects

Check the *Allow server redirects* check box to allow the WCS server to redirect requests from Surfer. Clear the check box to disallow server redirects. If a server validation fails, try clearing the *Allow server redirects* check box. The *Allow server redirects* option is only available when the *Type* is set to *WCS: Web Coverage Service* .

Validate New Server

The **Add Map Source** dialog contains a *Validate* button. The button is disabled when the dialog is first opened. After the *URL* is added for a new server, the information pane on the right displays *Server validation required*, and the *Validate* button is enabled. Click the *Validate* button to test the connection to the server. If the test is successful *Server validated!* is displayed in the information pane, as well as the server and layer information. If the server validation fails, an error message is displayed in the information pane. If you wish to stop the validation process, click the *Cancel validation* button to stop the server validation.



Server and Layer Information

The right pane of the **View Map Source** , **Edit Map Source** , and **Add Map Source** dialogs contains server information such as *Title*, *Description*, *WMS Version*, *Bounding Box*, etc. Click the ▷ to open a section. The arrow turns black ◢

when a section is opened. Different servers will return different information. When a data source contains more than one layer, the layer information is listed for each layer. Click the ▹ to open a layer section. The layer information includes items such as *Description* , *Bounding Box*, *Supported CRS*, etc.

Cancel or Finish

Click *Cancel* or the ✕ button to return to the **Download Online Maps** dialog, without making any changes. Click *OK* in the **View Map Source** to return to the **Download Online Maps** dialog, making any changes to the *Name, Type,* or *URL* in the dialog. Click *OK* in the **Add Map Source** dialog to return to the **Download Online Maps** dialog, and the data source is added to the *Select Data Source* list.

Help

Click *Help* to open the online help file with information about the open dialog.

Chapter 22 - Map Properties

Introduction to Common Map Properties

Map properties common to all layers are located in the Properties window when the *Map* object is selected in the Contents window. These properties include the *view*, *scale*, *limits*, *frame,* and *coordinate system*. If there are multiple map layers in a map frame, the changes made on the **View**, **Scale**, **Limits**, **Frame**, and **Coordinate System** pages apply to all layers. The attributes and options on the **Info** page apply only the map. Each layer can have separate attributes.

View

Maps can be displayed at any orientation using the View page. The *Projection, Rotation, Tilt*, and *Field of view* of a map are changed on the **View** page.

Scale

Map scale controls the size of a map on the printed page. This is accomplished by defining a correspondence between length on the map (in map units) and length on the printed page (in page units). For example, one mile on the map corresponds to one inch on the page. The map scale is set using the Scale page. The map scale for the X axis and the map scale for the Y axis can be set in unison (proportionally), or they can be set independently. On 3D wireframes and 3D surfaces, the map scale for the Z axis can also be set.

Limits

Map limits are defined as the X and Y extents of the map. When a map is first created, the map limits are set to the minimum and maximum X and Y coordinate values in the grid file, base map, or data file used to create the map. You can accept these as the limits for the map, or you can specify custom limits using the Limits page. Custom limits are useful when plotting only a portion of a map.

Frame

In addition to adding color to maps, a background can be added behind the map. The Frame page sets the background fill properties and the border line properties for a map. The map background limits coincide with the axis limits on a contour, base, post, and vector maps. On 3D wireframes, the map background refers to the bottom plane of the base. On 3D surfaces, the background color is blended with the surface.

Coordinate System

The map coordinate system is the target coordinate system for the map. A coordinate system has a defined projection and datum. If some map layers are using a different coordinate system, the layer is converted to the map coordinate system. The map's *Coordinate System* is the new coordinate system that you want to use to project your X, Y coordinate data. 3D surface maps and wireframe maps do not have a coordinate system associated with them. When the map *Coordinate System* changes, these maps do not change.

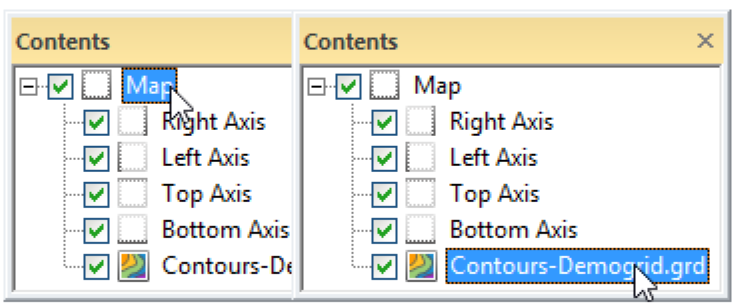
Info

Two different **Info** pages provide information for objects in a map or for a grid layer in a map: Info - Objects and Info - Grids. In addition to read-only information, comments can be added, deleted, and edited to clarify information contained in the map.

Map Properties

Properties common to all layers in a map are displayed in the Properties window when the *Map* object is selected. Specific properties of a map layer are controlled by the specific map layer properties.

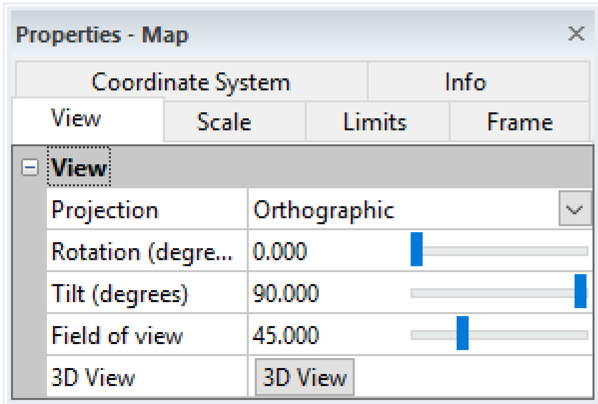
The properties of one or more maps can be opened by selecting the *Map* object (s) in the **Contents** window or by clicking on the *Map* object in the plot window.



Click on the Map object to open the map properties.

Click on the Contours object to open the contour layer properties.

The **Map Properties** control the View, Scale, Limits, Frame, Coordinate System, and Info of **Surfer** map frames.



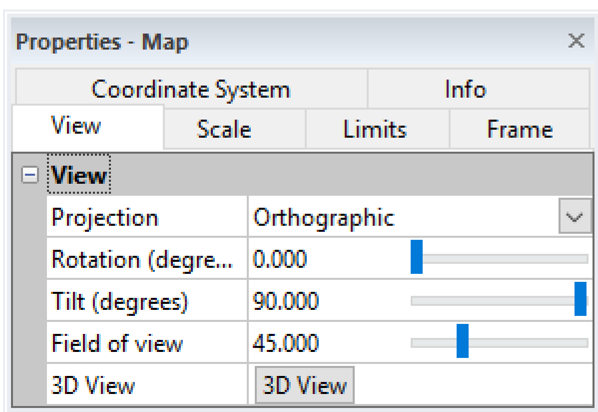
Specify the View, Scale, Limits*,* Frame*,* Coordinate Sys-tem*, and* Info *in the* ***Properties*** *window when a map is selected.*

View Properties

Maps can be displayed at any orientation using the map properties, **View** tab. The projection, rotation, tilt, and field of view of a map are changed on this page.

View Page

The **View** page is located in the Properties window. To open the map properties, click on the *Map* object in the Contents window.



Control the orientation of the map or maps on the ***View*** *page.*

Multiple Maps

When you have more than one map on the page, the **View** settings are applied to all selected maps. For example, this command is useful if you have a contour map and 3D wireframe based on the same grid file and displayed in the same plot window. You can select the map object for both the contour map and 3D wireframe. In the **Properties** window, click on the **View** tab to change the settings for both maps at one time.

Projection

The *Projection* group sets the type of projection for the selected maps. *Perspective* projections more closely simulate reality, but *Orthographic* projections maintain parallel lines and allow accurate measurements to be taken from the drawing.

Perspective Projection

The *Perspective* projection creates a visual effect whereby the size of the surface varies with the distance from the observer. When a *Perspective* projection is used, you can change the *Field of View* value to increase or decrease the perspective effect.

Orthographic Projection

The *Orthographic* projection creates a plot with a projection of the surface onto a plane, oriented perpendicular to the line of sight between the observer and the surface.

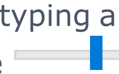
Rotation

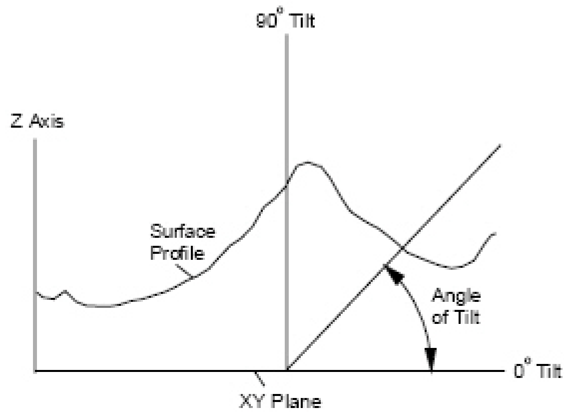
The *Rotation (degrees)* value controls the rotation of the map about the center. As the *Rotation (degrees)* value increases, the map rotates in a counterclockwise direction.

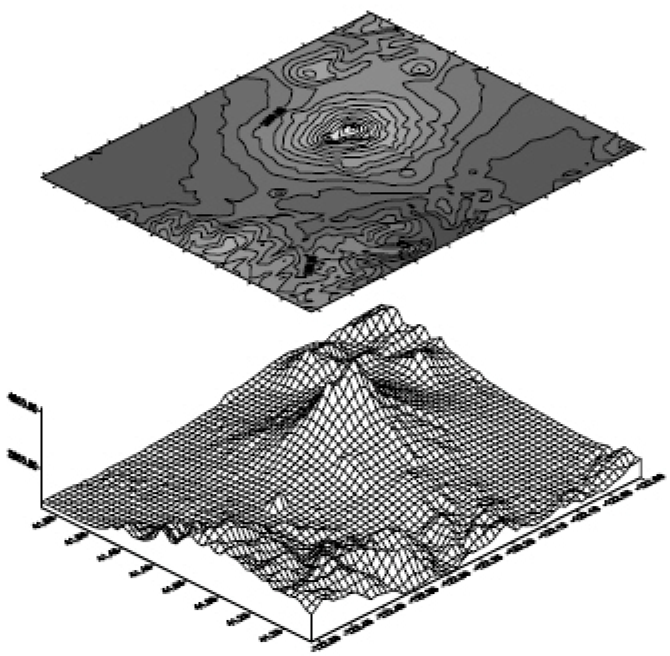
To change the *Rotation (degrees),* click in the box next to *Rotation (degrees)*. Highlight the existing value and type a new value. Values are always displayed with three digits after the decimal. Press ENTER on the keyboard after typing a new value to show the value on the map. Alternatively, click and drag the  to a new value. Values must range between zero and 360.000.

Tilt

Tilt refers to the orientation of the Z axis in the map display. A zero degree tilt means the Z axis lies in the plane of the page or screen. As the angle of tilt increases, the Z axis tilts toward the viewer. A 90.000 degree tilt means the XY plane lies in the plane of the page or screen and is orthogonal to the viewing direction. The default tilt is 30.000 degrees for 3D wireframes and 3D surfaces, and 90.000 degrees for all other map types. When you tilt three dimensional maps (i.e. 3D surface) to 90.000 degrees, the surface relief is no longer visible.

To change the *Tilt (degrees),* click in the box next to *Tilt (degrees)*. Highlight the existing value and type a new value. Values are always displayed with three digits after the decimal. Press ENTER on the keyboard after typing a new value to show the value on the map. Alternatively, click and drag the  to a new value. Values must range between zero and 90.000.

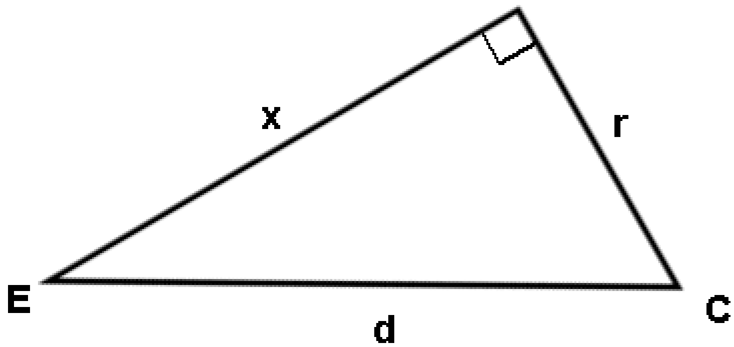




Maps can be displayed at any rotation angle and at any tilt angle from zero to 90 degrees. When more than one map is selected, the view parameters are applied to all selected maps.

Field of View

The *Field of View* controls the perspective effect of the *Perspective* projection option. The perspective effect is the visual effect of the map changing size relative to the distance from the observer. A small *Field of View* shows the map in perspective as the if the observer was far from the viewpoint, making the perspective more pronounced. A large *Field of View* shows the map in perspective as if the observer was very close to the viewpoint. The *Field of View* angle is formed by the left edge of the bounding box, the viewpoint, and the right edge of the bounding box. The image below illustrates the bounding sphere wedged into the viewing cross section.



The field of view cross section shows the position of the eye (E), the center of the surface (C), the distance from the eye to the point (d), and the radius of the bounding sphere around the surface plot (r).

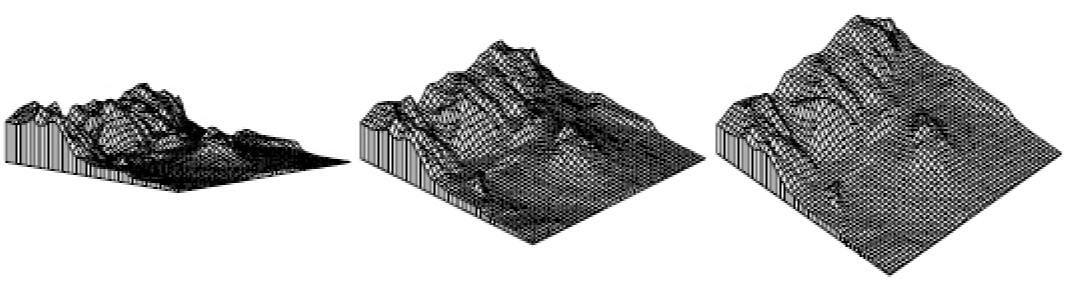
With an *Orthographic* projection the *Field of View* values have no effect. Note that the *Field of View* parameter does not influence the map scale.

To change the *Field of view,* click in the box next to *Field of view*. Highlight the existing value and type a new value. Values are always displayed with three digits after the decimal. Press ENTER on the keyboard after typing a new value to show the value on the map. Alternatively, click and drag the  to a new value. Values must range between one and 160.000.

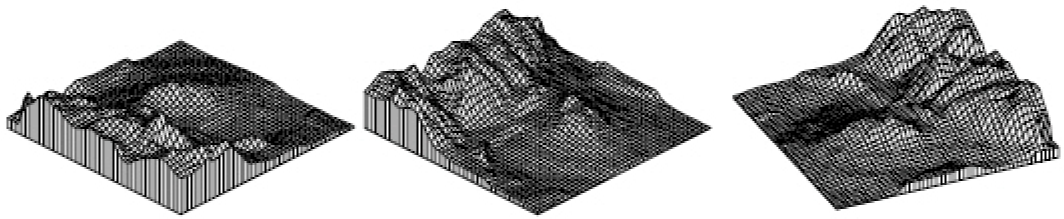
3D View

Click *3D View* to view the map in a new 3D view window. You can also create a new 3D view by selecting the map and clicking **Map Tools | View | 3D View**. The map must include at least one grid-based layer or point cloud map to create a 3D view.

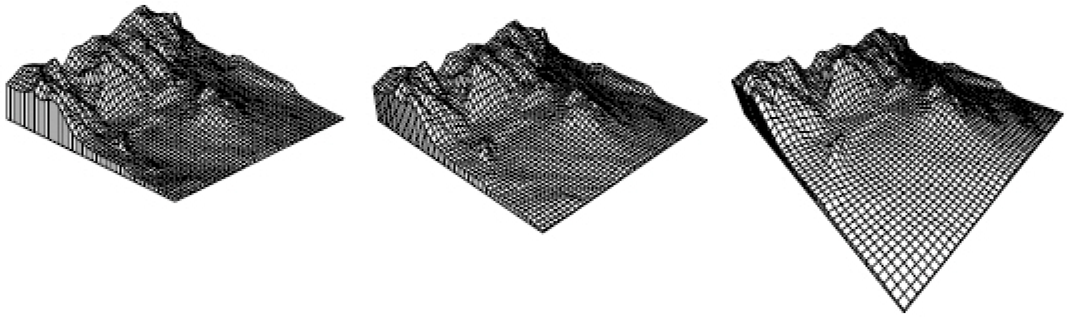
Example of Tilt, Rotation, and Projection Parameters



This is the same wireframe displayed at the same rotation angle but different tilt angles: 10, 30, and 45 degrees.



This is the same wireframe displayed at the same tilt angle but different rotation angles.



This is the same wireframe using different projection parameters. The map on the left uses an Orthographic projection. The map in the middle uses a Perspective projection with a Field of View value of 20 degrees. The map on the right uses a Perspective projection with a Field of View value of 75 degrees.

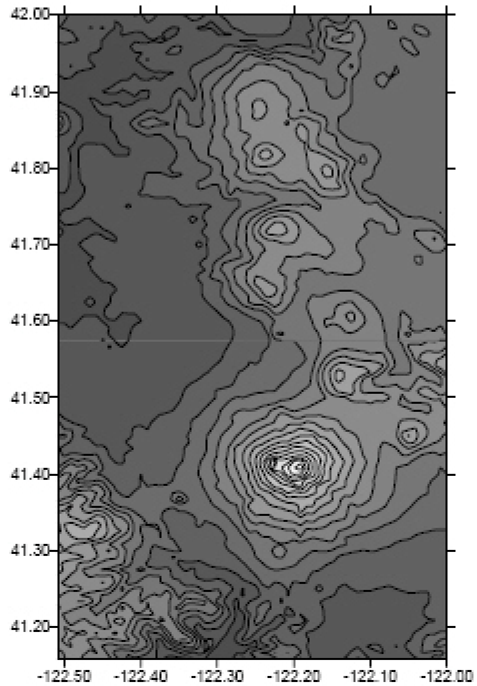
Scale Properties

The scale controls the size of a map or profile on the printed page. This is accomplished by defining a correspondence between lengths on the map (in map units) and lengths on the printed page (in page units). The map scale is specified using the **Scale** page in the Properties window. The scale for the X axis and the scale for the Y axis can be set in unison (proportionally), or they can be set independently. On three-dimensional maps (i.e. wireframes and surface maps), the scale for the Z axis can also be set.

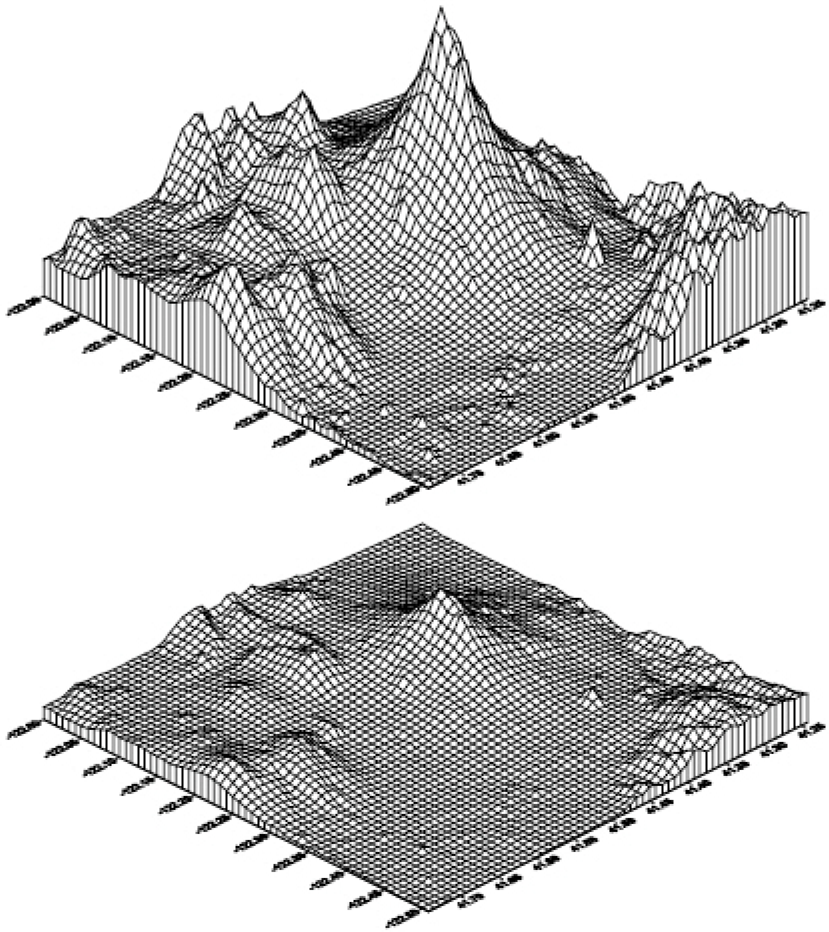
Axes are part of the map on which they are drawn. Therefore, when the scale is redefined, the axes are modified accordingly. Because **Surfer** calculates reasonable tick spacing based on the map lengths and map limits, redefining the scales can result in different tick spacing along the axis.

Default Scale

By default, map scales are defined proportionally in the X and Y dimensions, with the longest side measuring six inches on the printed page. For example, consider a map extending 60 map units in the X dimension and 40 map units in the Y dimension. Since the extent in the X dimension is longer than the extent in the Y dimension, the map is drawn six inches wide. The resulting scale is 10 map units per inch. The scale in the Y dimension is also 10 map units per inch, so the map is drawn four inches in height.



This map scale was customized. The Y dimension covers a greater extent than the X dimension in this map. By default, the Y axis would be six inches in length, and the X axis is proportionally shorter.



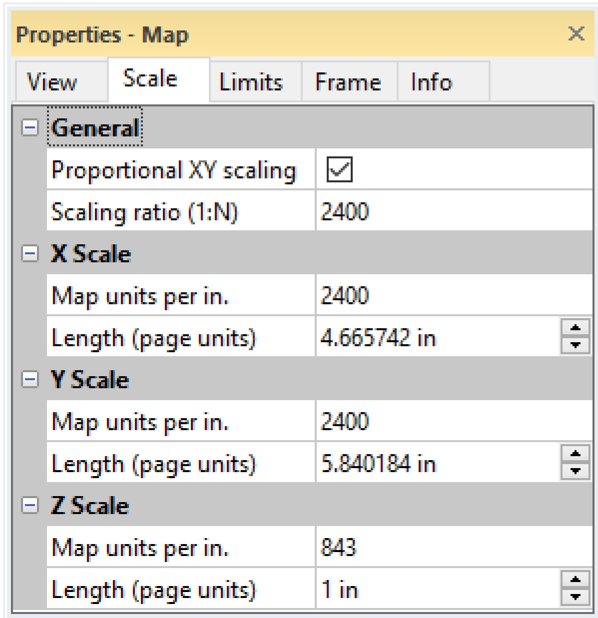
These are two 3D wireframe maps of the same grid, each using a different Z scale. The Z scale is exaggerated in the top view. Use the ***Scale*** *page to change the length of the Z axis.*

On three-dimensional maps the default scale in the Z dimension is defined such that the vertical extent of the map is one-fourth the maximum horizontal extent.

The default scales are a function of the map limits. Map limits are determined by the file used to create the map, or they can be defined using Limits page.

Scale Page

The **Scale** page is located in the map properties.



Specify the relative X, Y, and Z map scaling on the ***Scale*** *page in the* ***Properties*** *window.*

Proportional XY Scaling

The *Proportional XY scaling* check box ensures that the X and Y dimensions are scaled equally. When the box is checked, a change in either the X or Y dimension is reflected in the other dimension by a proportional amount. To scale the axes independently, uncheck the box.

Sizing a map with its selection handles rescales the map internally. To return to the proper scale, use the Home | Selection | Transform command to clear the transform, and then use the **Scale** page to scale the map.

When *Proportional XY scaling* is checked, the *Scaling ratio (1:N)* option is available. The *Scaling ratio (1:N)* property is hidden when *Proportional XY scaling* is not checked.

Scaling Ratio

Specify the X and Y map scale representative fraction by typing the desired value in the *Scaling ratio (1:N)* property field. The *Proportional XY scaling* check box must be checked to scale maps with by representative fraction. Press ENTER or click a different property to update the scaling values. Changing the *Scaling ratio (1:N)* automatically updates the *X Scale* and *Y Scale* sections *Map units per in (cm)* and *Length (page units)* property values.

The representative fraction scaling method shows the map scale as a ratio of distance on the map (1) to distance on the ground (N). For example, a representative fraction of 1:100000 means one centimeter on the map represents 100,000cm on the ground (1000m or 1km). Similarly on a 1:100000 scale map, 1 inch on the map represents 100,000 inches on the ground (8333.3ft or 1.578 miles).

When perspective, tilt, and rotation are applied to a map view, the ratio is honored for the center of the map in the X direction. The ratio may not be valid for other regions of the map, depending on the View page settings.

X, Y, and Z Scale

The *X Scale*, *Y Scale,* and *Z Scale* sections specify the scale along the indicated axis. Each axis can be scaled independently by clearing the *Proportional XY scaling* check box. When the *Proportional XY scaling* box is checked, any changes made to the scale in the X or Y dimension is automatically reflected in the other dimension. Z scaling is always independent of the X and Y scaling. Similarly, any change in the X or Y scale values automatically updates the *Scaling ratio (1:N)* value when *Proportional XY scaling* is checked.

Map Units Per In. (cm.)

The *Map units per in. (cm.)* option sets the number of map units per page unit in the associated dimension. For example, to draw a map at a scale of 1000 map units per inch, type the value 1000 into the box. The *Length (page units)* value is automatically updated to reflect the change. For example, if the map is 8000 units in the dimension you are setting, the map is 8 inches long in that dimension.

Length

The *Length (page units)* option sets the length of the map in the X, Y, or Z dimension. When the *Length (page units)* value is changed, the *Map units per In. (cm.)* box is automatically updated to reflect the change.

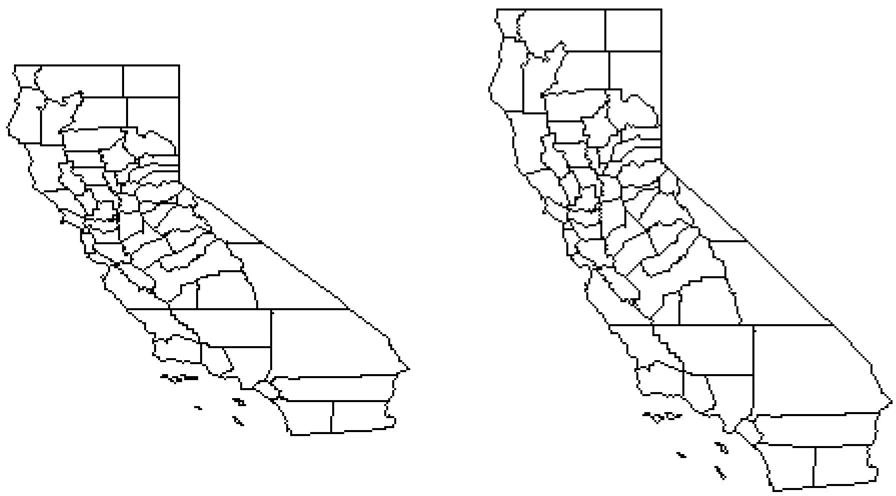
Using Different Scaling in the X and Y Dimensions

Under some conditions, different map units are used for the X and Y dimensions.

Consider a contour map of soil temperature as a function of depth and time. Depth is measured in feet, while time is measured in days. Clearly, one day is not equivalent to one foot. The two axes must be scaled differently because two different types of units are used for the two axes. Depth data might be collected every half foot from zero to four feet in depth, and the time data might be collected every day over a 30-day period. The depth axis extends four units and the time axis extends 30 units. Default scaling would produce a map six inches long in the time dimension by 0.8 inches high in the depth dimension. To create a square map, you must clear the *Proportional XY scaling* box on the Scale page and then set the map scaling so 30 units in the time dimension is equal to four units in the depth dimension. The map is then plotted at six inches by six inches.

Using Scaling to Minimize Distortion on Latitude/Longitude Maps

When you display maps based on latitude and longitude coordinates extending over a large region, they might appear somewhat distorted. This occurs because one degree of latitude is not equivalent to one degree of longitude. For example, consider a base map of the state of California (ca2010.gsb).



This shows the map of California before and after scaling. The map on the right is scaled up in the Y dimension so the map does not appear compressed.

As an approximation, the distance covered by one degree of latitude at the equator is equal to the distance covered by one degree of latitude at the poles, and is approximately 69 miles. This distance between degrees of latitude remains nearly constant over the globe, although it does vary slightly because the earth is not a perfect sphere. However, the distance between a degree of longitude decreases from the equator to the poles. For any latitudinal position, you can determine the length, in miles, between degrees of longitude based on the formula:

Distance covered by 1° of longitude (in miles) = cosine (latitude) x 69.172.

This equation assumes a Clark 1866 reference ellipsoid.

This table illustrates the change as you move from the equator to the poles.

Latitude	**Distance Covered by One Degree of Longitude**
0° (equator)	*69.172 miles*
30°	59.904 miles
60°	*34.586 miles*
90° (poles)	0 miles

Substitutions for units other than miles:

- For kilometers, you can substitute the number 111.321 for 69.172 in the formula above.
- For meters, you can substitute the number 111,321 for 69.172 in the formula above.
- For feet, you can substitute the number 365,228 for 69.172 in the formula above.

So, how can you put this information to use? Remember that you are plotting degrees of latitude and longitude, but what you really want to show on the map are the correct distances. You must scale the longitude values correctly for the correct distances to be represented on the map. The scaling factor to apply for maps is based on the cosine of the latitude for the area you are working on.

To determine the scaling factors:

1. Determine the latitude for the parallel through the center of the map, and obtain the cosine for this latitude value. The center parallel in ca2010.gsb is approximately 37.27 degrees. The cosine of 37.27 degrees is 0.80.

2. Click on the *Map* in the Contents window to select it. The map properties are displayed in the Properties window.

3. Click the Scale tab to set the X direction for the map. You can set either the *Length (page units)* value or the *Map units per in. (cm.)* value. The *X Scale* Map units is 1.72 inches in this example.

4. Uncheck the *Proportional XY scaling* box.

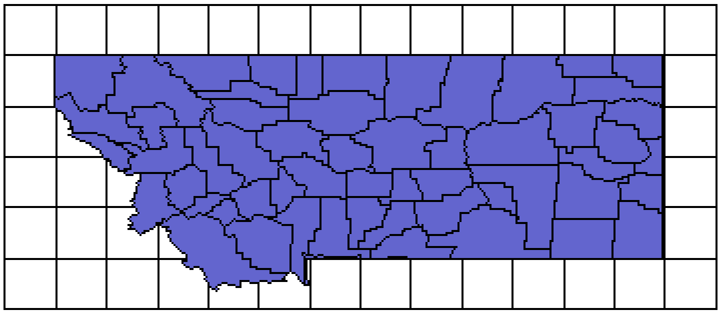
5. Multiply the *X Scale Map units* value by the cosine of the latitude, and enter this number into the *Y Scale Map units* field. 1.72 (*X Scale Map units*) X 0.80 (cosine of latitude) = 1.38 (*Y Scale Map units*). Enter 1.38 into the *Map units per in*. box under *Y Scale*.

The map is automatically updated.

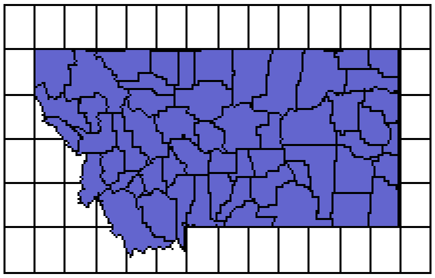
Example

Consider a map of the state of Montana. When you plot the map on a one to one scale, the map appears stretched in the east-west direction. To understand this problem, consider that for Montana the latitude ranges from 44.36° to 49°. The latitude for the center of the map is determined from this to be 46.68°. The cosine of 46.68° is 0.686. The distance covered by one degree of longitude at this latitude is only 0.686 times the distance covered by one degree of latitude. To reduce the distortion on this map, you must correct for this difference.

Let's say you are plotting the map at an X scale of 1" = 2 map units (longitude). For the map to be scaled appropriately, you would plot the Y scale at 1" = 1.372 map units (latitude, 2 x 0.686 = 1.372). This effectively stretches the map in the latitude (N-S) direction. Now the map distances are nearly the same in the longitude and latitude directions.



When the map is scaled so 1° of longitude equals 1° of latitude, map distances are distorted. Notice that the graticules form squares (equal spacing).

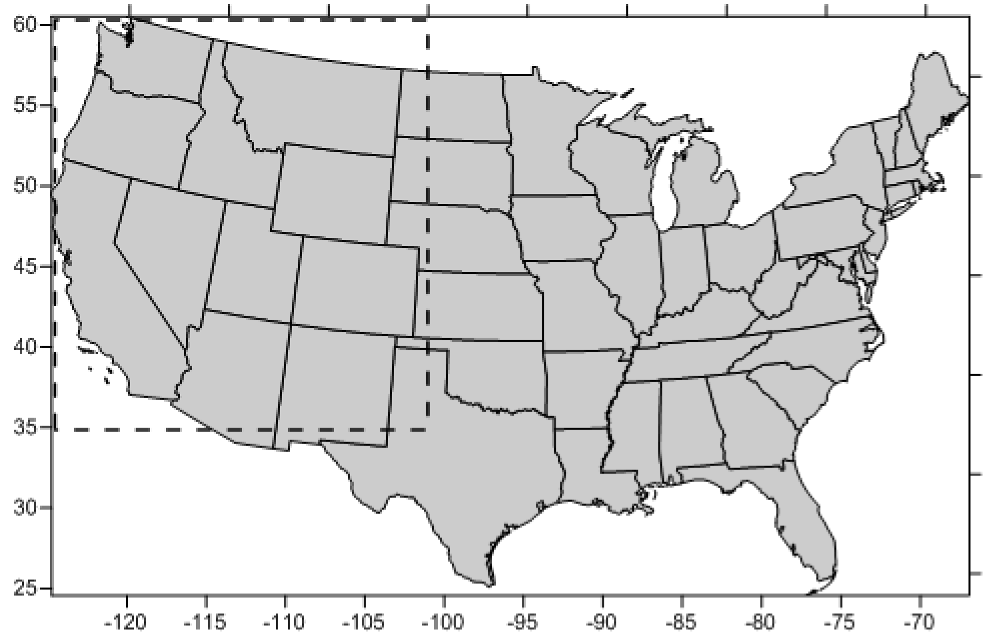


When you apply scaling factors, map distances are represented more accurately. The rectangular spacing of graticule lines indicate the difference in scaling.

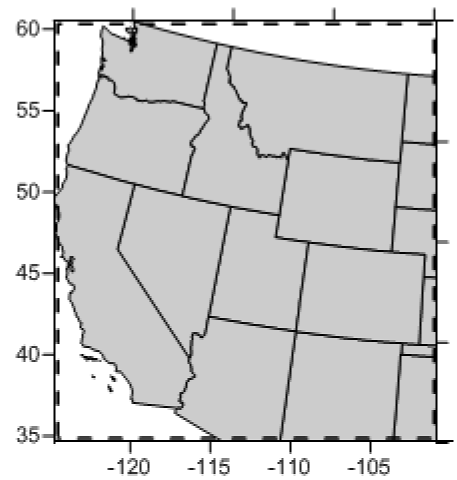
Limits Properties

The **Limits** page in the map properties lets you define the X and Y minimum and maximum map coordinates (specified as *xMin*, *xMax*, *yMin*, and *yMax* values). The **Limits** page is unavailable if there is a 3D wireframe layer in a multi-layer map.

When a map is created, the limits are automatically defined by the coordinate values contained in the file used to create the map. The limits of the selected map can be adjusted on the **Limits** page of the map properties or interactively with the Set Limits command.



The map limits of the map are set to the default limits as defined by the base map. The limits of a smaller area of interest are indicated with the dashed rectangle.



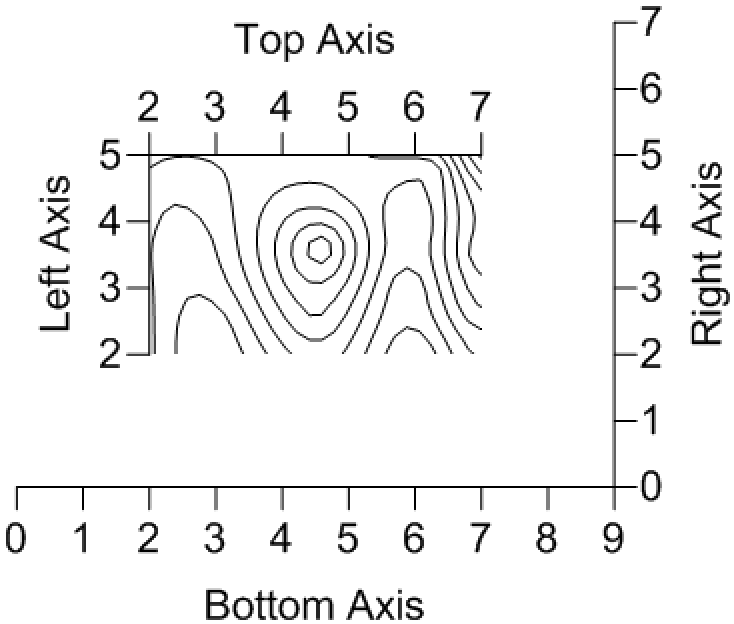
The map limits of the map have been adjusted to the limits of the area of interest.

Limits and 3D Wireframes

Map limits cannot be defined for 3D wireframes; this constraint includes composite maps containing 3D wireframe layers. These maps are limited to the extent of the grid file used to create the wireframe. To change the limits of a grid file for a wireframe, use Grids | Resize | Extract or create a new grid file with new limits. Alternatively, use a 3D surface map with mesh lines to simulate a wireframe map.

Limits and Axis Scaling

When you change map limits, axes that use automatic scaling are redefined to fit the new limits of the map. If you do not use automatic scaling for the axes, the axes are drawn over the same range as the original map and will not match the map using the new limits.



The map limits have been reset on this map. The Left Axis and Top Axis use automatic scaling and match the new map limits. The Right Axis and Bottom Axis do not use automatic scaling, and reflect the original limits of the map.

Limits and Map Scale

When new map limits are defined for a map, the map scale is not changed. For example, if the map uses a scaling of 1 inch = 1000 meters, the new map uses this same scale. This can result in maps much smaller or larger than the original map. In this case, click on the map and set the new scale on the Scale page.

Limits and Post Maps

Limits can be set on post maps or maps containing a post map layer. When a posted point lies outside the map limits, the posted symbol and the label are clipped from the map. When a posted point is at or inside the map limits, the entire symbol and posted label are printed, even if the symbol and label extend beyond the limits of the map.

Limits and Reversed Axes

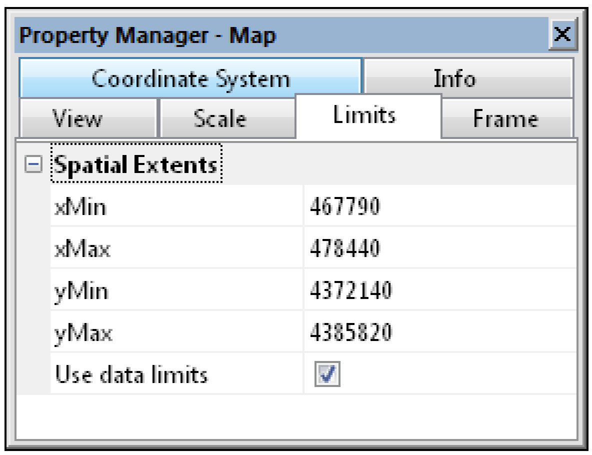
When an axis is reversed on the Scaling tab, the limits do not change. The xMin value must still be less than the xMax value. The reverse axis option only changes the display of the axes and map layers, not the actual limits.

Limits, Scale, and Adding Map Layers

When a new map layer is added to an existing map or a modified map layer exceeds the current map limits, a message appears prompting you to adjust the map limits to include all the layers. Select *Yes* to adjust the map limits. Select *No* to preserve the current map limits.

Limits Page

The **Limits** page is located in the map properties.



*Specify the map limits in the **Properties** window on the **Limits** page.*

Limits XY Min and Max

Set custom *xMin*, *xMax*, *yMin*, and *yMax* map limits in the *Spatial Extents* group.

Map limits can be set larger or smaller than the limits of the current map. For example, if you have a map that ranges from zero to 100 in the X dimension, but you only want to display the map from 25 to 50 in the X dimension, use 25 and 50 as the new minimum and maximum values on the **Limits** page.

When using date/time formats for any of the axis labels, the minimum and maximum on the **Limits** tab are entered in date/time format. To change the value, highlight the existing value and enter the minimum or maximum date/time value. For instance, 02/02/2014 12:00:00 AM can be entered into the *xMin* option.

xMin	6/10/2010 0:00:00
xMax	7/19/2013 0:00:00
yMin	4406800
yMax	4408800

This example shows a map with xMin *and* xMax *values in date/time format.*

Use Data Limits

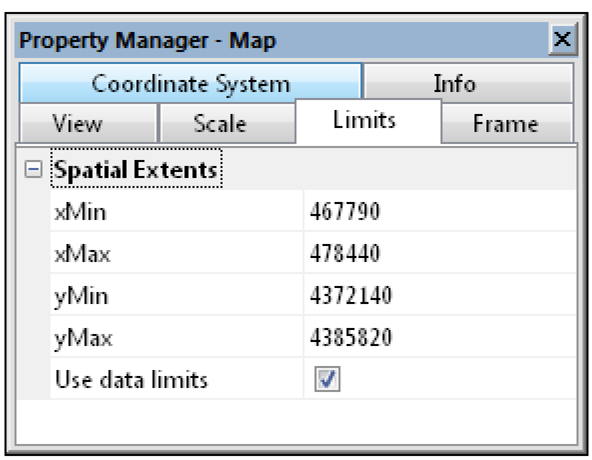
The *Use data limits* check box resets the map limits to the limits of the grid file, base map, or post map data file limits.

Setting Map Limits

Two methods exist for setting map limits. Map limits can be set in the plot window with the Set Limits command. The limits can be entered as values in the **Properties** window Limits page.

To set map limits on the **Limits** page:

1. Click once on the map object to select it.
2. The map properties appear in the Properties window.
3. On the Limits page, specify the X and Y minimum and maximum values for the map by entering the values into the *xMin*, *xMax*, *yMin*, and *yMax* boxes. The *Use data limits* check box resets the map limits to the limits of the grid file, base map, or posting data file limits.
4. The map is redrawn with the new limits. If the axes use automatic scaling, they are also redrawn to fit the new map limits.

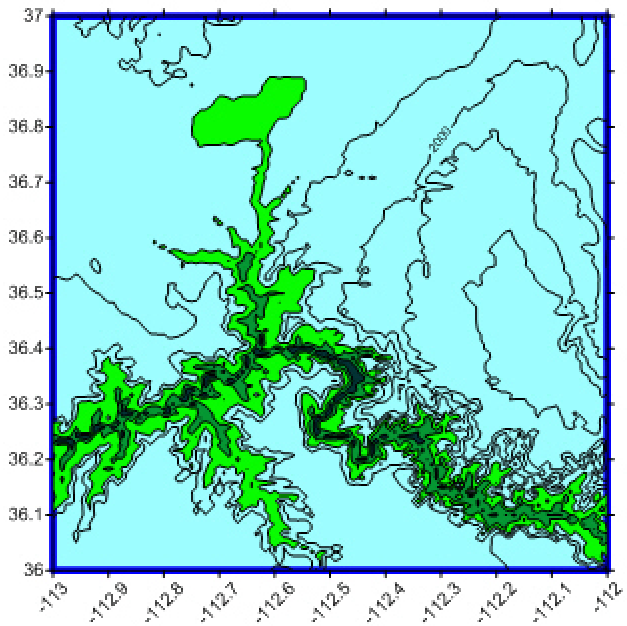


*Change map limits on the **Limits** page in the **Properties** window.*

Frame Properties

The map properties **Frame** page sets the map border line properties and the background fill properties for the selected map. The map background limits coincide with the axis limits on contour, base, post, 3D surface, and vector maps.

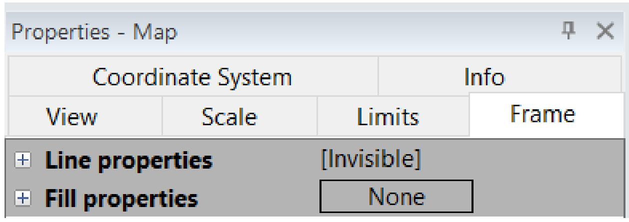
Map backgrounds lie underneath all other objects on the map. This means that the background can be obscured by other map features. For example, when you create color filled contour maps, the fill between contours obscures the map background wherever a solid or image pattern is used for the contour fill. Similarly, the outline of the map background can be obscured by the map axes.



This is a contour map displaying green filled contours. The remainder of the contours are 0% opaque, allowing the light blue background to be seen. A heavy dark blue map border has also been added to the map.

Frame Properties

The **Frame** page is located in the Properties window when a *Map* object is selected.



*Specify the map border and background properties in the **Properties** window on the **Frame** page.*

Line Properties

Click the ⊞ next to *Line Properties* to open the Line Properties section and customize the map border.

Fill Properties

Click the ⊞ next to *Fill Properties* button to open the Fill Properties section and customize the map background.

Reload Data

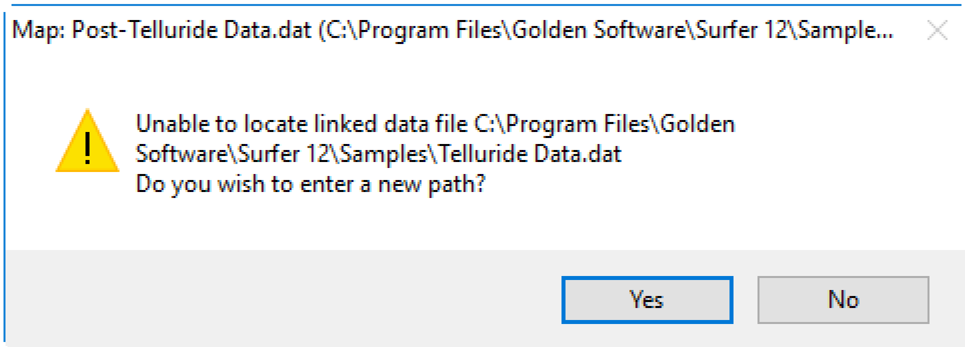
You can reload external map data for all maps in the currently loaded **Surfer** file with the **File | Reload Map Data** command or the button. For only the selected layer use the **File | Reload Layer Data** command or the button.

Surfer caches all data used by all maps within the .SRF file including the data points used in a post map and the grids used in grid based maps.

If the original data file or grid file is changed and unsaved, the modified file will not be used when the .SRF file is reopened. Use the **File | Reload Map Data** command to update data or grid file information from the saved version of the files.

Unable to Locate

If the data are not found a dialog with the name of the map layer is displayed. Click the *Yes* button to manually find the missing data file. Click the *No* button to skip the map layer and choose not to update the file that cannot be located.



This example shows data file used for a post layer that is not found when reloading the map data.

Updating a Single Map Layer

Map data can be reloaded for individual map layers by clicking the **File | Reload Layer Data** command or by clicking the button in the **Properties** window and re-selecting the data file.

Coordinate System Note

Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Chapter 23 - Map Features

Add to Map

The **Map Tools | Add to Map** command group contains commands for adding map layers, additional left, right, top, bottom, or Z axes, a scale bar, a viewshed, a graticule, or a profile to the existing map. Alternatively, right-click on an existing map and click **Add to Map** to add a map layer, axis, scale bar, or profile.

Map Layers

Add a Base Layer from Server, Base Layer, Empty Base Layer, Contour Layer, Post Layer, Classed Post Layer, Color Relief Layer, Grid Values Layer, 1-Grid Vector Layer (aspect and gradient are calculated), 2-Grid Vector Layer (X, Y or direction, magnitude), Watershed Layer,Point Cloud layer, Viewshed layer, 3D Wireframe Layer, 3D Surface Layer, Left Axis, Right Axis, Top Axis, Bottom Axis, Z Axis, Scale Bar, or Profile to the selected map.

Layers that cannot be added to the selected map are grayed out. For example, a 3D wireframe map layer cannot be added to a 3D surface map.

Layers can be dragged between maps in the **Contents** window. Layers can be combined with the Overlay Maps command to create a single *Map* object.

Select multiple files to add multiple layers to the map. One layer is added for each selected file. Multiple files cannot be selected when adding 2-Grid Vector layers.

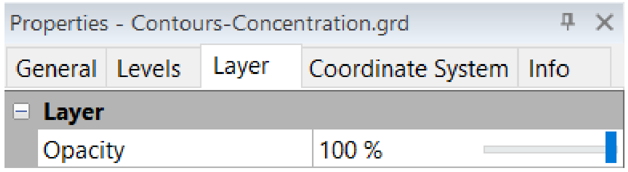
Map Limits

Adding a new layer to an existing map will not change custom scaling or map limits, with the exception of 3D wireframe and 3D surface maps. Adding a 3D wireframe or 3D surface to an existing map will change custom scaling or map limits.

If a map layer is added to a map frame and the map layer exceeds the current map limits, a **Surfer** warning dialog will be displayed allowing you to adjust the map limits to include all layers. Select *Yes* to adjust the map to include all layers. Select *No* to leave the current map limits.

Layer Properties

The **Layer** page in the Properties window for a base, post, classed post, grid values, contour, color relief, vector, peaks and depressions, point cloud, watershed, 3D wireframe, viewshed, or graticule controls the opacity of the entire map layer. Surface maps do not have a **Layer** page.



Adjust the Opacity *of the entire layer on the* ***Layer*** *page. This example shows the* ***Layer*** *page of a contour map in the* ***Properties*** *window.*

Opacity

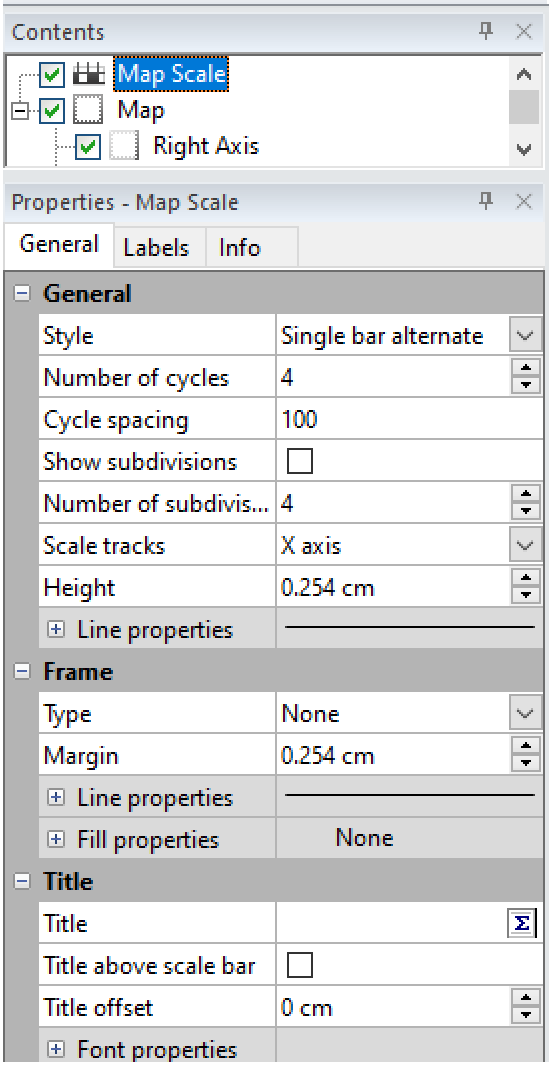
Change the opacity of the entire layer by entering an *Opacity* value from 0% (completely transparent) to 100% (completely opaque). Alternatively, click and drag the  to change the opacity percentage.

3D surface maps do not have the ability to change the opacity for the entire map.

Scale Bar

Scale Bar General Properties

The **General** page includes the style, spacing, cycles, frame, and title properties for the map scale bar. You have flexibility over the spacing of the cycles and the number of map units the cycles represent on the scale bar. With the default options, the scale bar is drawn relative to X dimension map units.



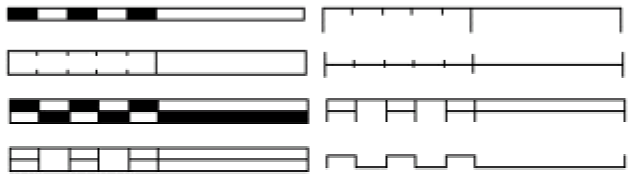
*Map scale bar properties are set in the **Properties** window.*

General Section

The *General* properties section contains the general display options for the scale bar.

Style

Select a scale bar style from the *Style* list. Available style options include *Single bar alternate*, *Single bar unfilled*, *Double bar alternate*, *Double bar unfilled*, *Comb*, *Double-sided comb*, *Rail*, and *Zigzag*.



The scale bar styles include (clockwise from the upper right): Comb, Double-sided comb, Rail, Zigzag, Double bar unfilled, Double bar alternate, Single bar unfilled, and Single bar alternate.

Number of Cycles

Specify the *Number of cycles* to use for the scale bar. This is the number of divisions the scale bar uses, and can be any value from one to 100.

Cycle Spacing

The *Cycle spacing* is the distance (length) of one cycle in the *Scale tracks* axis units. Usually the *Label increment* value is equal to the *Cycle spacing* value, although you can specify label increments different from the *Cycle spacing*.

Show Subdivisions

Subdivisions can be added to the first cycle by clicking the *Show subdivision* check box.

Number of Subdivisions

Specify the *Number of subdivisions* for the first cycle by typing a number in the box or clicking the buttons. The minimum *Number of subdivisions* is 2, and the maximum *Number of subdivisions* is 10.

Scale Tracks

Specify the axis units on which you want to base the scale bar. Click on the existing axis and select the appropriate option in the *Scale tracks* drop down. If the X and Y dimensions are scaled proportionally, the *X axis* and *Y axis* options have the same result. The *Z axis* option is only appropriate for 3D wireframes, 3D surface maps, layers on a wireframe or surface, and tilted post maps with 3D label lines.

Height

Specify the height of the scale bar in page units by typing a value in the *Height* field. Set the *Height* to a value between 0 and 4 inches (0 and 10.16 centimeters).

Line Properties

Specify the line properties for the scale bar by clicking on the next to Line Properties. The line property *Color* also controls the fill color of the cycles.

Frame Section

The *Frame* section specifies the properties to use for the border around the map scale bar. See the Frame Properties help topic for information on these common properties.

Title Section

The *Title* section contains the title text and options.

Title

Add a title to the scale bar by entering text into the *Title* field. Click in the *Title* field, and type a title. Press ENTER or click elsewhere, and the title will be added or updated. You can format the title in the *Title* field using math text instructions or by using the Text Editor. Click the button to open the **Text Editor**.

Title Above Scale Bar

Check the *Title above scale bar* check box to display the title text above the scale bar. Clear the *Title above scale bar* check box to display the title text below the scale bar. When labels and the title are displayed on the same side of the scale bar, the labels will be drawn nearer to the scale bar.

Title Offset

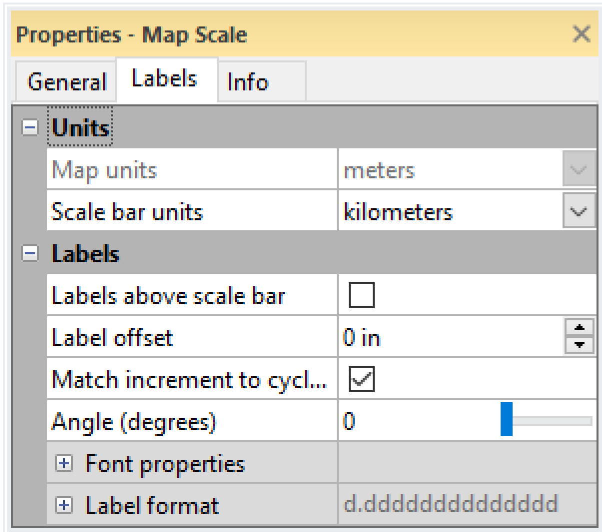
The *Title offset* value moves the title away from the scale bar or scale labels. The title moves down with increasing offset when *Title above scale bar* is not checked. The title moves up with increasing offset when *Title above scale bar* is checked. Type a value between 0 and 10 inches (0 and 25.4cm) in the *Title offset* field or click the to change the *Title offset*.

Font Properties

Specify the title font properties by clicking on the next to Font Properties.

Scale Bar Labels Properties

The **Labels** page includes the label units, placement, font, and format properties for the scale bar labels.



*Set the labels properties in the **Labels** page.*

Units Section

The *Units* section contains properties for setting the scale bar units. The *Map units* property displays the linear units for the map target coordinate system. The *Scale bar units* property sets the units for the scale bar labels. Select the desired units from the *Scale bar units* list. The Scale bar units property cannot be edited if the *Map units* is *Unknown* or *n/a (spherical coordinates)*.

Labels Section

The *Labels* section contains properties for the scale bar labels.

Labels Above Scale Bar

Check the *Labels above scale bar* check box to display the labels above the scale bar. Clear the *Labels above scale bar* check box to display the labels below the scale bar. When labels and the title are displayed on the same side of the scale bar, the labels will be drawn nearer to the scale bar.

Label Offset

The *Label offset* value moves the labels away from the scale bar in page units. Labels move down with increasing offset when *Labels above scale bar* is not checked. Labels move up with increasing offset when *Labels above scale bar* is checked. Type a value between 0 and 10 inches (0 and 25.4cm) in the *Label offset* field or click the to change the *Label offset*.

Match Increment to Cycle Spacing

Select the *Match increment to cycle spacing* property to lock the label increment to the *Cycle spacing* value on the General page.

Clear the *Match increment to cycle spacing* property to set the label increment independent of the scale bar cycle spacing. When the coordinate system linear units are defined the *Match increment to cycle spacing* property should be selected. Clear the *Match increment to cycle spacing* property to manually convert from unknown or lat/lon units to label units. Note that setting the label increment and scale bar cycle spacing to different values when the map units are defined and linear can lead to incorrect values for the scale bar.

Label Increment

The *Label increment* specifies the change in label value from the beginning of one cycle to the beginning of the next. The *Label increment*controls the actual label being displayed under each cycle. The*Label increment*can be different from the*Cycle spacing*, which controls the length of each cycle. You may want these different if the map is in feet and you want the scale bar to show meters for the labels.

Label Angle

The *Angle (degrees)* is the preferred orientation, in degrees, of the scale bar labels. Rotate the labels by entering a value into the *Angle* box. Positive values rotate the label counterclockwise. Values can be between 0 and 360.

Font Properties

Specify the label font properties by clicking on the ⊞ next to Font Properties.

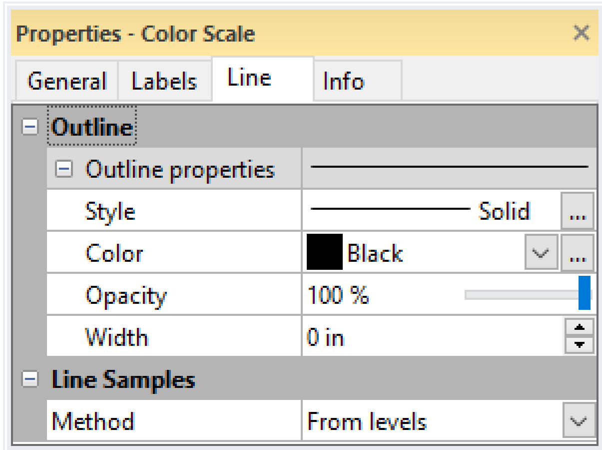
Label Format

Click the ⊞ next to Label Format to specify the numeric format used for the labels.

When the *Type* is set to *Date/time*, if *Invalid date* text is displayed instead of the actual date/time format, the value for the label is outside the defined date/time range.

Color Scale Bar Line Properties

The color scale bar **Line** page controls the line properties for the color scale bar outline in the 2D view. The line properties are also displayed for the levels in discrete (contour and wireframe) color scale bars.



Set the line properties for the color scale bar outline and levels.

Outline Properties

The *Outline* section includes the line properties for the color scale bar border. This line property is used for the outside of the color scale bar box. You can set *Style, Color, Opacity,* and *Width* of the line. The *Outline properties* can also be applied to the line samples by setting the *Method* to *Same as outline*.

Line Samples

The *Line Samples* section includes the method and line properties for the level line samples in discrete (contour and wireframe) color scale bars. The *Line Samples* section is only displayed for contour layers when the contour layer is color filled.

Method

The *Method* property sets the source for the line sample properties: *Same as outline*, *Uniform*, or *From levels*.

- *Same as outline* sets the line sample properties source to the *Outline properties*. Set the line properties for the outline and the line samples in the *Outline properties* section. Wireframe color scale bars use the *Same as outline* method by default.
- *Uniform* makes the line samples uniform regardless of the individual level line properties in the map. Set the level line sample line properties in the *Line properties* section displayed below the *Method* property.
- *From levels* sets the line sample properties source to the line properties for the levels in the map. Contour map color scale bars use the *From levels* method by default.

When a contour map is not color filled, the color scale bar uses the *From levels* method and the *Line Samples* section is not displayed.

Line Properties

When the *Method* is set to *Uniform*, a *Line properties* section is displayed. These properties specify the line properties for the color scale bar line samples.

Legend

Click the **Map Tools | Add to Map | Legend** command or the  button to add a legend to the plot. The legend can include any number of post, classed post, 1-grid vector, 2-grid vector, and base symbology layers from any of the maps in the plot. However once a layer is included in a legend, it cannot be added to any other legends. The total number of legends is limited to the total number of post, vector, and base layers in the plot, i.e. each layer can have its own legend. Once all of the post, classed post, 1-grid vector, 2-grid vector, and base layers are represented in a legend, the **Map Tools | Add to Map | Legend** command button is disabled. Every layer that is not already included in a legend is added automatically when a new legend is created. Use the *Layers in Legend* section of the Properties window Layers page to control which layers are included in the legend.

Base Symbology Legends

When a base layer includes a symbology, a legend can be displayed for the base layer. Unique values legends can include the value, fill, line, and symbol samples, and the legend order is determined by the order in the **Symbology** dialog. Unclassed colors legends can include fill value and fill samples. Unclassed symbols legends can include value and symbol samples. Classed colors legends can include the upper limit value, class name, and fill samples. Classed symbols legends can include the upper limit value, class name, and symbol samples. Pie chart symbols can be relatively sized and can include slice colors with labels.

Drillhole Layer Legends

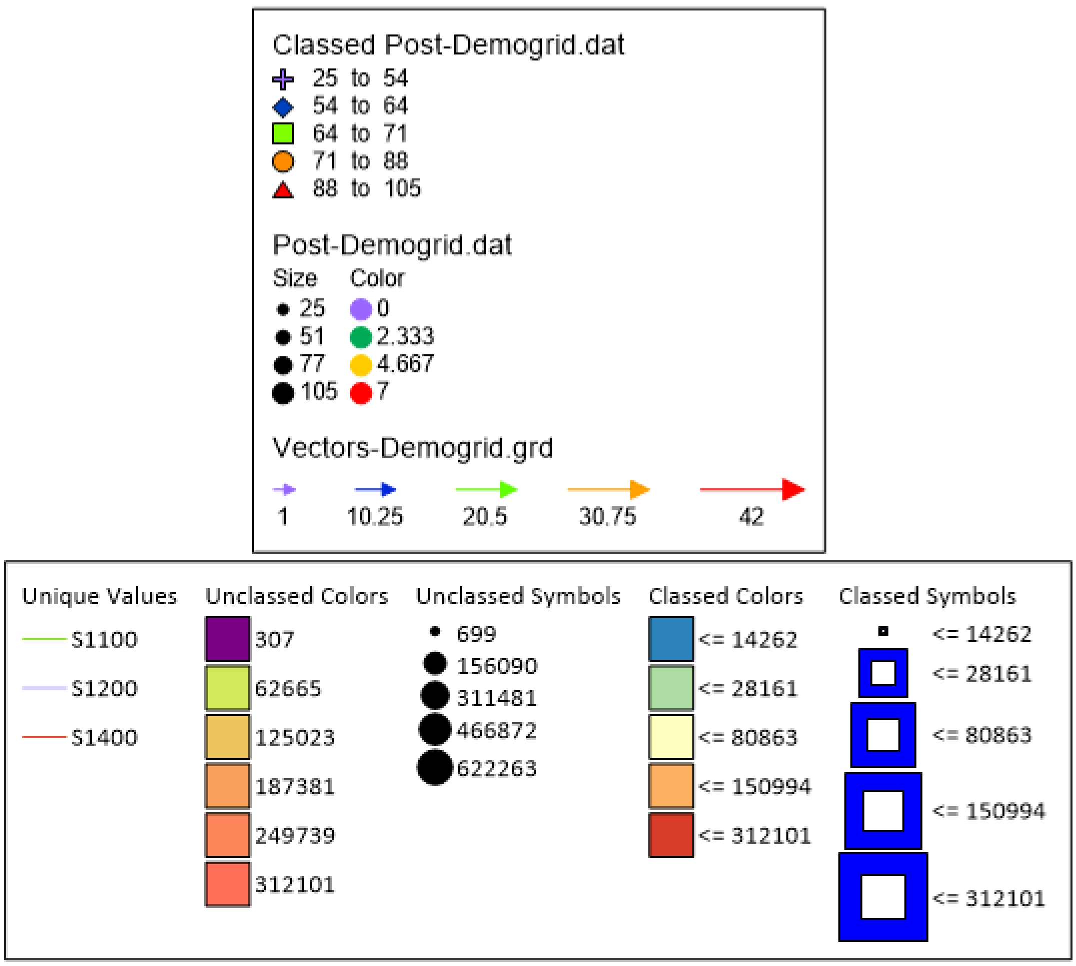
When a drillhole layer **Symbol** property for *Symbol method* in the **Hole Top Symbol** section is set to *By collar*, a legend can be added to a drillhole layer. The legend will display the hole top symbol and the hole id from the collars data. See Drillhole Map for more information.

Post Layer Legends

Legends can be displayed for post layers and classed post layers. The post layer legend displays symbols representing the symbol size when proportional sizing is specified. The post layer legend displays symbols representing the symbol color when a color column is specified and the *Color method* is *Numeric via colormap*. When both proportional sizing and the *Numeric via colormap* color method is used, the symbols representing size and color are displayed separately with "Color" and "Size" headers. When neither proportional sizing nor a *Numeric via* *colormap* color method is applied to the post layer, a single symbol is displayed. The classed post layer legend displays a symbol for each class and optional information about the class.

Vector Layer Legends

A legend for a vector layer displays scaled vectors with the appropriate value next to each vector. By default, the vector legend displays the minimum and maximum vector length as symbols along with a numeric value for each symbol. This is useful in determining what the vectors on a 1-grid vector or a 2-grid vector layer represent. The vector lengths are set in the *Magnitude (data)* section on the Scaling page. If the *Scaling method* on the Symbol page is set to *By magnitude*, the symbols in the legend are displayed with the minimum and maximum colors.



One of each layer or symbology legend type is displayed above.

Legend Properties

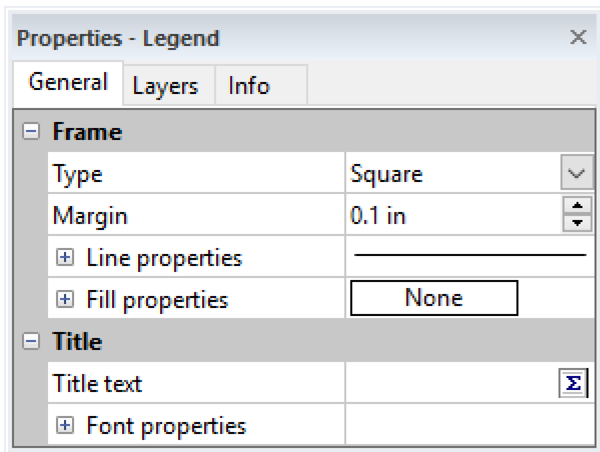
To change the legend properties, click once on the legend to select it. The properties for the legend are displayed in the **Properties** window. The legend properties include the following pages:

General

Layers
Info

General Properties

The **General** page of the legend properties includes frame and title properties.



*Set the frame and title properties in the **General** page.*

Frame

The *Frame* section specifies the properties to use for the border around the legend. See the Frame Properties help topic for information on these common properties.

Title

The *Title* section adds a title for the legend.

Title Text

Click in the box next to *Title text* and type the title for the legend. The title may include math text instructions. Type the title exactly as you want the text to appear in the legend.

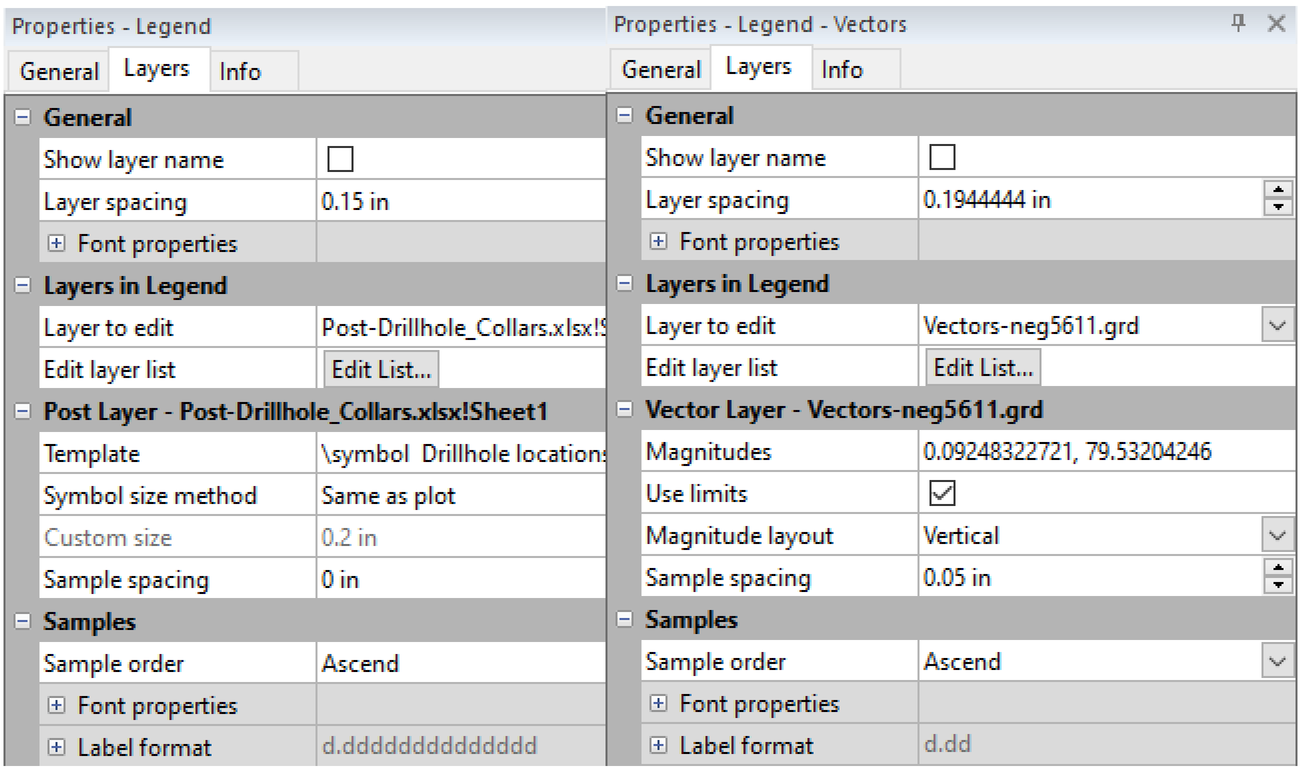
Titles can contain multiple lines. To create a multiple line title, click the Σ button. The Text Editor appears. Type the text in the dialog. To add a new line of text, press the ENTER key on the keyboard. To change the properties of the text, highlight the text and change the desired property. When finished, click *OK*. You are returned to the plot window.

Title Font

Click the ⊞ next to *Font Properties* to open the Font Properties section. Set the font properties to use for the legend title here. The legend title and legend class font properties may be set separately.

Legend Layers Properties

The **Layers** page in the Properties window controls which layers are included in the legend and the appearance of the legend.



*The **Layers** page controls which layers are included in the legend and the legend sample appearance. The image on the left displays the **Layers** page for a post layer legend while the image on the right displays the vector layer legend.*

General

The *General* section contains options for the display of layer names in the legend.

Show Layer Name

Check the *Show layer name* check box to display layer names above the layers' samples in the legend. Clear the *Show layer name* check box to display only samples in the legend.

Layer Spacing

The *Layer spacing* property sets the spacing between the layers in the legend in page units.

Font Properties

Expand the *Font properties* to control the font properties for the layer names in the legend. All of the layer names use the same font properties.

Layers in Legend

The *Layers in Legend* section controls which layers are included in the legend, the legend layout, the order of the layers in the legend, and which layer's samples are being edited in the *Layer* section below.

Layer to Edit

The *Layer to edit* selection determines which layer is moved by the *Layer position* property and which layer's properties are displayed in the *Layer* section. Click the current selection in the *Layer to edit* field and select the layer you wish to edit from the list. The list includes all of the layers in the legend.

Edit Layer List

Click *Edit list* in the *Edit layer list* field to add and remove layers in the legend with the Layers for Legend dialog. The **Layers for Legend** dialog can also be used to rearrange the layers in the legend.

Layer Layout

Change the overall layout of the legend by selecting *Vertical* or *Horizontal* in the *Layer layout* property field. The *Layer layout* property is only available when the legend contains two or more layers. Select *Vertical* to display layers arranged top to bottom. Select *Horizontal* to display layers arranged left to right. The *Layer layout* selection controls the buttons in the *Layer position* field.

Layer Position

The *Layer position* field changes the position of the *Layer to edit* selection. When *Layer layout* is set to *Vertical*, click *Move Up* to move the layer up one position in the legend, and click *Move Down* to move the layer down one position in the legend. When *Layer layout* is set to *Horizontal*, click *Move Left* to move the layer one position to the left in the legend, and click the *Move Right* button to move the layer one position to the right in the legend. You can also rearrange the layers in the Layers for Legend dialog.

Layer Section

The *Layer* section contains the properties for the appearance of the samples for the *Layer to edit* selection. The *Layer* section name includes the layer type and layer name formatted as *<Type> Layer - <Layer Name>*, for example *Post Layer - Post-Drillholes_Collars.xlsx!Sheet1* . The properties in the layer section depend on the layer type: post, classed post, 1-grid vector, 2-grid vector, or base.

Post, Classed Post, and Base Symbology Properties

The following properties are available when a post, classed post, or base layer is selected in the *Layer to edit* field.

Variable Layout

The *Variable layout* property controls the display of the size and color samples when both are displayed on the legend for a post layer. Proportional sizing must be used on the post layer, and a color column must be used with the *Numeric via colormap* color method to display both size and color samples in the legend. Set the *Variable layout* property to *Horizontal* to display the size and color samples side by side. Set the *Variable layout* property to *Vertical* to display the size samples above the color samples.

The *Variable layout* property is only available for the post layer legend.

Number of Samples

Set the number of samples for legend with the *Number of samples* property. The *Number of samples* controls both the size and color samples sections in the post layer legend. Type the desired number of samples in the *Number of samples* field or click the to change the value.

The *Number of samples* property is available for post layer, unclassed colors symbology, and unclassed symbols symbology legends.

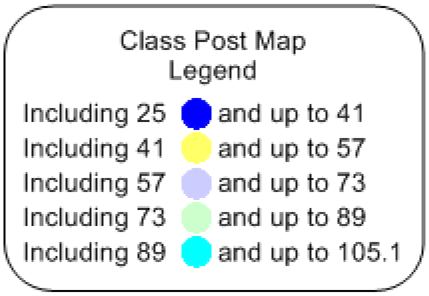
Template

The *Template* box customizes the format of the legend items. The specified template is applied to each class or sample in the legend. Normal text and math text instructions can be included within the template, in addition to the following special legend template directives:

Directive	Post	Classed Post	Unique Values	Unclassed Colors	Unclassed Symbols	Classed Colors	Classed Symbols	Pie Chart
\fill			fill sample	color sample		color sample		color sample
\line			line sample					

Directive	Post	Classed Post	Unique Values	Unclassed Colors	Unclassed Symbols	Classed Colors	Classed Symbols	Pie Chart
\lower		lower class limit						
\name		class name				class name	class name	
\symbol	symbol sample	symbol sample	symbol sample		symbol sample		symbol sample	
\upper		upper class limit						
\value	sample value		sample value	sample value	sample value	"<= " upper class value	"<= " upper class value	sample value

Any number of directives can be included in any order. The *\tab* directive can be included to insert a 0.25 inch tab character. As an example, the following template: *Including \lower \symbol and up to \upper* produces a classed post map legend that looks like:



A sample classed post map legend shows classes, symbols, and text in the class lines.

Symbol Size Method

The *Symbol size method* sets the size of the legend symbols. The *Symbol size method* property is only available for post and classed post layer legends.

- Selecting *Based on font* causes the symbols to be drawn using the size specified in the *Samples* section.
- Selecting *Same as plot* draws the symbols the same size as they appear on the map. This can cause symbols for different classes to be different sizes in a classed post layer legend. It's recommended you use *Same as plot* in the legend for a post layer with proportional symbol sizing.

- Selecting *Custom* allows a size to be entered for the legend symbols in the *Custom size* option.

When a legend is created for a post layer with proportional sizing, the *Symbol size method* is automatically set to *Same as plot* . Unclassed symbols and classed symbols symbology legends always display the symbols the same size as in the plot.

Fill Sample Size

The *Fill sample size* property controls the size of the fill sample square in the legend. Fill samples are available for base layers with unique values, unclassed colors, or classed colors symbology. Specify a size for the fill sample square by typing a value in page units in the *Fill sample size* field.

Sample Spacing

The *Sample spacing* property controls the spacing between samples. The *Sample spacing* can be set independently for each layer in the legend. Type a value in page units in the *Sample spacing* field to specify the space between legend samples.

1-Grid Vector and 2-Grid Vector Properties

The following properties are available when a 1-grid vector or 2-grid vector layer is selected in the *Layer to edit* field.

Magnitudes

If you would like to display different vector lengths in the legend, enter the new lengths in the *Magnitudes* box. The new vectors are scaled relative to the scaling set on the Scaling page in the vector map properties. You can display more than two vectors in the legend by entering a string of numbers separated by commas or spaces into the *Magnitudes* box. For example, 0.5, 10, 25, 50 would show four symbols in the vector legend.

Use Limits

If the *Use limits* box is checked, only two vectors are displayed in the vector length legend. These are the *Minimum* and *Maximum* values set on the **Scaling** page in the *Magnitude (data)* section. When checked, the *Magnitudes* property and legend will update to show the new values when the *Magnitude (data)* changes.

Magnitude Layout

The *Magnitude layout* options align the symbols vertically or horizontally in the legend. To change the *Magnitude layout*, click on the current option. Select either *Vertical* or *Horizontal* in the list.

Samples

The *Samples* section contains the options for the sample order within layers, and the sample font and label format properties.

Sample Order

The *Sample order* property specifies whether the samples are listed in ascending order (from smallest to largest) or descending order (from largest to smallest). Click the current selection and select *Ascend* or *Descend* from the list. The *Sample order* applies to the samples in all layers in the legend.

Font Properties

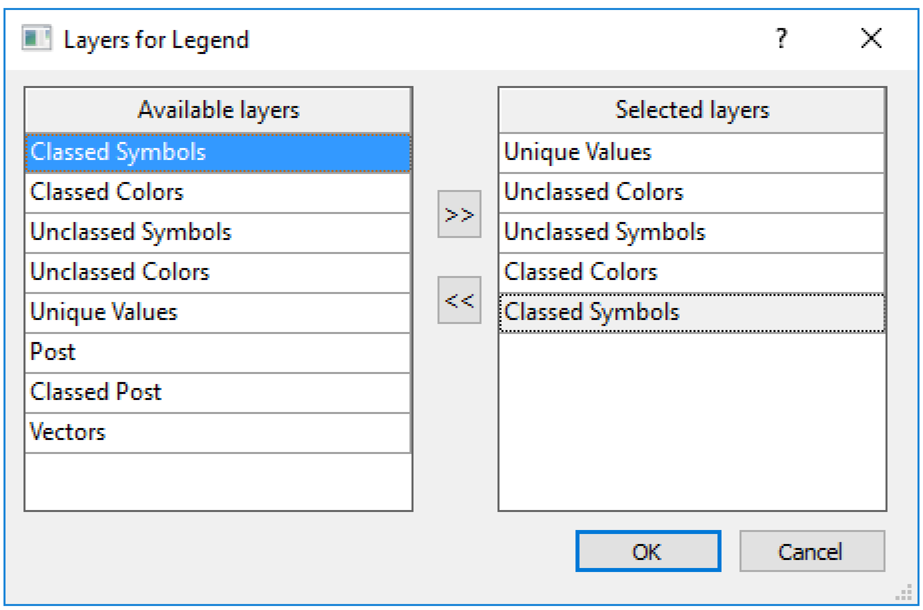
Control the sample font properties in the *Font properties* section. The font properties apply to the samples in all layers in the legend.

Label Format

Control the sample label format in the *Label Format* section. The label format properties apply to the samples in all layers in the legend.

Layers for Legend Dialog

Click the *Edit list* button in the **Properties** window Layers page to add, remove, and move map layers in the legend.



*Add, remove, and move layers in the **Layers for Legend** dialog.*

Available Layers

The layers that can be added to the legend are listed in the *Available layers* list. The *Available layers* list includes any layer that is not in a legend and the layers in the selected legend. One or more layers must be selected to add a layer to the legend. Select a layer by clicking on the layer name. Select multiple layers by holding CTRL while clicking on the layer names. Select a group of layers by clicking the first layer, holding SHIFT, and clicking the last layer. The first, last, and all layers between are selected. Deselect a layer by selecting another layer, or by holding CTRL and clicking the selected layer.

Add layers to the legend by selecting the desired layer or layers. Next click >> to add the selected layer(s) to the *Selected layers* list. The added layers are added to the end of the *Selected layers* list.

Selected Layers

The *Selected layers* list shows the layers that are included in the legend and the order in which they are displayed. Layers can be removed from the *Selected layers* list, and layers can be rearranged within the *Selected layers* list.

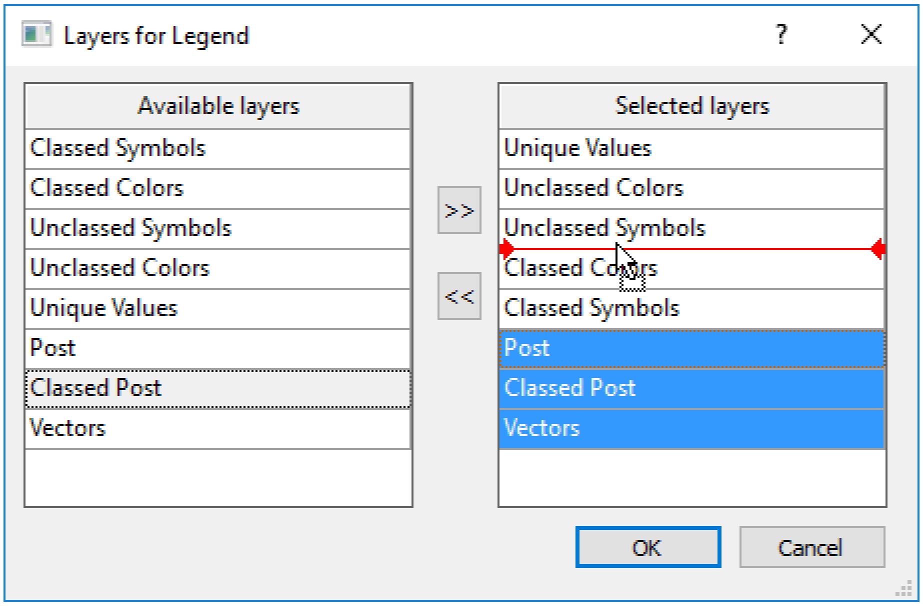
One or more layers must be selected to remove or move the layers in the legend. Select a layer by clicking on the layer name. Select multiple layers by holding CTRL while clicking on the layer names. Select a group of layers by clicking the first layer, holding SHIFT, and clicking the last layer. The first, last, and all layers between are selected. Deselect a layer by selecting another layer, or by holding CTRL and clicking the selected layer.

Removing Layers

Remove layers from the legend by selecting the desired layer or layers. Next click << to remove the selected layer(s) from the *Selected layers* list.

Rearranging Layers

Rearrange layers in the in the legend by selecting the desired layer or layers. Next click and drag the selected layer(s) to a new position in the list. While dragging the selection, the position to which the layers will be moved is indicated by a red line with red arrows at the ends. Release the left mouse button to move the layers.



The Post, Classed Post, and Vectors layers will be moved between the Unclassed Symbols and Classed Colors layers when the left mouse button is released.

OK and Cancel

Click *OK* to apply the changes to the legend. Click *Cancel* to close the **Layers for Legend** dialog without making any changes to the legend.

Color Scale Bar - Vector, Color Relief, 3D Surface, and Point Cloud Maps

A color scale bar is a legend that shows the fill assigned to vector symbols on a 1-grid vector map or a 2-grid vector map, or the colors used in a color relief map, 3D surface map, or point cloud map. Color scale bars can be added to 2D and 3D maps, but have slightly different options. Color scale bars for contour and wire-frame map layers are similar, but have slightly different options.

Click on the color scale bar to open the color scale properties in the Properties window.

Color Scale Bar Visibility

The color scale bar display can be turned on or off in the Contents window by checking or clearing the box to the left of the color scale bar name.

Adding a Color Scale Bar

To add a color scale bar to a vector map:

1. Click on the vector layer to select it.
2. In the **Properties** window, click on the Symbol tab.

3. Specify the fill properties to use for the vectors in the *Color Scaling* section.
4. Check the *Show color scale* box to activate the color scale bar.

To add a color scale bar to a color relief, surface, or point cloud map:

1. Click on the color relief, surface, or point cloud layer to select it.
2. In the **Properties** window, click on the **General** tab.
3. Check the *Show color scale* box to activate the color scale bar.

Removing a Color Scale Bar

A color scale bar can be removed by clicking on the color scale bar and pressing the DELETE key on the keyboard. Alternatively, you can clear the *Show color scale* box in the layer properties.

Using a Logarithmic Scale on a Color Scale Bar

The color scale bar reflects the scaling of the colormap. To set the colormap to use a logarithmic scale for a vector map:

1. Click on the vector map layer to select it.
2. In the **Properties** window, click on the **Symbol** tab.
3. Click the ⊞ next to *Color Scaling* to open the *Color Scaling* section.
4. Set the *Scaling method* to *By magnitude* or *By grid file*.
5. Click the ... next to the *Vector colors* colormap.
6. Check the *Logarithmic scaling* option and click *OK*.
7. The color scale bar is automatically updated to show logarithmic scaling.

To set the colormap to use a logarithmic scale for a color relief, surface, or point cloud map:

1. Click on the color relief, surface, or point cloud layer to select it.
2. In the **Properties** window, click on the **General** tab.
3. Click the ... next to the *Colors* colormap for a color relief map layer. Click the ... next to the *Upper* colormap for a 3D surface map layer. Click the ... next to *Colormap* for a point cloud layer.
4. Check the *Logarithmic scaling* option and click *OK*.
5. The color scale bar is automatically updated to show logarithmic scaling.

Editing a Color Scale Bar

To modify a color scale bar, click on the color scale bar to select it. The properties are listed in the **Properties** window.

General
Labels
Line
Info

Color Scale Bar - Contour and 3D Wireframe

A color scale bar is a legend that shows the line color, line style, fill color, and fill pattern assigned to each contour level on a contour map, or the colors assigned to levels in a 3D wireframe. Click on the color scale bar to open the color scale properties in the Properties window. Color scale bars for vector, color relief, and 3D surface map layers are similar, but have slightly different options.

Color Scale Bar Visibility

The color scale bar display can be turned on or off in the Contents window by checking or unchecking the box to the left of the color scale bar name.

Adding a Color Scale Bar

To add a color scale bar to a contour map:

1. Click on a contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Check the *Color scale* box to display the color scale bar.
4. Check the *Fill contours* box to fill the contours with color, if desired.

To add a color scale bar to a 3D wireframe:

1. Click on a wireframe map to select it.
2. In the Properties window, click on the Color Zones tab.
3. Click the *Edit Levels* button next to *Color zones*.
4. In the **Properties** dialog, check one or more of the *Apply zones to lines of constant* boxes.
5. Check the *Color Scale* box.
6. Click *OK* in the dialog to draw the wireframe with a color scale bar.

Removing a Color Scale Bar

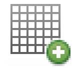
A color scale bar can be removed by clicking on the scale bar and pressing the DELETE key on your keyboard. Alternatively, you can clear the *Color Scale* check box in the **Properties** window or the **Properties** dialog.

Color Scale Bar Properties

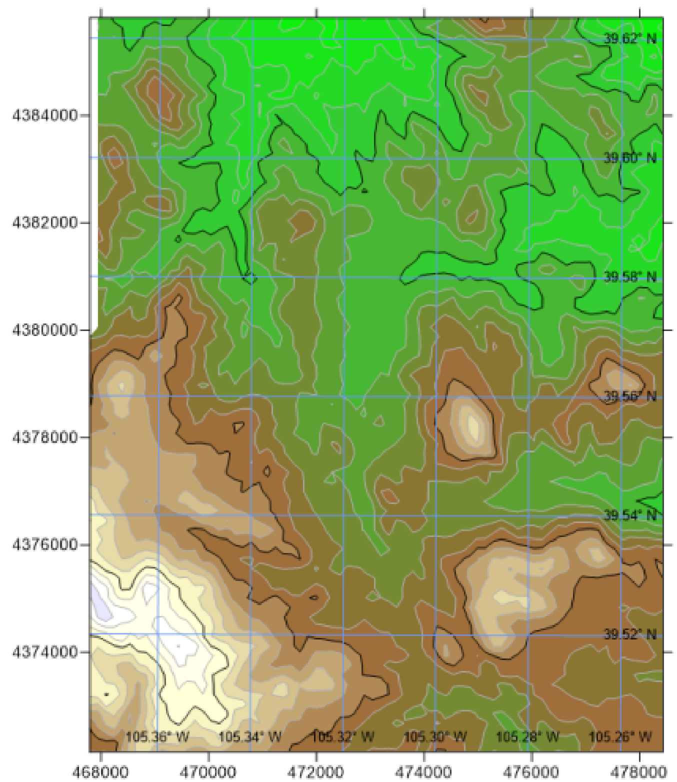
To modify a color scale bar, click on the color scale bar to select it. The properties are listed in the **Properties** window.

General
Labels
Line
Info

Graticule

Clicking the **Map Tools | Add to Map | Graticule** command or the  button adds a graticule to an existing map. A graticule is a network of lines representing the Earth's parallels of latitude and meridians of longitude. A graticule represents lines of latitude and longitude. A map grid represents lines in other units. The **Map Tools | Add to Map | Graticule** command creates either a graticule or map grid. An existing map or one of its child objects must be selected to enable the **Map Tools | Add to Map | Graticule** command.

The graticule *Units*, *Origin*, *Spacing*, and Label Format are updated automatically when changing the map target coordinate system.



A graticule has been added to this contour map to indicate lat/long coordinates in addition to the coordinate system values displayed on the left and bottom axes.

To add a graticule to a map:

1. In the Contents window or plot window, click on the map to which the graticule should be added.
2. Click the **Map Tools | Add to Map | Graticule** command. Alternatively, right-click the map and select **Add | Graticule**in the context menu.

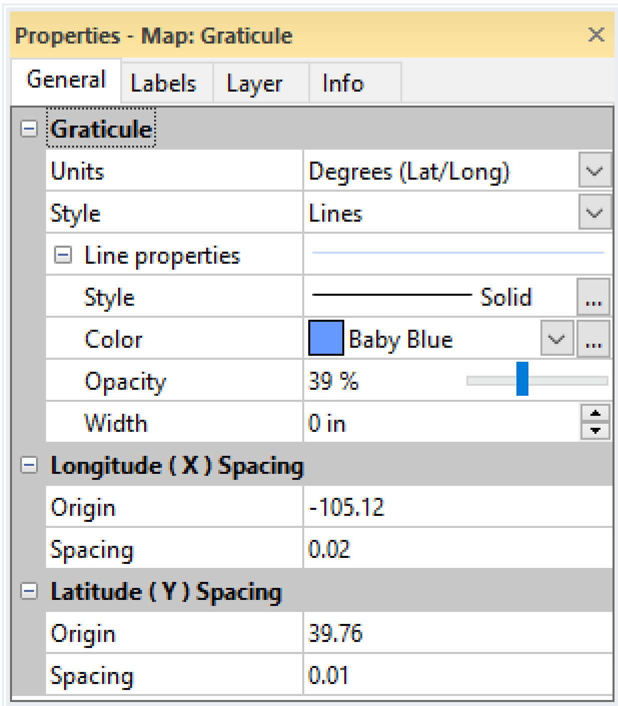
3. A graticule is added to the map. Select the graticule in the **Contents** window or plot window to view its properties in the Properties window.

Graticule Properties

The graticule properties are edited in the **Properties** window. The **Properties** window contains **General**, Labels, Layer, and Info pages.

General Page

The **General** page of the **Properties** window contains options for the graticule units, line style, and spacing.



*The **General** page of the **Properties** window controls graticule display options.*

Graticule Units

The *Units* property specifies the units for the graticule. The options in the *Units* list are dependent on the map coordinate system. If the map is projected, the available options include *Degrees (Lat/Long)* , *Millimeters*, *Centimeters*, *Meters*, *Kilometers*, *Inches*, *Feet*, *Survey Feet*, *Yards*, and *Miles*. If the map uses a known physical unit, the linear units in the preceding list are available. If the map has a datum, *Degrees (Lat/Long)* and *Unknown map units* are available. If the map is unreferenced, only *Unknown map units* is available. Click the current selection and select a new unit from the *Units* list to change the graticule units.

Graticule Style

The *Style* property specifies if *Lines* or *Crosses* are displayed to indicate the graticule. Click the current selection and select *None*, *Lines*,or *Crosses* from the list to change the graticule style. The *Cross size* property is enabled when the *Style* is set to *Crosses.* Change the *Cross size* by typing a value in page units into the *Cross size* field or clicking the buttons.

Graticule Line Properties

The *Line Properties* section controls the display for graticule *Lines* or *Crosses*. See the Line Properties help topic for more information on editing line properties. By default, the graticule is created with a *Baby Blue* line color and 39% opacity.

Graticule Line Origin and Spacing

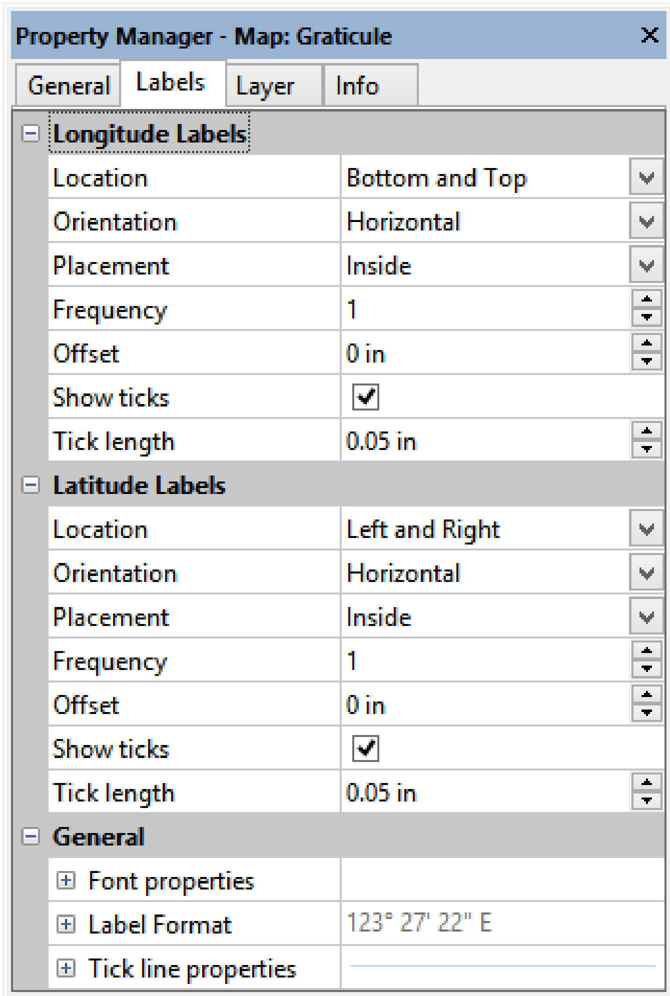
Graticule lines are drawn at a specified latitude and longitude and equally spaced from the origin. The *Longitude (X) Spacing* section controls the placement and spacing of longitude graticule lines. The *Latitude (Y) Spacing* section controls the placement and spacing of latitude graticule lines.

The *Origin* property determines the alignment point for the graticule lines. Specify the *Origin* by typing a value in the *Units* specified in the **Properties** windowinto *Origin* field. The *Origin* also controls which lines are labeled when the *Label Frequency* property on the **Labels** page is a value other than 1. The graticule line at the origin is always labeled.

The *Spacing* property specifies the distance between the graticule lines. Set the graticule *Spacing* by typing a value in the *Units* specified in the **Properties** window into the *Spacing* field. The minimum *Spacing* value is arbitrarily limited so that a maximum of 10000 graticule lines are drawn.

Labels Page - Graticule

The **Labels** page in the Properties window controls the display options and label formats for graticule labels.



*The **Labels** page of the **Properties** window controls graticule label display options.*

Longitude Label Location

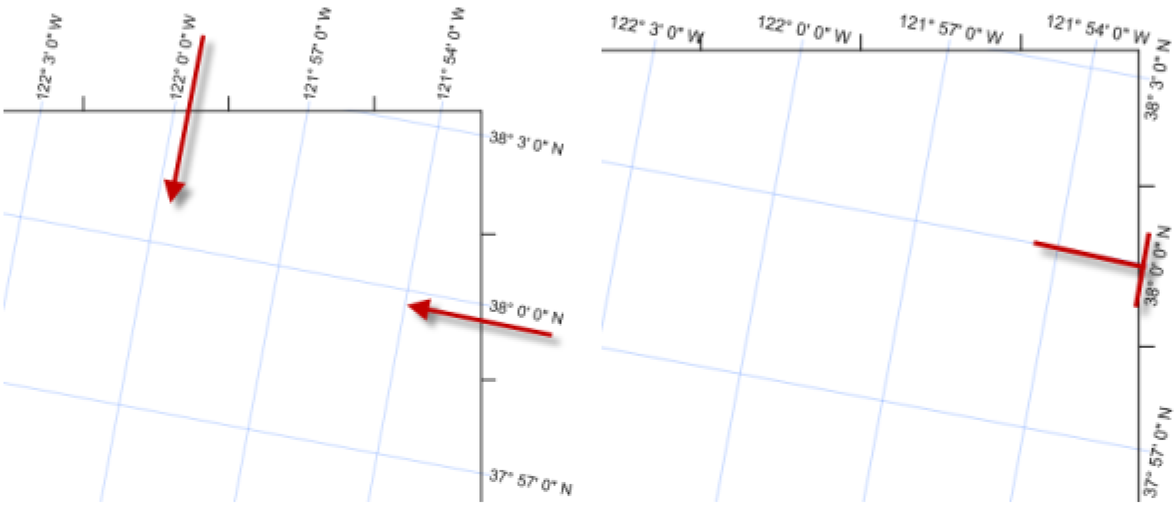
The *Location* property in the *Longitude Labels* section controls the label placement in reference to the graticule. Click the current selection in the *Location* field and select *None*, *Bottom*, *Top*, or *Bottom and Top* from the list. *None* disables longitude labels. *Bottom* displays the labels below the graticule. *Top* displays the labels above the graticule. *Bottom and Top* displays labels both above and below the graticule.

Latitude Label Location

The *Location* property in the *Latitude Labels* section controls the label placement in reference to the graticule. Click the current selection in the *Location* field and select *None*, *Left*, *Right*, or *Left and Right* from the list. *None* disables latitude labels. *Left* displays the labels to the left of the graticule. *Right* displays labels to the right of the graticule. *Left and Right* displays labels on both sides of the graticule.

Label Orientation

The *Orientation* property specifies the label orientation. Click the current selection in the *Orientation* field and select *Horizontal*, *Vertical*, *Along line*, or *Normal to line* from the list. *Horizontal* displays the labels horizontally. *Vertical* displays the labels vertically. *Along line* displays the labels parallel to the graticule lines. *Normal to line* displays the labels perpendicular to the graticule lines.



The image on the left shows the labels oriented parallel to the graticule lines when the Along line *option is selected for the Orientation property. The image on the right shows the labels oriented perpendicular to the graticule lines with the Normal to line option selected.*

Label Placement

The *Placement*property specifies the label placement in reference to the map frame. Click the current selection in the *Orientation* field and select *Outside* or *Inside* from the list. *Outside* displays the labels outside the map limits*. Inside* displays the labels inside the map limits.

3D Surfaces are limited to the extents of the map, so the *Placement* must be *Inside* for the graticule labels to be visible on a 3D Surface map.

Label Frequency

The *Frequency* property specifies for which graticule lines labels are displayed. The labels are displayed at every *n*th graticule line, starting from the origin. A *Frequency* value of 1 displays labels at every graticule line, 2 displays labels at every other line, 3 displays labels at every third line, etc. A label is always displayed at the origin value, unless *None* is selected in the *Location* field. Type a value of 1 or greater into the *Frequency* field, or adjust the value by clicking the buttons.

Label Offset

The *Offset* property specifies the label distance from either the map frame or tick marks. The *Offset* is the distance from the map frame when ticks marks are not displayed. The *Offset* is the distance from the end of the tick marks when *Show ticks* is checked. Type a value in page units into the *Offset* field or click the buttons to adjust the *Offset*. Increasing the offset moves the labels away from the map or ticks and in the direction of the *Placement* property. When *Placement* is set to *Outside*, increasing the *Offset* moves the labels away from the map. When *Placement* is set to *Inside*, increasing the *Offset* moves the labels into the map.

Tick Marks

The *Show ticks* property controls the display of label tick marks. When *Show ticks* is checked, label ticks are displayed in the direction specified by the *Placement* property. Label ticks are not displayed when *Show ticks* is unchecked. Click the *Show ticks* check box to check or uncheck the box.

Tick Length

The *Tick length* property specifies the length of the tick marks from the map edge. The *Tick length* property is enabled when *Show ticks* is checked. Type a value in page units or click the buttons to change the *Tick length* value.

Label Font

The *Font Properties* section of the **Labels** page controls the label font properties. See the Font Properties help topic for more information on changing font properties.

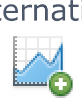
Label Format

The *Label Format* section of the **Labels** page controls the label format properties. See the Label Formats help topic for more information on changing format properties.

Tick Line Properties

The *Tick Line Properties* section of the **Labels** page controls the tick mark line properties. See the Line Properties help topic for more information on changing line properties.

Profile

The **Map Tools | Add to Map | Profile** command allows the user to digitize a line on the map. A profile, or cross section, is created automatically from the line and a base map layer added to the existing map. Alternatively, right-click on an existing map and click **Add to Map | Profile** or the  button to add a profile.

Profiles display distance traveled from the first point to the last on the X Axis and elevation on the Y Axis.

Adding a Profile

To add a profile by drawing a new polyline on a map:

1. Select the map or map layer and click the **Map Tools | Add to Map | Profile** command. The cursor changes to .
2. Draw a line by clicking in the locations where the profile should be located. See the *Polyline* topic for more information on drawing lines.
3. Double-click the last point or press ENTER on the keyboard to end the drawing mode.

The profile is automatically placed below the bottom axis and the profile line is added as a new base map layer. The base map layer is named *Base - Profile 1*. The *Base - Profile 1* will automatically assign the coordinate system from the Map coordinate system. The profile is named *Profile 1.* The 1 increments for both the profile name and the base layer name for each additional profile added to the map.

To add a profile using an existing polyline:

1. Select the polyline you wish to use for the profile. The polyline must be in a base layer in a map that also includes a grid-based layer.
2. Click the **Map Tools | Add to Map | Profile** command or right-click the polyline and select **Add Profile**.

The profile is automatically placed below the bottom axis. The profile number increments for each additional profile added to the map.

Editing a Profile

To make changes to the profile, click once on the profile in the **Contents** window or in the plot window to select it. The properties are displayed in the Properties window.

Reshaping the Profile Line

Profile lines can be altered with the reshape command. To edit the path of a profile, click on the *Polyline* object in the *Base - Profile* layer in the map to select it. Click the **Features | Edit Features | Reshape** command. You can then move existing points on the line by clicking and dragging the point to the desired location. Add new points to the line by holding down the CTRL key on the keyboard and clicking on the desired location. Delete points by clicking on the point and pressing the DELETE key on the keyboard. Press ESC on the keyboard when all edits are complete. The profile graph updates as the points are moved.

Moving the Profile Line

If a profile line needs to be moved on the map, click the *Polyline* object and move it to the desired location. The profile graph updates after the line is moved.

Resizing the Profile

Profiles can be resized by clicking on the *Profile* object in the **Contents** window. In the **Properties** window, click on the **Scale** tab. The *Length* of either axis can be changed.

Saving the Profile Line Coordinates

To save the coordinates of the profile line, click on the *Base - Profile* layer in the map to select it. In the **Properties** window, click on the **General** tab. Click the button next to the *Input file* option. Type a *File name* and click *Save.* Click *OK* in the options dialog and the .BLN file is created. The .BLN file can be used for other base maps or opened in the worksheet.

Saving the Profile Data

To save the elevations and distances along the profile, click the *Profile* object to select it. In the **Properties** window, click on the Plot tab. Click the *Save to File* button to save the information to a .DAT file.

Profiles with Multiple Map Layers

When multiple map layers are overlaid, a profile is created for each unique grid in the map. For example two profiles are created for a map that has three layers created from two grid files, i.e. a profile is created for each grid. Hide the display of individual profiles with the properties on the Plot page in the **Properties** window. Profiles are always created using the Map coordinate system.

When a new layer is added to a map with a profile, the new layer is automatically added to the profile.

Profiles and Vector Layers

Profiles are not created for vector layers. This is because the vector magnitudes are usually of a significantly different scale than the Z values in the grid file, and profiles of widely different scales cannot be reasonably plotted on the same set of axes. If you wish to create a profile of slope, gradient, or magnitude values from a vector layer, use the following process:

1. Use **Grid Calculus** Terrain Slope or Gradient Operator to create a grid of slope or gradient values if you are using a 1-grid vector layer. Use Grid Math with the function *sqrt(A^2 + B^2)*to create a grid of magnitudes if you are using a 2-grid vector layer with Cartesian coordinates. Use the magnitude

grid directly in step 2 if you are using a 2-grid vector layer with polar coordinates.
2. Use Grid Slice to slice the slope, gradient, or magnitude grid created in step 1 along the profile trace. The trace line can be a loaded from a file, an existing base (vector) layer, or drawn in an empty base map. Create an *Output BLN* file in the **Grid Slice** dialog.
3. Create a base map from the *Output BLN* file. This is the profile of the slope, gradient, or magnitude along the profile trace. You will likely need to adjust the profile scale.

Profiles and 3D Maps

Profiles can only be added to maps that are not tilted and in the orthographic projection. To temporarily change a map to this orientation:

1. Click on the *Map* object in the **Contents** window.
2. In the **Properties** window, click on the **View** tab. Change the *Projection* to *Orthographic*.
3. Click and drag the *Tilt (degrees)*  to 90.
4. Then, click the **Map Tools | Add to Map | Profile** command.

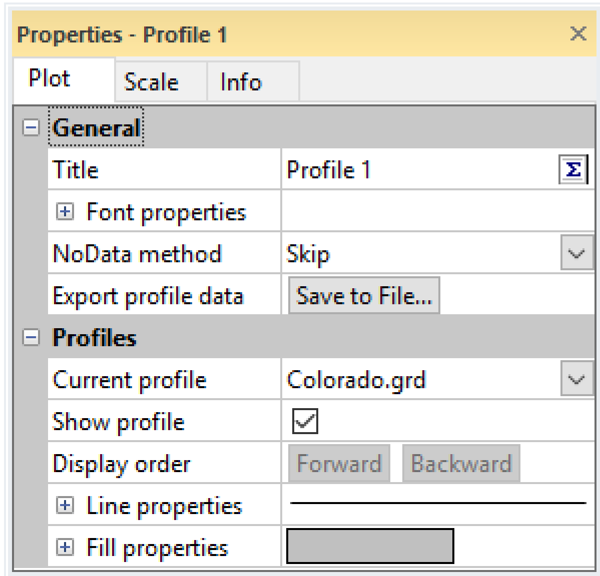
Once the profile is located on the map, the *Projection* and *Tilt (degrees)* properties can be changed back to their previous values.

Profile Plot Properties

Click the **Map Tools | Add to Map | Profile** command or right-click on a map and click the **Add to Map | Profile** to add a profile to an existing map.

Profile Properties

You have flexibility over the display and color of the profile. The **Plot** page controls the profile title and display order, line, and fill properties for the profile lines in the profile plot.



*The profile properties are set in the **Properties** window.*

General Section

The *General* section of the **Plot** page includes the title properties and *Export profile data* command.

Title

Profile titles can use any text properties including math text formatting instructions. Type the profile title exactly as you want it to appear on the *Title* line.

Titles can contain multiple lines. To create a multiple line title, click the Σ button. The Text Editor appears. Type the text in the dialog. To add a new line of text, press the ENTER key on the keyboard. To change the properties of the text, highlight the text and change the desired property. When finished, click *OK*. You are returned to the **Properties** window.

Font Properties

To edit all of the text properties for the entire title at once, click the ⊞ next to *Font properties*. You can then set any desired font settings, including font, size, and text color.

NoData Method

The *NoData method* property controls how NoData areas in the map are displayed in the profile. Available options are *Skip, Minimum,* and *Custom Value*. When the *NoData method* is set to *Skip*, the areas where NoData values are encountered are not displayed on the profile. When set to *Minimum*, the areas where NoData values are located are displayed with the minimum *Elevation* value of the profile. When set to *Custom Value*, the *Custom value* option becomes available. Type a value into the *Custom value* field, and the NoData areas are drawn in the profile with an elevation equal to the *Custom value*.

Export Profile Data

Click the *Save to File* button in the *Export profile data* field to save the profile elevation and distance data. In the Save As dialog, type a *File name* and click the *Save* button. A .DAT file is created.

The .DAT file contains one column of distance data and one or more columns of elevation data. The first column, Column A when opened in the worksheet, includes the accumulated distance along the profile line. The remaining columns include the elevation data for each profile in the profile plot. The DAT file also includes a header row with "Distance" in the first column and the profile name in the remaining columns.

Profiles Section

The *Profiles* section includes the display state, display order, and line and fill properties for each of the profiles in the profile plot.

Current Profile

Select the profile you wish to edit in the *Current profile* field. The selected profile is indicated in the profile plot with blue circles along the profile. Click the current selection and select the desired profile from the list. All profiles in the profile plot are included in the *Current profile* list. Profiles use the name of their corresponding grid file, when available. When a grid file name is not available, profiles use their corresponding layer name.

The *Current profile* list also indicates the order of the profiles in the plot. The profile at the back of the plot is listed first. The profile at the front of the plot is listed last.

Show Profile

Check the *Show profile* check box to display the *Current profile* in the profile plot. Clear the *Show profile* check box to hide the *Current profile* from the plot. Limits for the profile plot adjust automatically as profiles are turned on an off.

Display Order

Control the display order *Current profile* with the *Display order* property. Click *Forward* to move the *Current profile* forward one position in the plot. Click *Backward* to move the *Current profile* backward in position in the plot. When the profile plot contains only one profile, the *Forward* and *Backward* buttons are disabled.

Line Properties

Specify the line properties for the *Current profile* by clicking on the ⊞ next to Line Properties and setting the desired option. The line style, color, opacity, and width can be set.

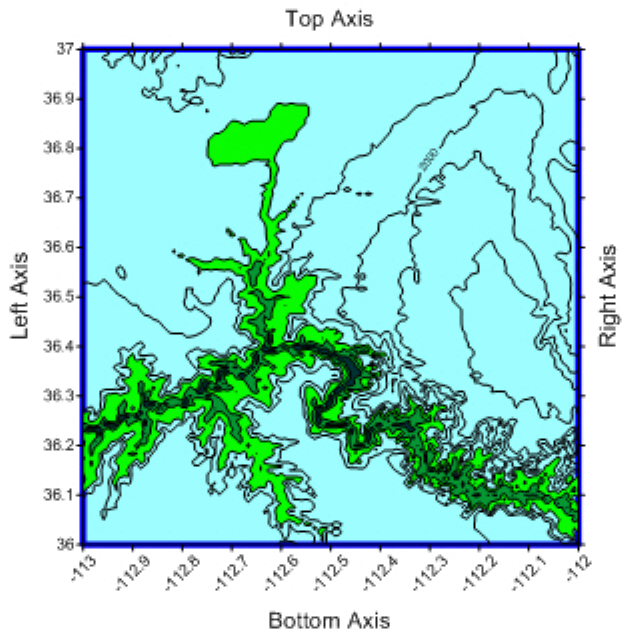
Fill Properties

Specify the fill properties for the Current profile by clicking on the ⊞ next to Fill Properties and setting the desired option. The fill pattern, color, and opacity can be set. The fill goes from the profile line to the X Axis. You may wish to use semi-transparent fill or no fill when profile lines intersect.

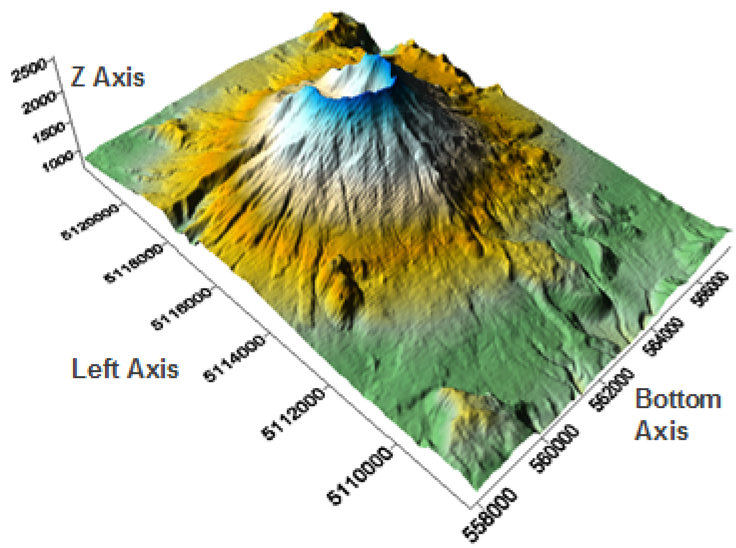
Map Axes

Four axes are automatically created for each map in **Surfer** . Axes are named by their relative positions on the map and are called the Left Axis, Right Axis, Top Axis, and Bottom Axis. The Left Axis and Right Axis are Y axes, and the Top Axis and Bottom Axis are X axes. 3D wireframe and 3D surface maps also have a single Z Axis. Variograms and profiles have two axes with the default names of X Axis and Y Axis.

By default, axis limits match the X, Y limits of the grid file, boundary file, or data file used to create the map. When a map is first created, the Left Axis and Bottom Axis display major ticks and tick labels, and the Top Axis and Right Axis display only major ticks. The tick spacing is automatically scaled so a reasonable number of ticks are drawn along each axis. Tick scaling and label format parameters can be independently specified for the axes on the map. By default, axes start with the lowest value at the bottom of Y axes and increase up and the lowest value at the left side of X axes and increase to the right. Axes can be reversed so that the small value is at the top of the Y axis or at the right side of the X axis. Axes can show numbers or date/time values.



This is a contour map with the four axes labeled with their default names.



This 3D surface map contains a Z axis in addition to the X and Y axes. 3D wireframe maps also contain a Z axis.

Editing Axes

To edit an axis, click on the axis to select it. The axis properties are listed in the Properties window on the General, Ticks, Scaling, Grid Lines, and Info tabs.

Axis Visibility

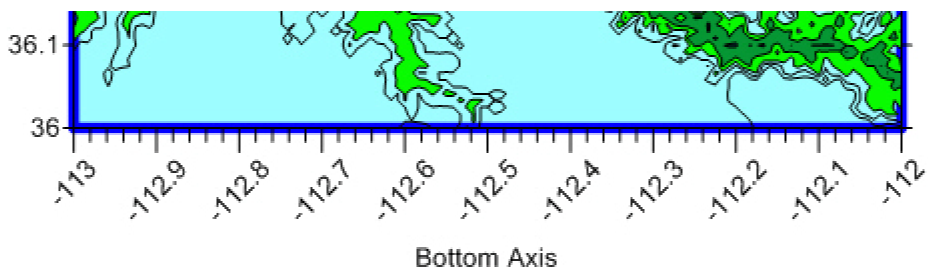
When the map, variogram, or profile is created, the axes are displayed automatically. You can turn off the display of any or all of the axes on the map by enabling the visibility of the axis with the Contents window. Click the check box adjacent to the axis name to turn on or off the axis. On a 3D wireframe or 3D surface, only the Left Axis, Bottom Axis, and Z Axis are initially visible.

Axis Position

Individual axes cannot be moved with the mouse, but you can position axes relative to the map layers on the Scaling page in the axis properties. Open the axis properties by clicking on the axis name in the **Contents** window.

Ticks

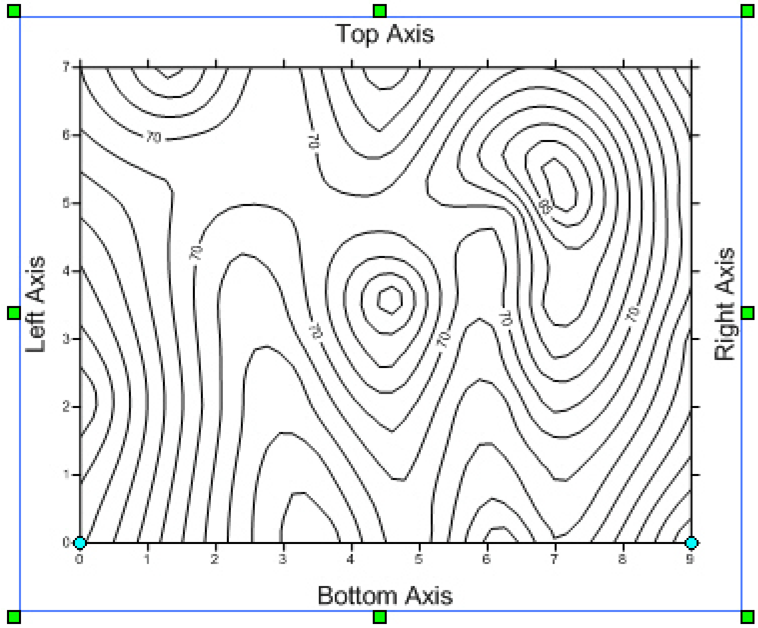
Map axes can display both major and minor ticks. Major ticks can display tick labels. Minor ticks are displayed at regular intervals between major ticks, and do not have associated tick labels. When tick labels are displayed, all major ticks along the axis are represented with a label. The display of tick marks is controlled from the Ticks page in the axis properties. The spacing of the major ticks along the axis is controlled from the Scaling page in the axis properties. Major tick labels are controlled by the *Labels* group on the General page in the axis properties.



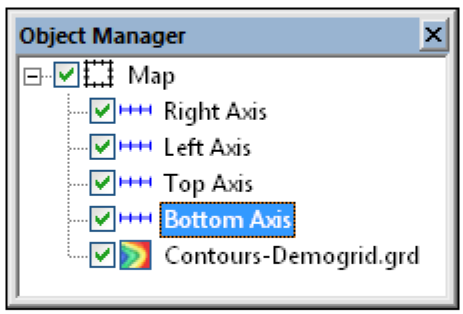
On this axis, four minor ticks are included with every major tick.

Selecting an Axis

Click an axis in the **Contents** window or plot window to select it. When selecting in the plot window, place the pointer on the tick marks or tick labels when clicking. Hollow blue circle handles appear at each end of the axis. Because the axes are part of the map on which they are drawn, the eight solid green square handles appear around the entire map. In addition, the name of the axis is displayed in the middle left side of the status bar. For example, if you select the bottom axis of a contour map, the text in the status bar appears as "Map: Bottom Axis." The axis is also highlighted in the **Contents** window. The properties for the axis are shown in the **Properties** window.



When an axis is selected, blue circle handles appear at the ends of the axis (shown on the bottom axis on this map) and green square handles appear around the entire map.



The selected map axis is highlighted in the Contents window. The axes checked are displayed on the map.

Other Axis Features

Other axis features are axis titles and grid lines. Axis titles can be placed on any of the axes and are drawn in relation to the associated axis. Grid lines are drawn at the same position as the axis tick marks.

Adding Additional Axes to Map

Click the **Map Tools | Add to Map | Axis** command or the  button to add any number of:

- **Left Axes**
- **Right Axes** ,
- **Top Axes** ,
- **Bottom Axes** , or
- **Z Axes** ,

to your current map. Alternatively, right-click on an existing map and click **Add to Map** to add a map layer, axis, or scale bar.

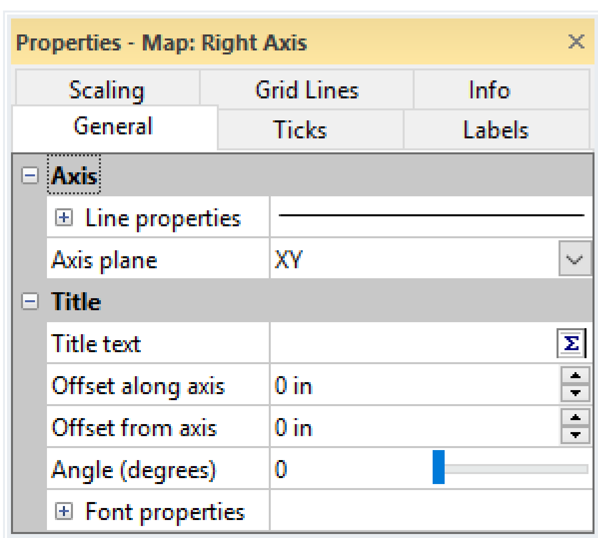
Click on the axis to display the axis properties in the **Properties** window. The position of the axis is changed by changing the *Crosses At* value on the Scaling page.

Axis General Properties

Add an axis title, change axis line properties, display labels, or change the axis plane to make an axis more informative and customized. These items are set on the **General** page for axis properties. To display the axis properties, click once on the axis. The properties are displayed in the Properties window.

General Page

The **General** page controls the display of axis titles, axis lines, tick labels, and the plane of the axis.



*Specify the axis title and line settings in the **Properties** window on the **General** page.*

Axis

The *Axis* section controls the axis line properties and the plane of the axis.

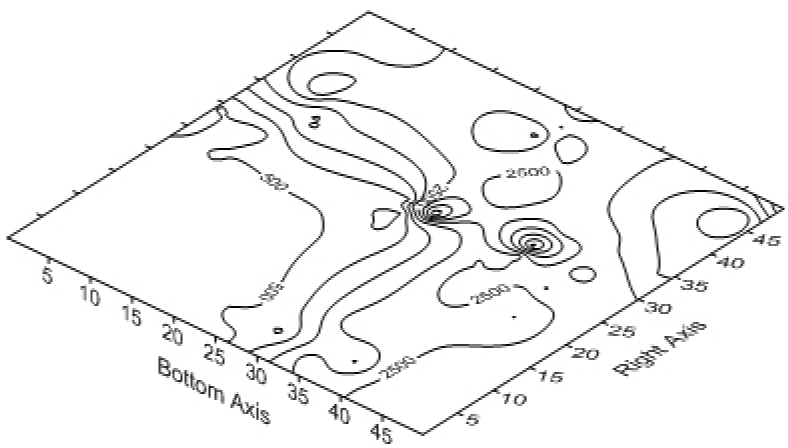
Line Properties

Click the ⊞ button next to *Line Properties* to display the Line Properties options. Choose the line *Style*, *Color*, *Width*, and *Opacity* for the selected axis.

Axis Plane

The *Axis Plane* option specifies the plane in which the tick marks and labels are drawn. For maps with a 90-degree tilt, axis tick marks and labels must appear in the XY plane to be seen. The*XY*option draws the tick marks and labels in the plane defined by the X and Y axes. For planar view maps (the default orientation for base, contour, post, etc. maps), use this setting.

The *XZ* option draws the tick labels in the plane defined by the X and Z axes. The *YZ* option draws the tick labels in the plane defined by the Y and Z axes. These options are used only when the tilt for the map is less than 90 degrees.



Axis tick labels can be drawn in different axis planes and can be rotated to make reading easier. In this example, the bottom axis is on the XZ Axis Plane *and the right axis is on the* XY Axis Plane.

Title

Axis titles can use any text properties including math text formatting instructions. Type the axis title exactly as you want it to appear on the *Title text* line.

Titles can contain multiple lines. To create a multiple line title, click the Σ button. The Text Editor appears. Type the text in the dialog. To add a new line of text, press the ENTER key on the keyboard. To change the properties of the text, highlight the text and change the desired property. When finished, click *OK*. You are returned to the **Properties** window.

Offset along Axis

The *Offset along axis* box controls the placement of the title along the axis. Positive offset values move the axis title in the positive direction along the axis. The offset values are set in page units.

Offset from Axis

The *Offset from axis* box controls the placement of the title towards or away from the axis. Positive offset values move the axis title away from the axis. The offset values are set in page units.

Angle (degrees)

The *Angle (degrees)* box controls the angle at which the title is drawn. Positive angles rotate the title in a counterclockwise direction. Values must be between 0 and 360. To change the *Angle (degrees)*, highlight the existing value and type a new number. Or, click and drag the bar to the desired location. The text moves in the plot window as the slider moves.

Font Properties

Click the ⊞ button next to *Font Properties* to specify the text properties for the axis title.

Axis Ticks Properties

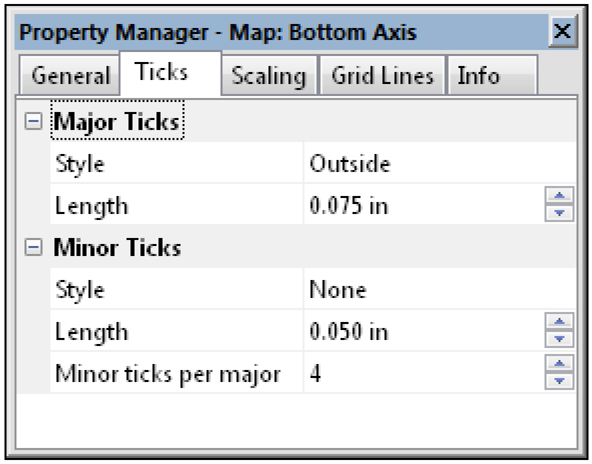
Axes can display both major and minor ticks. When a map is created, the display of major ticks is on by default. Major ticks display tick labels along the axis. Minor ticks on an axis are not represented with a tick label, and are not drawn on an axis by default. Minor ticks are defined as the number of minor ticks to be drawn between adjacent major ticks.

Axis Plane

The *Axis plane* option specifies the plane in which the tick marks and labels are drawn. The *Axis plane* property is located on the General page. For maps with a 90-degree tilt, axis tick marks and labels must appear in the XY plane to be seen. The *XY* option draws the tick labels in the plane defined by the X and Y axes. For planar view maps (the default orientation for base maps, contour maps and post maps), use this setting. The *XZ* option draws the tick labels in the plane defined by the X and Z axes. The *YZ* option draws the tick labels in the plane defined by the Y and Z axes. These options are used only when the tilt for the map is less than 90 degrees.

Ticks Page

The **Ticks** page controls the display of major and minor ticks, the position, and length of ticks.



Specify the axis tick properties in the ***Properties*** *window on the* ***Ticks*** *page.*

Major Ticks

The *Major Ticks* section controls the display of the major tick marks. Click the ⊞ next to *Major Ticks* to open the *Major Ticks* section.

Style

Click the *Style* option to display a list. In the list, select the desired tick mark location. Options are *Outside, Inside*, or *Cross* the selected axis. If you do not want to show ticks, select *None* from the list.

Length

The *Length* box controls the length of the major tick marks. The tick lengths are in page units.

Minor Ticks

The *Minor Ticks* section controls the display of the minor tick marks. Click the ⊞ next to *Minor Ticks* to open the *Minor Ticks* section.

Style

Click the *Style* option to display a list. In the list, select the desired tick mark location. Options are *Outside, Inside*, or *Cross* the selected axis. If you do not want to show minor ticks, select *None* from the list.

Length

The *Length* box controls the length of the minor tick marks. The tick lengths are in page units.

Minor Ticks Per Major

To specify the number of minor ticks to draw between adjacent major ticks, enter the number into the *Minor ticks per major* box. This should be one less than the number of minor divisions you want to appear between adjacent major ticks. Specify a value between 0 and 100.

For example, if you want the minor ticks to indicate 10 divisions, the *Minor ticks per major* value should be set to 9.

Axis Legend Properties

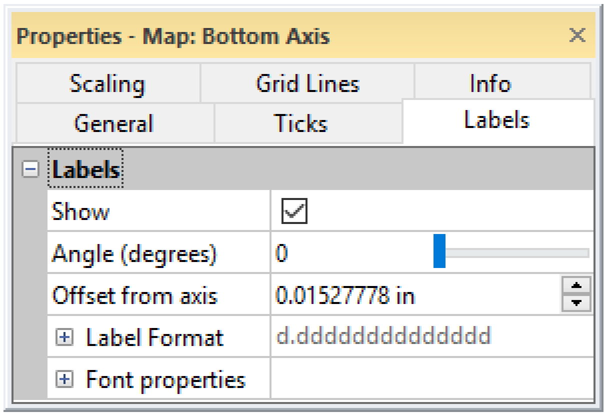
Tick labels are the numbers or text that appear at major tick mark locations. Tick labels indicate the axis values in data units. The **Labels** page includes properties to set the axes label position, format, and font. To display the axis properties, click once on the axis. The properties are displayed in the Properties window.

Axis Plane

The *Axis plane* option specifies the plane in which the tick marks and labels are drawn. The *Axis plane* property is located on the General page. For maps with a 90-degree tilt, axis tick marks and labels must appear in the XY plane to be seen. The *XY* option draws the tick labels in the plane defined by the X and Y axes. For planar view maps (the default orientation for base maps, contour maps and post maps), use this setting. The *XZ* option draws the tick labels in the plane defined by the X and Z axes. The *YZ* option draws the tick labels in the plane defined by the Y and Z axes. These options are used only when the tilt for the map is less than 90 degrees.

Labels Page

The **Labels** page controls the display of tick labels.



*Specify the axis tick label settings in the **Properties** window on the **Labels** page.*

Show

The *Show* check box controls the display of tick labels. When the *Show* box is checked, major ticks on the selected axis are displayed with a label.

Angle (degrees)

The *Angle (degrees)* box specifies the angle at which the tick labels are drawn. Positive angles rotate each axis tick label in a counterclockwise direction. Values must be between 0 and 360. To change the *Angle (degrees)*, highlight the existing value and type a new number. Or, click and drag the bar to the desired location. The text moves in the plot window as the slider moves.

Offset From Axis

The *Offset from axis* box controls how far away the tick labels are drawn from the tick marks. Positive offset values move the axis title away from the axis. The offset values are set in page units.

Label Format

Click the ⊞ button next to *Label Format* to display the Label Format options. This sets the numeric format used for the tick labels. Labels can be formatted with *Fixed*, *Exponential*, *Compact*, *Date/time*, or *DMS (Lat/Long) numeric formats*. See the Label Formats help topic for more information on the label format types and options.

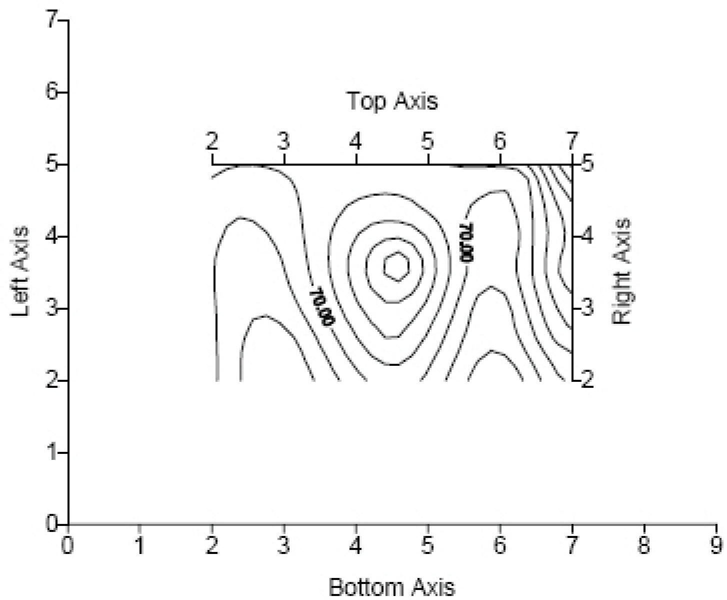
Font Properties

Click the ⊞ button next to *Font Properties to* display the text properties for the axis tick labels. The *Font, Size (points), Foreground color, Foreground opacity, Background color, Background opacity, Bold, Italic, Strikeout,* and *Underline* options are available.

Axis Scaling Properties

Axis scaling controls the axis limits, the major tick starting and ending values, and the position of the axis relative to the other axes in the map. To set axis scaling, click on the axis to select it. Click on the **Scaling** page in the Properties window.

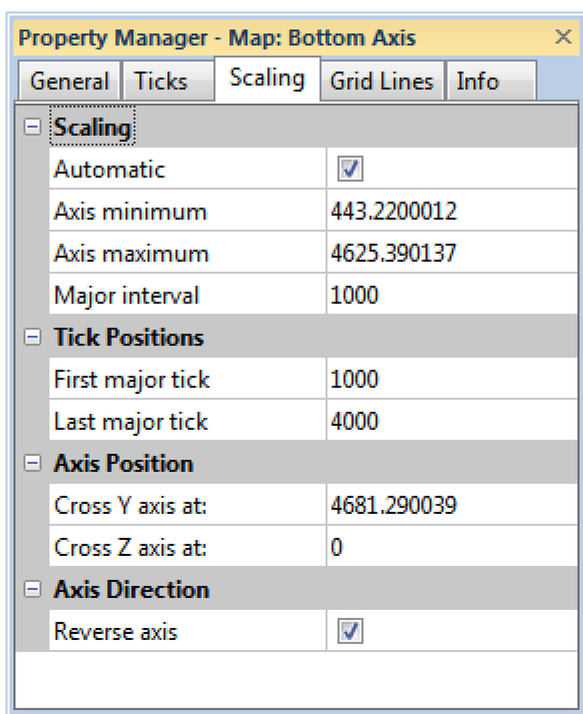
Axis scaling is different from setting the map scale. When axis scaling is set, the axis limits are defined but does not rescale the map. Axes that use different limits than the map are still drawn at the same scale as the map. Because of this, it is possible to have axes shorter or longer than the map. To change the map scale, click on the *Map* object and click the Scale tab in the **Properties** window.



In this example, the Bottom Axis and Left Axis are offset from the map, and the limits of the axes are set differently than the map limits.

Scaling Page

The **Scaling** page for an axis in the **Properties** window controls the axis scaling properties.



*Specify the axis scaling in the **Properties** window on the **Scaling** page.*

Automatic Scaling

The *Automatic* check box uses the limits of the map as the limits for the axes and uses a reasonable tick mark spacing along the axis. When any parameters are changed on the axis properties **Scaling** page, the *Automatic* check box is automatically unchecked. After making any changes, you can return to the automatic settings by checking the *Automatic* box.

Axis Minimum

The *Axis minimum* box specifies the minimum axis value. To change the minimum value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

When using date/time formats for the axis labels, enter the minimum date/time axis value. For instance, 02/02/2014 12:00:00 AM can be entered into the *Axis minimum* option.

Axis Maximum

The *Axis maximum* box specifies the maximum axis value. To change the maximum value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

When using date/time formats for the axis labels, enter the maximum date/time axis value. For instance, 10/31/2014 12:00:00 AM can be entered into the *Axis maximum* option.

Major Interval

The *Major interval* box contains the value of the spacing, in data units, between major ticks along the selected axis. The major tick marks occur at values

Tick = First major tick + n Major interval n* =0, 1, ..., *N*

where *N* is the largest integer such that *Tick ≤ Last major tick*. Thus, a tick mark is placed at the value of the *Last major tick* only if the interval (*Last major tick - First major tick*) is evenly divisible by *Major interval*. To change the interval, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

When using the date/time formats for the axis labels, click on the existing time scale (*Years, Weeks, Days, Hours, Minutes*, or *Seconds*) and select the desired time scale to select how often the labels are drawn. Then, highlight the existing value and type a new value to set how many of the desired time scales should occur before the next tick. Alternatively, click the  to increase or decrease the value. For instance, setting the *Major interval* to 3 Years displays a tick at the *First major tick* location. The second tick will occur 3 years after the first.

When date/time axis labels are displayed, the Major interval is set in date/time units.

First Major Tick

The *First major tick* box contains the major tick mark minimum value. This value is expressed in data units. The *First major tick* does not need to be the same as the *Axis minimum* but it cannot be less than the *Axis minimum*. To change the *First major tick* value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

When using date/time formats for the axis labels, enter the first major date/time axis tick mark value. For instance, 02/02/2014 12:00:00 AM can be entered into the *First major tick* option.

Last Major Tick

The *Last major tick* box contains the maximum allowed major tick mark value. This value is expressed in data units. The *Last major tick* does not need to be the same as the *Axis maximum* but it cannot be greater than the *Axis maximum.* To change the *Last major tick* value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

When using date/time formats for the axis labels, enter the last major date/time axis tick mark value. For instance, 10/31/2014 12:00:00 AM can be entered into the *Last major tick* option.

Axis Position

Axes are a part of the map on which they are placed. By default, axes are drawn at the edge of a map or surface. Map axes are placed in relation to the other axes on the map.

A contour map has two Y axes: the Left Axis and Right Axis. The placement of the Y axes are relative to the data units along the X axes (the Top Axis and Bottom Axis) on the map. By default, the left Y axis crosses the X axes at the minimum X axis value.

Cross Y Axis At

The *Cross Y axis at* box is only displayed when an X or Z axis is selected. This specifies the position of the X or Z axis along the Y axis. The value is entered in Y data units. To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

When using date/time formats for the axis labels, the *Cross X axis at, Cross Y axis at,* and *Cross Z axis at* must always be in numbers, not in date/time format.

Cross Z Axis At

The *Cross Z axis at* box specifies the position of the X or Y axis relative to the Z axis on a 3D wireframe or surface map. The value is entered in Z data units. To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. By default, the X and Y axes are drawn at the elevation of the base defined for the wireframe or surface map. For example, if the base for the wireframe is set to be 25, the X and Y axes cross the Z axis at an elevation of 25.

When using date/time formats for the axis labels, the *Cross X axis at, Cross Y axis at,* and *Cross Z axis at* must always be in numbers, not in date/time format.

Cross X Axis At

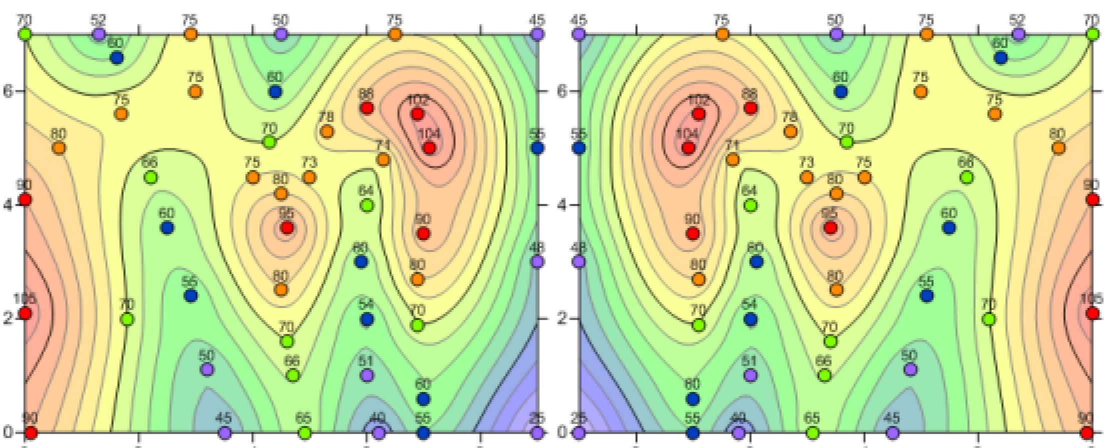
The *Cross X axis at* box is only displayed when an Y or Z axis is selected. This specifies the position of the Y or Z axis along the X axis. The value is entered in X data units. To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

When using date/time formats for the axis labels, the *Cross X axis at, Cross Y axis at,* and *Cross Z axis at* must always be in numbers, not in date/time format.

Axis Direction

To make an axis descending, check the box next to *Reverse axis*. Map layers are automatically updated when the axis is reversed. When the axis is normal (the *Reverse axis* box is unchecked), the axis minimum value is on the left side for an X axis and on the bottom for a Y axis. When reversed, the axis minimum is located on the right side for an X axis and the top for a Y axis. When one X axis is reversed (i.e. *Bottom Axis*), all X axes on the map are automatically reversed. When one Y axis is reversed (i.e. *Right Axis*), all Y axes on the map are automatically reversed.

Z axes and variogram axes cannot be reversed.



The two maps both display contours and classed post data from the same files. The map on the right has the Reverse axis option checked for the Bottom Axis . All map properties are reversed.

Setting the Axis Scale

Axis limits are specified in map coordinates, profile, or variogram units. Axis limits do not need to coincide with the map limits. Use these steps to change the axis scale:

1. Click on an axis to select it. The **Properties** window automatically shows the properties for the axis.
2. Click on the **Scaling** tab.
3. You can set any of the parameters listed below.

   a. The *Automatic* box uses the limits of the map as the limits for the axes and uses a reasonable tick mark spacing along the axis. When any parameters are changed in the axis properties, the *Automatic* box is automatically unchecked. After making any changes, you can return to the automatic settings by checking the *Automatic* box.
   b. The *Axis minimum* box specifies the minimum axis limit.
   c. The *Axis maximum* box specifies the maximum axis limit.

The axis is automatically updated with the new axis scale settings as the values are changed.

Edit Major Tick Spacing on an Axis

On the Scaling page in the axis properties there are three options that control the spacing of major ticks along the selected axis: the *First major tick*, *Last major tick,* and *Major interval* boxes.

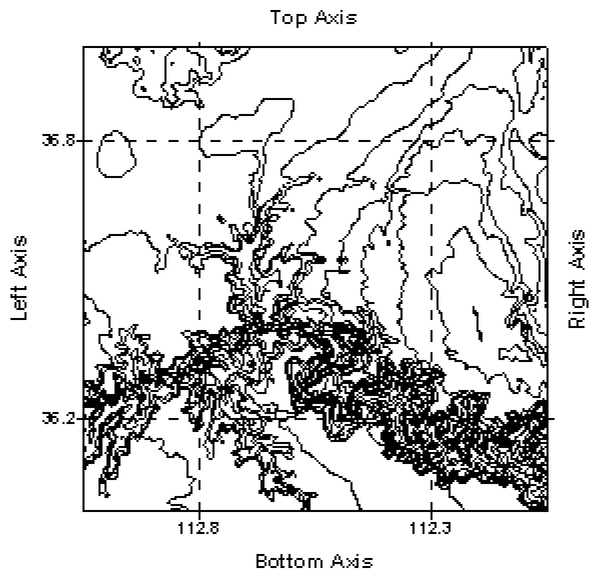
To edit major ticks on an axis:

1. Click once on the axis to select it.
2. In the Properties window, click the **Scaling** tab.
3. You can specify the position for the first and last major ticks, and the spacing in data units between each major tick along the axis.
4. The *First major tick* box contains the value of the minimum major tick mark. This value is expressed in data units. The *First major tick* does not have to be the same as the axis minimum but it cannot be less than the axis minimum.
5. The *Last major tick* box contains the value of the maximum allowed major tick. This value is expressed in data units. The *Last major tick* does not have to be the same as the axis maximum but it cannot be greater than the axis maximum.
6. The *Major interval* box contains the value of the spacing, in data units, between major ticks along the selected axis. The major tick marks occur at values,

   *Tick = First Major Tick + n * Major Interval* n=0,1,...,N

   where N is the largest integer such that Tick <= Last Major Tick. Thus, a tick mark is placed at the value of the *Last major tick* only if the interval (Last Major Tick - First Major Tick) is evenly divisible by *Major interval*.

7. The axis is redrawn with the selected tick parameters.



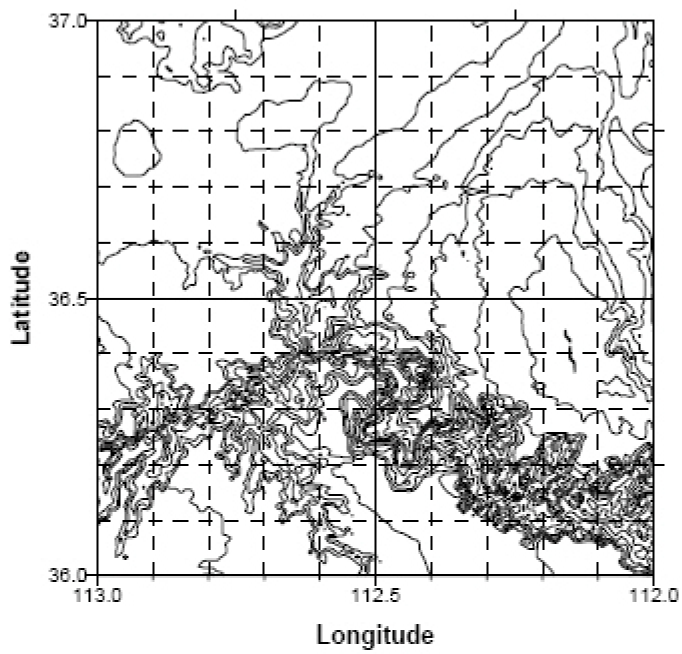
By setting the First major tick, Last major tick and Major interval values, you can control the position of major ticks along the axis.

Axis Grid Line Properties

Grid lines are a mesh over the top of the map. Along an axis, grid lines can originate from the major ticks, minor ticks, or both. The major and minor grid lines can use different line properties. When grid lines are added to 3D wireframe or 3D surface maps, the lines are drawn on the plane defined by the base of the map.

Grid lines are drawn over the extent of the map. For example, if you draw grid lines for an X axis (the Bottom Axis), the grid lines are drawn parallel to the Y axis (Left Axis) over the entire map. If your X axis is offset from the Y axis, the grid lines are not drawn beginning at the end of the X axis, but beginning at the minimum Y value on the map.

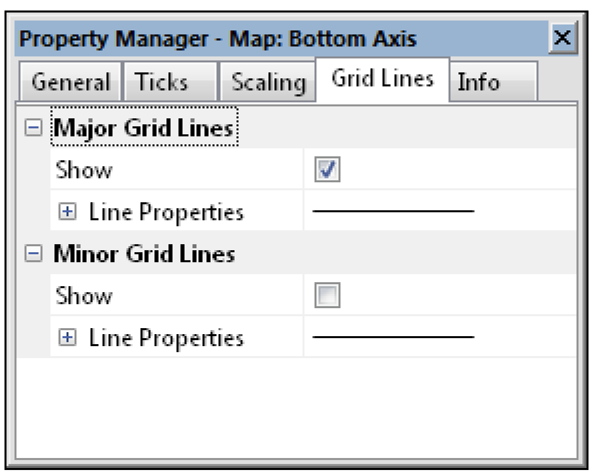
Specify the tick spacing to use for the grid lines on the Scaling page. The ticks do not need to be displayed for the grid lines to be drawn.



Grid lines originate from the tick marks. Different line properties can be used for the major and minor grid lines. In this example, major grid lines are solid and minor grid lines are dotted.

Grid Lines Page

The **Grid Lines** page controls the visibility and properties of major and minor grid lines.



Specify the axis grid line properties in the ***Properties*** *window on the* ***Grid Lines*** *page.*

Major Grid Lines

If you want grid lines to extend from the major ticks, check the *Show* box in the *Major Grid Lines* section. Click the ⊞ next to *Line Properties* to open the line properties section for the *Major Grid Lines*. Change the *Style, Color, Opacity,* and *Width* for the major grid lines.

Minor Grid Lines

If you want grid lines to extend from the minor ticks, check the *Show* box in the *Minor Grid Lines* section. Click the ⊞ next to *Line Properties* to open the line properties section for the *Minor Grid Lines*. Change the *Style, Color, Opacity,* and *Width* for the minor grid lines.

To Draw a Grid Covering the Map

1. Click on an axis to select it. The axis properties are automatically displayed in the Properties window.
2. Specify the tick spacing to use for the grid lines on the Scaling page. The ticks do not need to be displayed for the grid lines to be drawn.
3. Click the **Grid Lines** tab.
4. If you want grid lines to extend from the major ticks, check the *Show* box in the *Major Grid Lines* section.
5. Change the options under *Line Properties* for the *Major Grid Lines*. You can change the *Style, Color, Opacity,* and *Width* for the major grid lines.

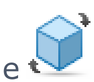
The grid lines are drawn on the map as you make changes in the **Properties** window.

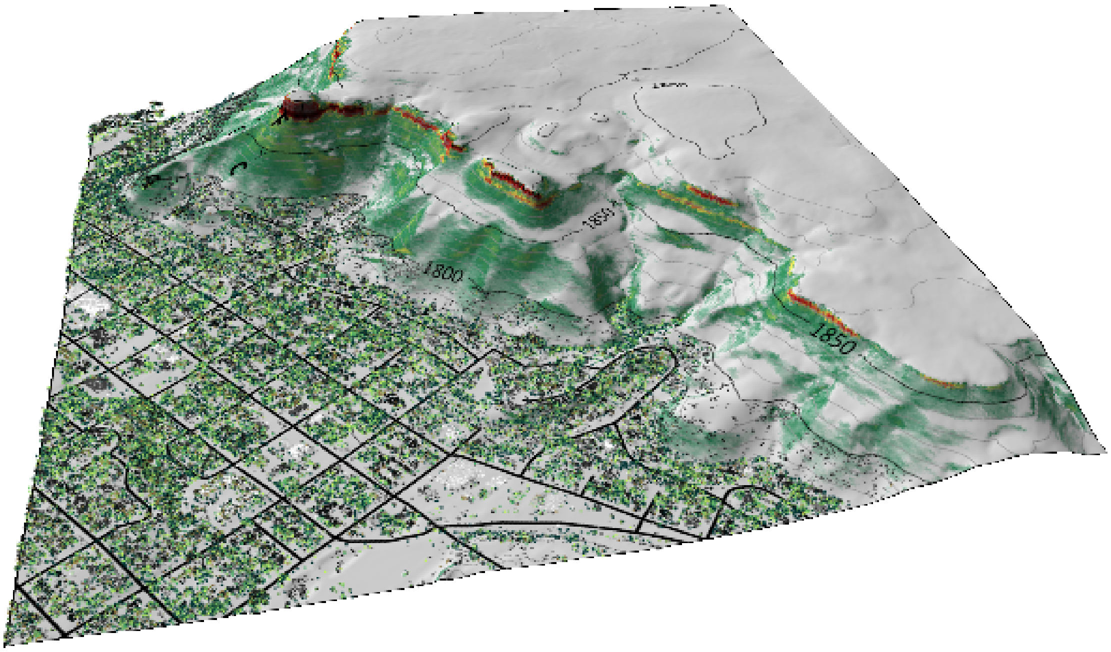
Chapter 24 - 3D View

3D View Window

The 3D view window displays a map in a three-dimensional view space. The input grids from grid-based layers are rendered as surfaces in the 3D view. Base (vector and raster), post, and classed post layers are overlaid on the surfaces as textures. Point data in base (vector) layers from data, post, and class post layers are also displayed in 3D. If the base (vector) layers have 3D polylines, 3D polygons or a 3D polymesh in them, then those 3D objects will display in true 3D in the 3D view. The visualizations from the grid-based maps are also overlaid on the surface. For example, contour lines or color relief layers will be overlaid on the surfaces. Point cloud layers are rendered as a 3D point cloud. Contour lines may be rendered as 3D polyline objects. However, contours are displayed as overlays on a surface and not as 3D polylines by default. If the base (vector) layers have 3D polylines, 3D polygons or a 3D polymesh in them, then those 3D objects will display in true 3D in the 3D view.

The 3D view window only displays layers that are visible in the map in the plot window. Show or hide surfaces, textures, and vector data in the 3D view by selecting or clearing the visibility check boxes in the 3D view **Contents** window. Completely remove a surface, texture, vector data, axis, or color scale from the 3D view by switching to the plot window and turning the visibility off for the associated map layer.

Click the **Map Tools | View | 3D View** command or the  button or the *3D View* button in the *Map* frame **View** properties page to open a 3D view window of the selected map. A new 3D view is created for the selected map. A 3D view can also be created by right-clicking a map or map layer and clicking **3D View** in the context menu. To create a 3D view, the map must include at least one of the following: a grid-based layer, a base from data, a post or classed post layer, a point cloud layer, or a base (vector) layer with a 3D object. The document tab includes the file name and view number. For example, when a 3D view is created for a map in the Plot1 plot window, the plot window tab name is *Plot1:1* and the 3D view window tab name is *Plot1:2*.



The 3D view displays the map in a three-dimensional space. This map includes contour, color relief, post, base, and point cloud layers.

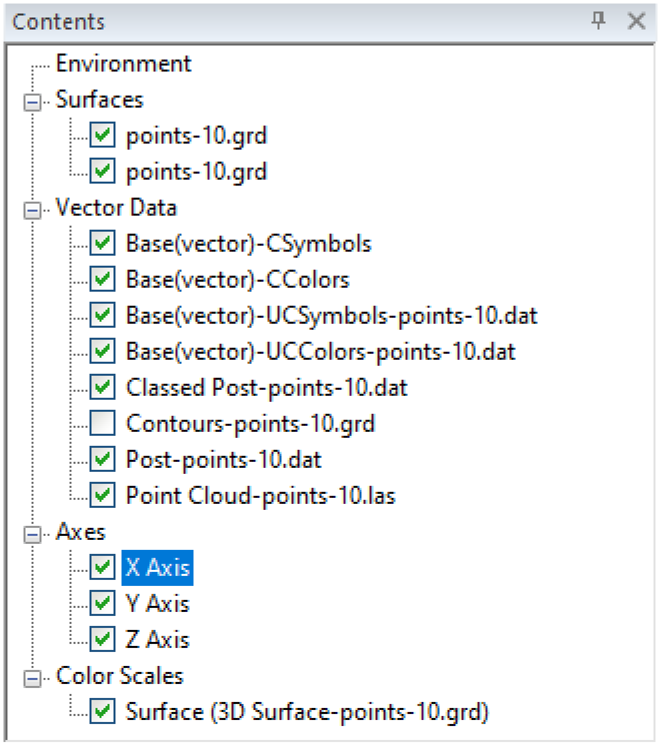
3D View Commands

The **3D View** ribbon tab contains commands for modifying the view and creating videos and images in the 3D view window.

Fit to Window	Zoom to fit the entire map in the 3D view window
Zoom In	Zoom in on the center of the 3D view window
Zoom Out	Zoom out from the center of the 3D view window
Realtime	Zoom by clicking and dragging the mouse
Pan	Pan the 3D view window
Trackball	Rotate the 3D view window
Walk	Walk along the surface in the 3D view window
Go to Home	Move the 3D view to the home position
Set Home	Set the 3D view home position
Create/Edit	Create or edit a fly-through
Play	Play the fly-through
Record	Create a video with the current fly-through settings
3D Digitize	Save XYZ coordinates to a text file
Copy to Clipboard	Copy an image of the 3D view window to the clipboard
Export Image	Export the current view in the 3D view window as an image
Export 3D	Export the map in the 3D view window to a 3D PDF model

Contents

The **Contents** window contains the *Environment* and the surface and vector objects in the 3D view. *Surfaces* are generated from the grids in the map, and *Vectors* are generated from the point cloud and contour layers in the map. Select the objects in the **Contents** window to change the properties of the 3D view and the map in the 3D view. A check mark next to the layer name indicates the layer is displayed in the 3D view. Clear the check box next to a layer to hide the layer in the 3D view.



*The **Contents** window includes the surfaces, vectors, axes, and color scales in the 3D View.*

Environment

The *Environment* object in the **Contents** window contains the properties for the 3D view. Select the *Environment* to modify the 3D view background, vertical exaggeration, water level, camera, and lighting. The *Environment* object has three Properties pages:

General
Camera
Lighting

The **Lighting** properties have no effect on the 3D view when a hill shaded or reflectance shaded color relief layer is present in the map. Turn off hill shading and reflectance shading for the color relief layer(s) to use the *Environment***Lighting** properties.

Surfaces Group

The *Surfaces* group in the **Contents** window contains the surfaces in the 3D view. A surface is created for each grid in the map. The *Surface* group includes properties for controlling the rendering quality of all the surfaces in the 3D view and their overlay textures. The *Surfaces* group has one **Properties** page, General.

Individual Surfaces

Each surface in the 3D view can have overlays from the map. The available overlay textures are listed under the General page in the **Properties** window for each surface. Available textures include the grid-based map visualizations, e.g. contours and color relief, as well as any base or post layers in the map. Individual overlays can be hidden or displayed on each surface by selecting the surface you wish to edit in the **Contents** window, and then selecting or clearing the visibility check boxes next to the overlay layer names in the *Textures to display* section of the General page.

By default, each surface shows the texture from any map layer that uses the same grid file as the surface, and the textures from any base or post layers. Use the **General** page to hide or display textures.

The overlay textures do not have any 3D view properties. To change the appearance of the overlays, modify their properties in the plot window. Switch to the plot window by clicking the plot window document tab.

Vector Data Group

The *Vector Data* group in the **Contents** window contains the 3D vector objects in the 3D view, i.e. point cloud layers and 3D polyline contours. There are no *Vector Data* group properties.

Contours

Contours are created for contour layers in the map. By default, the contour layer is displayed as a texture overlay on the surface. Vector contour lines are hidden by default. Select the check box next to the contours vector object in the **Contents** window to display the contours. Change the properties of the contours on the General page of the **Properties** window. Leave the contours the same as those assigned in the 2D plot by selecting *Same as plot* for *Color Method*. Or, select *Fixed* to change the contours to have the same color and width in the 3D view.

Vector Data

3D point vector data is displayed for post and classed post layers in the map. 3D point, polygon, and polyline vector data is displayed for base (vector) layers in the map. The post, classed and post layers have one **Properties** page, General; and the base(vector) layer has two Properties pages, **General** and Z coordinate.

Point Clouds

Select the individual point cloud layers within the *Vector Data* group to change the *Point size* for the point clouds. The point cloud layers have one **Properties** page, General. The number of points in the point cloud is limited by the *3D View maximum number of points* in the **Options** dialog Rendering page.

Drillhole Group

3D drillhole layers are created for drillhole layers in the map. The *Drillhole* group has no properties. See the 3D Drillhole Properties topic for **Points**, **Intervals**, **Path**, and **Label** properties for 3D drillhole layers.

Axes Group

Three axes are automatically created for a map in the 3D view in **Surfer**. The *Axes* group in the **Contents** window contains the *X*, *Y*, and *Z* axes for the 3D map. All the axes are displayed by default. Clear an axis to remove it from the 3D map. Each axis has four *Properties* pages. Click on the axis in the **Contents** window to change its properties in the **Properties** window:

General
Ticks
Labels
Scaling

Color Scales Group

The *Color Scales* group in the **Contents** window contains color scales in the 3D view. A color scale is created for 2D layers that have a colormap. The *Color Scales* group adds or hides a color scale on the plot. The color scales can include any number of post, classed post, 1-grid vector, 2-rid vector, contour, point cloud and base colormaps from any of the maps in the plot. The total number of color scales is limited to the total number of layers that have their own color scales. The *Color Scales* group has one **Properties** page, Color Scales.

Individual Color Scales

Each color scale in the 3D view can have overlays from the map. The available overlay color scale for each 2D layer are listed under the *Color Scales* page in the **Properties** window. Select the color scale you wish to display on the 3D view. Clear the color scale to remove it from the 3D view. Color scales that are displayed may be edited by selecting the color scale in the **Contents** window. Color scales have four Properties pages:

General

Title
Labels
Frame

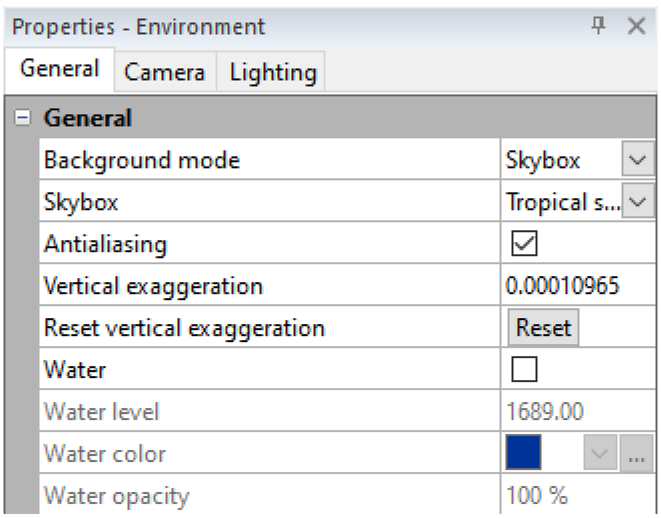
Color scales will not export to a 3D file but will export to images and be copied to the clipboard.

3D View Properties

The following sections describe the properties for the 3D view window and the various objects in the 3D view.

3D View Environment General Properties

The **Environment** page contains the 3D view window background, vertical exaggeration, antialiasing and water properties. Select the *Environment* object in the 3D view **Contents** window to modify the 3D view and 3D symbol properties.



*Set the 3D view background and water properties in the **General** page.*

Background mode

The *Background mode* property specifies the background style. Select *Color* to use a solid color for the 3D view window background. Select *Skybox* to use a skybox for the 3D view background. A skybox is a scene that creates the illusion of three-dimensional distant surroundings. The skybox is also useful for keeping track of the camera orientation when rotating the view.

Background Color

When *Background mode* is set to *Color*, the *Background color* property is displayed. Select the desired *Background color* from the color palette or click ... to create a custom color in the Colors dialog.

Skybox

When *Background mode* is set to *Skybox*, the *Skybox* property is displayed. Selected the desired *Skybox* from the list. Select *(none)* to use the previous solid color background.

Antialiasing

The *Antialiasing* property is selected by default to enable multisampling anti-aliasing. Copy-to-clipboard and export-to-image functions also employ the multisampled framebuffer when *Antialiasing* is selected. Older versions of Surfer and computers which do not have modern graphics cards will not show the *Antialiasing* property.

Vertical Exaggeration

The *Vertical exaggeration* property is a unitless scale factor for Z coordinates in the 3D view. The *Vertical exaggeration* scales the Z direction to maintain a proportional appearance in the 3D view when Z values are in different units than the XY coordinates or when the XY extents and Z values are in significantly different ranges. The default *Vertical exaggeration* scales the maximum Z extent to one-fourth the maximum X or Y range.

Increase the *Vertical exaggeration* to accentuate the differences in the surface Z values. Decrease the *Vertical exaggeration* to flatten the appearance of the objects in the 3D view. The slider can be used to decrease the *Vertical exaggeration* to one-tenth of the default value, and the slider can be used to increase the *Vertical exaggeration* to ten times the default value. To specify a *Vertical exaggeration* value outside this range, type the desired value directly in the *Vertical exaggeration* field.

The *Vertical exaggeration* is originally calculated based on the X, Y, and Z extents of data loaded into the 3D view when it is created. The *Vertical exaggeration* is recalculated when the extents of the visible data changes due to changing visibility for surfaces or vector data in the **Contents** window.

Water

Select the *Water* property to display a water level plane on the 3D view. Clear the *Water* property to remove the water level plane from the 3D view. A water level plane is an XY plane that extends across the extents of the map in the 3D view.

Water Level

The *Water level* property sets the Z value, or elevation, for the water level plane in the 3D view window. Type the desired *Water level* in the field or click and drag the slider to change the *Water level*.

Water Color

The *Water color* property sets the color of the water level plane. Select the desired *Water color* from the color palette or click ... to create a custom color in the Colors dialog.

Water Opacity

The *Water opacity* property sets the opacity of the water level plane. A value of 0% is completely transparent. A value of 100% is completely opaque. Type a value between 0 - 100 in the *Water opacity* field or click and drag the slider to adjust the water level plane opacity.

Semi-Transparent Water Level and 3D Surface Layers

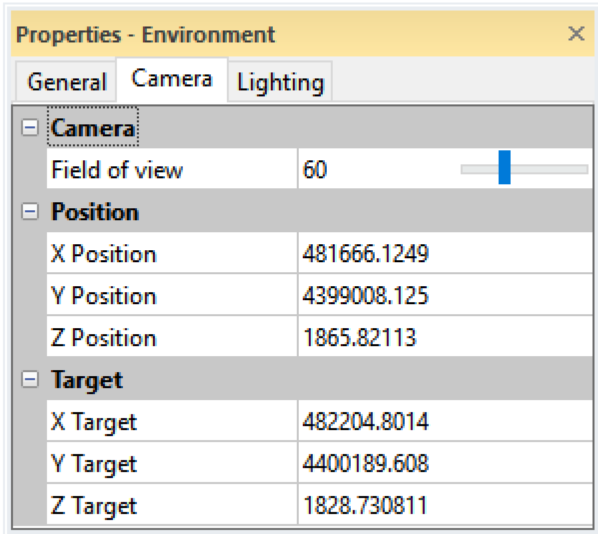
Semi-transparent water levels require certain settings. When applying a *Water opacity* less than *100%* to a 3D view, the following properties should be used:

1. Make sure *Show surface background* is selected on the surface General page.
2. Clear any 3D surface layer texture check boxes on the surface General page.
3. The surface should include at least one other layer texture.

If these options are not used, the water level opacity may be inconsistent. Note that this requires the 3D surface layer texture to be removed. Use a color relief layer instead of a 3D surface layer to maintain a colormap on the surface with a semi-transparent water level in the 3D view.

3D View Environment Camera Properties

The **Camera** page contains the camera field of view, location, and aim properties for the 3D view window.



*Set the camera FOV, position, and target in the **Camera** page.*

Field of View

The *Field of View* controls the perspective effect of the 3D view window. The perspective effect is the visual effect of the map changing size relative to the distance from the observer. The *Field of View* angle is the angle between the left edge of the 3D view window, the viewpoint, and the right edge of the 3D view window. A large *Field of view* increases the perspective effect, while a small *Field of view* decreases it. Type a value in the *Field of view* property field to change the *Field of view*, or change the *Field of view* by clicking and dragging the slider .

Position

The *Position* section includes the camera position properties: *X position*, *Y position*, and *Z position*. The *Position* properties update automatically as the camera is moved with the Pan, Trackball, **Zoom**, or Walk commands. You can also type coordinates into the *X position*, *Y position*, and *Z position* fields to move the camera.

Target

The *Target* section includes the camera aim properties: *X target*, *Y target*, and *Z target*. The *Target* properties update automatically as the camera is moved with the Pan, Trackball, **Zoom**, or Walk commands. The *X target*, *Y target*, and *Z target* values are the coordinates of the point at which the camera is pointing. The *Target* values also describe the point about which the view rotates when the **Trackball** command is used.

It is recommended that the camera position is saved with Set Home before modifying the *Target* properties. The Go to Home command can then be used to return the camera to the default state.

Lighting Properties

To edit a 3D surface map, click once on the surface map to select it. The properties for the surface map are displayed in the Properties window. To edit the **Lighting** in the 3D View, click *Environment* in the Contents window. Changes made to the environment lighting will be automatically applied to the lighting of point vector data in the 3D view. The 3D view window lighting properties are displayed in the **Properties** window.

The **Lighting** page controls the lighting for the entire multi-layer map. This includes the surface, any overlays that may have been combined with the surface, and point vector data in 3D. The light source is fixed relative to the surface, so if the surface is rotated, the light rotates with it.

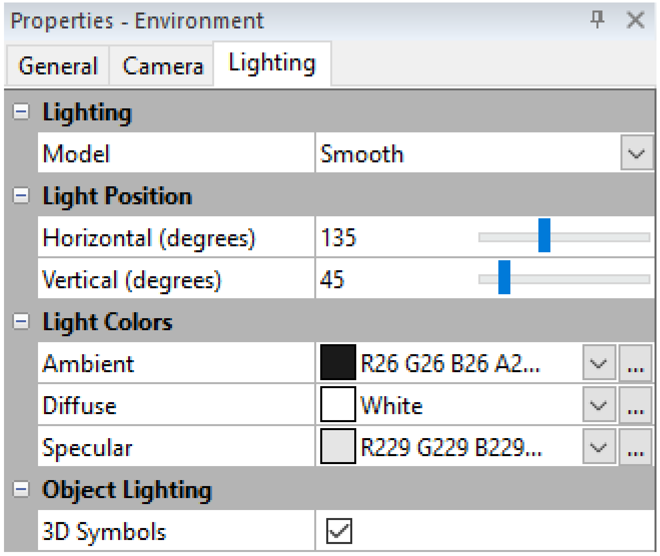
Hill and Reflectance Shading in the 3D View

The 3D view lighting is turned off when the map in the 3D view contains a hill shaded or reflectance shaded color relief layer. This is because the map is already shaded by hill shading or reflectance. The cumulative effect of both lighting systems would result in very dark regions of the map in the 3D view.

To use the 3D view **Lighting** properties instead of the color relief hill or reflectance shading, the *Terrain representation* option on the color relief layer General page must be set to *Color only*.

Lighting Properties

The following properties control the lighting appearance of the 3D surface map in the plot window or the lighting of the map in the 3D view.



*Change lighting properties in the **Properties** window on the **Lighting** page.*

Lighting

There are three lighting options. Click on the current lighting option next to *Model*. In the list, select the desired lighting option.

- *None* disables all lighting effects. The color shown is from the surface material color only.
- *Smooth* splits each grid cell is into two triangular polygons. *Gouraud* shading is used to interpolate colors within the triangles from the three vertices of each triangle. This results in smooth transitions across the triangles and the entire grid, but it is slightly slower than flat shading.
- *Flat* uses a single vertex (grid node) to define the shaded color for the entire polygon. Note that each grid cell is divided into two triangular polygons. This results in a faceted look since each triangle is only filled with a single color.

Light Position

The *Light Position* section specifies the orientation of the light source. The light source can be thought of as the sun shining on a topographic surface.

The *Horizontal (degrees)* box defines the direction for the light source in the horizontal plane. Zero degrees corresponds to the light source shining from due east toward the west. Positive angles rotate the light source counterclockwise. For example, a specified horizontal angle of 90 degrees places the light source north of the unrotated surface. 180 degrees places the light source west of the unrotated surface and shining east. 270 degrees places the light source south of the unrotated surface and shining north. The default horizontal angle is set at 135 degrees, or NW. To change the *Horizontal (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position.

The *Vertical (degrees)* box rotates the light source in the vertical plane. A vertical angle of zero degrees places the light source at the horizon and shining horizontally. An angle of 90 degrees places the light source directly overhead and shining down onto the map. 180 degrees places the light source at the opposite horizon and shining horizontally. 270 degrees places the light source directly below the map and shining up. The default vertical angle is 45 degrees. As the vertical angle approaches zero, shadows lengthen and the overall display shifts to the colors at the left end of the color spectrum. To change the *Vertical (degrees),* highlight the existing angle value. Type a new value or click and drag the  to the desired angle position.

Light Colors

There are three different types of light color, *Ambient*, *Diffuse*, and *Specular*. Note that these colors are used to represent reflectivity. White is 100% reflective and reflects the material color unaltered to the viewer. Black is 0% reflective, and causes all material color to be absorbed. Our perception of color is based on reflected and absorbed light. For example, a leaf appears green because it absorbs all colors in the light spectrum EXCEPT green. Since only green is reflected to your eye, the leaf appears green.

Surfer uses a pure white light source. The light "strikes" the surface and some of the light is absorbed based on the color of the surface material at the point the light ray struck it. Some light is reflected to the viewer according to the type of light (*Ambient*, *Diffuse*, and *Specular*), and the reflectivity color associated with each type of light specified in the *Light Colors* section.

- *Ambient* refers to light that has been scattered so evenly by the environment that its direction is impossible to determine. Increasing the ambient light component brightens the scene without casting shadows. The default *Ambient* color is 90% black which means that the ambient light contribution is fairly small.
- *Diffuse* refers to light coming from a particular direction and is brighter if aimed directly down on a surface than barely glancing off the surface. When diffuse light hits the surface, it is scattered uniformly in all directions so that it appears equally bright no matter where the eye is located. Increasing diffuse light intensifies shadow effects. The default *Diffuse* color is white, which is the maximum amount of reflectivity.
- *Specular* refers to light coming from a particular direction, and tends to bounce off the surface in a preferred direction. A shiny surface such as metal has a high specular component, while a surface like carpet has almost no specular component. Increasing the percentage of specular light results in strong shadow effects and more pronounced "shiny" or glare spots. The default *Specular* color is 90% black.

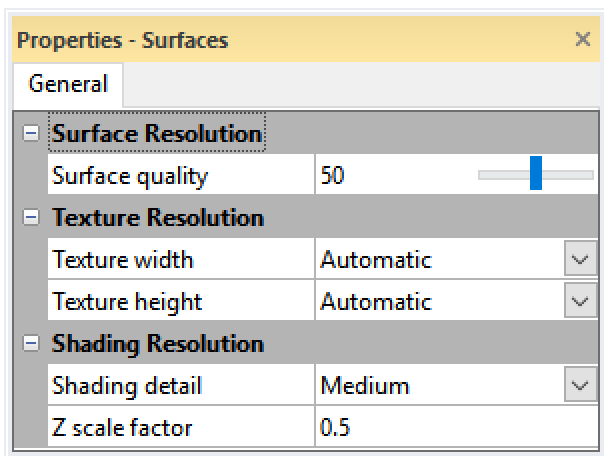
In general, these reflectivity colors should be specified as shades of gray in order to evenly reflect the surface material color components. However, special effects are possible by specifying non-gray colors for the reflectivity. For example, assume the *Ambient* reflectivity is set to pure red, and the *Diffuse* and *Specular* components are set to pure black. The *Diffuse* and *Specular* components are essentially disabled by setting their reflectivity color to black. The only light that is reflected to the viewer is red ambient light. Portions of the surface that lack a red component in the material color will appear black, since only red light is reflected to the viewer.

Object Lighting

The *3D Symbols* property is selected by default to enable the lighting to illuminate and cast shadows on point vector data as it does on the surface displayed in the 3D View window. When this property is cleared, shadows on 3D symbols remain static. The *Symbol method* for 3D vector point data must be set to *Symbols* and the symbol style either *Shaded Circle*, *Sphere* or *Cube*. The lighting *Model* must be either *Smooth* or *Flat*.

Surfaces Group General Properties

The **General** page contains the surface and texture resolution properties for all surfaces in the 3D view. Select the *Surfaces* group in the 3D view **Contents** window to modify the resolution of the surfaces and textures. Increase the surface and texture resolution to improve the appearance of the map. Decrease the surface and texture resolution to increase the drawing speed in the 3D view.



*Set the surface and texture appearance in the **General** page.*

Surface Resolution

The *Surface quality* property in the *Surface Resolution* section controls the quality of the surfaces in the 3D view. As the *Surface quality* is increased, more vertices are included in the surfaces in the 3D view. When *Surface quality* is set to 100, the surface is displayed at the full grid resolution. When the *Surface quality* is 50, half as many vertices in the X and Y directions are used to display the surface, i.e. the number of vertices is one-fourth the number of grid nodes. When *Surface quality* is set to 0, the minimum resolution is used to display the surface.

When the *Surface quality* is less than *100*, overlaid texture contour lines may appear to vary in elevation on the surface. If your computer is capable of rendering the surface at full resolution, a *Surface quality* of *100* is recommended when the surface in the 3D view includes a contour layer texture overlay.

Texture Resolution

The *Texture Resolution* properties control the appearance of the surface overlay textures. The *Texture width* is the number of pixels used to render the texture overlays in the X direction. The *Texture height* is the number of pixels used to render the texture overlays in the Y direction.

The *Texture width* and *Texture height* can be specified independently to account for different surfaces. For surfaces that are generally square, the *Texture width* and *Texture height* should be set to the same value. For surfaces that are rectangular, i.e. one side is significantly longer than the other, the texture appearance may be improved by increasing the resolution along the long side of the surface.

By default the *Texture width* and *Texture height* are set to automatic. When both *Texture width* and *Texture height* are set to *Automatic*, the surface resolution along the longer direction is set to 1024 and the surface resolution along the shorter side is set based on the ratio between the surface width and height. For example, a square surface will have 1024 pixels along the width and height while a long and skinny surface may only use 256 pixels along the shorter direction. When only one of the *Texture width* or *Texture height* values are set to *Automatic*, the *Automatic* value varies based on the specified value and the ratio between the surface width and height.

To change the *Texture width* or *Texture height*, select the desired resolution from the list. Select *Automatic*, *256*, *512*, *1024*, *2048*, *4096*, *8192*, or *16384* pixels.

Shading Resolution

The *Shading Resolution* properties control the amount and quality of the surface shading. The *Shading detail* specifies the amount of surface shading detail. The *Z scale factor* increases or decreases shading.

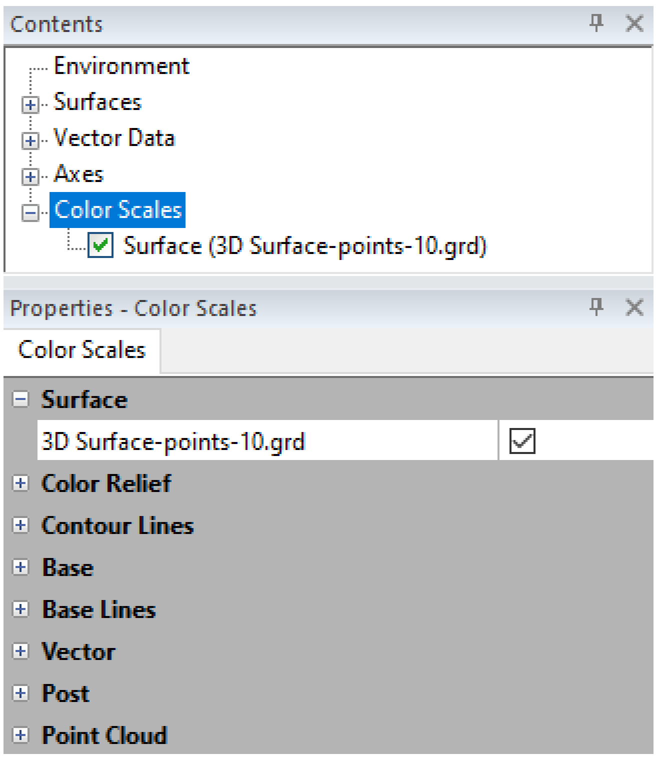
The *Shading detail* has three levels of shading quality:

- *High* increases the level of detail in the surface shading. High looks best for most surfaces, but consider lowering the level if your video card memory is limited.
- *Medium* is similar to the shading by the color relief layer, where some shading information is visible at every grid node.
- *Low* is best for computers with little memory or when displaying low resolution grids in the 3D view.

The *Z scale factor* property sets the Z scale to use for the surface. As you increase the *Z scale factor*, the surface becomes more exaggerated vertically. Increasing the factor enhances the shading effect, and can be useful for bringing out more detail, especially on relatively flat surfaces.

3D View Color Scales Group

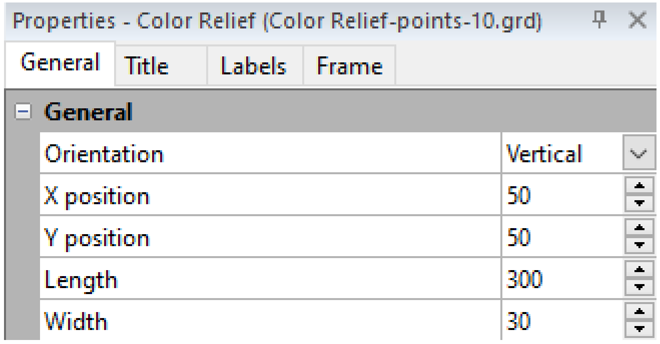
The **Color Scales** page in the **Properties** window contains the group of color scale bars for 3D layers. Colormap(s) created in 2D layers are converted to color scale bars in the 3D View. Select the *Color Scales* group in the 3D view Contents window to show the layers that have color scale bars in the **Properties** window. Click the ⊞ button to open the map type section on the **Color Scales** page to select the color scale to display adjacent to the map.



When the Color Scales group is selected in the ***Contents*** *window, the color scale bars for each layer are displayed in the* ***Properties*** *window.*

3D View Color Scale General Properties

The **General** page contains the properties of the individual color scale in the 3D view. Click once on the individual color scale in the *Color Scale* group in the 3D view **Contents** window to change the position of the color scale on the 3D map.



Change the position of the color scale in the 3D View on the General *page in the* **Properties** *window.*

Orientation

The *Orientation* section controls whether the color scale bar is set to Vertical or Horizontal.

X and Y Position

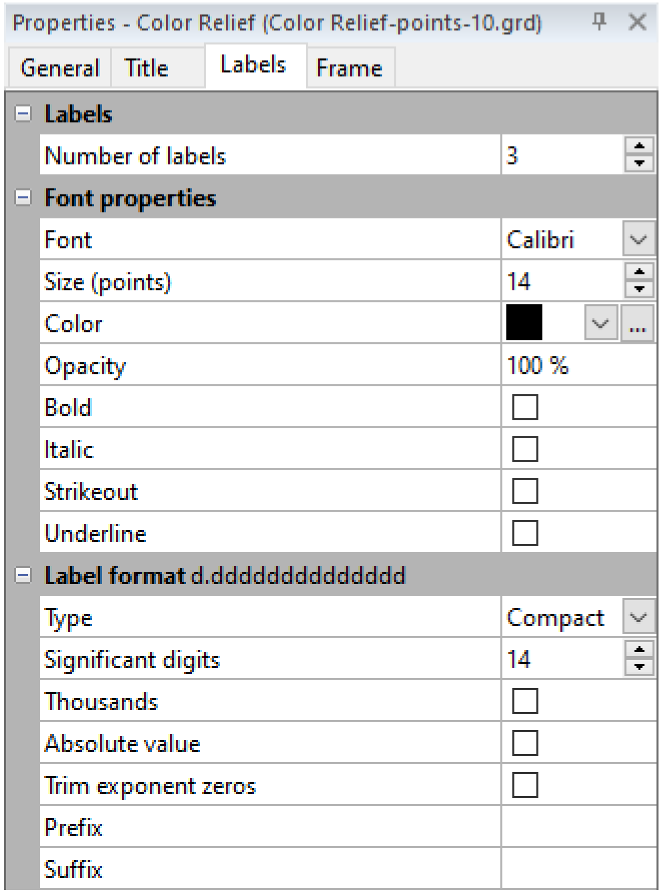
Use the *X position* and *Y position* to set the X, Y position on the page for the color scale. To change the location, highlight the existing value and type the desired value. Or click the buttons to increase or decrease the position.

Length and Width

Use the *Length* and *Width* controls to set the length and width of the color scale. To change the size, highlight the existing value and type the desired value. Or, click the buttons to increase or decrease the size.

3D View Color Scale Labels Properties

The **Labels** page for the 3D view color scale sets the number of labels that are included on the color scale, their appearance and format. Labels are created in the colormap editor of the 2D map. Click once on the map in the *Color Scale* group in the 3D view **Contents** window to select it.



The Labels *page controls the number of labels that are included on the color scale and the labels' appearance.*

Labels

The *Number of labels* section controls the number of labels that are displayed for the color scale.

Font Properties

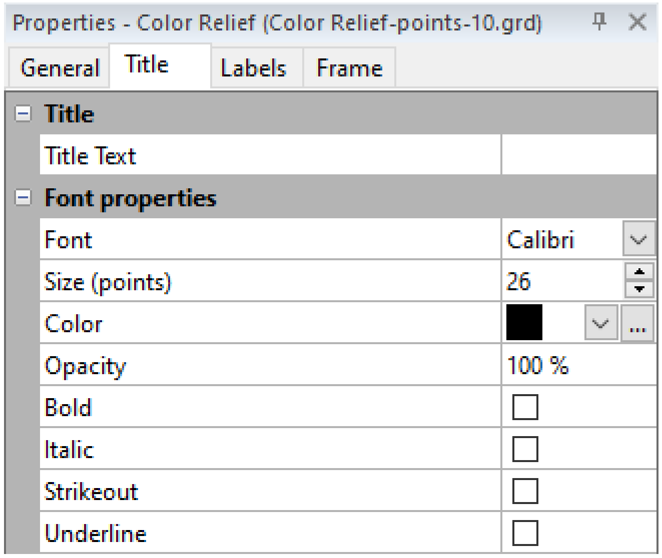
Set the font properties for the color scales labels in the Font properties section.

Label Format

Set the label format properties for the color scales in the Label format section.

3D View Color Scale Title Properties

The **Title** page contains the properties of the title for the 3D view individual color scale. Click once on the individual *Color Scale* in the 3D view **Contents** window to change the properties of the title of the color scale on the 3D map.



Edit the properties of the color scale title on the Title *page of the* **Properties** *window.*

Title Text

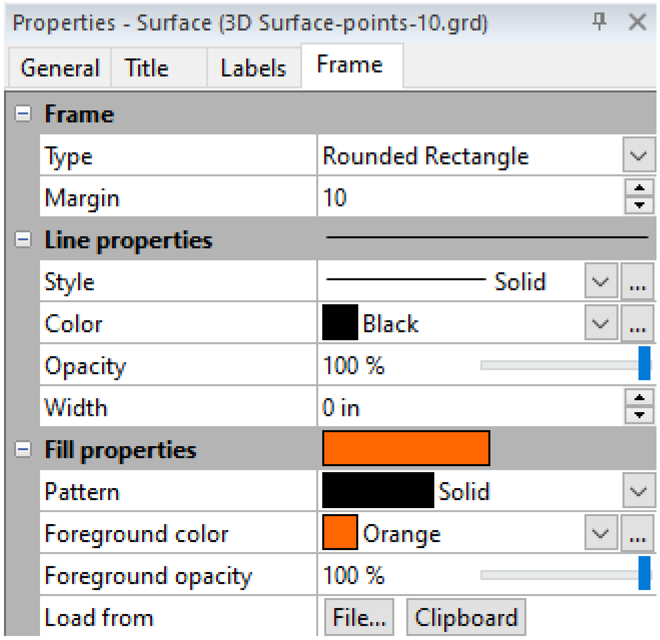
The *Title Text* section adds a title for the color scale. Click in the box next to *Title Text* and type the title exactly as you want the text to appear above the color scale.

Font Properties

Set the font properties for the color scale labels in the Font properties section.

3D View Color Scale Frame Properties

The **Frames** page for the 3D view color scale specifies the properties to use for the border and the foreground area around the color scale.



Edit the properties of the color scale frame on the Frame *page of the* **Properties** *window.*

Frame

The *Frame* section specifies the *Type* and *Margin* for the border around the color scale.

Type

The *Type* list specifies the type of border to use for the color scale. You can select *None*, *Rectangle*, or *Rounded Rectangle* for the frame type. Setting this value to *None* shows no border around the box. Fill properties cannot be applied to the legend frame when the *Type* is set to *None*. Setting the Type to *Rectangle* creates a rectangle at the edge of the color scale at the *Margin* distance. Setting Type to *Rounded Rectangle* creates a rounded rectangle at the edge of the color scale at the *Margin* distance.

Margin

Margin specifies the distance between the edge of the frame and the text or symbols within the legend. To set the *Margin* distance, highlight the existing value and type in the new value. Press ENTER on the keyboard or click anywhere else in the Properties window to make the change. Alternatively, click the buttons to increase or decrease the margin.

Line Properties

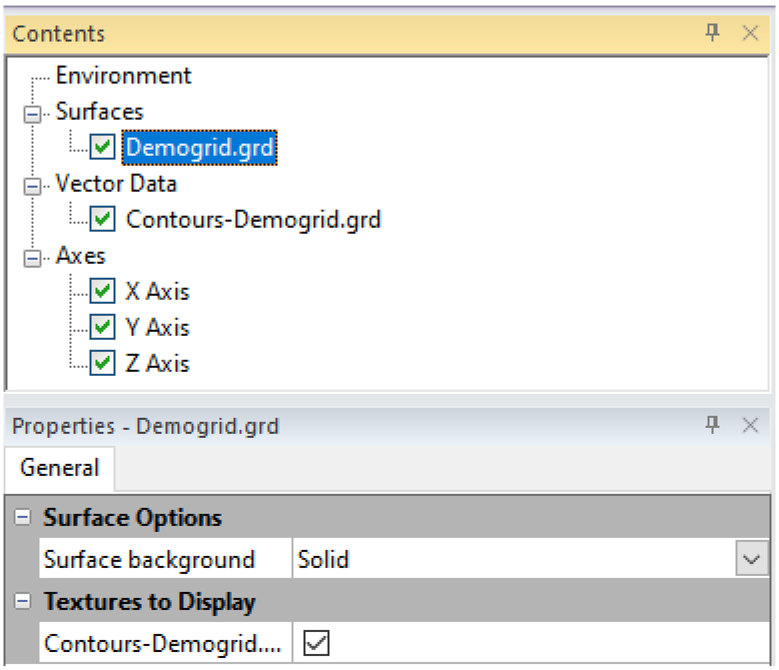
Click the ⊞ next to *Line Properties* to open the Line Properties section. Set the *Style* , *Color, Opacity*, and *Width* of the line to use for the color scale frame.

Fill Properties

Click the ⊞ next to *Fill Properties* to open the Fill Properties section. Set the *Pattern*, *Foreground Color*, *Foreground Opacity* to use for the foreground area of the color scales area. Use *Load from* to load an image from a file or the clipboard to the foreground area of the color scale.

Surface General Properties

The **General** page contains the available surface options and textures for each surface in the 3D view. Select a single grid in the *Surfaces* group in the 3D view **Contents** window to define its surface options and textures.



*Select the surface background and textures to display on the surface in the **General** page.*

Surface Background

For each surface, the *Surface background* property can be set to *Solid* or *None*. Select *Solid* to display a solid, gray scale surface with the textures displayed on top. Select *None* to not display a surface background. If the map contained 3D surface layers, the *Surface background* list also contains the 3D surface layers. Selecting one of the 3D surface layers for the surface background will apply the layer colors to the surface background.

Textures to Display

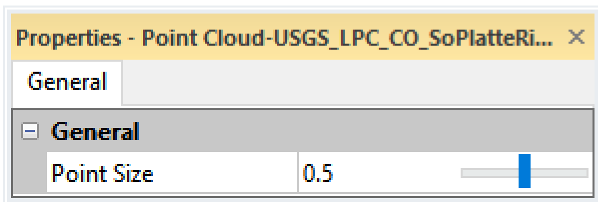
Each surface can show the texture from any map layer that uses the same grid file as the surface and can show the textures from any base or post layers.

The textures from the layers in the map are overlaid on the surface are listed in the *Textures to display* section of the **General** page. Select the check box next to the textures you wish to display on the surface. Clear the check box next to the texture name to remove the texture from the surface.

The associated layer must be visible in the 2D plot window for the texture to be included in the *Textures to display* list.

Point Cloud General Property

The **General** page contains the properties for the point cloud in the 3D view window. Select a point cloud in the *Vector Data* group in the 3D view **Contents** window to modify the point cloud properties. The number of points in the point cloud is limited by the *3D View maximum number of points* in the **Options** dialog Rendering page.



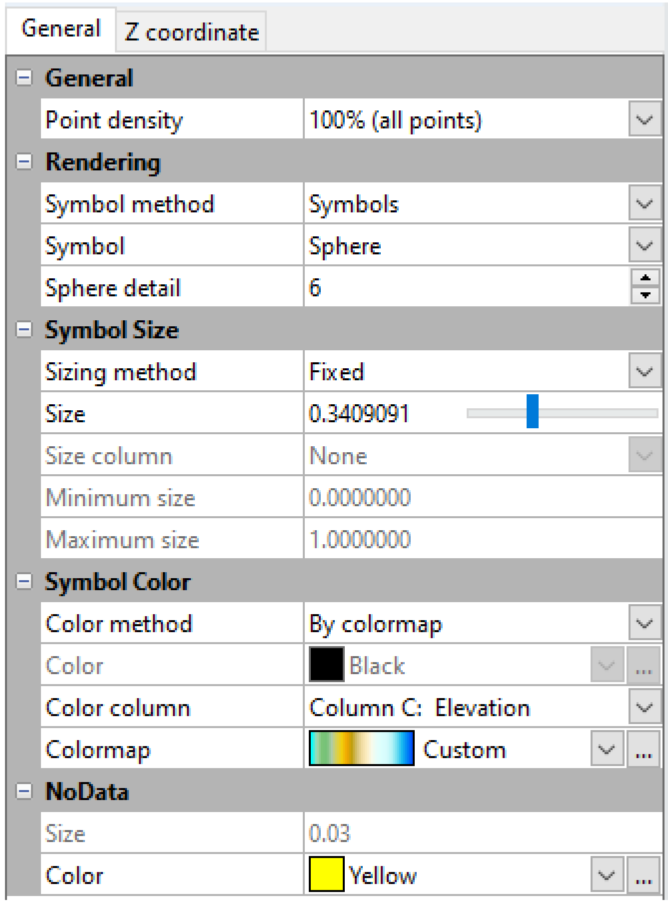
*Set the point cloud point size in the **General** properties page.*

Point Size

The *Point size* property sets the size of the point cloud points. Click and drag the slider  to the left to decrease the *Point size* value, or drag the slider  to the right to increase the *Point size* value. The *Point size* property sets the points to a constant-pixel diameter. However, the number of pixels for the *Point size* values depends on your PC video card.

Vector Data General Properties

The **General** page contains the properties for the point vector data. 3D point vector data is displayed for post and classed post layers and for base (vector) layers that include point features. Select a *Post*, *Classed Post*, or *Base* layer in the *Vector Data* group in the 3D view window to edit the point vector data properties.



Select the appearance of the 3D rendering.

General

The *General* section includes the properties for defining points and symbols.

Point Density

The *Point density* property specifies the number of displayed points. Select a value from the *Density* list to plot between 1% and 100% of points: *100% (all points)*, *50% (every other point)*, *33% (every 3rd point)*, *25% (every 4th point)*, *20% (every 5th point)*, *10% (every 10th point)*, *5% (every 20th point)*, or *1% (every 100th point).*

Rendering

The *Rendering* section includes properties that specify the style of the symbols.

Symbol Method

The *Symbol method* property defines the style of the point vector data symbols. Select *Symbols* in the *Symbol method* list to display the points as circles, squares, spheres, or cubes. Select *Same as plot* to display the points with the same symbol style as found the in map in the plot window.

Symbol

The *Symbol* property specifies the symbol style when the *Symbol method* property is set to *Symbols*. Select *Circle*, *Square*, *Cube*, *Shaded circle*, or *Sphere* from the list. The *Circle*, *Square*, and *Shaded circle* are two dimensional symbols, and the *Cube* and *Sphere* are three dimensional. If you have a large quantity of points, a *Shaded circle* might be a better option than the *Sphere* if you need to reduce rendering time and cannot reduce the point density.

Sphere Detail

If *Sphere* is the selected *Symbol*, use *Sphere detail* to set the level of smoothness of the rendered spheres. Possible values range from 1 to 10, with a default of 6. When Lighting is set to *Smooth*, higher *Sphere detail* values render smoothly rounded spheres. When Lighting is set to *Flat*, higher values render smaller facets than lower values. Lower *Sphere detail* values render more quickly than higher values. Other ways to reduce rendering time is to select a lower *Frequency* value or choose a 2D symbol, such as a *Square*, *Circle*, or *Shaded circle*.

Symbol Size

The *Symbol Size* section includes properties for setting the symbol size based on the plot, to a fixed value, or by proportional scaling.

Sizing Method

The *Sizing method* specifies the point vector data symbol size. Select the desired *Sizing method* from the list:

- *Same as plot* uses the same size symbols as the symbols in the map in the plot window.
- *Fixed size* uses the same size symbols throughout the 3D view. The *Size* property is available when the *Sizing method* is set to *Fixed.*
- *Proportional* uses different sized symbols for the point vector data based on a data value. The *Size column*, *Minimum size*, and *Maximum size* properties are available when the *Sizing method* is set to *Proportional*. The *Proportional* option uses linear scaling between the data minimum and maximum. For more scaling options, use the proportional scaling options in 2D map layers and set the *Sizing method* to *Same as plot*.

Size

The *Size* property is available when the *Sizing method* is set to *Fixed.* The *Size* property sets the size of the fixed symbols. Click and drag the slider to the left to decrease the *Size* value, or drag the slider to the right to increase the *Size* value.

Size Column

Select the data column or attribute for scaling the symbol sizes in the *Size column*. The *Size column* list includes data columns for post and classed post layers. The *Size column* list includes attributes from the base layer with numeric or mixed data. Attributes that include only text data are not included in the *Size column* list.

If a mixed data column is selected, point vector data with text data will not be displayed.

Minimum Size

Specify the size of the symbol that corresponds to the smallest data value in the *Minimum size* property. Set the *Minimum size* to a value between 0.00 and 1.00.

Maximum Size

Specify the size of the symbol that corresponds to the largest data value in the *Maximum size* property. Set the *Maximum size* to a value between 0.00 and 1.00.

Symbol Color

The *Symbol Color* section includes properties for coloring the symbols based on the plot, to a specific fixed color, or by a colormap.

Color Method

The *Color method* property specifies the symbol color for the point vector data:

- *Same as plot* uses the same symbol color(s) as the symbols in the map in the plot window.
- *Fixed* uses the same symbol fill color throughout. The *Color* property is available when the *Color method* is set to *Fixed.*
- *By colormap* assigns colors to the symbols based on a data value and colormap. The *Color column* and *Colormap* properties are available when the *Color method* is set to *By colormap*. If the post, classed post, or base (vector) layer already assigns colors by a colormap, you may wish to set the *Color method* to *Same as plot*.

Color

The *Color* property specifies the symbol color when the *Color method* is set to *Fixed.* Select a fill color for the scatter plot by clicking the current *Color* and selecting the desired color in the color palette. Click the ... button to define a custom color in the Colors dialog.

Color Column

Select the data column or attribute for the colormap in the *Color column* field. The *Color column* list includes data columns for post and classed post layers. The *Color column* list includes attributes from the base layer with numeric or mixed data. Attributes that include only text data are not included in the *Color column* list.

Colormap

The *Colormap* is the color spectrum that is mapped to the *Color column* data values. Select a predefined colormap from the *Colormap* list, or click the ... to modify or create a colormap in the Colormap Editor.

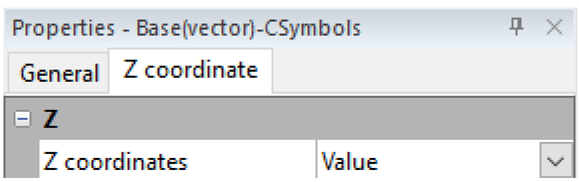
NoData

The *NoData* section includes properties for specifying the size or color of points where the *Size column* and/or *Color column* do not include data. Specify the NoData symbol size in the *Size* field. Specify the NoData symbol color in the *Color* field.

Z Coordinate Page - 3D View Base(vector) Layer

The **Z coordinate** page in the Properties window contains the attributes for the Z values for 3D polylines and 3D polygons. Select the Z coordinates for the base (vector) layer in the *Z coordinates* field. The *Z coordinates* list includes data columns for base(vector) layers. The *Z coordinates* list includes attributes from the base layer with numeric data. Attributes that include only text data are not included in the *Z coordinates* list.

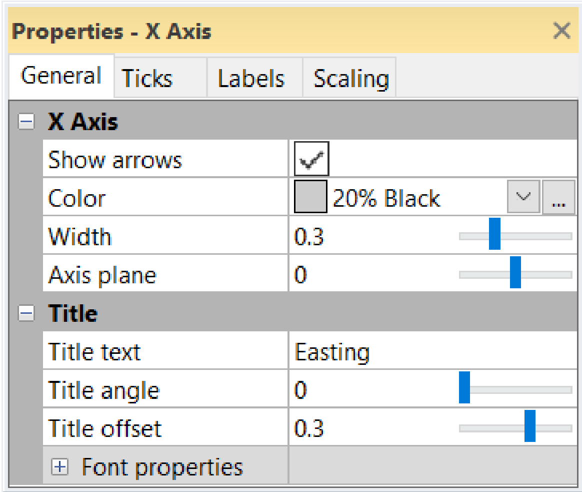
Select *[None]* to remove all 3D polyline and 3D polygon objects from the 3D View Window that have do not have any Z values.



*Select the Z coordinates for the base(vector) layer on the **Z coordinate** page in the **Properties** window.*

3D View Axis General Properties

The **General** page for the 3D view axis objects includes options for axis appearance, plane, and title.



*Set the appearance, plane, and title properties in the **General** page.*

Axis

Set the arrow and axis plane options in the *Axis* section.

Show Arrows

Select *Show arrows* to display arrows at the end of the axis.

Color

Select the axis color in the *Color* field. Select a color from the color palette or click ... to set a custom color in the Colors dialog.

Width

The *Width* property sets the width of the axis. Select a value between 0 and 1 for the axis width.

Axis Plane

The *Axis plane* property rotates the axis visual features, such as ticks, labels, and title, about the about the axis. Specify a value in degrees between -180 and 180.

Title

The *Title* section includes axis title properties.

Title Text

The *Title text* property sets the text that is displayed for the axis title. Type the desired title in the *Title text* field.

Title Angle

The *Title angle* property rotates the title in the axis plane. Set the *Title angle* to a value between 0 and 360 degrees.

Title Offset

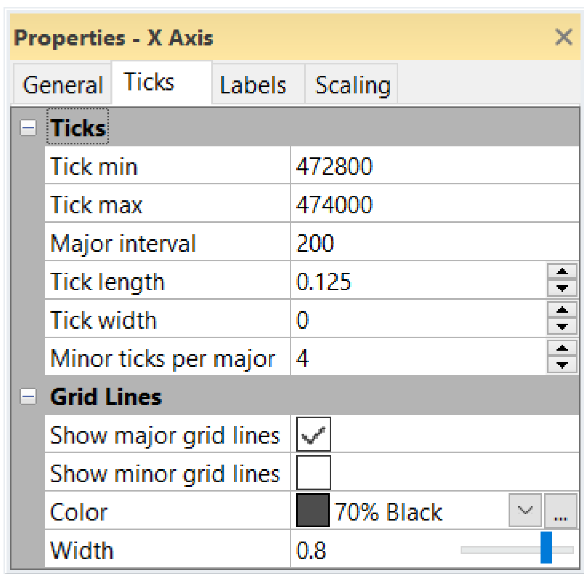
The *Title offset* property sets the distance between the title and axis. Positive values move the title away from the map region and negative values move the title into the map region. Set the *Title offset* to a value between -1 and 1.

Font Properties

Set the font properties for the title in the Font properties section.

3D View Axis Ticks Properties

The **Ticks** page for the 3D view axis objects includes options for axis tick and grid line spacing and appearance. The Ticks page properties can be reset to default values on the Scaling page.



*Set the tick and grid line properties in the **Ticks** page.*

Ticks

Set the tick size and spacing options in the *Ticks* section.

Tick Min and Tick Max

Specify the range over which tick mark and grid lines are displayed in the *Tick min* and *Tick max* property fields. Set the *Tick min* and *Tick max* values to the desired value in map units.

Perpendicular grid lines are drawn to the *Tick max* for an axis. For example, the *Tick max* property for an X axis controls which grid lines are drawn for the X axis and the grid line length for the Y axis and Z axis grid lines.

Major Interval and Minor Ticks Per Major

Specify the tick spacing in the *Major interval* field. Major ticks and major grid lines are drawn at the *Tick min* and then at every *Major interval* between the *Tick min* and *Tick max*. Minor tick marks and minor grid lines are controlled by the *Minor ticks per major* property. Specify the number of minor ticks and minor grid lines between the major ticks in the *Minor ticks per major* field. For example, with a *Major interval* of 50 units, a *Minor ticks per major* value of 4 places minor tick marks at 10, 20, 30, and 40 unit increments between the major ticks.

Tick Length

The *Tick length* property sets the length of the major tick marks. Set the *Tick length* to a value between 0 and 1. Set the *Tick length* to 0 to hide the tick marks. The minor tick mark length automatically adjusts to half the major tick mark length

Tick Width

The *Tick width* property sets the width of the tick marks. Set the *Tick width* to a value between 0 and 1. Both major and minor tick marks are controlled by the *Tick width* property.

Grid Lines

The *Grid lines* section includes the appearance properties for the grid lines.

Show Major Grid Lines and Show Minor Grid Lines

Select the *Show major grid lines* property to display grid lines at the major tick marks. Select the *Show minor grid lines* property to display grid lines at the minor tick marks. Clear the check boxes to remove the grid lines.

Color

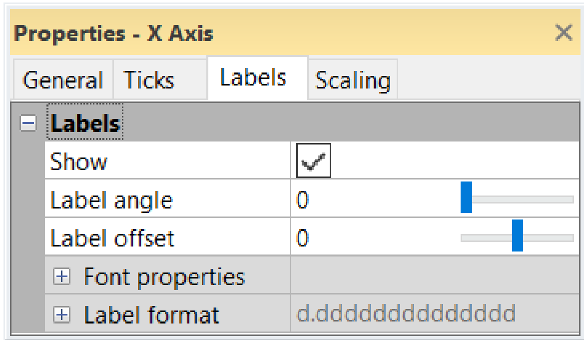
Select the axis color in the *Color* field. Select a color from the color palette or click ... to set a custom color in the Colors dialog.

Width

The Width property sets the grid line width. Set the Width property to a value between 0 and 1.

3D View Axis Labels Properties

The **Labels** page for the 3D view axis objects includes options for axis label appearance and position.



*Set the label properties in the **General** page.*

Show

Select *Show* to display labels at the end of the major tick marks.

Label Angle

The *Label angle* property sets the rotation of the labels in the axis plane. Set the *Label angle* to a value between 0 and 360 degrees.

Label Offset

The *Label offset* property sets the distance between the labels and the major tick marks. Set the *Label offset* to a value between -1 and 1. Positive values move the labels away from the map region. Negative values move the labels into the map region. A value of 0 places the labels at the end of the major tick marks.

Font Properties

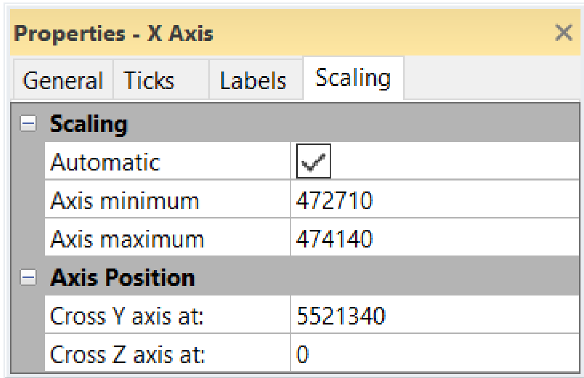
Set the font properties for the axis labels in the Font properties section. Note the label font *Size* property is in relative units rather than points (pt). The label increment is automatically updated as the label font size changes to ensure labels do not overlap, i.e. fewer labels are drawn as the label font size is increased.

Label Format

Set the label format properties for the axis labels in the Label format section.

3D View Axis Scaling Properties

The **Scaling** page for the 3D view axis objects includes the axis limits and position properties.



Set the axis limits and position properties in the ***Scaling*** *page.*

Scaling

The *Scaling* section includes axis limits properties and an option to reset the scaling to default values.

Automatic

Select *Automatic* to set the axis limits, axis position, and the Ticks properties to their default values. The *Automatic* property is automatically cleared after changing any of the related properties.

Axis Minimum and Axis Maximum

The *Axis minimum* and *Axis maximum* properties control the axis length. Set the *Axis minimum* and *Axis maximum* to the desired value in map units.

Axis Position

The *Axis Position* section includes two of the *Cross X axis at:*, *Cross Y axis at:*, and *Cross Z axis at:* properties for specifying the axis position. Which properties are displayed depends on which axis is selected. Specify a value in map units to move the selected axis in the plane with the other axis. For example when an X axis is selected, the *Cross Y axis at:* property moves the X axis in the XY plane and the *Cross Z axis at:* property moves the axis in the XZ plane.

3D Drillhole Properties

After a Drillhole Map is created, view the drillholes, wells, or boreholes in the 3D view by right clicking the drillhole layer and clicking *3D View*, by selecting the map and clicking **Map Tools | View | 3D View**, or by clicking the *3D View* button in the Map frame View Page page properties.

To define 3D drillhole properties, click the drillhole layer in the **Contents** window in the 3D view. The 3D drillhole **Properties** window appears.

This help topic has the following main sections that describe the 3D drillhole properties.

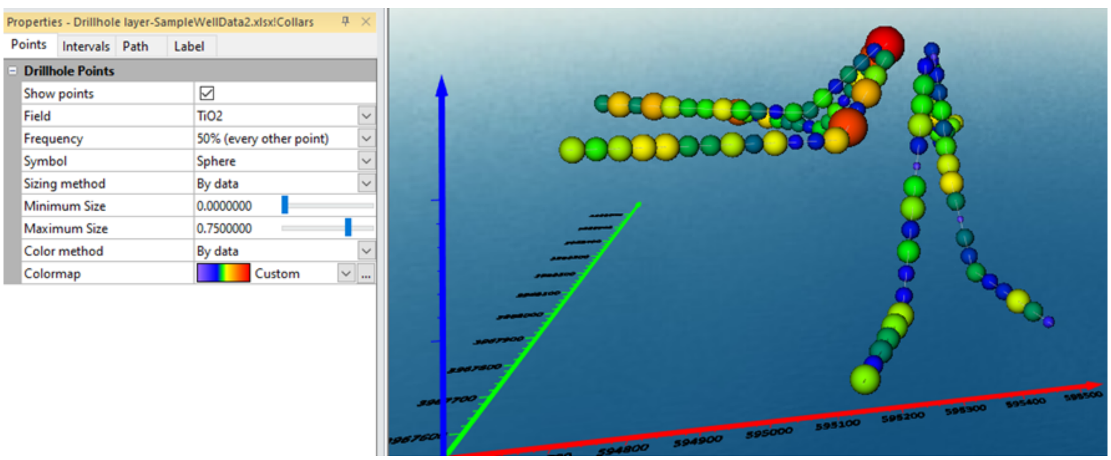
Points Page
Intervals Page
Path Page
Label Page
Drillhole Color Scales

Points Page

If points data were imported in the 2D map, these data can be displayed as point symbols in the 3D model. In the example below, the sample data has values in the TiO2 column in the points data (see C:\Program Files\Golden Software\Surfer\Samples\SampleDrillholeData.xlsx).

Show Points

To display points, click the *Show points* check box in the **Points** page in the **Properties** window.



*Example 3D Drillhole **Points** properties using the TiO2 field in the sample points data*

Field

Select the field to display as the point value from the *Field* list. Only fields imported from the points data in the **Drillhole Manager** can be selected.

Frequency

Set the *Frequency* of point symbols to display. For example, 50% displays half of the points.

Symbol

Select the symbol to use for the points from the *Symbol* list: *Circle*, *Square*, *Cube*, *Shaded circle*, or *Sphere*. The *Circle*, *Square*, and *Shaded circle* are two dimensional symbols, and the *Cube* and *Sphere* are three dimensional. If you have a large quantity of points, a *Shaded circle* might be a better option than the *Sphere* if you need to reduce rendering time and cannot reduce the point density.

Sizing Method

Select either *Fixed* or *By data* for a *Sizing method*.

- *Fixed* creates point symbols that are the same specified size. After selecting *Fixed*, use the *Size* property to define the point size.
- *By data* sizes point symbols using data in the *Field* property. After selecting *By data*, use the *Minimum Size* and *Maximum Size* properties to set the size range.

Color Method

Select either *Fixed* or *By data* for the *Color method*.

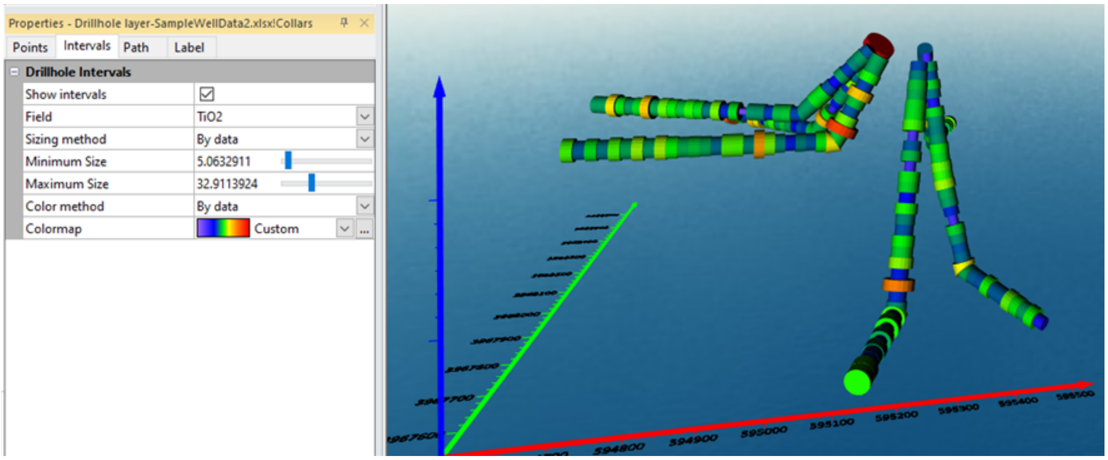
- *Fixed* makes all symbols the same specified color. After selecting *Fixed*, select a color for the point symbols by clicking the current color and selecting the desired color in the color palette, or by clicking the  button to define a custom color in the **Colors** dialog. See the Colors Dialog help topic for more information.
- *By data* uses the data in the *Field* property to apply a color scale to the point data. After selecting *By data*, select a color scale for the symbols by clicking the current color scale and selecting a new one, or by clicking the  button to modify or create a colormap in the Colormap Editor.

Intervals Page

If interval data were imported in the **Drillhole Manager** in the 2D map, these data can be displayed in colored intervals in the 3D model. In the example below, the sample data has values in the TiO2 column in the interval data (see C:\Program Files\Golden Software\Surfer\Samples\SampleDrillholeData.xlsx).

Show Intervals

Check the *Show intervals* check box to display the intervals.



*Example 3D Drillhole **Intervals** properties using the TiO2 field in the sample intervals data*

Field

Select the field to display as the interval value from the *Field* list. Only fields imported from the interval data in the **Drillhole Manager** can be selected.

Sizing Method

Select either *Fixed* or *By data* for a *Sizing method* for the width of the tube. The size (radius of the path cylinder) is 0-100, where the units are proportional to the map's x-axis range.

- *Fixed* makes all intervals the same specified size. After selecting *Fixed*, define a size in the *Size* control.
- *By data* uses the data in the *Field* property. After selecting *By data*, use the *Minimum Size* and *Maximum Size* properties to set the size range.

Color Method

Select either *Fixed* or *By data* for the *Color method*.

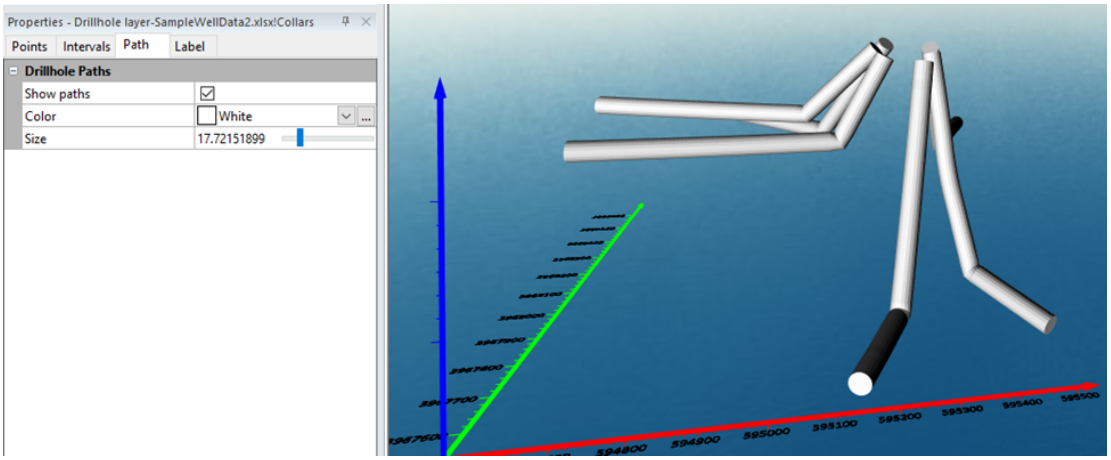
- *Fixed* makes all symbols the same specified color. After selecting *Fixed*, select a color for the symbols by clicking the current color and selecting the desired color in the color palette, or by clicking the ... button to define a custom color in the **Colors** dialog. See the Colors Dialog help topic for more information.
- *By data* uses the data in the *Field* property to apply a color scale to the interval data. After selecting *By data*, select a color scale for the symbols by clicking the current color scale and selecting a new one, or by clicking the ... button to modify or create a colormap in the Colormap Editor.

Path Page

These properties define the path of the drillhole using the depth, azimuth, and inclination values specified in the collars and survey tables.

Show Paths

Check the *Show paths* check box to display the path of the drillholes. Drillhole paths can still be displayed, even if point and interval data are not imported.



*Example 3D Drillhole **Path** properties showing paths*

Color

Select the *Color* of the path. Click the current color and select the desired color in the color palette or click the  button to define a custom color in the **Colors** dialog. See the Colors Dialog help topic for more information.

Size

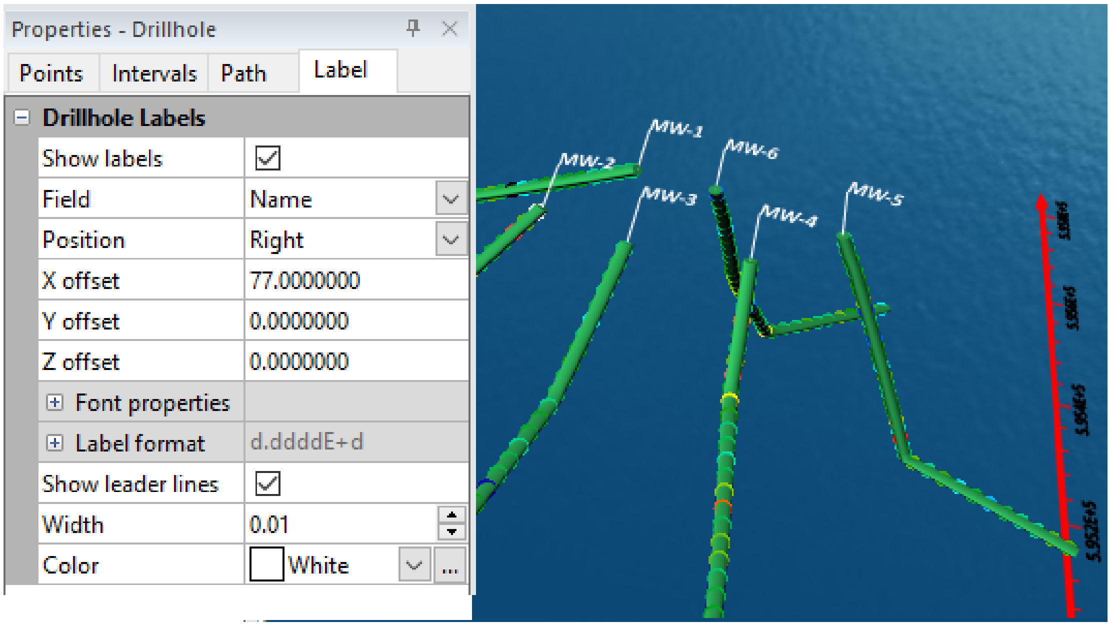
Set the *Size* of the path. The path is the radius of the path cylinder in a 0-100 scaling, where the units are proportional to the map's x-axis range.

Label Page

The **Label** page for the 3D view drillhole layer includes options for drillhole label appearance and position.

Show Labels

Check the *Show labels* check box to display labels using a field from the imported collars table.



*Example 3D Drillhole **Label** properties showing labels*

Field

Select the field to display as the label from the *Field* list. Only fields imported from the collars data in the **Drillhole Manager** can be selected.

Position

In the *Position* list select a *Default*, *Left*, *Center*, or *Right* position of the label relative to the drillhole. Use the *X offset*, *Y offset*, and *Z offset* boxes to enter a specific, user-defined position.

Font

Use the **Font** properties to define the font of the label. See the Text and Font Properties help topic for more information.

Label Format

Use the **Label format** properties to define the format of the labels in the map. See the Label Properties help topic for more information.

Leader Lines

When a value is entered for the *X*, *Y*, or *Z offset*, check the *Leader lines* check box to display leader lines that connect the label to the top of the drillhole. If the *Leader lines* check box is checked, properties appear to define the *Width* and *Color* of the leader lines.

Drillhole Color Scales

Click the **Color Scales** object in the **Contents** window to display the **Properties** for the color scales. Color scales are available for drillhole points and intervals when the drillhole layer uses *By data* for the point or interval *Color method*. When checked, the selected color scale will be displayed. When unchecked, the color scale will not be displayed. See the 3D View Color Scales Group help topic for more information.

3D View Commands

The following sections describe the commands in the 3D view window.

Walk

Click the **3D View | View | Walk** command or the button to enter walk mode. The **Walk** camera position and orientation is saved when you walk mode and restored when you reenter walk mode. While in walk mode, the keyboard and mouse are used to walk along the surface in the 3D view window. The 3D view window must contain at least one surface to enter walk mode.

Walk Mouse Controls

Click and drag in the 3D view window to move and rotate the view.

- Drag the mouse up to move forward, or towards where the view is pointed.
- Drag the mouse down to move backward, or away from where the view is pointed.
- Drag the mouse right to rotate to the right, or clockwise.
- Drag the mouse left to rotate to the left, or counterclockwise.
- Roll the mouse wheel forward to increase the elevation of the camera.
- Click and drag down with the mouse wheel or middle-mouse button to increase the elevation of the camera.
- Roll the mouse wheel backward to decrease the elevation of the camera.
- Click and drag up with the mouse wheel or middle-mouse button to decrease the elevation of the camera.

Walk Keyboard Controls

Press the ARROW or WASD keys to move and rotate the view. Use SHIFT and CTRL to change the camera elevation.

- Press UP ARROW or W to move forward, or towards where the view is pointed.
- Press DOWN ARROW or S to move backward, or away from where the view is pointed.
- Press RIGHT ARROW or D to rotate to the right, or clockwise.

- Press LEFT ARROW or A to rotate to the left, or counterclockwise.
- Press SHIFT to increase the camera elevation.
- Press CTRL to decrease the camera elevation.

Walk with Multiple Surfaces

The **3D View | View | Walk** command may be used with a 3D view that includes more than one surface. When the surfaces are contiguous, you may walk from one surface to another while in walk mode.

It is not recommended to use the **Walk** command in a 3D view with two or more surfaces that occupy the same XY extents but have significantly different Z values. When this is the case, the camera elevation follows the highest surface (largest Z values).

Go to Home

Click the **3D View | Home | Go to Home** command or the button to reset the camera position, rotation, and zoom to the last saved home position.

The home position is set with the 3D View | Home | Set Home command. After setting the view to the desired rotation, zoom, and position, click the **3D View | Home | Set Home** command to save this position.

The home position is only saved while the plot window is open. The 3D view can be opened and closed, and the home position will be remembered. However, the home position is forgotten when the plot window is closed.

Set Home

Click the **3D View | Home | Set Home** command or the button to store the current camera rotation, zoom, and position to the home position. The new home position is stored for each 3D view individually. The home position is only saved while the plot window is open. The 3D view can be opened and closed, and the home position will be remembered. However, the home position is forgotten when the plot window is closed. The next time the plot is opened in the plot window and 3D view window, the **Surfer** default view position is used.

To set the home position,

1. Use the Pan, Trackball, and zoom commands, or the Camera properties, to set the view to the desired rotation, zoom, and position.
2. Click the **3D View | Home | Set Home** command to save this position.

Only one home position may be saved at any time. Click the **Set Home** command to overwrite the previous home position.

The home position is not set by default. The **Set Home** command must be used before the Go to Home command can be used to change the camera position.

Create/Edit Fly-Through

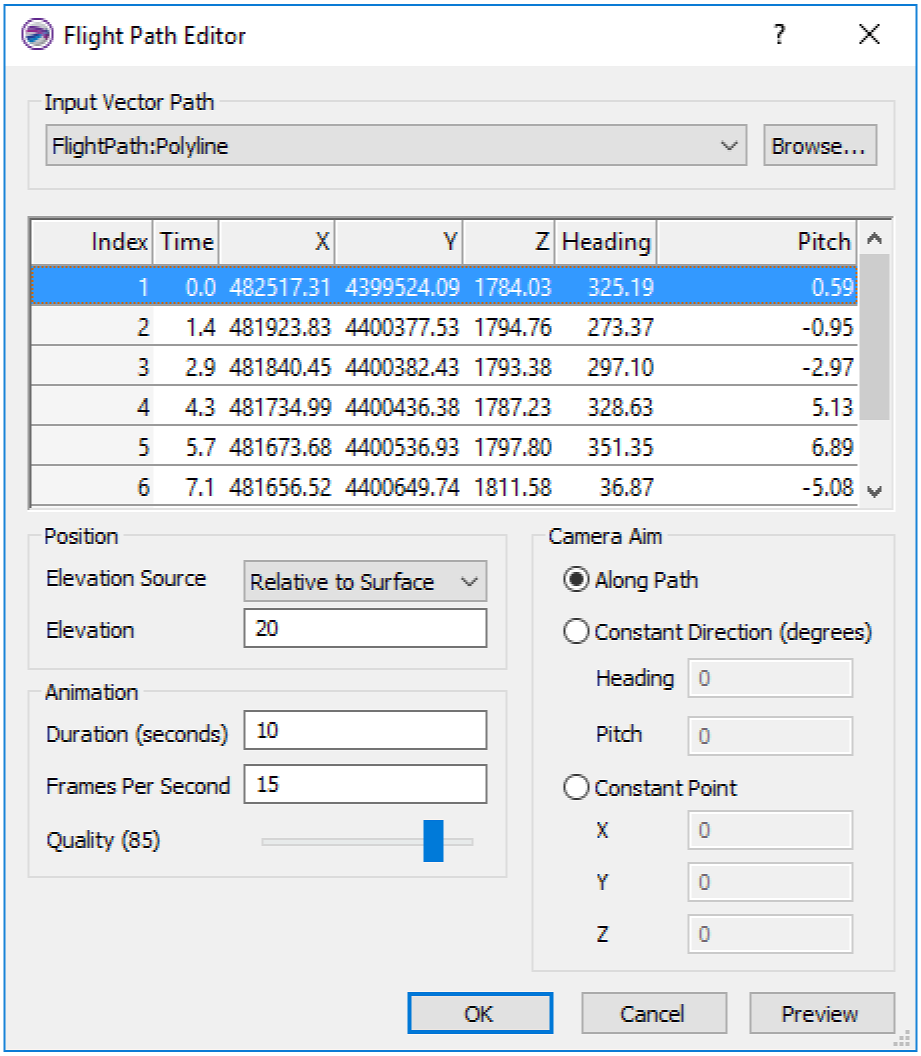
Click the **3D View | Fly-Through | Create/Edit** command or the  button to create or edit a fly-through. A fly-through moves the camera along a path specified by the vertices of a vector object. The **Flight Path Editor** is displayed when the **3D View | Fly-Through | Create/Edit** command is clicked.

Fly-throughs can be played from the 3D view or recorded and saved as an AVI file once a fly-through has been created.

A flight path is defined by key frames. A key frame is a rendering of the 3D view at a specific time with a specific camera position and aim. The camera position and aim are interpolated between the key frames to generate the remaining frames for the fly-through animation. The key frame positions are listed in the *Key Frame* list in the **Flight Path Editor**.

Flight Path Editor

The **Flight Path Editor** defines the path, camera aim, duration, and quality for the flight path.



*Create or edit the flight path in the **Flight Path Editor**.*

Input Vector Path

Define the vector feature for the flight path in the *Input Vector Path* section. Select a vector feature from a layer in the map from the *Input Vector Path* list. The features are listed by layer name and feature name, separated by a colon. For example, in the image above *FlightPath* is the base (vector) layer name and *Polyline* is the feature name. The *Key Frame* list is updated after a vector feature is selected.

Click *Browse* to load a vector flight path from a file. If the file includes more than one feature, only the first feature will be used to define the *Input Vector Path*.

Key Frame List

The *Key Frame* list displays the *Index*, *Time*, camera position, and camera aim for the key frames in the flight path.

- The *Index* value shows the order of the key frames.
- The key frame *Time* is determined by the *Duration* and the distance between key frames.
- The key frame camera X and Y positions are specified by the *Input Vector Path* feature's vertices.
- The camera's Z position is defined by the properties in the *Position* group.
- The key frame camera *Heading* and *Pitch* are defined by the properties in the *Camera Aim* group.

Position

The *Position* group controls the Z position of the camera for the key frames. The *Elevation Source* determines whether the Z position is relative to the surface or an absolute elevation.

Select *Relative to Surface* to specify an elevation for the key frame Z positions above or below the surface in the *Elevation* field. Negative values position the camera below the surface, and positive values position the camera above the surface. The *Key Frame* list is updated when a value is typed in the *Elevation* field.

Select *Absolute Height* to set a single Z position for all key frames in the *Elevation* field. Type the Z position for the key frames in the *Elevation* field.

Camera Aim

The *Camera Aim* group controls the camera aim for the key frames. Three *Camera Aim* options are available: *Along Path*, *Constant Direction*, *Constant Point*. The *Camera Aim* options control the *Heading* and *Pitch* for the key frames.

Heading is the direction the camera is pointing, and it is greater than or equal to 0 and less than 360 degrees. A heading of 0 degrees is pointing parallel to the increasing y-axis direction, i.e. North. Heading increases as the direction rotates clockwise. A heading of 90 degrees is parallel to the increasing x-axis direction, i.e. East.

Pitch is the rotation of the camera about its lateral axis. In other words, it is a measure of how much the camera is pointing up or down. The *Pitch* value must be greater than -90 degrees and less than 90 degrees. A *Pitch* of 0 degrees is looking directly ahead. A positive pitch value has the camera pointed up, and a negative pitch value has the camera pointed down.

Along Path

Select *Along Path* to orient the camera along the flight path at each key frame.

Constant Direction

Select *Constant Direction (degrees)* to set a single *Heading* and *Pitch* for all key frames.

Constant Point

Select *Constant Point* to specify a target X, Y, and Z position for the camera aim. The key frame *Heading* and *Pitch* values will be updated where the camera is aimed at the target position at each key frame. Type the desired target coordinates in the *X*, *Y*, and *Z* fields.

Animation

The *Animation* group controls the fly-through *Duration*, *Frames per second*, and *Quality*.

Duration

The *Duration (seconds)* option defines the length of the fly-through in seconds. The camera moves at a constant velocity along the flight path. The *Duration (seconds)* value and flight path distance defines the time between each key frame. The flight *Duration (seconds)* value must be greater than or equal to 1.

Frames per Second

The *Frames per second* option specifies the number of frames generated per second in the fly-through. A large *Frames per second* value generates a smoother fly-through, but it also requires more processing time and generates a larger AVI file. With small *Frames per second* values, the fly-through will appear choppy, but it will also process faster and generate a smaller file. The *Frames per second* value must be greater than or equal to 2.

Preview and Save Changes

Click *Preview* to view a preview of the fly-through with the current **Flight Path Editor** settings. Once the preview is complete, the **Flight Path Editor** is displayed again. Click *OK* to save the changes to the fly-through. Click *Cancel* to close the **Flight Path Editor** without saving any changes.

The Play and Record commands are enabled after the fly-through is created.

Play Fly-Through

Click **3D View | Fly-Through | Play** or the  button to play the current fly-through. A fly-through must be created by clicking the 3D View | Fly-Through | Create/Edit command before the **Play** command is available.

Once a fly-through has been created, click **3D View | Fly-Through | Play** to view the fly-through, click **3D View | Fly-Through | Record** to record the fly-through, or click **3D View | Fly-Through | Create/Edit** to modify the fly-through.

Record Fly-Through

Click **3D View | Fly-Through | Record** or the  button to play and record the current fly-through. A fly-through must be created by clicking the 3D View | Fly-Through | Create/Edit command before the **Record** command is available.

Once a fly-through has been created, click **3D View | Fly-Through | Record** to view the fly-through. Specify a file name and path in the **Save As** dialog and click *Save* to record the fly-through. The **Record** command creates an AVI video file of the fly-through.

Click **3D View | Fly-Through | Play** to play the fly-through or click **3D View | Fly-Through | Create/Edit** to modify the fly-through.

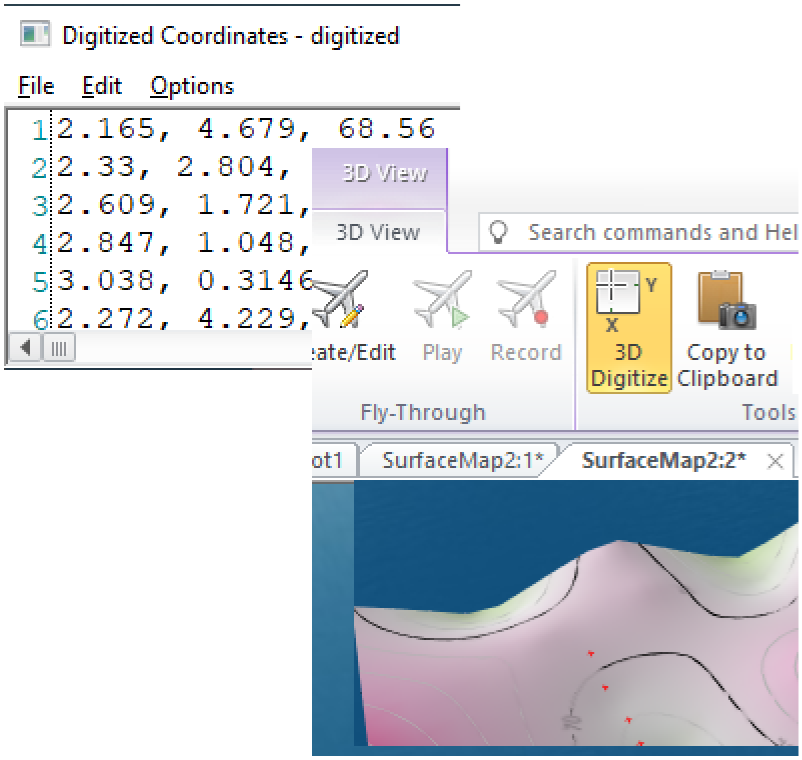
3D Digitize

The **3D View | Tools | 3D Digitize** command allows you to write coordinates from a 3D view to the **Digitized Coordinates** dialog. Digitizing 3D coordinate data might be useful for identifying spot locations and writing their XYZ coordinates to a file. For example, these could be surface or subsurface points, such as 3D drillhole or borehole paths that you could later use to create a contour map to add to the drillhole 3D view.

Note: Use the **Map Tools | Layer Tools | Digitize** command to write 2D map coordinates from the 2D plot window to a data file. That digitizing tool is a good tool to digitize boundaries from maps and easily create a blanking file from the digitized information.

Digitized Coordinates Window

From a 3D view, click the **3D View | Tools | 3D Digitize** command to enter into digitize mode. Left-click on a specific point over an object (e.g. surface, drillhole, or 3D geometry) in the 3D view, and the **Digitized Coordinates** window is displayed.



*Digitized X, Y, and Z values are written to the **Digitized Coordinates** window.*

Each time the map is clicked, a small, temporary red symbol is drawn at the clicked location, and the XY and Z coordinates at the red symbol's location are written to the **Digitized Coordinates** window.

The **Digitized Coordinates** window stays open until it is closed by clicking the button in the top right corner of the **Digitized Coordinates** window. This allows convenient editing of the data and allows digitizing to start and stop multiple times and be recorded in the same window.

Note: For higher accuracy when digitizing 3D surfaces, increase the *Surface quality* property in the **Surface Resolution** section of the **Surfaces General** properties.

Menu Commands

The **Digitized Coordinates** window has three menus: **File**, **Edit**, and **Options**.

File

The **File** menu allows you to save or open a .DAT data file or a Golden Software Blanking .BLN file.

Edit

The **Edit** menu command allows for common editing, such as undo, redo, cut, copy, paste, delete, find, replace, and select all. These commands work in much the same way as the commands in the Home and Data tabs for a worksheet.

Options

The **Options** menu allows the user to set the numeric format with the **Options | Coordinate Format** command. This opens the Label Format dialog, where the numeric format can be set. The period is always used as the decimal separator. No prefix, suffix, or thousands comma can be displayed in the **Digitized Coordinates** window.

The **Options** menu also allows the user to specify whether the Z value is included when digitizing. Toggle the **Options | Include Z coordinate** command to include or not include Z values.

When digitizing to create a blanking file, the **Options** menu also determines if the digitized blanking file assigns the NoData value inside or outside the defined points. By default, the **Options | NoData Inside Region** option is checked. When saving the digitized coordinates, a 1 is written to the BLN file blanking flag, and the area inside polygons is assigned the NoData value. Click the **NoData Inside Region** option to change the setting. When **NoData Inside Region** is unchecked, a 0 is written to the BLN file blanking flag, and the region outside polygons is assigned the NoData value. By default, this setting is remembered during each **Surfer** session.

Digitizing Information from the 3D View

To digitize information from a 3D view, follow these steps:

1. Create a 3D view and click the **3D View | Tools | 3D Digitize** command. The cursor becomes a cross hair to indicate digitize mode.
2. Click the left mouse button in the 3D view to write the current coordinates to the **Digitized Coordinates** window. Continue adding points in this manner. Digitized points appear as temporary small red 3D plus-signs on the 3D view.
3. When you are finished digitizing points, click on any tool button or command or press the ESC key on the keyboard to exit digitize mode.
4. To save the data in the **Digitized Coordinates** window, click the **File | Save As** command in the window. You are prompted to save the data as *Blanking Files (*.bln)* or *Data Files (*.dat)*. Type a *File name* and click *Save* to save the data. Click *Cancel* to not save the data and return to the **Digitized Coordinates** window.
5. Click the  button in the top right corner of the window to close the **Digitized Coordinates** dialog. The dialog will stay open until closed in this manner.

Copy to Clipboard

Click the **3D View | Tools | Copy to Clipboard** command or the  button to copy an image of the current view in the 3D view window to the clipboard. The size of the copied image is the same as it appears in the current 3D view window. Paste the image in a **Surfer** plot window with the Paste command. The image can also be pasted in another application with that application's **Paste** command or usually by pressing CTRL+V.

Changing the Image Size

Control the size of the copied image by changing the size of the **Surfer** application window or the 3D view window.

The following steps describe one method to reduce the size of the image:

1. Click the View | Windows | Cascade command.
2. Resize the 3D view window to the desired size.
3. Click the **3D View | Tools | Copy to Clipboard** command.
4. Paste the image in the desired **Surfer** plot window or other application.
5. Maximize the 3D view window to return Surfer to the tabbed document view.

The following steps describe one method to maximize the size of the image:

1. Maximize the **Surfer** application window.
2. Hide the Contents and Properties windows.
3. Minimize the ribbon.
4. Click the **3D View | Tools | Copy to Clipboard** command.
5. Paste the image in the desired **Surfer** plot window or other application.
6. Return **Surfer** to the desired layout by showing the **Contents** and **Properties** windows, changing the application window size, and restoring the ribbon. You can also clear all application customizations if desired by clicking the View | Windows | Reset Windows command.

3D View Export Image

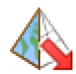
Click the **3D View | Tools | Export Image** command or the  button to export the current view in the 3D view window to an image. The Export Image dialog is displayed. Only image file formats are available in the **Export Image** dialog *Save as type* list:

- BMP Windows Bitmap
- EPS Encapsulated Postscript
- GIF Image
- JPG JPEG Compressed Bitmap
- JPEG-2000 Bitmap
- PNG Portable Network Graphics

- PNM Image
- RGB SGI-RGB Image
- SUN Sun Raster Image
- TGA Targa Truevision
- TIF Tagged Image
- X AVS X-Image

The default size of the image is set to the current size of the 3D view window, but the exported image size can be changed in the **Export Options** dialog Size and Color page. The PNG and TIF export file formats support background transparency, i.e. the image will be exported without a background or skybox. As always, this option requires a 32-bit RGBA color depth and must be selected in the **Export Options** dialog.

3D View Export 3D

Click the **3D View | Tools | Export 3D** command or the  button to export the current view in the 3D view window to a 3D image. The **Export 3D** dialog is displayed. 3D PDF and WRL VRML 3D Vector Model file formats are available in the **Export 3D** dialog *Save as type* list. Specify export options in the Vector PDF Options or VRML Export pages of the **Export Options** dialog. The grid surface quality may be specified in the Quality page of the **Export Options** dialog. The grid surface quality directly affects the image quality and file size of the export.

Chapter 25 - Map Tools

Map Tools Tab Commands

The **Map Tools** tab in the plot document has the following commands:

Layer	Adds a Base, Base from Server, Base from Data, Empty Base, Contour, Post, Classed Post, 3D Surface, 3D Wireframe, Color Relief, Grid Values, Watershed, Viewshed, 1-Grid Vector, or 2-Grid Vector layer to the selected map
Scale	Adds a scale bar to the selected map
Legend	Adds a legend to the plot
Graticule	Adds a graticule or grid lines to the selected map
Profile	Adds a profile to the selected map
Axis	Adds a top or bottom (X), left or right (Y), or Z axis to the selected map
Trackball	Rotates and zooms the selected map in 3D
3D View	View the selected map in a new 3D view window
Set Limits	Set the map limits by drawing a rectangle
Measure	Measure distance and area on the selected map
Overlay Maps	Combine selected maps to use the same map properties
Stack Maps	Aligns two or more selected maps
Convert Layer	Changes one grid-based layer type to another
Digitize	Displays the coordinates of the selected map in map units and collects data points from the map
Break Apart	Removes a map layer from the selected multi-layer map
Georeference Image	Georeference or warp an image in a base layer
Convert Coordinates	Convert vector layer coordinates from an unknown system to another unknown or known system
Open Attribute Table	Opens the Attribute Table for the objects in the selected layer

Open Drillhole Manager	Open the **Drillhole Manager** to manage or view drillhole data
Track Cursor	Track cursor location across plot, worksheet, and grid editor windows for a maps, data files, and grids
Query Objects	Selects objects returned from a specified query
Join Attributes	Import a data file to add attributes to the existing attribute table
Export Contours	Export contours as lines in a DXF, SHP, or TXT file
Grid	Opens the selected layer's grid in the **Grid Editor**
Data	Opens the data for a selected layer in a worksheet window
Contour Labels	Edits individual contour labels of the selected map
Layer Labels	Edits individual post map labels of the selected map

Scale Bar

The **Map Tools | Add to Map | Scale** command or the  button places a scale bar on the map. Alternatively, right-click on an existing map and select **Add to Map | Scale Bar** to add a scale bar. Scale bars are divided into equally spaced cycles. Any number of cycles between one and 100 can be used. Each cycle is labeled based on map units.

To place a scale bar on a map:

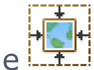
1. Select the map and click the **Map Tools | Add to Map | Scale** command.
2. The scale bar is automatically placed below the bottom axis.
3. To make changes to the scale bar, click once on it to select it. The properties are displayed in the Properties window.

Scale Bar Properties

The scale bar properties are displayed in the following pages:

General
Labels
Info

Set Limits

Click the **Map Tools | Map Tools | Set Limits** command or the  button to interactively set the map limits. When **Set Limits** is enabled, the map limits can be adjusted by clicking and dragging the sides or corners of the map limits rectangle or by drawing a new rectangle. A map, layer, or object within the map or layer must be selected before clicking the **Set Limits** command.

The plot window appearance changes when the **Set Limits** command is clicked. The areas outside the map limits are shaded. Regions of the plot window within the map limits are displayed normally. The map limits are represented by a blue rectangle with yellow selection handles. Additionally, the **Map Tools | Map Tools | Set Limits** command button is highlighted when **Set Limits** is active.

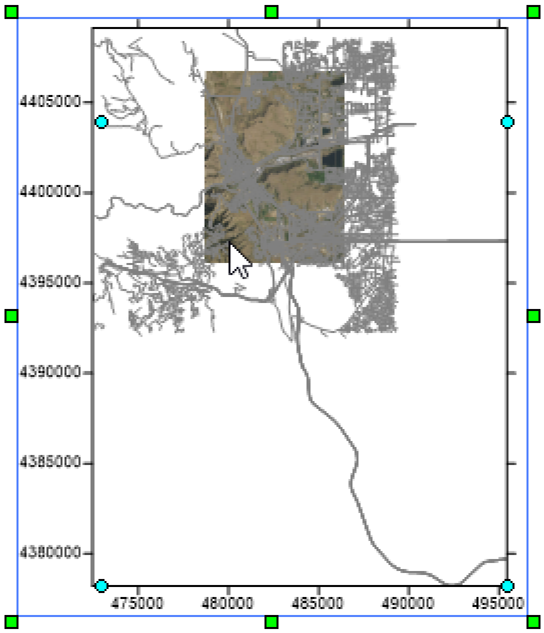
Set Limits Tips

The following keyboard and mouse actions control the **Set Limits** command mode.

- Click and drag the yellow selection handles to move the map limits.
- Press and hold CTRL, then click and drag to draw a rectangle to specify the map limits. The cursor will change to a crosshair to indicate draw mode.
- Double-click in the plot window or press ENTER to update the map with new map limits and exit **Set Limits** mode. Alternatively, right-click in the plot window and click **Save** to update the map and exit **Set Limits** mode.
- Press ESC to exit **Set Limits** mode without making any changes to the map. Alternatively, right-click in the plot window and click **Cancel** to exit **Set Limits** mode without making changes.
- Use the mouse wheel or scroll bars to zoom and pan the plot window.
- The Undo command will undo changes to the map limits after you press ENTER or double-click the plot window. However, using the **Undo** command while **Set Limits** is activated will undo actions made prior to activating **Set Limits** mode. If you wish to make changes to the map limits while still in **Set Limits** mode, click and drag the selection handles or press CTRL and draw a new rectangle.
- The **Set Limits** command cannot be used with 3D surfaces or wireframes.
- The **Set Limits** command cannot be used with rotated, tilted, or sheared maps. Most maps have no rotation, tilt, or shear by default. If the **Set Limits** command is unavailable, set the *Rotation* to 0 and the *Tilt* to 90 on the Map frame View page and set the *Shear* to 0 with the Home | Selection | Transform command

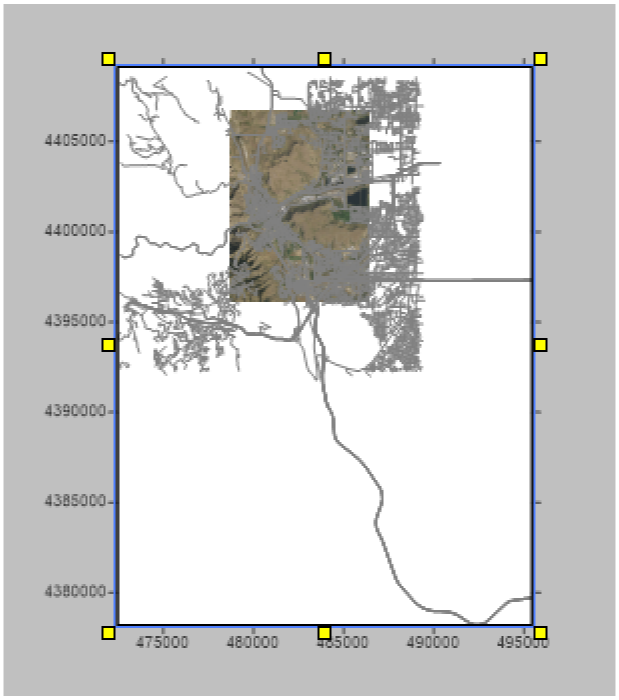
Example

In the following example, the map limits will be set by clicking and dragging the selection handles and by drawing a rectangle. The example map includes a color relief layer, a base (raster) layer, and a base (vector) layer. The vector base layer extents are significantly larger than the extents for the other layers. We will set the map limits to a region within the image base layer extents.



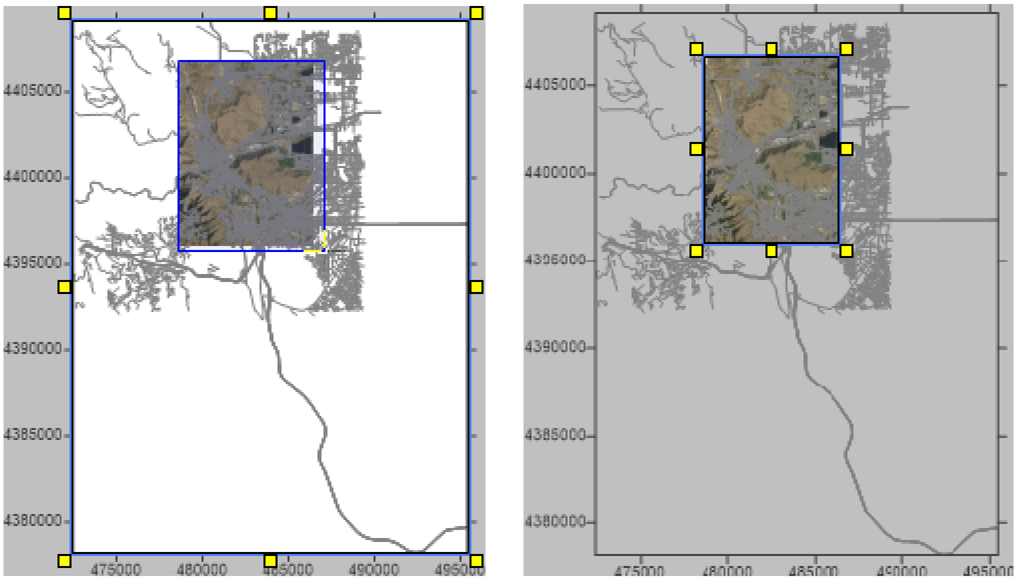
The map limits cover the entire data extents.

1. Click the map in the plot window to select it.
2. Click the **Map Tools | Map Tools | Set Limits** command. The **Set Limits** mode is activated.



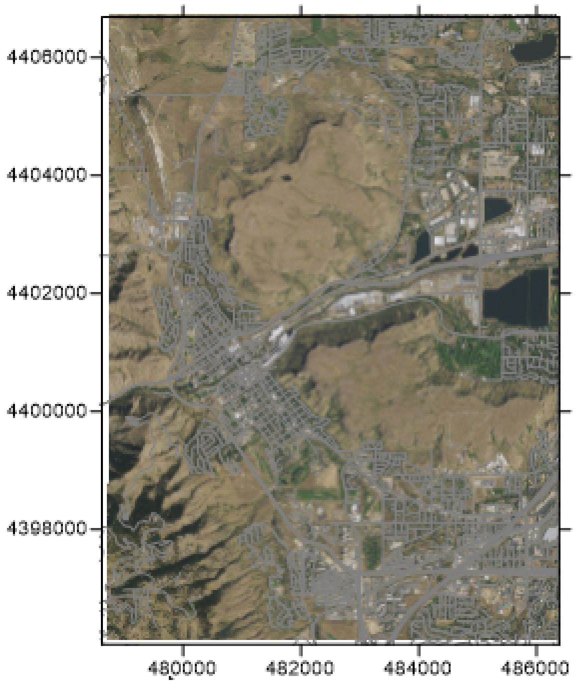
The map limits are represented by the blue rectangle.

3. Press and hold the CTRL key.
4. Click and drag in the plot window to draw the new map limits.



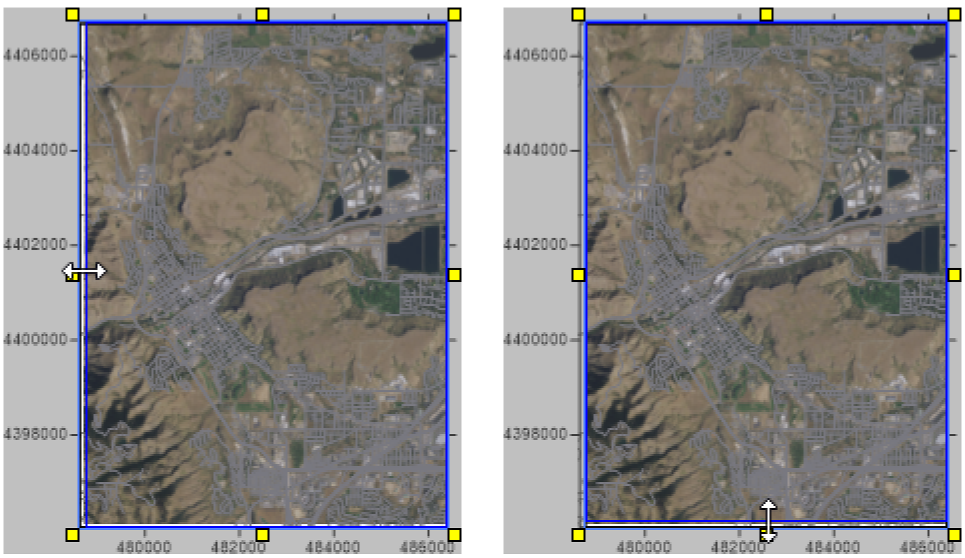
The new rectangle was drawn around the image base layer.

5. Double-click in the plot window, or press ENTER, to save the new limits.
6. When making large changes to the map limits, as in this example, it may be necessary to change the map scale on the **Properties** window Scale page.



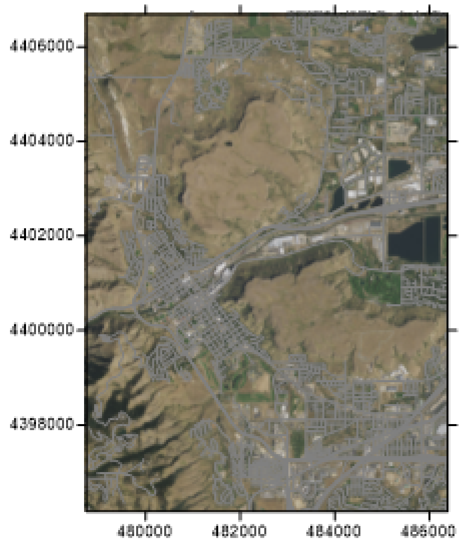
The map limits and scale have been updated.

7. The map limits still need to be reduced on the left and bottom. Click the **Map Tools | Map Tools | Set Limits** command.
8. Click and drag the left and bottom selection handles to the desired location.



Limits can also be set by clicking and dragging the selection handles.

9. Double-click the plot window to save the new map limits.



The map limits are now in the desired location.

Now the map limits have been updated.

Trackball

The **Map Tools | View | Trackball** command rotates, tilts, and changes the field of view in the plot window. The **3D View | View | Trackball** command also rotates and tilts the view in the 3D view.

Trackball in the Plot Window

Click the **Map Tools | View | Trackball** command or the button or right-click on the map and click **Trackball** to *rotate*, *tilt*, and change the *field of view* of a selected map in the plot window. The cursor will change to a , indicating you are in trackball mode. The **Map Tools | View | Trackball** command is similar to using the properties in the map View tab in the Properties window.

Tilt and Rotation

In trackball mode, left-click in the map's bounding box. While holding down the left mouse button, drag the mouse up and down to change tilt, and drag the mouse left or right to change rotation. The rotation and tilt values are shown in the status bar when the left mouse button is held down.

Field of View

In trackball mode, when you right-click in the map's bounding box, dragging up increases the field of view and changes the *Projection* to *Perspective*. Dragging down while right-clicking decreases the field of view, and the *Projection* changes to *Orthographic* if the field of view is decreased to 1°. The field of view value is shown in the status bar when the right mouse button is held down.

Exit Trackball Mode

Press the ESC key on your keyboard to exit trackball mode. Alternatively, click the **Map Tools | View | Trackball** command again or select another command mode, such as select, draw, or reshape.

To modify the map orientation with **Trackball**:

1. Select one map.
2. Click the **Map Tools | View | Trackball** command, or right-click on the map and click **Trackball**.
3. Left-click in map's bounding box. While holding down the left mouse button, drag the mouse up and down to change tilt, and drag the mouse left or right to change rotation. The rotation and tilt values are shown in status bar.
4. Right-click in the map's bounding box. While holding down the right mouse button, drag the mouse up to increase the field of view and changes the *Projection* to *Perspective*. Drag the mouse down while right-clicking to decrease the field of view, and change the *Projection* to *Orthographic* if the field of view is decreased to 1°. The field of view value is shown in status bar.
5. Press the ESC key on the keyboard to end trackball mode.

Trackball in the 3D View

Click the **3D View | View | Trackball** command to rotate and tilt the field of view of the 3D view window. The cursor will change to a  , indicating you are in trackball mode. The 3D view can also be changed by adjusting the values in the *Environment* properties Camera page.

Trackball mode is the default mode for the 3D view window. Press ESC while in another tool mode to return to the trackball mode. While in trackball mode in the 3D view, pan the view by clicking and dragging with the middle mouse button.

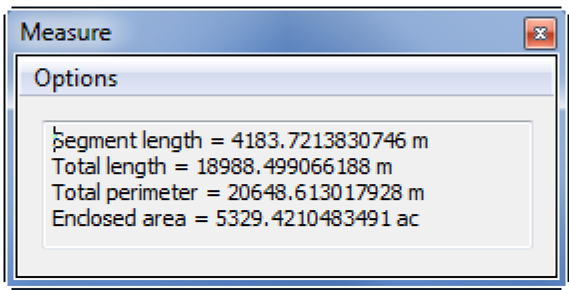
Tilt and Rotation

In trackball mode, left-click in the 3D view window. While holding down the left mouse button, drag the mouse to change the tilt and rotation. How the view is tilted and rotated is determined by the location you first click and the direction you drag. For example, clicking near the left side of the 3D view window and dragging the mouse up tilts the left side of the map up and right side down. While clicking near the center of the view and dragging the mouse up tilts the front of the map up and the back of the map down.

Measure

Click the **Map Tools | Map Tools | Measure** command or the button or right-click and select **Measure** to calculate the distance and area of a highlighted section of a map. After clicking the command, click on the screen around the edge of the line or area you wish to measure. The *Segment length, Total length, Total perimeter,* and *Enclosed area* are displayed in the **Measure** dialog in map units. Double-click or press ENTER on the keyboard to finish the current measurement. The dialog will stay open until you click the button in the top right of the dialog or click on the dialog and press ESC on the keyboard.

Units are in map units. Use the **Options** command to change the units. The length and area are calculated only using the X and Y coordinates (planar length and area). The Z values are not used to calculate the length or area of the measured polyline or polygon. When using latitude and longitude, the lengths and areas are calculated using a Cartesian coordinate system, not using a great circle or the curvature of the earth.



*The area and length are displayed in the **Measure** dialog.*

Drawing Points

Click the left mouse button to add points to the measuring object. If the CTRL key is pressed while clicking points, the points are constrained to 45-degree angles. Double-click the left mouse button or press ENTER on the keyboard to end adding points to the measuring object. Press ESC to exit measure mode. Points can be added inside or outside of the selected map layer and the map frame.

Removing Points

Right-click to remove the last drawn point. This can be done repeatedly.

Panning and Zooming

Click and drag the scroll bars to move the view up or down while in measure mode. Alternatively, hold down the wheel button straight down, and the cursor will turn to a closed hand. When the cursor is a , drag the mouse to pan the plot window. Be sure to click straight down with the scroll wheel.

You can use a wheel mouse to zoom in and out while in measure mode. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out.

3D Maps, Tilt, and Perspective

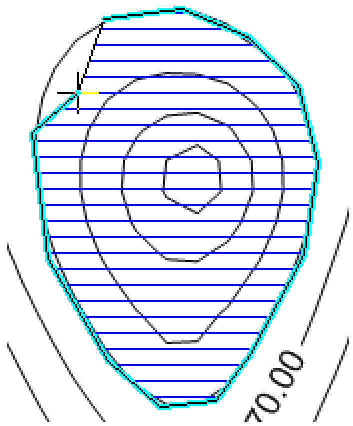
A 2D non-tilted, non-perspective must be selected. If a 3D surface map layer or 3D wireframe map layer is included, the **Measure** command will be unavailable. Select the 3D surface map layer or 3D wireframe map layer and press DELETE on the keyboard or click the Break Apart command.

When all 3D maps have been removed, click on the *Map* in the **Contents** window to select it. Click on the View tab in the **Properties** window. Set the *Projection* to *Orthographic* and the *Tilt (degrees)* to 90.

Measurements

The **Measure** dialog displays four measurements derived from the drawn polygon or polyline:

- The *Segment length* is the length of the last segment drawn. To indicate a new segment, click the left mouse button. The *Segment length* starts again at zero and increases in any direction as the mouse is moved.
- The *Total length* is the length of all of the individual line segments. Each *Segment length* is a positive value and added together. This is the sum length of all cyan colored lines.
- The *Total perimeter* is similar to the *Total length*, except it includes the last unfinished segment that would return the digitized area to the original point. In the diagram below, this is the light black line.
- The *Enclosed area* is the area of the shaded region being displayed. If a polygon intersects itself, the area from all sub-polygons will be added together.



The cyan lines are included in the Total length*. The* Total perimeter *includes all cyan lines and the black line. The* Enclosed area *is represented by the area filled by blue horizontal lines.*

Copying Measurements

To copy the information from the **Measure** dialog, right-click in the dialog and select *Select All*. Right-click again and select *Copy* . The information can then be pasted into the plot window or another program.

Measure Options

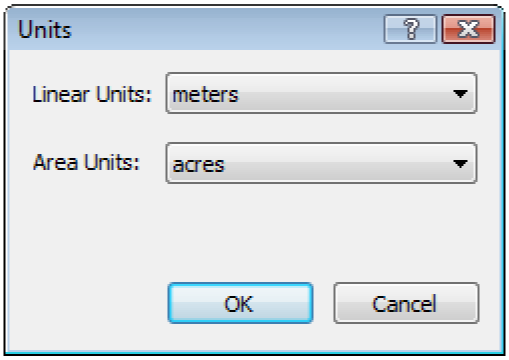
Click the **Options** menu in the **Measure** dialog to modify how the measurements are displayed and to save or export the measurement geometry.

Numeric Format

The numeric format for the length and area values can be changed by clicking the **Options | Numeric Format** command. The Label Format dialog opens. Set the desired format and click *OK* and the formats in the **Measure** dialog update.

Units

The units displayed in the **Measure** dialog are in *Map* units. By default, the *Map* units are controlled by the Coordinate System tab in the Properties window for the *Map* object. To change the units, click the **Options | Units** command in the **Measure** dialog. In the **Units** dialog, select the appropriate *Linear Units* and *Area Units*. Click *OK* to return to the **Measure** dialog.



Select the Linear Units *and* Area Units *for the measurements.*

The *Linear Units* are used for the *Segment length, Total length,* and *Total perimeter*. The *Area Units* are used for the *Enclosed area*.

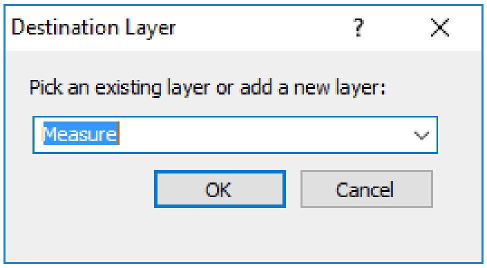
Note that the units cannot be changed if the map is not referenced or if the *Map* coordinate system is set to display latitude and longitude. Set the layer source coordinate system and the map target coordinate system and the units can be changed.

The default *Linear Units* and *Area Units* can be changed with the *mtDistUnit* and *mtAreaUnit*Default Settings. The following table lists the default setting value for the various options.

Linear Units	mtDistUnit Value	Area Units	mtAreaUnit Value
Millimeters	0	Square millimeters	0
Centimeters	1	Square centimeters	1
Meters	2	Square meters	2
Kilometers	3	Square kilometers	3
Inches	4	Square inches	4
Feet	5	Square feet	5
Survey feet	6	Square survey feet	6
Yards	7	Square yards	7
Miles	8	Square miles	8
Nautical miles	9	Square nautical miles	9
Chains	10	Square chains	10
Rods	11	Square rods	11
Links	12	Square links	12
Fathoms	13	Square fathoms	13
		Hectares	14
		Acres	15

Save Geometry to Base Layer

Click **Options | Save Geometry to Base Layer** in the **Measure** dialog to add the polyline or polygon used for measurement to a new or existing base layer. After clicking **Save Geometry to Base Layer** the **Destination Layer** dialog is displayed.



Select or create a destination layer for the measure geometry.

By default, measurement geometries are saved to the *Measure* base layer. If the *Measure* base layer does not exist, one will be created. Click the ▾ to select a layer from the list if you wish to add the geometry to an existing base layer. Type a layer name in the **Destination Layer** dialog to create a new base layer for the measurement geometry.

Export Geometry

Click **Options | Export Geometry** in the **Measure** dialog to export the measurement geometry to a vector file. After clicking the **Export Geometry** command, the Export dialog is displayed. Note that only vector file types will be displayed in the *Save as type* list.

Other Methods to View Area and Length

Other methods of displaying length and area are to digitize an object and create a base map from the object or add an empty base map to an existing map and draw the objects in the base map. Both of these methods are described below. The perimeter and area are displayed on the Info page in the **Properties** window when the object is selected.

Digitize

To digitize an object, click on an existing map layer and click the Digitize command. Click on the screen in the desired locations. The coordinates appear in the **Digitized Coordinates** window. Save the coordinates to a .BLN file by clicking the **File | Save As** command in the window. A new base map can be created by clicking the **Home | New Map | Base** command or the **Home | Add to Map | Layer | Base** command, if an existing map is selected. Once the base layer exists, click on any object in the base layer to see the length and area information on the Info tab in the **Properties** window.

Drawing Objects on an Existing Map

To draw objects on an existing map, click on the map to select it. Click the **Home | Add to Map | Layer | Empty Base** command to add an empty base map layer to the map. Click on the *Base* object in the **Contents** window to select the new empty base layer. Use the Features tab commands to add polylines, spline polylines, polygons, rectangles, rounded rectangles, or ellipses to the base map layer. Click on any object in the base layer to see the length and area information on the Info tab in the **Properties** window.

Overlay Maps

It is possible to combine several maps by overlaying the maps. You can overlay any combination of contour, base, post, color relief, vector, or surface maps. Overlays can contain only one wireframe, however. When you overlay maps, the overlays use a single set of X, Y, and Z axes and the maps are positioned according to the *Map* coordinate system. If two or more maps use the same limits, they will overlay on top of one another. If maps cover adjacent areas, overlaying maps places them in the correct position relative to one another and creates a single set of axes that span the entire range. Overlaid maps become a single object and are moved and scaled as a single entity.

Consider a contour map and a base map that displays the outline of a lake on the contour map. The limits of the base map are the XY extents of the lake and are not the same as the contour map limits. If you create both the base map and the contour map in a single plot window, they do not overlay correctly because by default the maps are scaled differently. In addition, each map uses a different set of XY axes. If you select both maps and then click the **Map Tools | Map Tools | Overlay Maps** command or the  button, the contour map and the base map are rescaled and combined into a single map using a single set of axes. The lake is now correctly positioned on the contour map.

Overlays and Wireframes

When you overlay contour, post, or base maps on a wireframe, the maps are draped over the wireframe. The wireframe is drawn in the usual fashion but the base, vector, or contour maps are "molded" over the top of the wireframe lines. Hidden lines are not removed from maps overlaid on wireframes. For example, contour lines are not hidden when the contour map lies over a wireframe.

Overlays and Surfaces

When you overlay maps on top of surface maps, hidden lines are removed and the maps are "molded" on the surface. Surface maps and images, vector files, and even other surface maps can be overlaid. The Overlays tab in the surface map properties contains options for handling color in these cases.

Overlay Exceptions

The **Map Tools | Map Tools | Overlay Maps** command overlays all selected maps. Most combinations of map types can be overlaid. The exceptions are combining wireframes and surfaces, overlaying raster maps or point clouds with wireframes, and overlaying multiple wireframes. Raster maps include color relief maps, surfaces, and base maps containing an image.

Overlaying Maps

Map layers from different map objects can be overlaid with the **Overlay Maps** command or with the Contents window. Click and drag map layers into other map objects to overlay maps with the **Contents** window. To overlay maps with the **Map Tools | Map Tools | Overlay Maps** command:

1. Create the maps in a single plot window by choosing the **Home | New Map** commands. When maps are first drawn in the plot window, they may or may not overlay directly, and each map is an independent object with its own set of axes.
2. Select the maps you want to combine. When selecting multiple objects you can:
    a. Hold down the SHIFT key and click each map layer to be combined in the plot window.
    b. Hold down the CTRL key and click each map to be combined in the **Contents** window.
    c. Click **Home | Selection | Select All** or press CTRL+A on the keyboard to select all the maps in the plot window.
    d. Click **Home | Selection | Select All | Block Select** to surround the maps to be combined.
3. Choose **Map Tools | Map Tools | Overlay Maps** to combine all the selected maps. The maps are combined in the correct position based on their coordinates and limits.

Editing Overlays

To edit individual overlays in a composite map, click on the desired map layer in the **Contents** window. Make the changes in the Properties window. The map layer is automatically updated with the changes.

Hiding Overlays

After combining maps, it is possible to hide one or more of the overlays. To temporarily hide a map, uncheck the visibility box next to the map layer name in the **Contents** window. The map is redrawn without the selected overlay. To make the overlay visible again, recheck the visibility box. Note that if a surface is made invisible, the overlays are also made invisible.

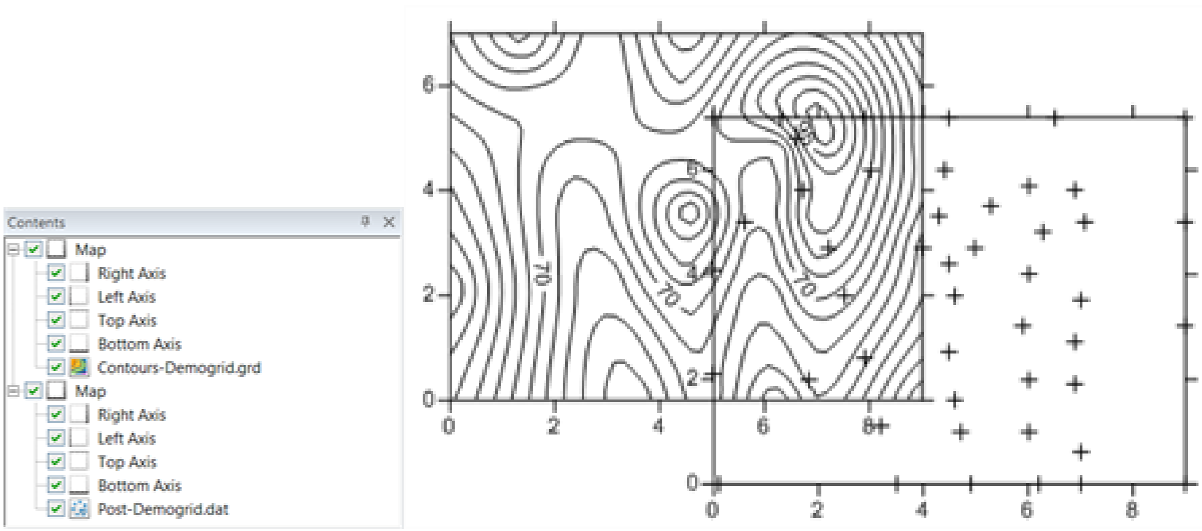
Deleting Overlays

To permanently remove an overlay from a map, select the map layer and press the DELETE key on the keyboard.

Combining Maps

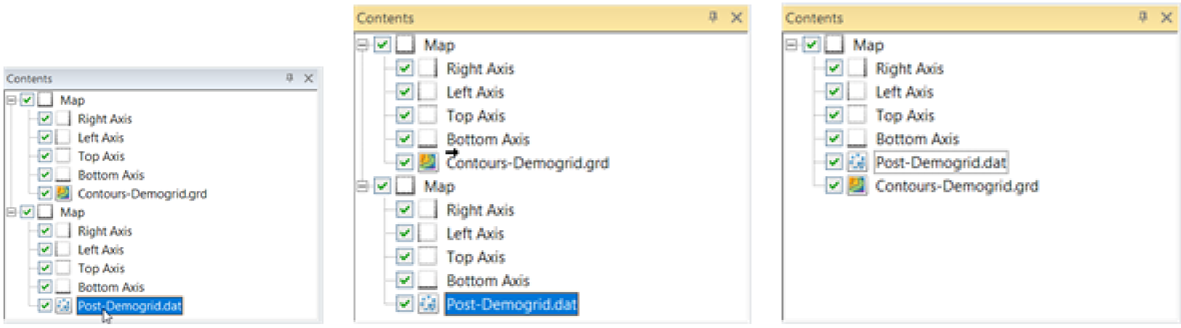
Overlay maps in **Surfer** by dragging the map layer to a new map frame in the Contents window, by using the **Map | Add** command, or by using the Map Tools | Map Tools | Overlay Maps command. To combine two maps by dragging in the **Contents** window,

1. Start with two separate map objects. In this example, there is a post map and a contour map. Note that each map has an independent set of axes.



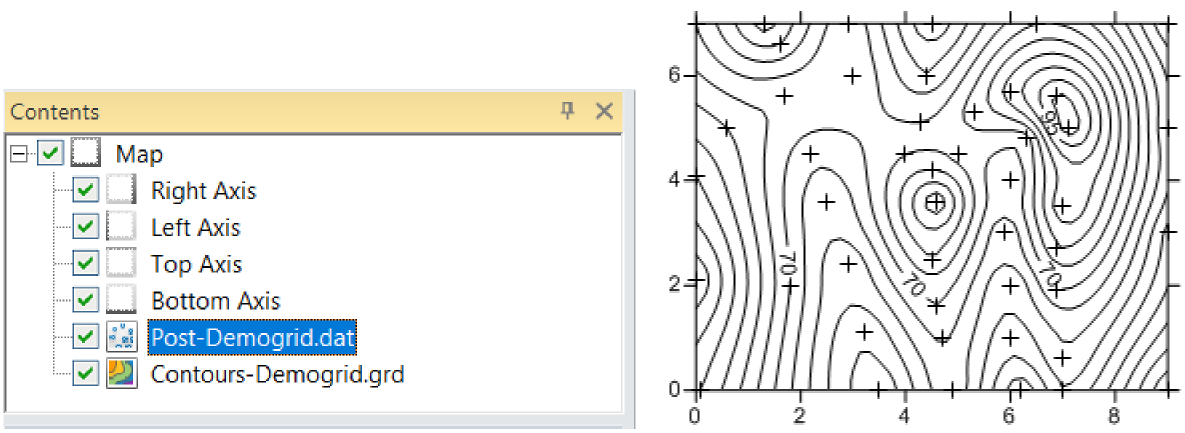
*The contour map layer and the post map layer are displayed in separate map objects in the **Contents** window and the plot window.*

2. Select the post map layer in the **Contents** window and drag it to the contour map object. To do this, left-click and hold the left mouse button while you drag the map layer to a new map frame. When the cursor changes to a horizontal arrow, release the left mouse button and the map layer is added to the new map frame. The post map will now be overlaid on the contour map.



First left-click and select the post map layer (left), then drag the post map layer to the other map object. When the cursor is a horizontal arrow (middle), release the mouse button to drop the map layer in the new location (right).

3. An empty map frame may remain after removing the last map layer from the map object, depending on your options. If an empty map frame exists, select the empty map frame and press DELETE on the keyboard to remove the empty map frame. The end result is a single map object with two map layers; a post map layer and a contour map layer. Additional map layers can be added with the Add to Map commands.



The result of this method is one Map object with two map layers.

Overlaying Maps on an Existing Map Frame

Maps can be overlaid on an existing map frame. Select the existing map and use the **Map | Add** command to overlay a new map on the selected map. See Introduction to Map Layers.

Removing a Map Overlay

Select the map layer and use the Break Apart Layer command to remove a map layer from a map object. Alternatively, right-click on the map layer and select **Break Apart Layer**.

Deleting a Map Layer

To delete a map layer from a map frame, select the map layer in the **Contents** window and press the DELETE key on the keyboard. Alternatively, you can select the map layer and use the Home | Clipboard | Delete command, or right-click the map layer and select *Delete*.

Overlaying Two Existing Maps

If two maps already exist, you can move (or overlay) a map layer from one map frame into the other map frame by dragging and dropping in the Contents window or by selecting both maps with **Home | Selection | Select All** and overlaying the maps with the **Map Tools | Map Tools | Overlay Maps** command.

Deleting Empty Map Layers

When the last map layer has been removed from a map, there is an option to automatically delete the existing map frame and axes. Click File | Options and click on the *General* page. Check the *Delete empty maps* option. Click *OK*. When the last map layer is moved from a map frame, the map frame and axes are automatically deleted. When this option is not checked, empty map frames and axes remain after the last map layer is moved. By default, this option is checked so that empty map objects are automatically deleted.

Stack Maps

The **Map Tools | Map Tools | Stack Maps** command aligns individual maps horizontally so that their coordinates align on the page. The **Stack Maps** command was designed to stack maps using similar coordinates on each map. This command is useful for keeping two or more maps separated on the page. The vertical position of individual maps on the page is not changed by this command.

When using **Map Tools | Map Tools | Stack Maps** the same View should be used for each map. This ensures that points on the stacked maps are aligned properly. The **View** parameters are applied to all selected maps. Stacking maps with perspective projections can be problematic since the Z axis is not vertical on the page. It is better to use the orthographic projection when stacking maps with different Z ranges.

The relative vertical placement of maps on the page defines how far apart the maps are after they are stacked. Before using the **Stack Maps** command, position the maps in the approximate location on the page where you want each map to be drawn. The **Stack Maps** command only moves maps horizontally on the page, not up or down.

The Stacked Maps.SRF sample file displays a contour map stacked on a 3D surface map.

Stacking a Contour Map over a 3D Surface Map

You can produce a plot of a contour map over a 3D surface map using **Map Tools | Map Tools | Stack Maps**. Usually a contour map is stacked over a 3D surface of the same grid file, but you can also stack different grid files one over another. If different grid files are used for stacking, the X and Y coordinates for both the grid files should be similar.

To stack a contour map over a 3D surface map:

1. Create a 3D surface and a contour map with the **Home | New Map** command.
2. Select both maps with the Home | Selection | Select All command.
3. With both maps selected, the map properties appears in the Properties window.
4. Click the **View** tab.
5. Set *Projection* to *Orthographic*.
6. Set the *Rotation (degrees)* and *Tilt (degrees)* to the desired values by dragging the . As you drag the slider next to the *Rotation (degrees)* and *Tilt (degrees)* commands, the maps automatically update.
7. Position the surface and contour maps vertically on the page where you want them to appear. For example, you might move the surface to the bottom half of the page, and move the contour map to the top half of the page.
8. Select both the surface and the contour map with the **Home | Selection | Select All** command and click the **Map Tools | Map Tools | Stack Maps** command or the button. The X and Y coordinates of the two maps are aligned horizontally on the page.
9. To show vertical lines connecting the maps, use Home | Insert | Polyline to draw the lines at the corners of the map.

Creating Several Maps in the Same Plot Window

When you create several maps in one plot window, the default map size often results in overlapping maps even after they are carefully placed on the page.

To create several maps in the same plot window:

1. In an empty plot window, create a map using the commands available in the **Map** menu.

2. If the default scaling is not satisfactory, use the map properties Scale page to adjust the map size.

3. Select each map and drag it to the position on the page where you want it to appear, or enter a specific *X* and *Y* location in the Position/Size toolbar.

4. Repeat the previous steps as many times as needed.

Aligning Several Maps on the Same Page

Surfer provides several ways to align maps on the same page.

- Use the mouse to manually drag the maps on the page. Use the arrow keys on the keyboard to nudge the map. This method is rather imprecise, but provides complete control over the map position.

- Use Map Tools | Map Tools | Stack Maps to align maps placed one above the other on the page. Maps are moved horizontally on the page so their coordinate systems and origins are aligned. Both maps must be selected to activate the **Stack Maps** command.

- Use Layout | Arrange | Align to align selected objects relative to the bounding box surrounding the objects. The objects may be aligned both vertically and horizontally. Map alignment done in this way is based on the map's bounding box and not on the coordinate system for the maps. The bounding box for a map is indicated by the selection handles.

- Use the Rulers and Grid page in the **File | Options** command to select the *Snap to Ruler* option. When the *Snap to Ruler* option is on, the upper left corner of the object's bounding box is snapped to the nearest ruler division when the object is moved.

- Use the *X* and *Y* fields in the Position/Size toolbar to enter the same X and/or Y locations for multiple maps as needed to create the desired alignment.

Masking with Background

Polygons can also obscure the background of a base map. For example, consider the sample map of California. The fill properties for the counties (sub-polygons) on the base map are set to solid white, and the map background uses a gray background.

To create a map with a masked background:

1. Create a base map containing polygons. Use the **Home | New Map | Base** command and select the sample file ca2010.gsb. Alternatively, there are many boundary files available for download on the Golden Software website.

2. Click on the Map object in the Contents window to open the map properties in the Properties window.

3. Click the Frame tab.

4. Click the ⊞ next to *Fill Properties* to open the fill properties section.

5. Set the *Pattern* to *Solid.*

6. Set the *Foreground Color* to *30% Black.* The entire map is drawn with a light gray color.

7. Click on the *Base* object in the **Contents** window to open the base properties in the **Properties** window.

8. On the **General** page, click the ⊞ next to *Fill Properties* to open the fill properties section.

9. Set the *Pattern* to *Solid.*

10. Set the *Foreground Color* to *White.* The map is now drawn with white counties on a gray background.

11. The fill properties of individual counties can also be adjusted if needed. To do so, click on a *Polygon* in the *Base* object. The properties listed in the **Properties** window are for that individual county.

Extract Grid or Data from Map

Surfer has the ability to extract a .GRD file from a grid based map or a data file from a post map. This is useful if you have lost the original .GRD or data file used to create a **Surfer** map and want to alter the grid or data to update the map.

To extract a grid from a grid based map:

1. Click once on the map layer in either the plot window or the Contents window to select the map layer.

2. With the map layer selected, click in the Properties window on the **General** tab.

3. Next to *Grid file*, click the button.

4. In the **Save Grid As** dialog, type in a *File name*. Set the *Save as type* to the desired grid format. Click *Save*.

5. The grid is saved to the specified location.

To extract a data file from a post or classed post map:

1. Click once on the post or classed post map layer in either the plot window or the **Contents** window to select the map layer.
2. With the map layer selected, click in the **Properties** window on the **General** tab.
3. Next to *Data file*, click the  button.
4. In the **Save As** dialog, type in a *File name*. Set the *Save as type* to the desired worksheet format. Click *Save*.
5. The data file is saved to the specified location.

Chapter 26 - Layer Tools

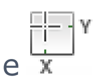
Digitize

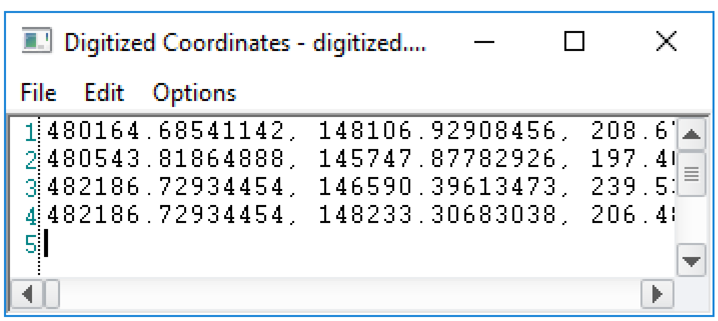
The **Map Tools | Layer Tools | Digitize** command allows you to write map coordinates to a data file. As you move the pointer across the selected map, the X, Y, and Z map coordinates for the current mouse position are shown in the status bar. Left-click on a map to write digitized points to the **Digitized Coordinates** dialog.

To use the **Digitize** command, you must select a horizontal planar map. To ensure that your map is horizontal planar, click on the *Map* object in the Contents window. Click on the View tab in the Properties window. Change the *Tilt (degrees)* to 90.

Note: Use the **3D View | Tools | 3D Digitize** command to digitize coordinates in the 3D view. Creating 3D digitized coordinate data might be useful for identifying spot locations and writing their XYZ coordinates to a file.

Digitized Coordinates Window

Click the **Map Tools | Layer Tools | Digitize** command or the  button or right-click on a map and select **Digitize** to enter into digitize mode. Left-click on a map, and the **Digitized Coordinates** window is displayed.



*The digitized X, Y, and Z values are written to the **Digitized Coordinates** window.*

The coordinates for the clicked point are written to the **Digitized Coordinates** window. Each time the map is clicked, a small, temporary red symbol is drawn on the map, and the map coordinates for the current mouse position are written to the **Digitized Coordinates** window. In this way, you can digitize boundaries from maps and easily create blanking files from the digitized information. You can also create a blanking file with **Map Tools | Layer Tools | Digitize**.

The **Digitized Coordinates** window stays open until it is closed by clicking the  button in the top right corner of the **Digitized Coordinates** window. This allows convenient editing of the data and allows digitizing to start and stop multiple times and be recorded in the same window.

Menu Commands

The **Digitized Coordinates** window has three menus: **File**, **Edit**, and **Options**.

File

The **File** menu allows you to save or open a .DAT data file or a Golden Software Blanking .BLN file.

Edit

The **Edit** menu command allows for common editing, such as undo, redo, cut, copy, paste, delete, find, replace, and select all. These commands work in much the same way as the commands in the Home and Data tabs for a worksheet.

Options

The **Options** menu allows the user to set the numeric format with the **Options | Coordinate Format** command. This opens the Label Format dialog, where the numeric format can be set. The period is always used as the decimal separator. No prefix, suffix, or thousands comma can be displayed in the **Digitized Coordinates** window.

The **Options** menu also allows the user to specify whether the Z value is included when digitizing a grid based map. Click the **Options | Include Z coordinate** command. The points you digitize after selecting this command will have a Z value when digitizing a grid based map. When digitizing other maps, such as base maps, or when you are digitizing outside the gridded area, the Z value is not included.

When multiple grid based maps are overlaid, the Z value is determined by the *Track map coordinates of* option in the Options dialog. When this option is set to *Topmost map/layer*, the map layer that is above all others in the Contents window is used for the Z value. When the *Track map coordinates of* option is set to *Selected map/layer,* the digitized Z value is determined by the current map layer that is selected in the **Contents** window. If the *Track map coordinates of* option is set to *Selected map/layer* and you select a different map layer while digitizing, the Z value will come from the newly selected map layer for new digitized points.

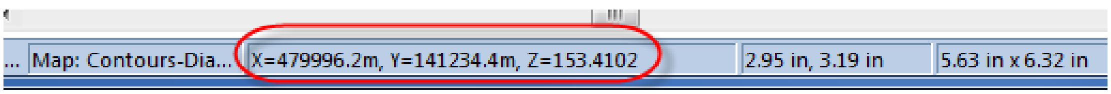
The **Options** menu also determines if the digitized blanking file assigns the NoData value inside or outside the defined points. By default, the **Options | NoData Inside Region** option is checked. When saving the digitized coordinates, a 1 is written to the BLN file blanking flag, and the area inside polygons is assigned the NoData value. Click the **NoData Inside Region** option to change the setting. When **NoData Inside Region** is unchecked, a 0 is written to the

BLN file blanking flag, and the region outside polygons is assigned the NoData value. By default, this setting is remembered during each **Surfer** session.

Digitizing Information from a Map

To digitize information from a map, follow these steps:

1. Create a map in the plot window and select the map. You can select an overlay, although the overlay cannot contain a wireframe or surface, or be tilted at an angle other than 90 degrees.
2. Click the **Map Tools | Layer Tools | Digitize** command or right-click and select **Digitize**. The cursor becomes a cross hair to indicate digitize mode.
3. As you move the cross hair cursor within the plot window, the map coordinates for the position are displayed in the status bar. If the map is a grid based map, like a contour map, the Z values are also displayed in the status bar.



The status bar displays the X, Y, and Z coordinates before you click on the map.

4. Click the left mouse button in the plot window to write the current coordinates to the **Digitized Coordinates** window. Continue adding points in this manner. Digitized points appear as temporary small red plus-signs on the map.
5. When you are finished using the **Digitize** command, click on any tool button or command, or press the ESC key on the keyboard.
6. To save the data in the **Digitized Coordinates** window, click the **File | Save As** command in the window. You are prompted to save the data as *Blanking Files (*.bln)* or *Data Files (*.dat).* Type a *File name* and click *Save* to save the data. Click *Cancel* to not save the data and return to the **Digitized Coordinates** window.
7. Click the  button in the top right corner of the window to close the **Digitized Coordinates** dialog. The dialog will stay open until closed in this manner.

Coordinate System Information

If the map that you are digitizing from is referenced with a coordinate system on the **Coordinate System** page, the digitized .BLN or data file automatically creates a .GSR2 file. The points digitized and the .GSR2 file contains the map's target coordinate system information. This is the map's coordinate system, not the map layer's coordinate system.

If you want to digitize information in a map layer's source coordinate system, you will need to change the map's target **Coordinate System** information to match the map layer's coordinate system.

Creating a Blanking File with the Digitize Command

One of the uses of the Digitize command is to create a Golden Software Blanking .BLN file. When creating a blanking file in this manner the header information is added to the file automatically. The file automatically assigns the NoData inside (1) flag to the polygon. If you wish to assign the NoData outside flag (0) to the polygon, click **Options | NoData Inside Region** in the **Digitized Coordinates** window to clear the **NoData Inside Region** option.

To create a blanking file with the **Digitize** command:

1. Create a map in the plot window.

2. Click once on the map layer to select it. This can be a map layer in an overlay, although the map cannot contain a 3D surface or 3D wireframe, or be tilted at any angle other than 90 degrees.

3. Click the **Map Tools | Layer Tools | Digitize** command or right-click on the map and select **Digitize**. The cursor becomes a cross hair.

4. Click the left mouse button in the plot window to write the current coordinates to the **Digitized Coordinates** window. Continue adding points in this manner. Digitized points appear as temporary small red plus-signs on the map.

5. If you wish to create multiple polylines, click in the **Digitized Coordinates** window. Manually insert a blank line between the coordinates that make up each polyline. Each group of contiguous coordinates will form a separate polyline in the blanking file.

6. If you wish to create a closed polygon, ensure the first and last coordinate are identical. You may need to copy the first line and paste it as the last line.

7. When you are finished defining the polyline(s) or polygon(s), select theFile | Save Ascommand in theDigitized Coordinateswindow. Save the file using a .BLN extension.

8. Press ESC on the keyboard or choose any toolbar button to exit digitize mode.

9. Click the  button in the **Digitized Coordinates** window to close the window.

Change a Map Layer Type

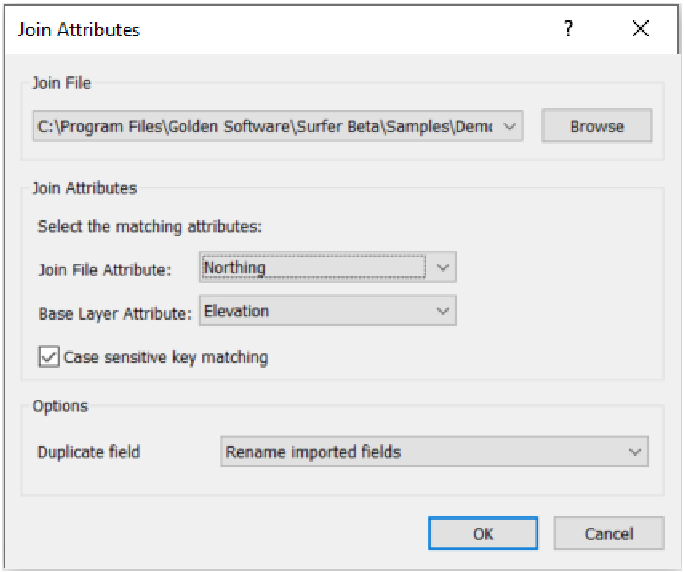
The **Map Tools | Layer Tools | Convert Layer** command changes the layer type from one to another. Select the grid-based map layer in the Contents window and click the **Map Tools | Layer Tools | Convert Layer** command to change the map layer to Contour, 3D Surface, 3D Wireframe, Color Relief, Grid Values, Watershed, Viewshed, or 1-Grid Vector.

Tips

- The **Convert Layer** command and button are only active if a grid-based map layer is selected. Maps that cannot be added to the selected map are grayed out. For example, 2-grid vector maps cannot be converted and a color relief layer cannot be converted to another color relief layer.
- Colormaps are retained when converting to Contour, Surface, Color Relief, or 1-Grid Vector maps.
- Do not convert a geotransformed layer to a Surface layer. This will likely result in the layer being outside the map frame and invisible.

Join Attributes Dialog

Click the button to import a data file that will add attributes to the existing Attribute Table. As long as a single object is in common between the existing layer of attributes and the new file, Surfer will import all the columns from the new file. Select the file and attributes to import in the **Join Attributes** dialog.



*Use the **Join Attributes** dialog to add attributes to objects by selecting a file that shares an attribute with an attribute from the base layer map.*

Join File

Browse to the data file containing the attributes that are being added and select the file to import.

Join Attributes

To accurately merge new data with existing data in Surfer, *Join Attributes* specifies the common objects between the two files. *Join Attributes* assigns the new attributes to the correct objects.

Join File Attribute

Select the name of the column from the Excel file being imported that contains the objects in common with the Surfer data file. The imported file contains the new attributes.

Base Layer Attribute

Select the name of the column in the Surfer data file that contains the objects in common with Excel file being imported.

Case Sensitive Key Matching

Select the *Case sensitive key matching* option when the attribute in the *Join File Attribute* and *Base Layer Attribute* must have the same case.

Options

Duplicate Field Handling

Select *Rename imported fields* when two columns share a name but the data is different. Surfer will add an extension with the imported file's name to the new data field in the updated **Attribute Table**.

Select *Overwrite existing fields* to delete existing data and import new data in fields that are identical between the two files.

Select *Ignore duplicate fields* to retain the data in the existing field and ignore the data in the file being imported.

Edit Data from the Map Layer

The **Map Tools | Edit Layer | Data** command or the button opens a worksheet window to display the worksheet that contains the data for a map layer. Select the map layer you wish to edit in the **Contents** window and then click **Map Tools | Edit Layer | Data**. Or, right click on the map and click **Edit Data** to open the worksheet window that contains the data.

Break Apart Layer

The **Map Tools | Layer Tools | Break Apart Layer** command breaks apart a layer from the map. Select the map layer in the Contents window and click the **Map Tools | Layer Tools | Break Apart Layer** command or the button to create a separate map object containing the selected map layer. Alternatively, right-click on the map layer in the **Contents** window and click **Break Apart Layer**.

The **Break Apart Layer** command is only active if a map layer is selected.

Delete Map Layer (Break Apart Overlay)

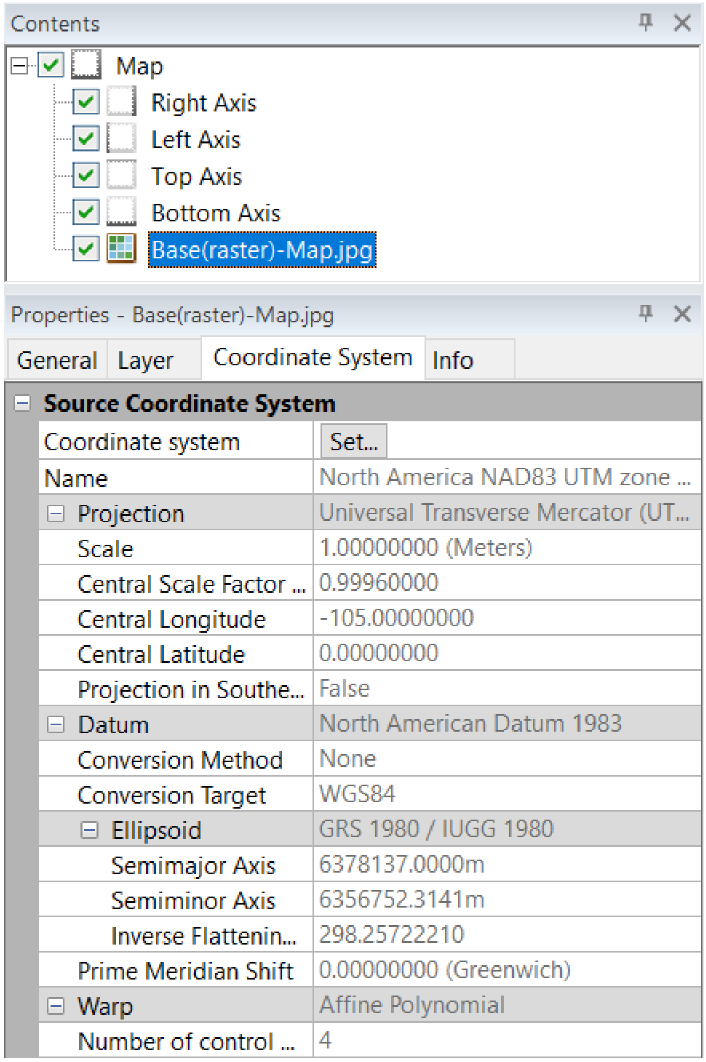
To delete a map layer (overlay) from a composite map, select the map layer in the **Contents** window and press the DELETE key on your keyboard. Alternatively, select the map in the **Contents** window, right-click and select *Delete*.

Assigning Coordinates to a Base (raster) Layer

When you import a georeferenced image as a base map, the image will be located according to its referenced coordinates. If your image is not georeferenced, you can assign coordinates to the image. You can redefine image coordinates when the image is part of a base layer, i.e. the image must be added using the Home | New Map | Base or **Home | Add to Map | Layer | Base** command (not the Home | Insert | Graphic command). By default, image files that are not georeferenced are assigned a coordinate system that corresponds to the rows and columns of the raster image. Under most circumstances the pixel coordinates or page coordinates do not match the coordinate systems used on other types of maps. In order to overlay this base map with other maps, you will need to redefine the coordinates for the image. After the image coordinates are redefined, the image base maps can be used in overlays with other types of maps. The image coordinate options are grayed unless the base map consists of a single image.

To change the base map image coordinates:

1. Click the **Home | New Map | Base** command. The **Import** dialog opens.
2. Select the image to create the base map.
3. Click *Open*.
4. In the Contents window, click on the base layer object to open the base layer properties in the Properties window.



Click on the Base(raster) object to open the base map properties in the ***Properties*** *window.*

5. You may wish to specify the coordinate system for the base layer under the following conditions:

    - You know the coordinate system you will be using to georeference the base layer, AND
    - You plan to overlay the base layer with layers in different coordinate systems, OR
    - You plan to transform the map to a different coordinate system,

If so, set the source coordinate system on the Coordinate System tab of the **Properties** window. Otherwise, leave the coordinate system set to *Unreferenced local system*.

6. Click the **Map Tools | Layer Tools | Georeference Image** command. The Georeference Image window is displayed.

7. Add control points to the image, specify the control point target coordinates, specify the warp method, and update the map with the **Georeference Image** window.

8. Verify the map looks correct in the **Surfer** window. Add, remove, or change control points or change the warp method if the map does not appear correct.

9. When you are satisfied with the georeferenced image, close the **Georeference Image** window.

The image is updated using the new coordinates.

Remarks

- When first georeferencing an image, or when making large changes to a georeferenced image, it is likely you will have to adjust the *Map* object's Limits and Scale to see the image.

    1. Click on the *Map* object in the **Contents** window.

    2. Click on the **Limits** tab. Check the *Use data limits* box.

    3. Click on the **Scale** tab. Change the scale properties, if necessary.

- Image files load as base maps in pixel coordinates unless the image is referenced.

- If you did not set the source coordinate system before georeferencing the image, you may wish to set the coordinate system. Set the base layer's coordinate system on the Coordinate System tab.

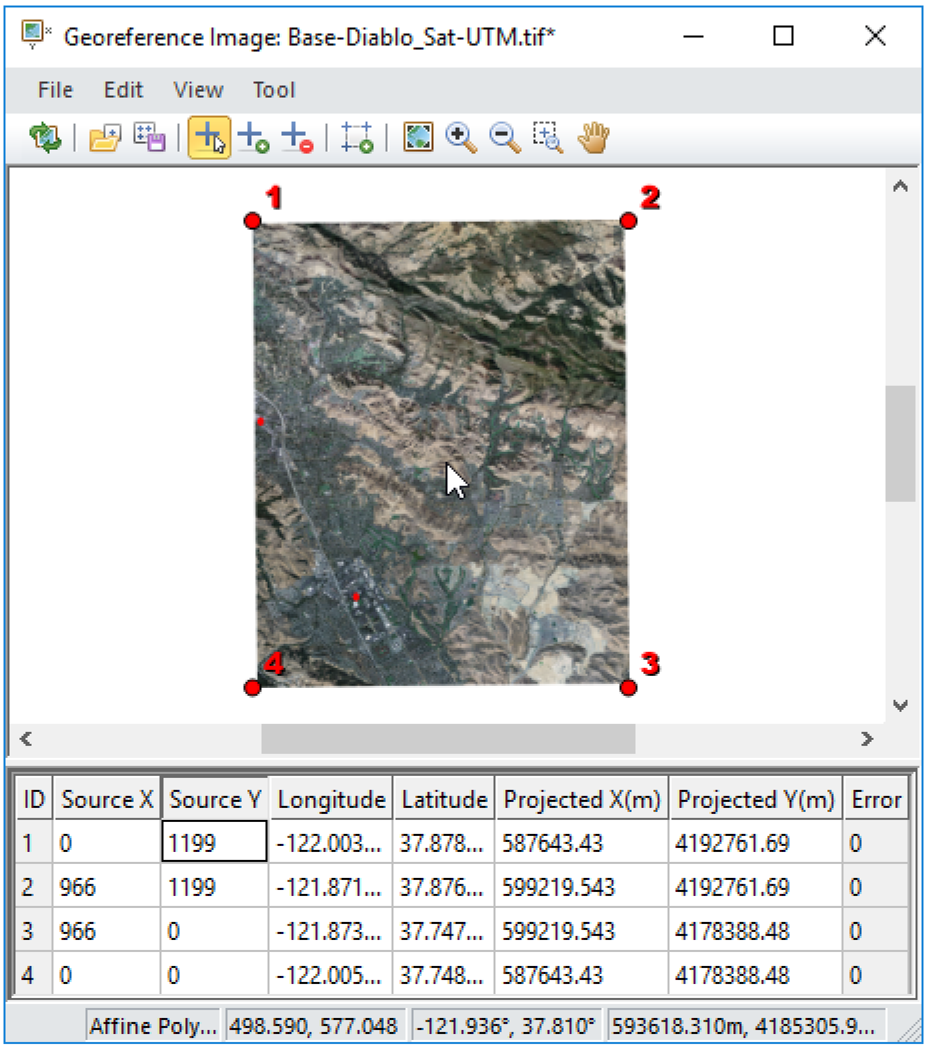
Georeference Image

An image is often used for a base map. Georeferencing is necessary to place unreferenced images in the correct relative position in the coordinate system when overlaying multiple map layers. Occasionally you may need to adjust an already georeferenced image. Georeference an image by selecting the base (raster) layer and clicking the **Map Tools | Layer Tools | Georeference Image** command or the button. You can also right-click the base layer in the **Contents** window or plot window and click **Georeference Image** in the context menu. Finally, you can georeference an image by clicking *Georeference Image* in the **Properties** window General page for the base (raster) layer.

Images must be in a base layer to be georeferenced, and the image must be at least 2x2 pixels to be georeferenced. Images are georeferenced with the **Georeference Image** window. Use the Home | New Map | Base or **Home | Add to Map | Layer | Base** command to import an image as a base layer. The **Georeference Image** command cannot be used with images imported with File | Insert | Graphic.

Georeference Image Window

The **Georeference Image** window is opened by clicking **Map Tools | Layer Tools | Georeference Image**, right-clicking the base layer or image object in the **Contents** window and clicking **Georeference Image**, or clicking *Georeference Image* in the **Properties** window General page for the base (raster) layer.



*Calibrate the image and set the warping method in the **Georeference Image** window.*

The **Georeference Image** window is a separate window from the **Surfer** application. You can move back and forth between the windows to verify your changes are correct and to test different warping methods. You can also minimize, maximize, restore, move, and resize the **Georeference Image** window.

When the **Georeference Image** window is minimized, the title bar is moved to the lower left corner of your screen. You can click the **Restore** or **Maximize** button to restore the window. Alternatively, click the **Map Tools | Layer Tools | Georeference Image** command or *Georeference image* button in the **Properties** window to restore the window.

The **Georeference Image** window displays the image name in the title bar. The **Georeference Image** window has four menus with various commands and a command toolbar. The window has two panes: a viewer pane that displays the image and control points, and a table that hold georeferencing information about the control points. Click and drag the dividing line between the table and viewer to adjust the relative sizes within the window. The entire window can also be resized by clicking and dragging a side or corner. A status bar is displayed at the bottom of the **Georeference Image** window.

Menu Commands and Toolbar

The **Georeference Image** window has four menus: **File**, **Edit**, **View**, and **Tool**. The **File** menu commands are used to apply changes to the map and save or load control points. The **Edit** menu commands edit the control points and sets the warp method. The **View** menu commands change the view in the view pane. The **Tool** menu commands change the tool mode for the mouse and cursor.

File Menu Commands

Update Map	Applies the image georeferencing to the map in the **Surfer** window.		CTRL + U
Load Points	Loads control points from a Georeference GEOREF file to the control points table.		CTRL + L
Save Points	Saves the control points to a GEOREF file.		CTRL + S
Close	Closes the **Georeference Image** window.		

Edit Menu Commands

Undo	Undoes the previous action in the **Georeference Image** window.		CTRL + Z
Redo	Redoes the previously undone action.		CTRL + Y
Delete	Deletes the selected control point or points.		DEL
Select All	Selects all control points.		CTRL + A

Deselect All	Deselects all control points.		
Set Warp Method	Sets the warp method		
Add Control Points At Corners	Adds control points to the four corners of the image. Appends the control points to the end of the control points table.		

View Menu Commands

Image	Adjust the view to fit the entire image in the view pane.	
Zoom In	Changes cursor to zoom in mode.	
Zoom Out	Changes cursor to zoom out mode.	
Zoom Selected	Zooms in to the selected point or points	
Pan	Changes the cursor to pan mode.	
Antialiasing	Use automatic mode, or turn antialiasing on or off.	

Tool Menu Commands

Select Mode	Changes the cursor to select mode.	
Add Mode	Changes the cursor to add point mode.	
Delete Mode	Changes the cursor to delete point mode.	

View Pane

The view pane displays the source image and control points. The view pane is dynamically linked to the control points table. Selecting, adding, changing, or removing a control point in the view pane automatically selects, adds, changes, or removes the control point in the control point table. Use the **View** menu commands, toolbar buttons, or mouse wheel to change the view in the view pane.

Antaliasing

By default, the **View | Antialiasing | Automatic** option is active. When zoomed out, antialiasing is enabled to smooth lines in the image. When you zoom in to the point at which one pixel in the image is 10x10 pixels or larger on the screen, antialiasing is turned off to provide a sharper picture.

However, if you wish to keep antialiasing on or off regardless of zoom level, click the **View | Antialiasing | On** or **View | Antialiasing | Off** command.

Control Point Table

The control point table contains information about the source and target coordinates for the control points. At least three control points are required to warp the image. Most warp methods require at least four control points, but there is no limit to the number of control points you can add. At least six columns are displayed in the control point table. Each row represents a single control point. The control point table is dynamically linked to the view pane. Changes made in the control point table are automatically reflected in the view pane. Any missing values or values that result in errors are highlighted in yellow in the control point table.

ID	Source X	Source Y	Longitude	Latitude	Projected X(m)	Projected Y(m)	Error
1	0	-1	-105.256944	39.7420726	477985	4399162	0
2	1003	-1	-105.116883	39.7422981	489985.499	4399162	0
3	1003	1249	-105.117111	39.8769763	489985.499	4414109.5	0
4	0	1249	-105.257446	39.8767497	477985	4414109.5	0

Enter the control point information in the control point table.

- The *ID* column contains the ID number for the control point. The ID helps you associate rows with points in the view pane.
- The *Source X* and *Source Y* columns contain the pixel location of the control points on the image. The 0,0 point is located at the lower left corner of the image. The maximum values are determined by the number of pixels in the image. The maximum point is located at the upper right corner of the image.
- The *Target X* and *Target Y* columns contain the desired coordinates of the control points. The *Target X* and *Target Y* columns are displayed when the base layer is assigned an unreferenced local coordinate system. Type the coordinate values for the control point in the *Target X* and *Target Y* fields.
- The *Longitude* and *Latitude* columns contain the desired coordinates of the control points. The *Longitude* and *Latitude* columns are displayed when the base layer is assigned a geographic or projected coordinate system. Type the coordinate values in degrees for the control point in the *Longitude* and *Latitude* fields. The values in the *Projected X* and *Projected Y* columns are automatically updated when you make a change to the values in the *Longitude* or *Latitude* column.
- The *Projected X* and *Projected Y* columns contain the desired coordinates of the control points. The *Projected X* and *Projected Y* columns are displayed when the base layer is assigned a projected coordinate system. Type the coordinate values in the coordinate system's units for the control point in the *Projected X* and *Projected Y* fields. The values in the *Longitude* and *Latitude* columns are automatically updated when you make a change to the values in the *Projected X* or *Projected Y* column.
- The *Error*column displays the RMS error value for each calibration point between the source image and referenced image.

Click a cell to select it. Begin typing to replace the selected cell's contents. Double-click on a cell to enter cell edit mode and edit the current cell contents. When you are finished replacing or editing the cell's contents, press enter, click on another cell, or navigate to another cell with the ARROW keys.

Status Bar

The status bar is displayed at the bottom of the **Georeference Image** window. The status bar includes three to five sections depending on the base layer's source coordinate system, from left to right:

- A tool tip for the current tool or the command over which the cursor is hovering.
- The selected warp method. No warp method is displayed if **Edit | Set Warp Method | Automatic** is selected.
- The cursor location in pixel units for all images.
- The cursor location in longitude and latitude units for images with a geographic source coordinate system.
- The cursor location in the coordinate system's units for images with a projected source coordinate system.

You may need to resize the **Georeference Image** window and/or the status bar sections to see all the status bar sections. Individual sections of the status bar can also be resized by clicking and dragging the separators.

Adding, Removing, Modifying, and Saving the Control Points

Control points are added, removed, and modified with the various commands, actions with the mouse, and/or key strokes. The **Georeference Image** window must contain at least three control points to warp the image.

Adding a New Control Point

There are three methods for adding control points to the image.

- Click the **Tool | Add Mode** command or  button to add points to the image with the mouse. The cursor changes to a crosshair to indicate you are in add mode. Click on the image and a control point is created. Zoom in with the mouse wheel or **Zoom In** command to precisely place control points.
- Click the **Edit | Add Control Points At Corners** command or  button to add a control point at each corner of the image. Note that aerial photographs may have distortion around the edges, and therefore using control points that are not on the edges may generate a more accurate calibration. However, this depends on if or to what extent there is distortion along the edges and the extents of the image coverage.

- Click the **File | Load Points** command or button to add control points from a GEOREF file. Loading points from a georeferencing file replaces any existing control points.

Selecting Control Points

Click the **Tool | Select Mode** command or button or press ESC to enable select mode. Click a control point to select it. Click and drag in the view pane to block select one or more control points. Click the **Edit | Select All** command or press CTRL + A to select all control points. Selected points are represented by green symbols and text, while deselected points are represented by red symbols and text. When points are selected in the view pane, the rows with the points are highlighted in the control point table.

Select a control point by clicking anywhere in the control point's row in the table. Select multiple control points by holding CTRL while clicking in different rows. Select a contiguous group of points by clicking the first row you'd like to select, holding SHIFT, and clicking the last row you'd like to select. The two rows you clicked and all rows between are selected.

Moving Control Points

Control points can be moved in the view pane or with the control point table:

- Select one or more control points to move the point or points to a different location on the image. Once you have selected the point you wish to move, click and drag the point to the desired location. All selected points will move the same direction and distance from their original starting location if you have more than one point selected. The *Source X* and *Source Y* values in the control point table update automatically for the moved point or points.
- Type a new value in the *Source X* and/or *Source Y* column to move a control point. The control point moves in the view pane automatically to reflect the new *Source X* and *Source Y* values.

Deleting Control Points

Control points can be deleted by clicking the control point when delete mode is active. Click **Tool | Delete Mode** or the button to activate delete mode.

Alternatively, select one more control points with any of the methods listed in the *Selecting Control Points* section of this topic. Next click the **Edit | Delete** command or press DEL to remove the control point or points. The point or points are removed from the control point table and view pane.

Saving Calibration Points

Click the **File | Save Points** command or button to save the calibration points to a GEOREF file. The GEOREF file can be used with the image source file to quickly georeference the image in other **Surfer** projects. Alternatively you can export the georeferenced image to a file type that includes georeference information, such as GeoTIFF.

Warp Method

The warp method determines how the image is warped to the control points. Available options are: Affine Polynomial, First Order Polynomial, Second Order Polynomial, Third Order Polynomial, Thin Plate Spline, Natural Cubic Spline, Marcov Spline, Exponential Spline, Rational Quadratic Spline, and Inverse Distance Squared.

Click the **Edit | Set Warp Method** command and click the desired method from the list to change the warp method. If too few points exist for a warp method, the method is disabled in the **Edit | Set Warp Method** menu. The number of required control points is displayed before the warp method name. *Affine Polynomial* requires at least three control points. *Second Order Polynomial* requires at least six control points. *Third Order Polynomial* requires at least ten control points. All other methods require at least four control points. Refer to each specific warp method topic for the transformation information and an image demonstrating the warp.

Automatic Warp Method

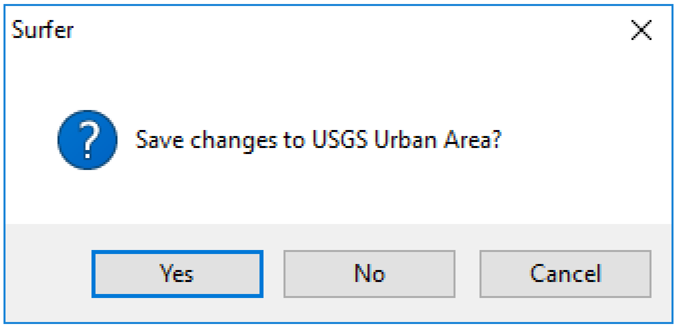
The **Edit | Set Warp Method | Automatic** option is checked by default. The default option is determined by the number of control points when the *Automatic* option is active. When there are three or more control points, the default method is *Affine Polynomial*.

Turn off automatic warp method selection by clicking the **Edit | Set Warp Method | Automatic** command to remove the check mark. The status bar will display *Unknown* and the **File | Update Map** command will be unavailable if the selected warp method requires more control points.

Applying the Georeferencing to the Map

Click the **File | Update Map** command or button to update the map with the image georeferencing. Click on the main **Surfer** window to view the map and verify the base layer image is satisfactory. When first georeferencing an image, or when making large changes to a georeferenced image, it is likely you will have to adjust the *Map* object's Limits and Scale to see the image. If the image is not satisfactory, you can change, add, or remove control points or change the warp method. Once the base layer image looks correct, you can close the **Georference Image** window by clicking the **File | Close** command or × button.

If you click the **Close** command with unapplied changes to the control points, a **Surfer** dialog is displayed.



Click *Yes* to apply your recent changes to the map and close the **Georeference Image** window. Click *No* to close the **Georeference Image** window without applying the changes. Click *Cancel* to return to the **Georeference Image** window.

Undoing Changes to the Map

If you wish to revert the changes to the map after clicking the **File | Update Map** command, the changes must be undone with the plot window Undo command. The action is added to the plot window undo list, because the change happens in the plot window rather than the **Georeference Image** window. After clicking the **Undo** command in the plot window, the map will revert back to its previous state. The control points and image in the **Georeference Image** window will not be changed.

Spatial Transformations

You can georeference an image with one of several spatial transformation methods, called warp methods. Image warping involves the process of mapping source coordinates to destination coordinates. This process requires that several points with known coordinates are located in the image. These points are known as control points. With these known points and the selected warp method, **Surfer** maps the control points to the desired coordinates. The points cannot fall into a straight line.

The spatial transformation methods correct for translation, rotation, and differential scaling. Spatial transformation is analogous to stretching and pinning a rubber sheet. The sheet is pinned down in various locations (control points) and is consequently stretched and contracted between these points. Spatial transformations can stretch the project in several different directions at one time. Therefore, it is beneficial to define more control points where distortion is greatest.

Spatial Transformation Methods

A generalized discussion of the spatial transformation methods is included in this help file. For mathematical details, refer to one of the references. A graphical illustration of each method is included. Keep in mind that the results depend upon the mapping from the source control points to the destination control points and the spatial transformation selected. In this case, the graphics are exaggerated for detail.

If a selected spatial transformation method is incompatible with the number of control points, **Surfer** replaces the method with the best method available. If you are unsure of which method to use, select *Affine Polynomial*.

Affine Polynomial
First Order Polynomial
Second Order Polynomial
Third Order Polynomial
Thin Plate Spline
Natural Cubic Spline
Marcov Spline
Exponential Spline
Rational Quadratic Spline
Inverse Distance Squared

Affine Polynomial

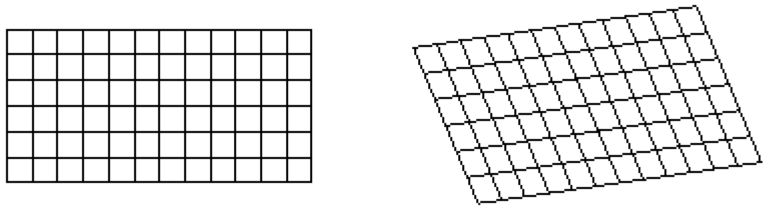
The most common transformation is *Affine Polynomial*. Affine transformations are a subset of bilinear transformations. Bilinear transformations account for rotation, shift, and differential scaling in X and Y. The X and Y axis orthogonality may change, but parallel lines remain parallel. Affine polynomial transformations of a plane change squares into parallelograms and change circles into ellipses of the same shape and orientation. The affine polynomial transformation results may not be as good as when using a higher order polynomial.

The minimum number of control points required for this transformation is three, though four are required to calculate an RMS value.

The general form for affine transformations is:

$$X = a_{0_x} + a_{1_x} x + a_{2_x} y$$
$$Y = a_{0_y} + a_{1_y} x + a_{2_y} y$$

This is called the six parameter affine polynomial transform.



The left graphic is the file before using the affine polynomial transformation. The right graphic is the file after an affine polynomial transformation. Note that lines remain parallel.

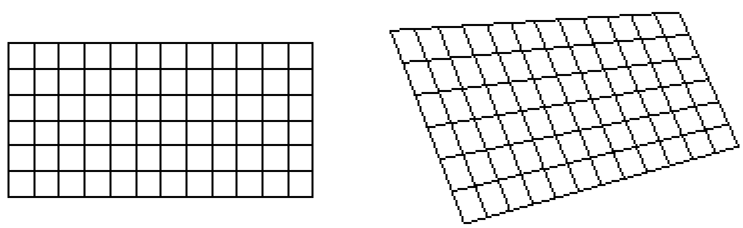
First Order Polynomial

The *First Order Polynomial* transformation preserves equally spaced points along a line but does not preserve diagonal lines. The equation is of the form:

$$X = a_{0_x} + a_{1_x} x + a_{2_x} y + a_{3_x} xy$$

$$Y = a_{0_y} + a_{1_y} x + a_{2_y} y + a_{3_y} xy$$

The minimum number of control points required for this transformation is four.

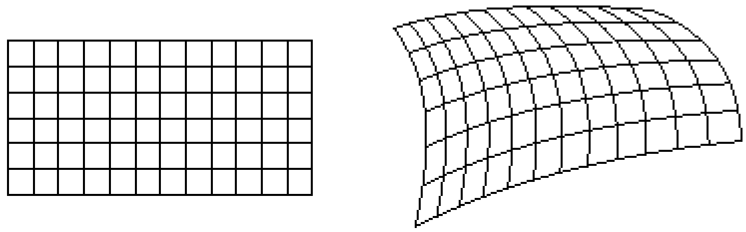


The left graphic is the file before using the first order polynomial transformation. The right graphic is the file after using a first order polynomial transformation. Note that lines are no longer parallel.

Second Order Polynomial

The minimum number of control points required for the *Second Order Polynomial* transformation is six. The equation is of the form:

$$X = a_{0x} + a_{1x} x + a_{2x} y + a_{3x} xy + a_{4x} x^2 + a_{5x} y^2$$
$$Y = a_{0y} + a_{1y} x + a_{2y} y + a_{3y} xy + a_{4y} x^2 + a_{5y} y^2$$

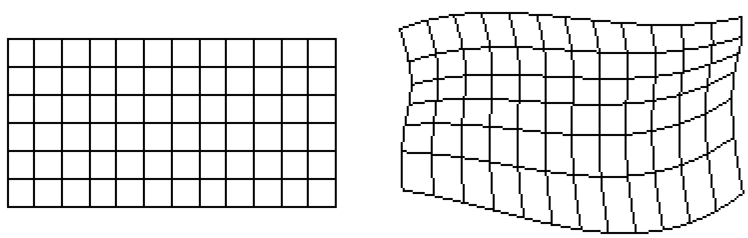


The left graphic is the file before using the second order polynomial transformation. The right graphic is the file after using second order polynomial transformation.

Third Order Polynomial

The minimum number of control points required for the *Third Order Polynomial* transformation is ten. The equation is of the form:

$$X = a_{0x} + a_{1x} x + a_{2x} y + a_{3x} x^2 + a_{4x} xy + a_{5x} y^2 + a_{6x} x^3 + a_{7x} x^2 y + a_{8x} xy^2 + a_{9x} y^3$$
$$Y = a_{0y} + a_{1y} x + a_{2y} y + a_{3y} x^2 + a_{4y} xy + a_{5y} y^2 + a_{6y} x^3 + a_{7y} x^2 y + a_{8y} xy^2 + a_{9y} y^3$$



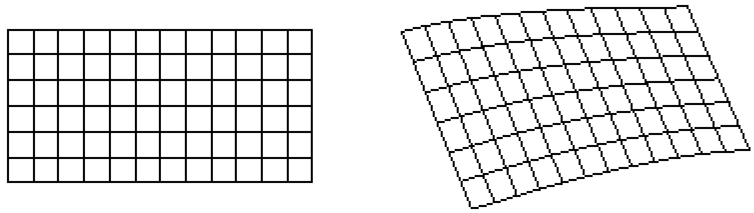
The left graphic is the file before using the third order polynomial transformation. The right graphic is the file after using third order polynomial transformation.

Thin Plate Spline

Radial basis functions include the *Thin Plate Spline*, *Natural Cubic Spline*, *Marcov Spline*, *Exponential Spline*, *and Rational Quadratic Spline*. These methods are best for correcting local distortions. RMS values are not reported when using radial basis function methods as these methods are perfect interpolators at the control points.

The minimum number of control points required for the *Thin Plate Spline* transformation is four. The equation is of the form:

$$\Phi(r_i) = \frac{1}{2} r_i^2 \log r_i^2$$



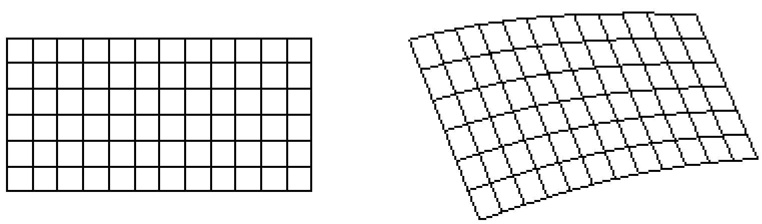
The left graphic is the file before using the thin plate spline transformation. The right graphic is the file after using a thin plate spline transformation.

Natural Cubic Spline

Radial basis functions include the *Thin Plate Spline*, *Natural Cubic Spline*, *Marcov Spline*, *Exponential Spline*, and *Rational Quadratic Spline*. These methods are best for correcting local distortions. RMS values are not reported when using radial basis function methods as these methods are perfect interpolators at the control points.

The minimum number of control points required for the *Natural Cubic Spline* transformation is four. The equation is of the form:

$$\Phi(r_i) = r_i^3$$



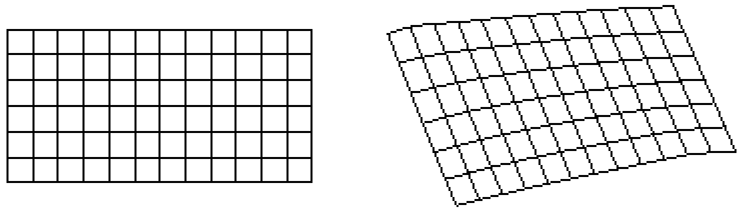
The left graphic is the file before using the natural cubic spline transformation. The right graphic is the file after using a natural cubic spline transformation.

Marcov Spline

Radial basis functions include the *Thin Plate Spline*, *Natural Cubic Spline*, *Marcov Spline*, *Exponential Spline*, and *Rational Quadratic Spline*. These methods are best for correcting local distortions. RMS values are not reported when using radial basis function methods as these methods are perfect interpolators at the control points.

The minimum number of control points required for the *Marcov Spline* transformation is four. The equation is of the form:

$$\Phi(r_i) = -r_i$$



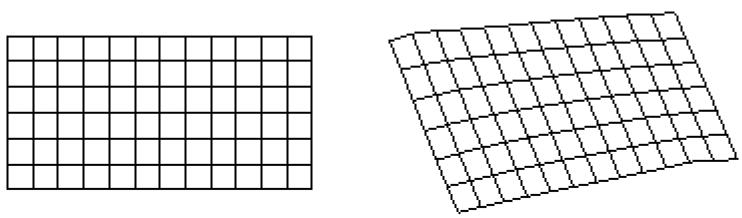
The left graphic is the file before using the Marcov spline transformation. The right graphic is the file after using a Marcov spline transformation.

Exponential Spline

Radial basis functions include the *Thin Plate Spline*, *Natural Cubic Spline*, *Marcov Spline*, *Exponential Spline*, and *Rational Quadratic Spline*. These methods are best for correcting local distortions. RMS values are not reported when using radial basis function methods as these methods are perfect interpolators at the control points.

The minimum number of control points required for the *Exponential Spline* transformation is four. The equation is of the form:

$$\Phi(r_i) = e^{-r_i}$$



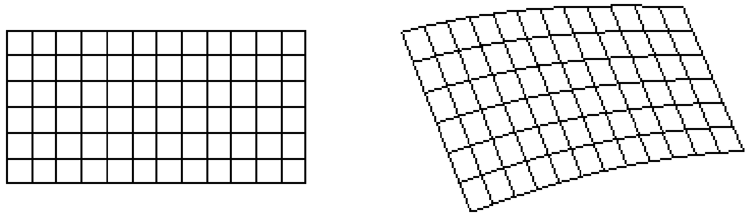
The left graphic is the file before using the exponential spline transformation. The right graphic is the file after using an exponential spline transformation.

Rational Quadratic Spline

Radial basis functions include the *Thin Plate Spline*, *Natural Cubic Spline*, *Marcov Spline*, *Exponential Spline*, and *Rational Quadratic Spline*. These methods are best for correcting local distortions. RMS values are not reported when using radial basis function methods as these methods are perfect interpolators at the control points.

The minimum number of control points required for the *Rational Quadratic Spline* transformation is four. The equation is of the form:

$$\Phi(r_i) = -\frac{r_i^2}{1 + r_i^2}$$



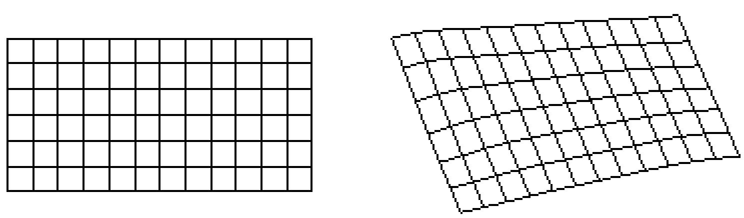
The left graphic is the file before using the rational quadratic spline transformation. The right graphic is the file after using a rational quadratic spline transformation.

Inverse Distance Squared

The *Inverse Distance Squared* method is an inverse distance to a power method. The minimum number of control points required for this transformation is four. A RMS value is not reported for this method because this method is a perfect interpolator at the control points. The equation is of the form:

$$X = a_{0_x} + a_{1_x} x + a_{2_x} y + \sum_{i=0}^{N-1} w_i(x, y) g_{i_x}$$

$$Y = a_{0_y} + a_{1_y} x + a_{2_y} y + \sum_{i=0}^{N-1} w_i(x, y) g_{i_y}$$



The left graphic is the file before using the inverse distance squared transformation. The right graphic is the file after using an inverse distance squared transformation.

Spatial Transformation Methods References

Chalermwat, Prachya, *High Performance Automatic Image Registration for Remote Sensing*, http://science.gmu.edu/~prachya/thesis/#_Toc467903485, July 30, 2001.

Crane, Randy (1996), *A Simplified Approach to Image Processing: Classic and Modern Techniques in C*, Prentice Hall, 336 pp.

Fogel, David N., *Image Rectification with Radial Basis Functions: Application to RS/GIS Data Integration*, http://www.ncgia.ucsb.edu/conf/SANTA_FE_CD-ROM/sf_papers/fogel_david/santafe.html, July 30, 2001.

Gong, Peng, *Remote Sensing and Image Analysis, 4.4 Georeferencing (Geometric Correction)*, http://www.cnr.berkeley.edu/~gong/textbook/chapter4/html/sect44.htm, July 30, 2001.

Gong, Peng, *Remote Sensing and Image Analysis, 4.5 Image Resampling*, http://www.cnr.berkeley.edu/~gong/textbook/chapter4/html/sect45.htm, July 30, 2001.

Glynn, Earl F., *Image Processing*, http://www.efg2.com/Lab/Library/ImageProcessing/Algorithms.htm#ImageRegistration, July 30, 2001.

Wolberg, George, (1990) *Digital Image Warping*, IEEE Computer Society Press Monograph, 340 pp.

Convert Coordinates

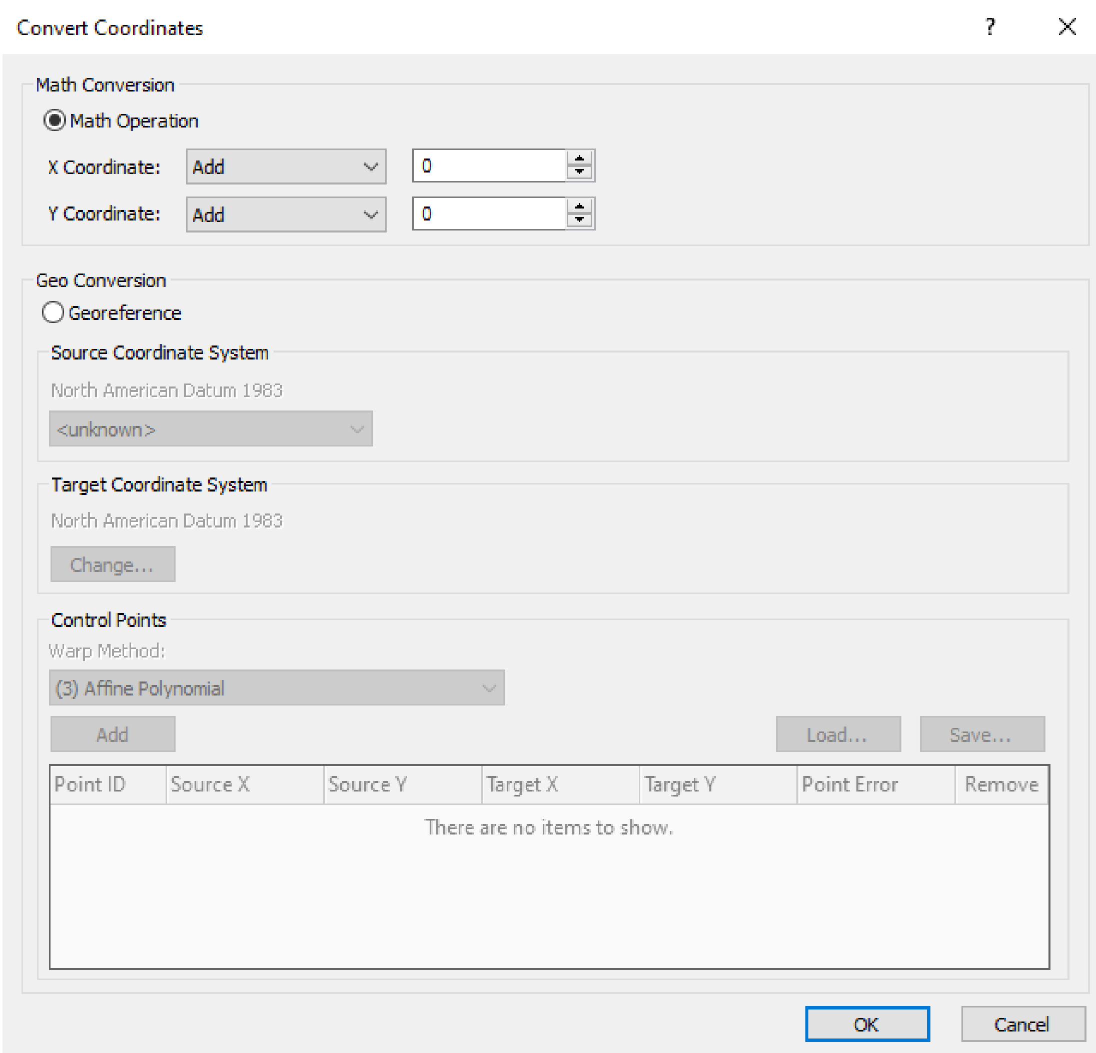
When vector objects or data points in a base layer are not spatially referenced correctly, the **Map Tools | Layer Tools | Convert Coordinates** feature can be used to reference the objects correctly.

This feature is primarily used to convert a base layer (either vector layer or data points layer) from an unknown coordinate system to either a known or an unknown system. For example, the feature can be used to convert from a local system to a UTM system. Or, the feature can be used to convert from one local system to another local system.

If you need to spatially reference an image being used as a base layer, the **Map Tools | Layer Tools | Georeference Image** can be used, instead.

Convert Coordinates Dialog

The **Convert Coordinates** dialog is opened by clicking **Map Tools | Layer Tools | Convert Coordinates** or by right-clicking the base layer in the **Contents** window and clicking **Convert Coordinates...**.



*In the **Convert Coordinates** window, either a Math Operation or Geo Conversion can be performed.*

The **Convert Coordinates** window contains two primary sections for the two kinds of coordinate conversions this feature can do, *Math Operation* and *Georeference*.

Math Operation

The *Math Operation* option applies a mathematical operation to the X and/or Y coordinates. This option can be used to adjust for offsets or errors in the data, like if the coordinates were off 50 meters in one direction. This option can also be used to convert units. For example, if units were measured in meters and they should be converted to feet, select the *Multiply by* operation and enter *3.28084* in the value field. The following math operations are available:

Add	Adds the value in the value field to the target coordinate(s)

Subtract	Subtracts the value in the value field from the target coordinate(s)
Multiply by	Multiplies the target coordinate(s) by the value in the value field
Divide by	Divides the target coordinate(s) by the value in the value field
Mirror	Mirrors the data across the midpoint of the target coordinate(s). When *Mirror* is selected, the value field is disabled.

Georeference

The *Georeference* option can be used to re-calibrate coordinate information for the layer based on at least three points on the map with known real-world coordinates and a chosen *Warp Method*. For example, if data was collected in a local coordinate system, it could be converted into a known geographic coordinate system.

The feature can be used by toggling the *Georeference* option, selecting a *Warp Method*, and clicking *Add* to add control points to the Base map. Once points have been added, target values for the X and Y coordinates can be entered in the *Target X* and *Target Y* fields. Once finished, click *OK* to complete the conversion.

Source Coordinate System

The **Source Coordinate System** is the coordinate system assigned to the layer being edited. This cannot be edited in the **Convert Coordinates** window. The linear units for the source data can be defined using the drop-down box. If the units for the source data are lat/long, the field should be left as *<unknown>*.

Target Coordinate System

The **Target Coordinate System** is the coordinate system the data will be converted to. The **Target Coordinate System** can be adjusted by clicking the *Change...* button. This opens the **Assign Coordinate System** dialog.

Control Points

The *Control Points* section of the window allows the *Warp Method* to be adjusted, control points to be added, removed, and edited, and georeferencing files to be loaded or saved.

Control points can be added manually or by loading a georeferencing file. To add points manually, click the *Add* button, bring your cursor over the desired area on the Base map, and left-click to add a point to that area. Points can also be added manually by clicking the *Add* button and manually entering a value for the *Source X* and *Source Y* fields. To add points using a georeferencing file, click the *Load...* button. In the **Open** dialog, navigate to your georeferencing file, select it, and click *Open*. The points are loaded automatically.

Control points can be moved by selecting the row for the desired point in the control points table and left-clicking on a new location of the map. Control points can also be moved by adjusting the Source X and Source Y values. Once the values are adjusted in the control points table, the control point marker on the Base map will move to the new location.

Control points can be deleted by clicking the icon in the *Remove* column of the control points table.

To save control points for future use, click the *Save...* button. In the **Save As** dialog, name the file, select a file path of your choosing, and click *Save*. The georeferencing file can be loaded for future conversions.

Control Points Table

The control point table contains information about the source and target coordinates for the control points. At least three control points are required to warp the image. Most warp methods require at least four control points, but there is no limit to the number of control points you can add. At least seven columns are displayed in the control point table. Each row represents a single control point. Any missing values or values that result in errors are highlighted in yellow in the control point table.

Point ID	Source X	Source Y	Target X	Target Y	Point Error	Remove
1	9593.79437	1459.15308	480000	142000	5.82076609E-11	
2	9678.06705	1486.49687	484000	148000	2.91038305E-11	
3	9742.16808	1424.63714	487500	142100	8.23180635E-11	

Enter the control point information in the control point table.

- The *Point ID* column contains the ID number for the control point. The ID helps you associate rows with points in the plot window.
- The *Source X* and *Source Y* columns contain the source coordinate of the control points in the base layer.
- The *Target X* and *Target Y* columns contain the desired coordinates of the control points. The *Target X* and *Target Y* columns are displayed when the base layer is assigned an unreferenced local coordinate system. Type the coordinate values for the control point in the *Target X* and *Target Y* fields.
- The *Point Error* column displays the RMS error value for each calibration point between the source image and referenced image.
- The *Remove* column hosts a button that can be clicked to remove a row from the table.

Click a cell to select it. Begin typing to replace the selected cell's contents. Double-click on a cell to enter cell edit mode and edit the current cell contents. When you are finished replacing or editing the cell's contents, press enter or click on another cell.

Warp Method

The warp method determines how the image is warped to the control points. Available options are: Affine Polynomial, First Order Polynomial, Second Order Polynomial, Third Order Polynomial, Thin Plate Spline, Natural Cubic Spline, Marcov Spline, Exponential Spline, Rational Quadratic Spline, and Inverse Distance Squared.

Click the **Warp Method** drop-down and click the desired method from the list to change the warp method. If too few points exist for a warp method, more control points must be added before the conversion can be completed. The number of required control points is displayed before the warp method name. *Affine Polynomial* requires at least three control points. *Second Order Polynomial* requires at least six control points. *Third Order Polynomial* requires at least ten control points. All other methods require at least four control points. Refer to each specific warp method topic for the transformation information and an image demonstrating the warp.

Applying Changes to the Map

When the steps for the desired coordinate conversion are completed, the conversion can be applied to the Map by clicking *OK*. If the coordinate conversion has changed the limits of the Map, you will need to either allow Surfer to update the limits of the map automatically or manually adjust the limits to new, appropriate values.

Undoing Changes to the Map

If you wish to revert the changes to the map after using the command, the changes can be undone with the plot window Undo command. After clicking the **Undo** command in the plot window, the map will revert back to its previous state.

Note, using the Reload Layer Data feature on a layer whose coordinates have been converted using this feature will return the layer to its original, unconverted state. This is because the layer source still references the original file. This file path can be updated by saving a copy of the new layer, either using **File | Export** or the **Save File** command for the layer, and then setting the *Input path* property on the General page for the base (vector) layer to the new file.

Attribute Table

The **Attribute Table** contains all the attributes for all the objects in a base layer. Use the attribute table to add, rename, or remove the attributes, or metadata, associated with the objects in a base map. The attribute table can also be used to edit attribute values for multiple objects without selecting the objects in the **Properties** window.

Opening the Attribute Table

A base layer or one of the objects in a base layer must be selected to open the attribute table.

To open the attribute table, click the **Map Tools | Layer Tools | Open Attribute Table** command or the button. Alternatively, right-click the base layer, or one of its child objects, and click **Open Attribute Table** in the context menu. When one of the objects in a base layer is selected before opening the attribute table, the attribute table is opened with the selected object highlighted.

The attribute table can be kept open as long as desired. It is unnecessary to close the attribute table to continue using **Surfer**. An attribute table can be open for each base layer in the document, but only one attribute table can be open for a each base layer.

Automatic Updating

Changes made in the attribute table are immediately displayed in the **Properties** window Info page. Changing the attribute value in the **Info** page automatically updates the attribute table. Changes and selections made in the Contents window are immediately displayed in the attribute table. Adding or deleting objects in the **Contents** window or plot window is reflected in the attribute table.

However, adding, renaming, and deleting attributes in the **Info** page is not reflected in the attribute table. Close and reopen the attribute table to see the changes made to attribute fields in the **Info** page.

Attribute and Object Order

The attribute table includes each object in the base layer and the attributes for each object. The objects are listed by row in the same order as in the **Contents** window. As mentioned above, changing the object order in the **Contents** window will automatically update the object order in the attribute table. Click on the column header once to sort the column in ascending order and twice in descending order.

Selected Object Indication

When an object is selected in the **Contents** window or plot window, the active object is highlighted by a sky blue background in the attribute table. When a cell is clicked in the attribute table, the associated object is selected in the window and plot window.

Undo and Redo

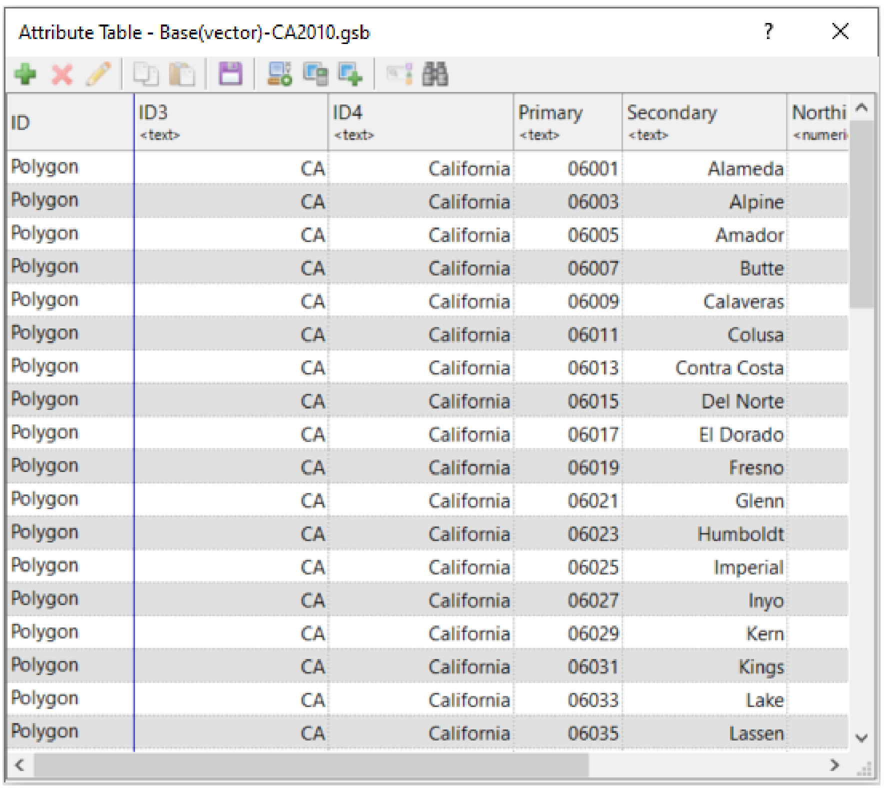
Click the **Home | Undo | Undo** command, button, or press CTRL+Z to undo the last action in the attribute table. Click the command, button, or press CTRL+Y to redo the last action in the attribute table.

Selecting Cells

Select a single cell by clicking once on the cell. Select multiple cells by clicking and dragging from one cell to another. Select non-adjacent cells by holding CTRL and clicking once on the cell. The selected cells are indicated with a blue background and white text.

Attribute Table Dialog

The attribute table consists of object rows, the *Id* column, and attribute fields. For each object and attribute, a cell displays the attribute value for the object.



ID	ID3 <text>	ID4 <text>	Primary <text>	Secondary <text>	Northi <numeri
Polygon	CA	California	06001	Alameda	
Polygon	CA	California	06003	Alpine	
Polygon	CA	California	06005	Amador	
Polygon	CA	California	06007	Butte	
Polygon	CA	California	06009	Calaveras	
Polygon	CA	California	06011	Colusa	
Polygon	CA	California	06013	Contra Costa	
Polygon	CA	California	06015	Del Norte	
Polygon	CA	California	06017	El Dorado	
Polygon	CA	California	06019	Fresno	
Polygon	CA	California	06021	Glenn	
Polygon	CA	California	06023	Humboldt	
Polygon	CA	California	06025	Imperial	
Polygon	CA	California	06027	Inyo	
Polygon	CA	California	06029	Kern	
Polygon	CA	California	06031	Kings	
Polygon	CA	California	06033	Lake	
Polygon	CA	California	06035	Lassen	

*Edit or add attributes and attribute values for each object in the **Attribute Table** dialog.*

Add Attribute Field

Click the button in the top left section of the dialog to add a new attribute field. The attribute field is also added to the **Info** page for all objects in the base layer. Clicking the button opens the New Attribute Name dialog. Type the new attribute name in the *Attribute name* field and click *OK*. A new attribute field can also be added by right-clicking on any node and selecting *Add Field* from the context menu.

Delete Attribute Field

Click the attribute field title or any node within the field, then click the button in the top left section of the dialog to delete the attribute field. The attribute field is also removed from the **Info** page for all objects in the base layer. Alternatively, delete an attribute field by right-clicking any node within the field and selecting *from the context menu.*

Rename Attribute Field

Click the attribute field title or any node within the field, then click the button in the top left section of the dialog to rename the attribute field. Clicking the button opens the **New Attribute Name** dialog. Type the new attribute name in the field and click *OK*. The attribute field is also renamed in the **Info** page for all objects in the base layer. Alternatively, rename an attribute field by right-clicking any node within the field and selecting *Rename field* from the context menu.

Copying and Pasting in the Attribute Table

Attribute values can be copied from the attribute table and pasted into **Surfer** or another program. Copy the selected cells by pressing CTRL+C. The **Home | Clipboard | Copy** command copies the selected object(s) in the plot window. The must be selected and the keyboard command must be used to copy values in the attribute table.

Text can be pasted into a single cell by clicking the button or pressing CTRL+V. The **Home | Clipboard | Paste** command pastes the clipboard contents in the plot window, not the attribute table.

Click the button or press CTRL+S to save the attributes to a data file in the Save As dialog.

Add Geometry Attributes

Click the button to add geometry attributes to the attribute table. Select the units and which geometry attributes are added to the attribute table in the Add Geometry Attributes dialog.

Calculate Attributes

Click the button to modify an existing attribute or create a new attribute with mathematical functions and existing attribute values in the Calculate Attributes dialog.

Join Attributes

Click the button to add attributes to the existing attribute table. As long as a single attribute is in common between the existing layer of attributes and the new file, **Surfer** will import all the columns from the new file. Select the file to import and attributes from the file in the Join Attributes dialog.

Query Objects

Click the button to Query Objects to select, deselect, move, copy, or exports objects that satisfy a user-defined query. Queries are performed on the objects in a single base layer.

Find and Replace

Click the button to find and replace text in the attribute table. The attribute table version of the Find and Replace dialog is displayed.

Id Column

The *Id* field displays the object name. The object name is edited with the Rename Object command. When an object is renamed, the object name is updated in the *Id* column of the attribute table. When objects are rearranged in the Contents window, the object order is updated in the attribute table. Object names cannot be edited in the attribute table.

Attribute Fields

Click once on a cell and type a value to change the attribute value for the object. Press ENTER to update the attribute value. Press one of the ARROW keys to update the attribute value and move the active cell selection to another cell.

Double-click on a cell to enter cell edit mode. While in cell edit mode the ARROW keys move the insertion point. The insertion point is where the next characters from the keyboard will appear. Type the desired value and press ENTER to change the attribute value. While in cell edit mode, you can paste text into the cell by pressing CTRL+V.

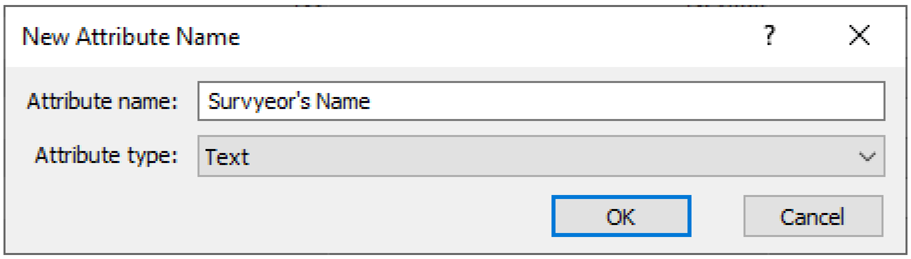
ZLEVEL

Surfer automatically links ZLEVEL attribute values and the Z coordinate values for Point features. Changing either the Z coordinate value or ZLEVEL attribute value for a point will automatically update the other value. If a Z coordinate is added to a point, a ZLEVEL attribute will be created for that point. However, the

Z coordinate value for 3D Polyline and 3D Polygon vertices is not linked to the ZLEVEL attribute, as each vertex can have its own Z coordinate value.

New Attribute Name Dialog

The **New Attribute Name** dialog is opened after the *Add* button is clicked in the *Attributes* section of **Properties** window Info Page or the button in the Attribute Table.



Enter the new attribute name in the ***New Attribute Name*** *dialog.*

Attribute Name

Type the name for the new attribute in the *Attribute name* field.

Attribute Type

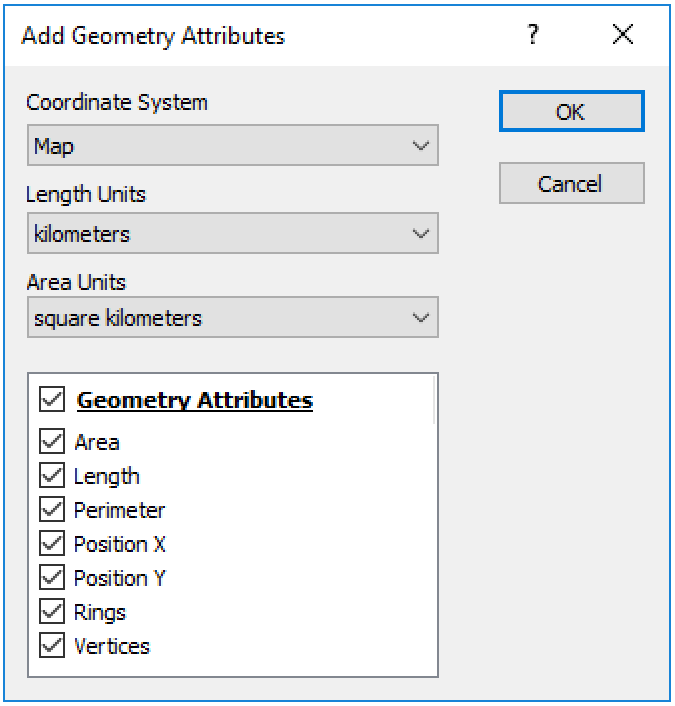
Specify the data type for the attribute in the *Attribute type* field. Select *Text* or *Numeric*.

OK or Cancel

Click *OK* and the new attribute is added to the object. Click *Cancel* and the **New Attribute Name** dialog is closed without adding a new attribute.

Add Geometry Attributes Dialog

Click the button in the Attribute Table to open the **Add Geometry Attributes** dialog. Select which attributes you wish to add and their units in the **Add Geometry Attributes** dialog.



*Select which attributes to add in the **Add Geometry Attributes** dialog.*

Coordinate System

Select the coordinate system to use for calculating the geometry values in the *Coordinate System* list.

- Select *Page* to use page units and paper space.
- Select *Local* to use the layer coordinate system.
- Select *Map* to use the map coordinate system.

Area, *Length*, and *Perimeter* attributes cannot be added if the selected *Coordinate System* uses spherical coordinates. If for example, your map layer uses the *World Geodetic System 1984* coordinate system, you must transform the map to a projected coordinate system and select *Map* in the *Coordinate System* list. Conversely, if your map uses a spherical coordinate system and your map layer uses a projected coordinate system, you must select *Local* in the *Coordinate System* list. *Position X*, *Position Y*, *Rings*, and *Vertices* attributes can be added regardless of the coordinate system type. Any attribute can be added with the *Page* coordinate system selected.

Length Units

Select the units for the *Length* and *Perimeter* attributes in the *Length Units* list. Options include *centimeters*, *chains*, *fathoms*, *feet*, *inches*, *kilometers*, *links*, *meters*, *millimeters*, *nautical miles*, *rods*, *survey feet*, and *yards*.

Area Units

Select the units for the *Area* attribute in the *Area Units* list. Options include *acres*, *hectares*, *square centimeters*, *square fathoms*, *square feet*, *square inches*, *square kilometers*, *square links*, *square meters*, *square miles*, *square millimeters*, *square nautical miles*, *square rods*, *square survey feet*, and *square yards*.

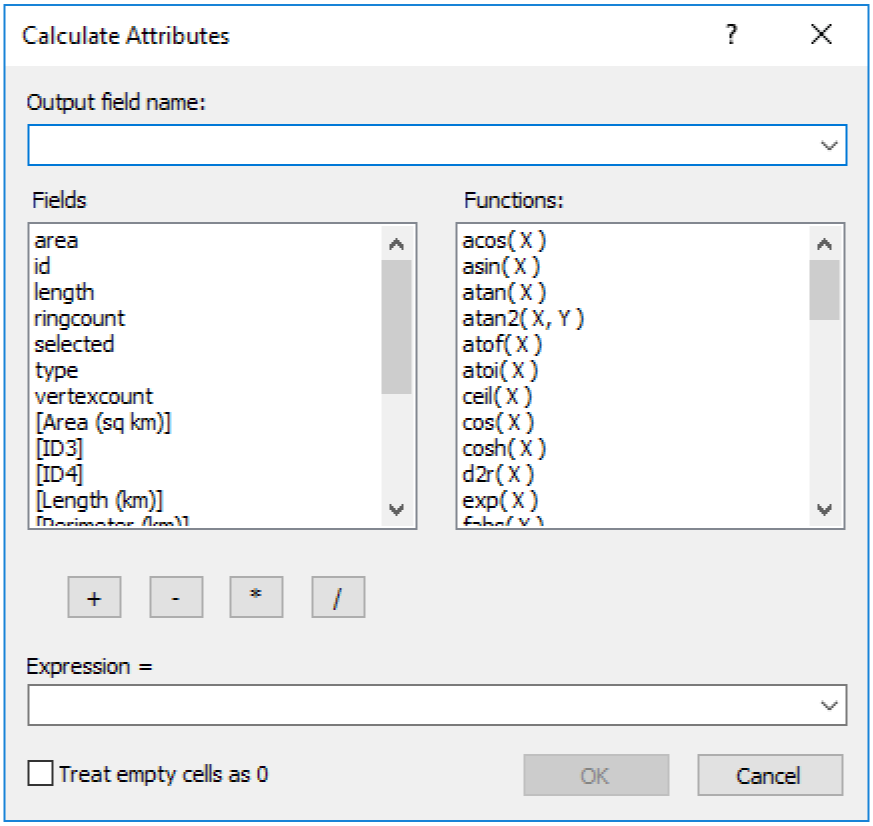
Geometry Attributes

Select which attributes to add to the **Attribute Table** in the *Geometry Attributes* section. The geometry attribute options include *Area*, *Length*, *Perimeter*, *Position X*, *Position Y*, *Rings*, and *Vertices*. Select the *Geometry Attributes* option to select all attributes. Clear the *Geometry Attributes* option to clear all attributes.

Attributes are added with the geometry name and units, if applicable. If an attribute with one of the names already exists, the created attribute will include an index value (1, 2, etc). Existing attributes will not be overwritten.

Calculate Attributes Dialog

Click the  button in the Attribute Table to open the **Calculate Attributes** dialog. Modify an exiting attribute or create a new attribute from a mathematical function an existing attributes in the **Create Attributes** dialog.



*Create a new attribute in the **Calculate Attributes** dialog.*

Output Field Name

To create a new attribute field, type the name for the new attribute in the *Output field name* box. To modify an existing attribute field, select an attribute from the *Output field name* list.

Fields and Functions

The *Fields* and *Functions* lists include attribute fields and mathematical functions that can be added to the *Expression*. Double-click an item in either list to add it to the *Expression*.

Operators

Click the add, subtract, multiply, or divide operator buttons to add the operator to the *Expression*.

Expression

The Expression is the function used to calculate the attribute values. The *Expression* is developed from the fields, functions, and operators.

Treat Empty Cells as Zero

Select the *Treat empty cells as 0* option to treat features with no attribute value as 0 when evaluating the *Expression* . Clear the *Treat empty cells as 0* option to skip any features when the *Expression* includes an attribute with no value.

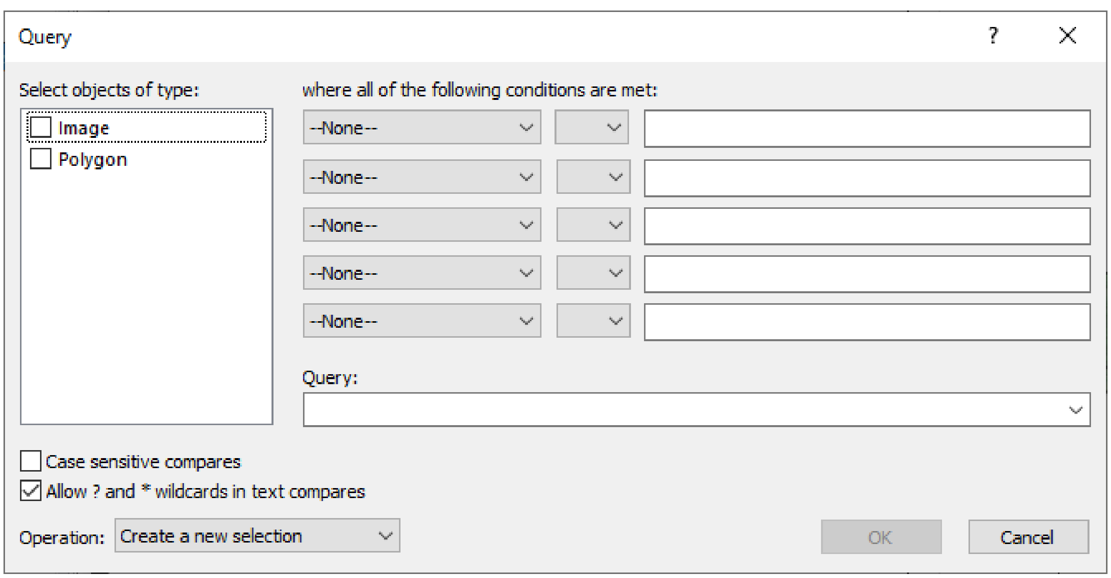
Query Objects

The **Map Tools | Layer Tools | Query Objects** command selects, deselects, moves, copies, or exports objects that satisfy a user-defined query. Queries are performed on the objects in a single base layer.

Click the **Map Tools | Layer Tools | Query Objects** command or the  button or right-click the base layer and click **Query Objects** to open the **Query** dialog and perform a query.

Query Dialog

Click the **Query Objects** command to create a query in the **Query** dialog. Create the query by selecting object types and specifying conditions using the controls in the dialog or by typing a query into the *Query* field. The controls and the *Query* field are synchronized, so control selections are reflected in the *Query* field. When possible, changes in the *Query* field are reflected in the controls. The controls and options are discussed in detail below.



Create a query with the ***Query*** *dialog by selecting object types and applying conditions or by typing a query into the Query field. Select options for text comparisons and the desired Operation for the objects returned by the query.*

Object Types

Specify the objects to query in the *Select objects of type* list. If you wish to query all object types, do not make any selections in the *Select objects of type* list. Click the check box next to an object type to add the object type to the *Query* field. When selecting multiple object types, the objects are combined with "or" and enclosed in parenthesis. The *Select objects by type* list contains the objects in the selected base layer. The base layer can contain one or more of the following object types: *Point*, *Polyline*, *Polygon*, *Spline Polyline* , *Rectangle*, *Rounded Rectangle*, *Ellipse*, *Text*, *Metafile*, and *Image.* Any number and combination of object types can be selected.

Control Clauses

Each of the five lines in the *where all of the following conditions are met* section is an independent control clause. Use the control clauses to further refine the object selection returned by the **Query Objects** command. Each clause consists of a *Field*, *Operator*, and *Value* from left to right. Specify a control clause by making a selection in the *Field* and *Operator* lists and typing a value into the *Value* field. By default multiple clauses are combined with the "and" logical operator. You can manually change the AND operator to another logical operator, such as OR, by typing into the *Query* field.

The *Field* list contains items and attributes that can be queried, and it includes: *Type*, *ID*, *Length*, *Area*, *VertexCount*, *RingCount*, and all attributes in the Attribute Table. *Length* is the length of a polyline or perimeter of a polygon. Select *--None--* to clear the *Operator* and *Value* fields, and remove the clause from the query. The selection is added to the *Query* field when it is clicked in the*Field* list.

The *Operator* compares the values returned by the *Field* selection and the specified *Value*. Choose equals (=), does not equal (*!*=), less than (<), less than or equal to (<=), greater than (>), or greater than or equal to (>=) from the *Operator* list. The selected *Operator* is added to the *Query* field.

The *Value* is the user-specified number or text to used in the query constraints. Mathematical Functions can also be used in the *Value* field. Numbers and text typed in the *Value* field are added to the *Query* field. Text is added with double quotes, so enclosing text with quotes in the *Value* field is not necessary.

Specify up to five constraints for the query using the control clauses. If more than five constraints are needed, or if more complex comparisons need to be performed, generate the query by typing directly into the *Query* field.

Query Field

The *Query* field contains the query used to return objects and perform the specified *Operation*. Click the arrow to view or select one of the last ten queries. Typing a query updates the *Select objects of type* and *where all of the following conditions are met* selections, when available. Complex queries and queries requiring more than 5 constraints must be created by typing in the *Query* field. Also, AND operators generated by the control clauses can be changed manually in the *Query* field. Mathematical Functions can be used in queries. The following list explains the syntax rules for queries:

- Text must be enclosed in double quotes. For example, type="Ellipse" or ID="Denver"
- User-defined attributes must be enclosed in square brackets. For example, [dip]=60
- The application-defined attributes *Type*, *ID*, *Length*, *Area*, *VertexCount*, and *RingCount* are not enclosed in square brackets.

Query Options

The *Case sensitive compares* and *Allow ? and * wildcard in text compares* options control how text is compared in the query.

- When the *Case sensitive compares* check box is checked, the text returned by the *Field* selection and the specified text in the *Value* field must have the same case.
- When the *Allow ? and * wildcard in text compares* check box is checked, a *?* can be used to represent any single character and an * can be used to represent a group of characters.

Operation

Select the desired action for the objects returned by the query in the *Operation* list.

- *Create a new selection* selects only the objects returned by the query.
- *Add to current selection* adds the objects returned by the query to the currently selected objects.
- *Remove from current selection* removes the returned objects from the current selection.
- *Select from current selection* selects the objects returned by the query that are part of the current selection. Objects not included in the current selection will not be selected with *Select from current selection*, even if the objects outside the current selection satisfy the query.
- *Copy to new layer* copies the objects returned by the query to a new empty base layer. A warning is displayed and no layer is created if no objects are returned by the query.
- *Move to new layer* moves the objects returned by the query to a new empty base layer. A warning is displayed and no layer is created if no objects are returned by the query.
- *Export to file* exports the objects returned by the query to a new file. Specify the path, file name, and file type in the Export Query Results dialog. A warning is displayed and no file is created if no objects are returned by the query. To maintain the spatial references for the objects, change the *Scaling source* to the map in the **Export Options** dialog Scaling page when saving the file.

OK and Cancel Buttons

Click *OK* to perform the query. Click *Cancel* to close the **Query** dialog without performing the query.

Track Cursor

Cursor tracking provides connections between the worksheet, plot windows, and grid node editor windows. Tracking occurs between multiple maps in the plot window. Links are graphical connections that connect what you see on a map to the same location on another map, and to the original data in the worksheet or grid in the grid node editor. This can be useful for finding bad data points, and for showing the same point on multiple maps that have the same X and Y data range.

Track Cursor in the Plot Document

The **Map Tools | Layer Tools | Track Cursor** command allows you to enable or disable the cursor tracking between multiple maps. Tracking is done on the current map frame's coordinate system. Left-click a map to see the location of the cursor click on additional maps. The cursor tracking is updated with every left-click.

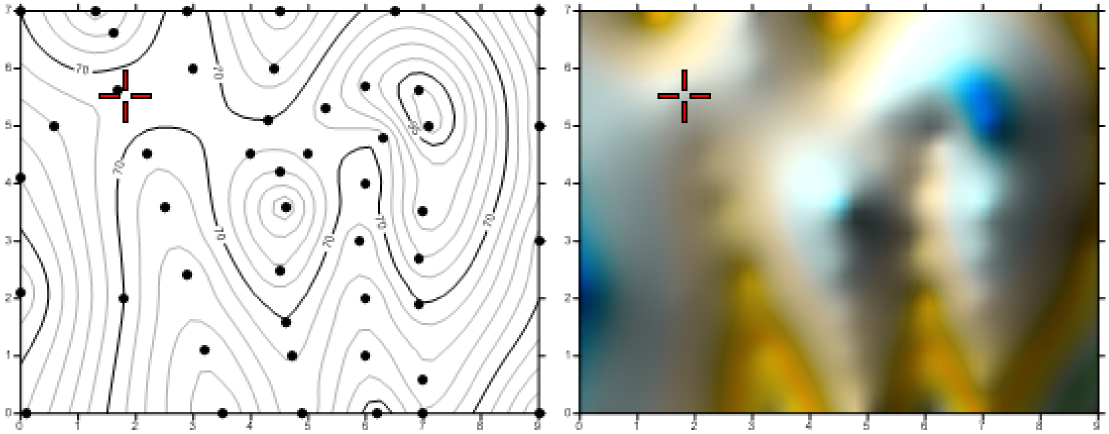
This command can also be used to link map layers to the worksheet or grid. Click a location in the plot window, and the tracking cursor will be displayed in the same location in the plot window and grid editor. The closest point in the worksheet will be highlighted.

3D surface and 3D wireframe maps cannot be used as the source map for cursor tracking. A 3D map can be rotated to a birds eye view (field of view = 45° , rotation = 45° , tilt = 90°) to see the tracked cursor in the correct location.

Example in the Plot

1. Click the **Home | New Map | Contour** command or the button.
2. In the **Open Grid** dialog, select the *Demogrid.grd* sample file (located in the Samples folder), and click *Open*. The contour map is created.
3. Click the **Home | New Map | Color Relief** command.
4. In the **Open Grid** dialog, select the *Demogrid.grd* sample file (located in the Samples folder), and click *Open*. The color relief map is created.
5. Click on the color relief map and drag it to the right to arrange the maps to be side by side.
6. Click the **Map Tools | Layer Tools | Track Cursor** command or the button to enable cursor tracking. A check mark next to **Track Cursor** indicates cursor tracking is on.
7. With the track cursor command enabled, left-clicking on one map will display the tracking cursor on the other map in the appropriate location.

8. To disable cursor tracking, click the **Map Tools | Layer Tools | Track Cursor** command.



The contour map (left) and color relief map (right) were both created with the sample file, Demogrid.grd. With the track cursor command on, clicking on one map, displays the tracking cursor on the other map in the appropriate location.

Track Cursor in the Worksheet Document

The **Data | Tools | Track Cursor** command allows you to enable or disable the cursor tracking between worksheet data, maps in the plot window and grids in the grid node editor. Select cells in the worksheet to position the tracking cursor on the map or grid. Alternatively, in the plot window or grid node editor, click on a location that contains the same XY range as the data file. The closest data point to the click will be highlighted in the worksheet.

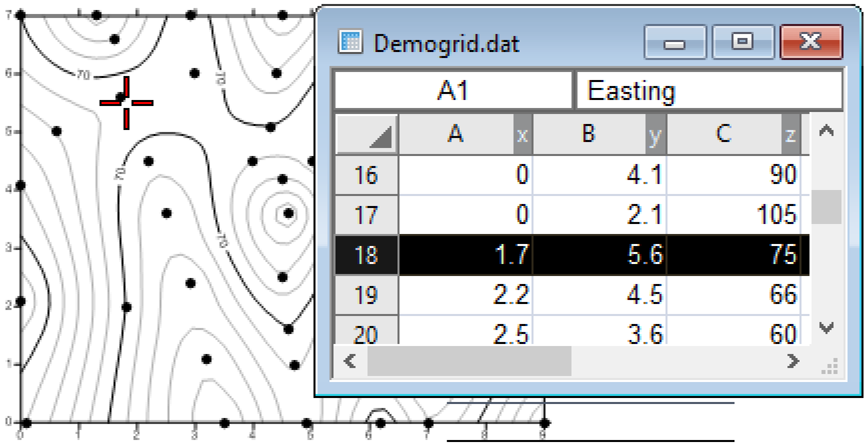
Cursor tracking must be enabled, and the worksheet containing the selected row must have the X and Y data in the current X and Y columns. By default this is column A for the X coordinates, and column B for the Y coordinates. If your X and Y post map data columns are not in the default columns, click the Assign XYZ Columns command to assign new columns.

The coordinates in the worksheet row must be included within the map extents for the cursor to appear in the plot window.

Example in the Worksheet

1. Using the example from above, select the contour map.
2. Click the **File | Open** command to open the *Demogrid.dat* file. The data file is displayed in the worksheet.
3. Click the **Data | Data | Assign XYZ Columns** command to ensure the X and Y coordinate columns are specified correctly. Make changes if necessary.
4. Click the **Data | Tools | Track Cursor** command to enable cursor tracking.
5. Click on a cell in the worksheet. The tracking cursor will move to the selected location on the maps in the plot window. Use the tabs to toggle between the plot and worksheet.

6. Alternatively, click on the map in the plot window and the closest data point will be highlighted in the worksheet.



In the plot, click near a post point.

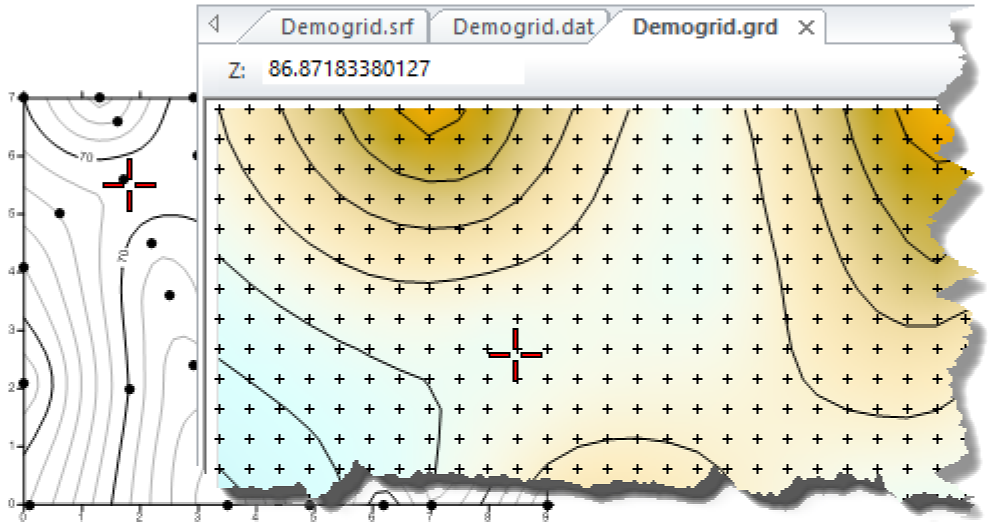
In the worksheet, the nearest post point is highlighted.

Track Cursor in the Grid Editor

The **Grid Editor | Options | Track Cursor** command allows you to enable or disable the cursor tracking between worksheet data, maps in the plot window and grids in the grid editor. Click in the grid editor to position the tracking cursor. When clicking in the grid editor, the nearest grid node will also be selected. The tracking cursor is displayed in the same XY location on the map, and the closest point in the worksheet is highlighted. Click in the plot window or worksheet, and the tracking cursor will be displayed at the same XY location in the grid editor. However when clicking in the plot window or worksheet, the selected node will not change.

Example in the Grid Editor

1. Using the example from above, select the contour layer.
2. Click the **Map Tools | Edit Layer | Grid** command.
3. Click the **Grid Editor | Options | Track Cursor** command to enable cursor tracking.
4. Click on the grid in the grid editor. The tracking cursor will move to the selected location in the grid editor and plot window. The nearest grid node will be selected if the Select tool is active. Use the tabs to toggle between the plot and grid editor.
5. Next, click on the map in the plot window. The tracking cursor will move to the new selected location. Use the tabs to toggle between the plot and grid editor. Click the tracking cursor in the grid editor if you wish to select the nearest grid node.



Click in the map in the plot window.

In the grid node editor, the tracking cursor moves, but the selected node remains the same.

Export Contours

To retain the Z information for contour lines for use in other mapping programs, click the **Map Tools | Layer Tools | Export Contours** command or the button. The **Export Contours** command exports the currently selected contour map as an AutoCAD .DXF, 2D Esri .SHP, 3D Esri .SHP, or a .TXT text format.

Surfer prompts to save the coordinate system and projection information, if one is specified for the map. The Export Options dialog appears with the option to save the coordinate system information. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in Surfer, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original data file, but the .GSR2 is required to define the coordinate system.

Export vs Export Contours

The File | Export command can be used to export Z information to an attribute field for BLN, BNA, DXF, GSB, GSI, KML, KMZ, MIF, and SHP files. The **Export** command can also export 2D or 3D BLN, DXF, SHP, or XYZ Points files. However, the **Export** command will also export any other visible items in the map, including contour labels. The **Export Contours** command automatically removes the contour labels from the output and closes the gaps where the labels were, and the **Export Contours** command only exports the contours lines, regardless of which items are visible in the map.

Export File Types

The following file types are available for export when using the **Export Contours** command.

AutoCAD DXF

The contours are saved as elevated polylines. The POLYLINE entities in the DXF have X and Y coordinates for each vertex, and the Z coordinate is stored as an attribute for the lines. The polylines are clipped to the map limits and scaled to the current map scale. Axes, labels, contour fills, line properties (width, style, and color), etc. are not exported. The contours are not affected by 2D page transformations or rotations. The coordinates in the AutoCAD .DXF file are saved with 64-bit floating-point precision.

Use the **File | Export** command with the *Write LWPOLYLINE entities even when 3D data is available* export option cleared to export the contours as POLYLINE entities with the Z values written for each vertex. Use the **File | Export** command with the *Write LWPOLYLINE entities even when 3D data is available* export option checked to export the contours as LWPOLYLINE entities with the Z values as an attribute for each line.

2D SHP

The 2D .SHP exports a shape 2D polyline type where each vertex along the line comprises an X and Y coordinate. The Z coordinate for each vertex is stored in the associated .DBF file.

3D SHP

The 3D .SHP exports a shape polyline Z type where each vertex along the line comprises an X, Y, and Z coordinate. The Z coordinate for each vertex is also stored in the associated .DBF file.

Text Format

The text format .TXT exports each contour line as a series of X, Y, Z points. Each point is on a separate line in the .TXT file. This file type is similar to the DAT or CSV XYZ Points file type.

To export 3D contour lines

1. Create a contour map by clicking the **Home | New Map | Contour** command. Select the .GRD file and click *Open*.
2. Select the contour map by clicking on *Contours* in the Contents window. It does not matter if the contour map is overlaid with other maps as long as only the contour map is selected.
3. Click **Map Tools | Layer Tools | Export Contours** to display the **Save As** dialog. Enter the *File name*, specify the *Save as type*, and click *Save*. The file

is exported with the specified extension. The selected file format and a .GSR2 and a .PRJ file are created.

If you would like to export the contours as part of a complete map, choose one of the export options in File | Export.

Chapter 27 - Coordinate Systems

Coordinate Systems

A coordinate system is method of defining how a file's point locations display on a map. Different types of coordinate systems exist that control how the coordinates are shown on the map. In **Surfer**, a map can be unreferenced in local coordinates, referenced to a geographic latitude and longitude coordinate system, or referenced to a known projection and datum. Each data set, grid, map layer, and the map frame can have an associated coordinate system. All coordinate systems for individual layers are converted "on the fly" to the map's target coordinate system. This allows maps with different coordinate systems to be easily combined in **Surfer**.

A local coordinate system generally is considered unreferenced. A local system has a location that begins numbering at an arbitrary location and increments numbers equidistant in the X and Y directions from this location. This is frequently referred to as a *Cartesian coordinate system*. The distance units can be specified for an unreferenced local system in the **Assign Coordinate System** dialog.

A *Geographic* coordinate system uses a spherical surface to define locations on the earth. Geographic coordinate systems are commonly called unprojected lat/long. **Surfer** has several predefined geographic coordinate systems available. Each system has a different datum. The same latitude and longitude value will plot in different locations depending on the datum.

A *Projected* coordinate system consists of a projection and a datum. Each projection distorts some portion of the map, based on the ellipsoid and datum specified. Coordinates can be lat/long, meters, feet, or other units. Different projections cause different types of distortion. It is recommended that you do not use projected coordinate systems if you do not need to convert between coordinate systems or if all your data are in the same coordinate system.

Map Coordinate System Overview

In **Surfer**, data, grids, map layers, and maps can have an associated coordinate system. All coordinate systems defined by the data, grids, and map layers are converted "on the fly" to the map's target coordinate system. This allows maps with different coordinate systems to be easily combined in **Surfer**.

The standard procedure for creating maps in a specific coordinate system is as follows:

1. Create the map by clicking on the appropriate **Home | New Map** command.
2. In the **Open Grid(s)** dialog, select the file to open and click *Open*.

3. In the **Contents** window, click on the map layer to select it.
4. In the **Properties** window, click on the **Coordinate System** page.
5. If the *Coordinate system* is not correct, click the *Set* button next to *Coordinate System*. The **Assign Coordinate System** dialog opens. This is the initial coordinate system for the map layer, i.e. the coordinate system for the source data. Select the correct coordinate system in the dialog. When finished making changes, click *OK*.
6. To change the target coordinate system for the map, click on the *Map* object in the **Contents** window. In the **Properties** window, click on the **Coordinate System** tab. This is the coordinate system in which you want the map to display.
7. Click on the *Change* button next to *Coordinate System* to set the desired target coordinate system. When finished, click *OK*.
8. All of the map layers are converted on the fly to the target coordinate system. The entire map is now displayed in the desired coordinate system.

Surfer does not require a map coordinate system be defined. Maps can be created from unreferenced data, grid, and map layers. As long as all map layers have the same X and Y ranges, coordinate systems do not need to be specified. If you do not specify a source coordinate system for each map layer, it is highly recommended that you do not change the target coordinate system. Changes to the target coordinate system for the map can cause the unreferenced map layers to appear incorrectly or to not appear.

3D surface maps and wireframe maps cannot be converted to a new coordinate system.

Source Coordinate System Properties

Maps can be created from data, grids, or base map files in any coordinate system. The *Source Coordinate System* is the coordinate system for the original data, grid, or base map used to create a map layer. Each map layer can reference a different projection and datum. If some map layers are using a different source coordinate system than what you want the map to display, the map layer is converted to the map's *Target Coordinate System*.

3D surface maps and wireframe maps do not have an associated coordinate system and cannot be converted to a different coordinate system. When a layer with a coordinate system is overlaid onto either a surface or wireframe map, the layer's coordinate system is removed and the layers are displayed in Cartesian coordinates.

When a surface map or wireframe map is created first, the *Map* object and *Surface* layer or the *Map* object and *Wireframe* layer are set to an unreferenced coordinate system. Any map layers subsequently added to the *Map* must be in the same units as the surface or wireframe layer.

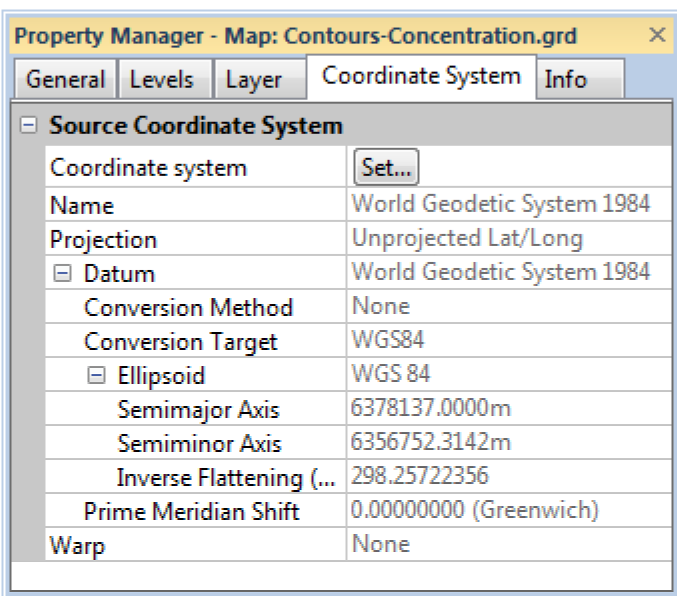
When a surface layer or wireframe layer is added to an existing map, the target coordinate system of the existing map must first be set to the same coordinate system as the surface or wireframe map to be added. To determine the coordinate system for the surface or wireframe layer, you may need to create a contour map from the grid file, first. Match the *Map* coordinate system to this grid file. Then, add the surface map or wireframe map.

When a vector map coordinate system is changed, only the location of the vectors changes. The components of the grids, including the direction of the vectors, are not projected. It is assumed that the grid components are in the map coordinate system units.

The target coordinate system can be applied to the *source coordinate system* for all unreferenced map layers. The *Assign to unreferenced layers* operation is located in the **Coordinate System** page of the *Map* object properties. For example, you may have overlaid map layers all created from the same unreferenced local system with specific units. You can create a new local coordinate system to define the units for the map, and apply the new system to the map's *target coordinate system*. Next use *Assign to unreferenced layers* to assign the custom local coordinate system (the map's *target coordinate system*) to the *source coordinate system* for all the map layers simultaneously.

The Coordinate System Page

The **Coordinate System** page is located in the Properties window when a map layer object is selected.



*Specify the map layer coordinate system on the **Coordinate System** page.*

Coordinate System

Click the *Set* button next to *Coordinate system* to open the Assign Coordinate System dialog. This dialog lets you set the source coordinate system. This is the coordinate system for the original data, grid, or base map.

Target Coordinate System Properties

Maps can be displayed in any coordinate system. The map is displayed in the coordinate system defined as the *Target Coordinate System*. When a map layer uses a different *Source Coordinate System* than the map's *Target Coordinate System*, the map layer is converted to the map's *Target Coordinate System*.

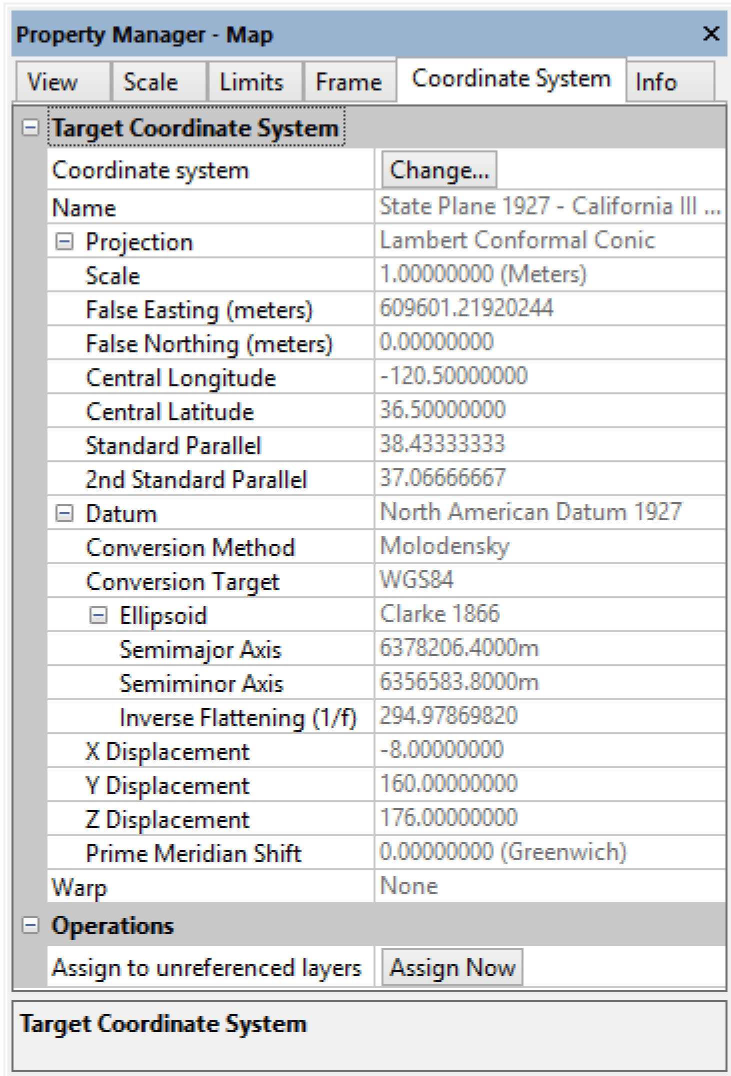
3D surface layers and wireframe layers do not have a coordinate system associated with them and cannot be converted to a different coordinate system. For this reason, there is not a **Coordinate System** tab in the **Properties** window for the surface layer, the wireframe layer, the *Map* object, or any other map layer when either a surface layer or wireframe layer exists in the *Map*.

When a surface map or wireframe map is created, the *Map* object and *Surface* layer or the *Map* object and *Wireframe* layer are set to an unreferenced coordinate system. Any map layers subsequently added to the *Map* must be in the same units as the surface layer.

When a surface layer or wireframe layer is added to an existing map, the coordinate system of the existing map must first be set to the same coordinate system as the surface or wireframe map to be added. To determine the coordinate system for the surface or wireframe layer, you may need to create a contour map from the grid file, first. Match the *Map* coordinate system to this grid file. Then, add the surface map or wireframe map.

Coordinate System Page

The **Coordinate System** page is located in the Properties window when a *Map* object is selected.



Specify the map coordinate system on the Coordinate System page.

Coordinate System

Click the *Change* button next to *Coordinate system* to open the Assign Coordinate System dialog. This dialog lets you set the desired target coordinate system. This is the coordinate system in which you want the map to be displayed.

Assign to Unreferenced Layers

Click the *Assign Now* button in the *Assign to unreferenced layers* field to apply the map target coordinate system to the source coordinate system for all map layers with an unreferenced coordinate system. The *Assign to unreferenced layers* operation only changes the *source coordinate system* for current map layers. If more map layers are added with unreferenced coordinate systems, then the

Assign Now button must be clicked again to apply the target coordinate system to the new map layers.

Displaying Data with Different Coordinate Systems in a Single Map

Surfer allows you to display map layers from any coordinate system in a single map. To do so, you need to assign the coordinate system for each map layer. Then, change the desired target coordinate system for the map.

The standard procedure for creating maps in a specific coordinate system are:

1. Create the map by clicking on the appropriate **Home | New Map** command.
2. Click on the map layer to select it. In the Properties window, click on the Coordinate System tab.
3. If the *Coordinate System* is not correct, click the *Set* button next to *Coordinate System* . The **Assign Coordinate System** dialog opens.
4. Make any changes in the dialog. This is the existing coordinate system for the map layer. When finished making changes, click *OK*.
5. Add the second map layer to the map using the appropriate **Map Tools | Add to Map | Layer** command.
6. After the second map layer appears, click once on it to select it. In the **Properties** window, click on the **Coordinate System** tab.
7. If the *Coordinate System* is not correct, click the *Set* button next to *Coordinate System* . The **Assign Coordinate System** dialog opens.
8. Make any changes in the dialog. This is the existing coordinate system for the second map layer. This coordinate system can be different from the first map layer and has no effect on the first map layer. When finished making changes, click *OK*.
9. To change the coordinate system for the map, click on the Map object. In the **Properties** window, click on the Coordinate System tab.
10. If the *Coordinate System* is not correct, click on the *Change* button next to *Coordinate System* to set the desired target coordinate system. When finished, click *OK*.
11. The entire map is now displayed in the desired target coordinate system.

Coordinate System Notes

Currently, imported metafiles do not change coordinates. When the coordinate system changes from the source coordinate system, the metafile is removed from the map view. When the coordinate system changes back to the metafile's coordinate system, the metafile reappears. To have metafile coordinate system changed, the metafile needs to be broken apart. To do this, click once on the metafile to select it. Click the **Features | Group | Ungroup** command. The metafile becomes a collection of lines, text, polygons, and images.

3D surface layers and wireframe layers do not have a coordinate system associated with them and cannot be converted to a different coordinate system. For this reason, there is not a **Coordinate System** tab in the **Properties** window for the surface layer, the wireframe layer, the *Map* object, or any other map layer when either a surface layer or wireframe layer exists in the *Map*.

When a surface map or wireframe map is created, the *Map* object and *Surface* layer or the *Map* object and *Wireframe* layer are set to an unreferenced coordinate system. Any map layers subsequently added to the *Map* must be in the same units as the surface layer.

When a surface layer or wireframe layer is added to an existing map, the coordinate system of the existing map must first be set to the same coordinate system as the surface or wireframe map to be added. To determine the coordinate system for the surface or wireframe layer, you may need to create a contour map from the grid file, first. Match the *Map* coordinate system to this grid file. Then, add the surface map or wireframe map.

Text is stretched slightly in the horizontal and vertical directions (before rotation) so that the text will occupy the same overall space as it did before being geotransformed. This can cause the individual characters to be spaced slightly closer together or further apart than the characters appeared before the coordinate system conversion. The text is still in the correct geographic location as it appeared before the transformation.

Coordinate System Frequently Asked Questions

Q: Do I need to specify a coordinate system for every map layer in Surfer?

A: You do not need to specify a coordinate system for your map layers in **Surfer** if you are not going to be converting between different coordinate systems or displaying maps from different coordinate systems. If all your map layers are in the same system, you do not need to specify the coordinate system.

However if all map layers are in the same coordinate system and you wish to specify the coordinate system, the map's Target Coordinate System can be applied to the layers' Source Coordinate Systems simultaneously with the *Assign now* button in the **Properties** window.

Q: Why would I want to set the coordinate system for a map?

A: If your data, grids, and base maps are in different coordinate systems, you will want to set the coordinate system for each map layer and the entire map. If you want to change the projection of your data, grid, or base map, you will want to set the coordinate system.

Q: Is there an easy way to define the coordinate system for my map and all layers at once?

A: Yes, but only when the unreferenced map layers all use the same coordinate system. If applicable, define the Source Coordinate System for any layers that do not share the same coordinate system as the other layers. Next verify that the *Source Coordinate System* is set to *Unreferenced local system* for the layers that you wish to set to the same coordinate system. Finally, set the Target Coordinate System for the map to the coordinate system shared by the unreferenced layers, and then click the *Assign now* button in the **Properties** window.

Q: How do I set the coordinate system for a data file?

A: Set the data file coordinate system using the worksheet's Data | Coordinate System | Assign Coordinate System command. The spatial reference information is saved in an external GSR2 file.

Q: How do I set the coordinate system for a grid?

A: Set the grid file coordinate system using the Grids | Edit | Assign Coordinate System command. The spatial reference information is saved in an external GSR2 file.

Q: If my data file has a coordinate system defined, do I need to define a different coordinate system for the grid?

A: If the grid was created from the data file after the coordinate system was defined, you do not need to define a coordinate system for your grid separately. **Surfer** automatically carries the coordinate system definition from the data to the grid and to any maps created from the data or grid.

Q: My map disappears after defining a coordinate system! Now what can I do?

A: Most likely, the map layer that disappeared did not have a defined coordinate system. In the **Contents** window, click on the map layer that has disappeared. In the **Properties** window, click on the **Coordinate System** tab. Click the *Set* button and define the map layer's coordinate system. Click *OK*. The map should now appear correctly. See the Source Coordinate System page for more information.

Q: My axes do not show the correct coordinates. How can I make them show the coordinates I want?

A: If the map axes are showing incorrect coordinates, you may need to change your Target Coordinate System. Click on the *Map* object in the **Contents** window. In the **Properties** window, click on the **Coordinate System** tab. Click the *Change* button to define the map's target coordinate system. After making the change, click *OK*. The map axes will now appear in the desired coordinates.

Q: When I import a map from another program, does it import the projection information?

A: If the map contained a reference file, **Surfer** will automatically read the reference file and apply any information it finds to the map layer.

Q: When I change the target coordinate system, my map limits change.

Surfer retains the map size, but the limits change when the target coordinate system changes.

Q: I need additional information about coordinate systems. Do you have any references that can help?

A: A good starting place is the references on the Projection References page. Many other good resources are available either online or in printed documentation.

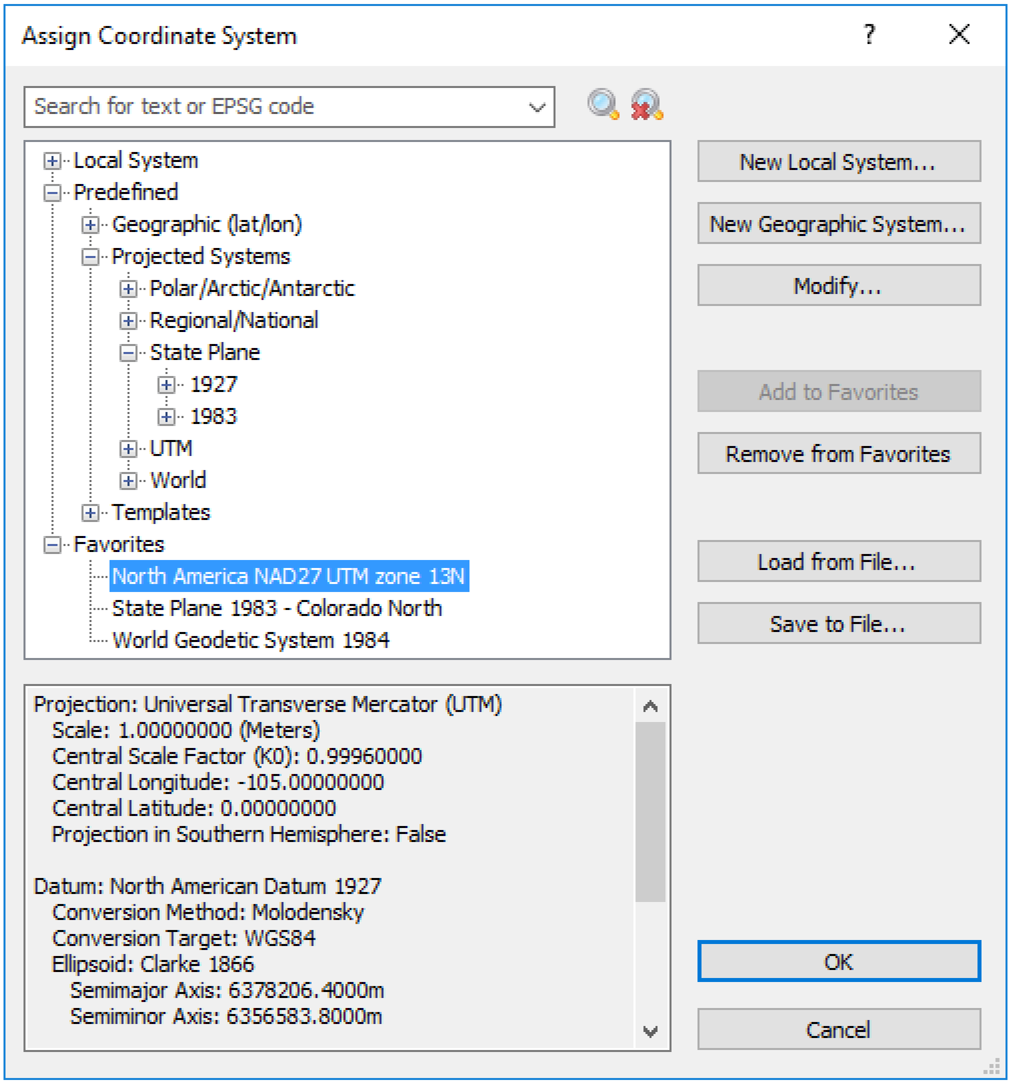
Assign Coordinate System

The **Assign Coordinate System** dialog is accessed from several locations. Often it is opened by clicking the *Set* button in the layer Coordinate System **Properties** page or the *Change* button in the map Coordinate System **Properties** page. It links a file, map layer, or map to a specific coordinate system. Once a coordinate system is defined for a file, a Golden Software Georeference .GSR2 file is created. This file contains all the relevant coordinate system information that **Surfer** needs to load the file in the proper coordinate system in the future.

When a .GSR2 file is created for a data file, it is read when creating grids or maps from the data file. The resulting grid or map layer has the same coordinate system as the original data file. The projection information can be saved with the grid file using the Spatial References options. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in **Surfer**, as the GSR2 retains all of the information needed.

Assign Coordinate System Dialog

Assign a coordinate system to a map, layer, or file in the **Assign Coordinate System** dialog.



Assign a projection to your file or map layer in the ***Assign Coordinate System*** *dialog.*

Search for Coordinate Systems

Surfer has over 2500 coordinate systems in the **Assign Coordinate System** dialog. To search for a specific coordinate system type a partial name, complete name, or EPSG code into the *Search for text or EPSG code* field. Next click the button or press ENTER to search for the coordinate system. The number of returned search results will be displayed below the search bar. The search results will replace the full coordinate system list. Navigate through the search results by clicking the ⊞ button to expand the categories in the **Assign Coordinate System** dialog. Click the button to clear the search results and display all of the coordinate systems in the **Assign Coordinate System** dialog.

When searching in the **Assign Coordinate System** dialog, the search string must exactly match a portion of the desired coordinate system name or EPSG code. However, the search string does not need to be the complete name or EPSG code. For example, searching for *System 1984* will return the *World Geodetic System 1984* coordinate system, but searching for *World 1984* returns no results.

Projection Categories

Click the ⊞ button to expand the options in the **Assign Coordinate System** dialog. Click the ⊟ button to collapse the options.

Local System

Expand *Local System* to select the *Unreferenced local system,* which contains a *Projection* of None, a *Datum* of None, and a *Warp* of None. For example, you may have a data set with an arbitrary coordinate system (i.e. not real world X, Y coordinates). You would assign this data to a *Local System* , if a coordinate system is necessary.

Predefined

Expand the *Predefined* section by clicking the + button. The *Predefined* section includes all coordinate systems that have been predefined for **Surfer**.

Geographic (lat/lon)

Expand *Geographic (lat/lon)* to select a Latitude/Longitude coordinate system and datum that fits your needs. Detailed information about each projection is listed at the bottom of the dialog when the system is selected. If your data are currently in a form of lat/lon, you would want to select one of the options in *Geographic (lat/lon)*.

Projected Systems

Expand *Projected Systems* to select a predefined *Polar/Arctic/Antarctic, Regional/National, State Plane, UTM,* or *World* coordinate system. Detailed information about each projection is listed to the right when the datum is selected.

Templates

Expand the *Templates* section to select a predefined template. Click the desired template and press the *Modify* button to change the properties. Once modified, the new coordinate system is added to the *Custom* section.

Favorites

Select a coordinate system and click the *Add to Favorites* button to add a coordinate system to your *Favorites* list. Alternatively, right-click on a selected coordinate system and choose *Add to Favorites* from the menu. Select a coordinate system and click the *Remove from Favorites* button to remove a coordinate system from your *Favorites* list. By default, no favorite coordinate systems are specified. It is recommended that systems that you use frequently and *Custom* systems that you intend to use more than once be added to the *Favorites* section for ease of navigation.

Custom

Expand *Custom* to see the custom coordinate system you have defined for the current file. *Custom* systems can be defined by clicking the *New* button. By default, no custom coordinate systems are specified. Custom coordinate systems are only listed in the dialog when the file using the custom coordinate system is open.

If a *Custom* coordinate system is defined and intended to be used more than once, it is highly recommended that the system be added to the *Favorites* section by clicking the *Add to Favorites* button.

New Local Coordinate System

The linear units and offsets can be specified for local coordinate systems. Click the *New Local System* button to open the Define Unreferenced Coordinate System dialog and define a custom local unreferenced coordinate system. Alternatively, click the *Modify* button while *Unreferenced local system* is selected in the *Select coordinate system* list to open the **Define Unreferenced Coordinate System** dialog.

New Geographic Coordinate System

Click the *New Geographic System* button to open the Define Coordinate System dialog and define a custom projection and datum. Alternatively, right-click on an existing coordinate system and choose *New Geographic System* to create a new projection based off the selected projection. The new custom projection and datum will be listed in the *Custom* section with the name you defined.

Modify Coordinate System

Select any coordinate system from the *Select a coordinate system* list. Click the *Modify* button to open the Define Coordinate System or Define Unreferenced Coordinate System dialog. Alternatively, right-click on the coordinate system and choose *Modify* from the menu. Modify any properties and click *OK*. The modified coordinate system is added to the *Custom* list.

If the *Modify* button is unavailable, make sure that a specific coordinate system is selected. If a category of systems, such as *Predefined* or *Geographic (lat/lon)* is selected, the *Modify* button is unavailable.

Add to Favorites

Click the *Add to Favorites* button to add a projection to your *Favorites* list. Alternatively, right-click on a coordinate system and choose *Add to Favorites.* This is very useful if you frequently use the same projection, such as World Geodetic System 1984. This is also useful for *Custom* systems that will be used on multiple project files. Adding the projection to the *Favorites* list makes selecting the projection easier in the future.

If the *Add to Favorites* button is unavailable, make sure that a specific coordinate system is selected. If a category of systems, such as *Predefined* or *Geographic (lat/lon)* is selected, the *Add to Favorites* button is unavailable.

Remove from Favorites

Select a coordinate system in the *Favorites* list and click the *Remove from Favorites* button to delete the system. Alternatively, right-click on a coordinate system in the *Favorites* section and choose *Remove.* The coordinate system is removed from the *Favorites* section.

If the *Remove from Favorites* button is unavailable, make sure that a specific coordinate system is selected in the *Favorites* list. If a category of systems, such as *Predefined* or *Geographic (lat/lon)* is selected or if a coordinate system is selected in a category other than *Favorites*, the *Remove from Favorites* button is unavailable.

Load from File

Click *Load from File* to load a coordinate system from a spatial reference file with the Open dialog. The following file types are supported for import:

- Golden Software Reference .GSR2
- Golden Software Reference .GSR
- Esri Projection .PRJ
- Well Known Text .WKT
- Esri Auxiliary .AUX.XML

Save to File

Click *Save to File* to save a predefined or custom coordinate system to a spatial reference file with the Save As dialog. The following file types are supported for export:

- Golden Software Reference .GSR2
- Esri Projection .PRJ
- Well Known Text .WKT
- Esri Auxiliary .AUX.XML

Example 1: Select a Predefined Coordinate System (i.e. UTM)

1. In the **Assign Coordinate System** dialog, click the ⊞ button to the left of *Predefined*.
2. Click the ⊞ button to the left of *Projected Systems.*
3. Click the ⊞ button to the left of *UTM*.
4. Click the ⊞ button to the left of *WGS84*.
5. Select *WGS84 UTM zone 13N*.
6. Click *OK* . The projection for this data is now set to WGS84 UTM zone 13N.

Example 2: Create and Select a Custom Coordinate System (i.e. Lambert Conformal Conic)

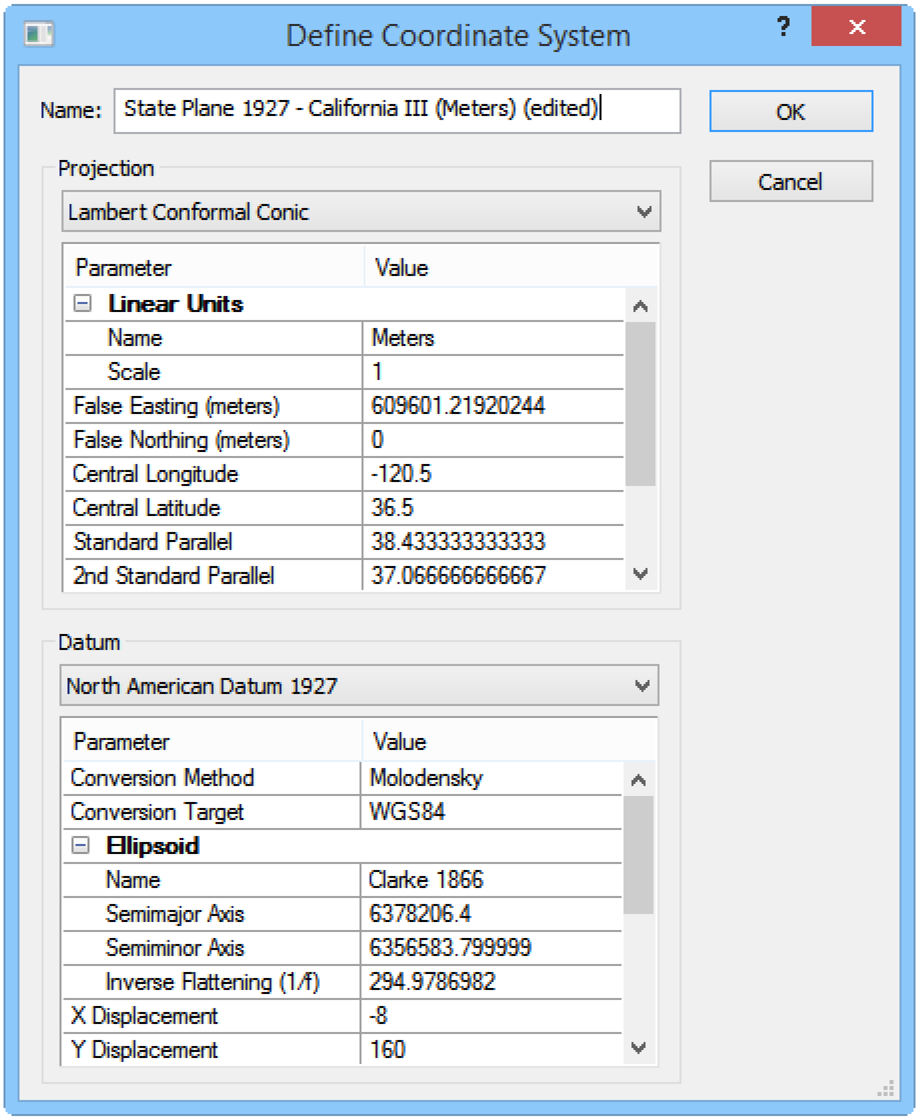
1. In the **Assign Coordinate System** dialog, click the *New* button to open the **Define Coordinate System** dialog.
2. Change the *Name* to *Lambert Conformal Conic Custom*.
3. From the *Projection* list, select *Lambert Conformal Conic*.
4. Make any necessary changes to the *Parameter* or *Values* in the *Projection* section.
5. Make any necessary changes to the *Datum* section.
6. Click *OK* and the *Lambert Conformal Conic Custom* projection is added to the *Custom* section.
7. Click the custom projection and click *OK* to apply the projection to the data.

Example 3: Saving a Custom Coordinate System

1. In the **Assign Coordinate System** dialog, click the ⊞ button to the left of *Custom.*
2. Select the coordinate system in the *Custom* section and click the *Add to Favorites* button.
3. Click the ⊞ button to the left of *Favorites* to open the *Favorites* section. The custom projection is saved here.
4. Click *OK* to close the dialog.

Define Coordinate System

Click the *New Geographic System* or *Modify* buttons in the Assign Coordinate System dialog to open the **Define Coordinate System** dialog. The **Define Coordinate System** dialog allows you to create a *Custom* coordinate system. Select a *Projection*, enter the projection *Parameters*, and specify a *Datum*. Click *OK* and the new coordinate system will be added to the **Assign Coordinate System** dialog *Custom* list for future use.



Define a custom coordinate system in the ***Define Coordinate System*** *dialog.*

Name

Specify a custom name for the new coordinate system in the *Name* box. When editing a predefined coordinate system, the name is automatically appended with *(edited)* after the existing name.

Projection

Select a projection from the *Projection* list. Specify custom *Parameter* and *Value* options. *Unprojected Lat/Long* does not have *Parameter* and *Value* options to specify.

The projection options include: *Unprojected Lat/Long, Albers Equal Area Conic, Azimuthal Equidistant, Bonne, Cassini, Eckert IV, Eckert VI, Equidistant Conic, Equidistant Cylindrical, Gnomonic, Hotine Oblique Mercator, Lambert Azimuthal Equal Area, Lambert Conformal Conic, Mercator, Miller Cylindrical, Mollweide, Oblique Mercator, Orthographic, Polyconic, Robinson, Robinson-Sterling, Sinusoidal, Stereographic, Transverse Mercator, Universal Transverse Mercator (UTM),* and *Van Der Grinten.*

Datum

Select a datum from the *Datum* list. Specify custom *Parameter* and *Value* options to fit your needs.

OK or Cancel

Click *OK* to create your new custom coordinate system. The coordinate system will be listed in the *Custom* section of the Assign Coordinate System dialog for this file. If you want to save the custom coordinate system for future use, add the custom coordinate system to the *Favorites* section. Click *Cancel* to return to the Assign Coordinate System dialog without creating a custom coordinate system.

Example 1: Defining a Custom Coordinate System

1. In the **Assign Coordinate System** dialog, click the *New Geographic System* button to open the **Define Coordinate System** dialog.
2. Change the *Name* to *Lambert Conformal Conic Custom*.
3. From the *Projection* list, select *Lambert Conformal Conic*.
4. Make any necessary changes to the *Parameter* or *Values*.
5. Make any necessary changes to the *Datum* section.
6. Click *OK* and the *Lambert Conformal Conic Custom* projection is added to the *Custom* section.
7. In the **Assign Coordinate System** dialog, select the custom projection and click *OK*.

Example 2: Saving a Custom Coordinate System

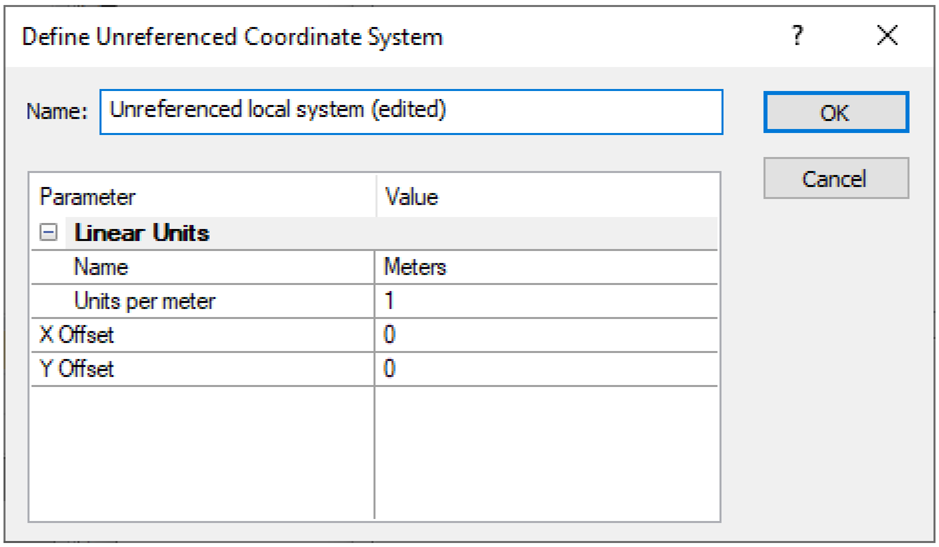
1. In the **Assign Coordinate System** dialog, click the ⊞ button to the left of *Custom.*
2. Select the coordinate system in the *Custom* section and click the *Add to Favorites* button.

3. Click the ⊞ button to the left of *Favorites* to open the *Favorites* section. The custom projection is saved here.

4. Click *OK* to close the dialog.

Define Unreferenced Coordinate System

Click the *New Local System* button, or *Modify* button with a local coordinate system selected, in the Assign Coordinate System dialog to open the **Define Unreferenced Coordinate System** dialog. The **Define Unreferenced Coordinate System** dialog creates a *Custom* coordinate system. Specify the linear units for the unreferenced coordinate system. Click *OK* and the new coordinate system will be added to the **Assign Coordinate System** dialog *Custom* list for future use.



Define a custom local coordinate system in the ***Define Unreferenced Coordinate System*** *dialog.*

Name

Specify a custom name for the new coordinate system in the *Name* box. When editing a predefined coordinate system, the name is automatically appended with *(edited)* after the existing name.

Linear Units

Specify the units for the new coordinate system in the *Name* field. Click the current selection and select the desired units from the list.

Alternatively, specify custom units by typing the number of units per meter in the *Units per meter* box. For example, to use the uncommon unit UK Metric Foot (defined as 300mm), type *3.33333333* into the *Units per meter* field.

Offset

Apply an offset by typing a value in the *X Offset* or *Y Offset* fields.

OK or Cancel

Click *OK* to create your new custom coordinate system. The coordinate system will be listed in the *Custom* section of the **Assign Coordinate System** dialog for this file. If you want to save the custom coordinate system for future use, add the custom coordinate system to the *Favorites* section. Click *Cancel* to return to the **Assign Coordinate System** dialog without creating a custom coordinate system.

Example 1: Defining Units for an Unreferenced Coordinate System

1. In the **Assign Coordinate System** dialog, click the *New Local System* button to open the **Define Unreferenced Coordinate System** dialog.
2. Change the *Name* to *Local Meters (no offset)*.
3. From the *Name* list in the *Linear Units* section, select *Meters*.
4. Click *OK* and the *Local Meters (no offset)* coordinate system is added to the *Custom* section.
5. In the **Assign Coordinate System** dialog, select the custom projection and click *OK*.

Example 2: Saving a Custom Coordinate System

1. In the **Assign Coordinate System** dialog, click the ⊞ button to the left of *Custom.*
2. Select the coordinate system in the *Custom* section and click the *Add to Favorites* button.
3. Click the ⊞ button to the left of *Favorites* to open the *Favorites* section. The custom projection is saved here.
4. Click *OK* to close the dialog.

Assign Coordinate System - Worksheet

The **Data | Coordinate System | Assign Coordinate System** command or the button links a data file to a specific coordinate system. Once the coordinate system is defined for the data file, a Golden Software Georeference .GSR2 file is created. This file contains all the relevant projection information that **Surfer** needs to load the data in the proper projection.

When a .GRD file is created using the Grids | New Grid | Grid Data command, the .GSR2 file for the data is read. The projection information can be saved with the grid file using the Spatial References options. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in **Surfer**, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original data file, but the .GSR2 is required to define the coordinate system. When a map is created from either the data file or the .GRD file, the .GSR2 file is read and the map layer automatically has the correct coordinate system.

Assign Coordinate System - Grid

The **Grids | Edit | Assign Coordinate System** command or the button links a grid file to a specific coordinate system. Once the coordinate system is defined for the grid file, a Golden Software Georeference .GSR2 file is created. This file contains all the relevant projection information that **Surfer** needs to load the grid in the proper projection.

Use the Data | Coordinate System | Assign Coordinate System command to link a data file to a .GSR2 file. When a .GRD file is created using the Grids | New Grid | Grid Data command, the .GSR2 file for the data is read. The projection information can be saved with the grid file using the Spatial References options. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid file in **Surfer**, as the GSR2 retains all of the information needed. The grid has the same coordinate system as the original data file, but the .GSR2 is required to define the coordinate system. When a map is created from either the data file or the .GRD file, the .GSR2 file is read and the map layer automatically has the correct coordinate system.

If the data file did not have an assigned coordinate system, or the grid file was provided directly from another source, the **Grids | Edit | Assign Coordinate System** can be used to link the grid with a .GSR2 file.

When any grid operation is performed on the referenced grid file, the **Export Options** dialog appears with the option to create a new .GSR2 file for the new output grid. For instance, if you use the Grids | Edit | Filter command and smooth the grid, the output smoothed grid will contain a .GSR2 file with the same coordinate system information as the original grid. When a map is created from the smoothed grid file, the .GSR2 file is read and the map layer automatically has the correct coordinate system. For commands that use multiple files, such as Grids | Edit | Assign NoData or Grids | Resize | Mosaic, all files should be in the same reference system, otherwise unexpected results may occur. The .GSR2 file in these cases is assigned by the first .GRD file specified.

What is a Map Projection?

Maps are usually seen in a flat, two-dimensional medium such as a drawing on paper or an image on a computer screen. Since the surface of the Earth is curved, or three-dimensional, the visual elements on the surface must somehow be transformed from three dimensions to two in order to display a map of the Earth's surface. Projections are a mathematical process by which the visual elements are transformed from three dimensions to two.

One of the simplest forms of projection is analogous to shining a light through a translucent globe onto a piece of paper and tracing the outlines. Other forms of projection may involve dozens of complex mathematical equations. Since no two-dimensional representation of a three-dimensional surface can be accurate in every regard, a variety of different projections have been developed to suit different purposes. Some projections are accurate in terms of area but not in scale, some are accurate in terms of scale but not in shape, and so on. The selection of an appropriate projection for a map depends on which characteristics of a map are most important or most desirable for a given project or audience.

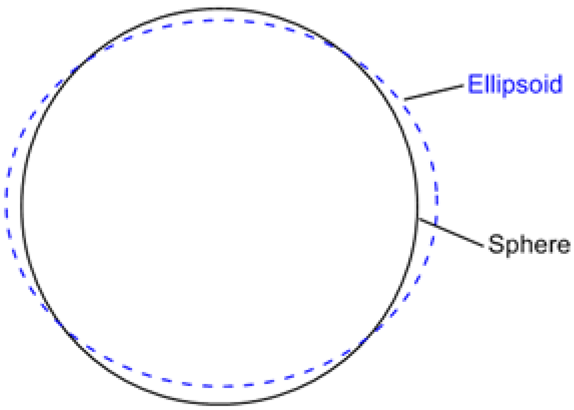
Surfer supports several of the projections that are most often used in modern cartography and related fields. The **Surfer** worksheet allows you to load data and define the projection using the Data | Coordinate System | Assign Coordinate System command. You can then convert the data to another coordinate system using the Data | Coordinate System | New Projected Coordinates command to project the data in another projection, ellipsoid, or datum. **Surfer**'s plot window allows you to assign a coordinate system to a grid file using the Grids | Edit | Assign Coordinate System command. You can also load maps, assign map coordinate systems using the Source Coordinate System for each map layer, and convert the entire map to another system using the Target Coordinate System.

There are many excellent textbooks and publications on this subject, and we do not attempt to explain projections in full detail here. If you need or want more information, you might consider reading the references that provide good introductory discussions of map projections.

Ellipsoids

For maps of the Earth where accuracy is not of particular concern, we can safely assume that the Earth is perfectly spherical in shape. For small-scale maps, the difference between a sphere and ellipsoid is not detectable on the map.

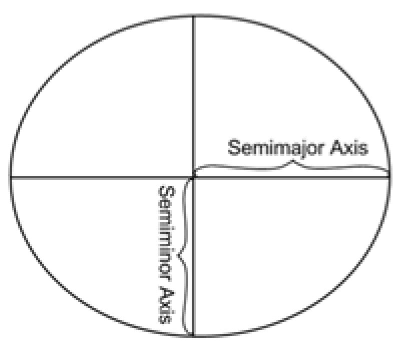
However, the Earth is actually somewhat ellipsoidal (or egg-shaped), approximately 1/300th wider than it is tall, assuming a vertical orientation with respect to the axis of rotation. This shape needs to be taken into account to produce larger scale maps of any significant accuracy.



This is an example of an ellipsoid (dashed blue line) superimposed on a sphere (solid black line).

While an ellipsoid is a closer approximation of the Earth's shape than a sphere, the Earth's surface is not entirely uniform in curvature, so any ellipsoidal representation of the Earth is still only an approximation. This being the case, cartographers have historically used a number of slightly different ellipsoidal representations in attempts to produce more accurate maps of different regions of the Earth.

Ellipsoids are defined by the ellipse being used and by the amount of rotation of the ellipse. An ellipse has two axes. The longer axis is the *Semimajor Axis*. The shorter axis is the *Semiminor Axis*. Rotating the ellipse around the semiminor axis creates the ellipsoid.



The diagram shows the semimajor and semiminor axes on an ellipse.

In **Surfer**, the ellipsoid can be defined by the *Semimajor Axis* and the *Semiminor Axis* or by the *Semimajor Axis* and the *Inverse Flattening (1/f)* value. The flattening value ranges from zero to 1, so the *Inverse Flattening (1/f)* value must be larger than 1. The flattening value is determined by:

f = (Semimajor Axis - Semiminor Axis)/Semimajor Axis

In the Assign Coordinate System dialog, click the *New* button to define a new coordinate system. In the Define Coordinate System dialog, the *Datum* group allows you to specify parameters for the datum. The *Ellipsoid* can be customized in terms of *Name, Semimajor Axis, Semiminor Axis,* and *Inverse Flattening (1/f)*. The *Semimajor Axis, Semiminor Axis,* and *Inverse Flattening* ratio should be set in meters. If you do not understand ellipsoids and datum definitions, it is recommended you use the defaults. For more information on these subjects, see the projection references.

Datums

Since coordinates on the Earth's surface can be recorded under widely varying assumptions about the shape and size of the Earth and the locations of the poles and prime meridian, cartographers have developed a standard for identifying the frame of reference for a coordinate system. This standard is called the *datum*. Because the frames of reference differ, a coordinate recorded in one datum usually has slightly different latitude and longitude values from the same point recorded in any other datum.

Some datums are designed to provide a marginally accurate representation of coordinates spanning the entire Earth, while other datums are designed to provide more accurate results in a particular region at the expense of lesser accuracy in other parts of the world. For example, the *South American Datum of 1969 (SA69)* is tailored to provide good results for maps of the South American continent and surrounding areas, but, as a consequence, provides poor results for the rest of the world.

When combining data from multiple sources into a single map, it is important that all of the coordinate systems being combined specify the projection and datum accurately. Since each datum has slightly different latitude and longitude values for the same coordinates, mixing coordinates from multiple datums together without fully defining the datum introduces inaccuracies into the map. **Surfer** will automatically convert different source coordinate systems from different datums to the target coordinate system.

A *datum conversion* can be used to convert coordinates from one datum to another using the **Data | Coordinate System | New Projected Coordinates** command. Click the ... next to the *Target Coordinate System* to open the **Assign Coordinate System** dialog. Click *New* to define a new projection and datum. The Define Coordinate System dialog has the *Conversion Method* and *Ellipsoid* parameters necessary to allow you to define a coordinate system with a custom datum.

Several different *Conversion Methods* may be used for converting coordinates from one datum to another:

Molodensky	The *Molodensky* method is the most widely used method of datum conversion. It adjusts latitude and longitude coordinates by taking into account the displacement between two datum's ellipsoids on all three axes. It does not take into account any rotational differences between the two ellipsoids.
Bursa-Wolfe	The *Bursa-Wolfe* method is similar to the *Molodensky* method, but in some instances it produces more accurate results because it takes into account both displacement and rotational differences between two ellipsoids. **Surfer** supports the *Bursa-Wolfe* method for conversions from the WGS84 datum to the following datums: World Geodetic System 1972, DHDN-1, DHDN, Australian Geodetic 1984, ANS84, MRT - Everest Modified, Switzerland - CH1903, NTF France - Paris Meridian, and Pulkovo 1942 - Hungary.

Surfer supports the NTv2 datum conversion method. NTv2 is the Canadian government's officially sanctioned method of converting Canadian map data from the old NAD27 datum to the NAD83 datum. NTv2 is based on a hierarchical database of interpolation grids of different resolutions for different regions of the country. NTv2 datum conversions cannot be performed unless an NTv2 grid shift file is installed in the same folder/directory as the **Surfer** program. Visit Natural Resources Canada's Geodetic Reference Systems page on the web for more information about this datum and obtaining an NTv2 grid shift file. See the **Golden Software**How to convert from NAD27 to NAD83 using NTv2 help topic for detailed steps on using the NTv2 datum conversion.

Surfer supports conversions for over 200 different predefined datums.

Custom Datum Definition

Click the *New* button in the Assign Coordinate System dialog to open the Define Coordinate System dialog. Use the *Datum* group to define a custom datum and specify the exact datum parameters and values.

Datum

The *Datum* list contains predefined datums. Select a datum to populate the *Parameter* and *Value* columns. Customize the parameters and values as needed.

Conversion Method

The *Conversion Method* controls the method of datum conversion and the conversion parameters. The conversion methods include *Molodensky*, *Bursa-Wolfe*, and *None*.

- The *Molodensky* method is the most widely used method of datum conversion. It adjusts latitude and longitude coordinates by taking into account the displacement between two datum's ellipsoids on all three axes. It does not take into account any rotational differences between the two ellipsoids.

- The *Bursa-Wolfe* method is similar to the *Molodensky* method, but in some instances it produces more accurate results because it takes into account both displacement and rotational differences between two ellipsoids.
- Choose *None* if the predefined methods do not suite your purpose. If you select *<custom>* from the *Ellipsoid Name* list, you can specify your own ellipsoid model parameters.

Conversion Parameter and Values

You can edit the conversion parameter and values directly by selecting the value you want to edit and typing a new value over it. Parameters vary depending on the selected *Conversion Method*.

Ellipsoid

The *Ellipsoid* group contains options for defining the ellipsoid. Use these settings to define the ellipsoid model that best approximates the curvature of the Earth's shape in the map region. If you do not understand ellipsoids and datum definitions, it is recommended you use the defaults.

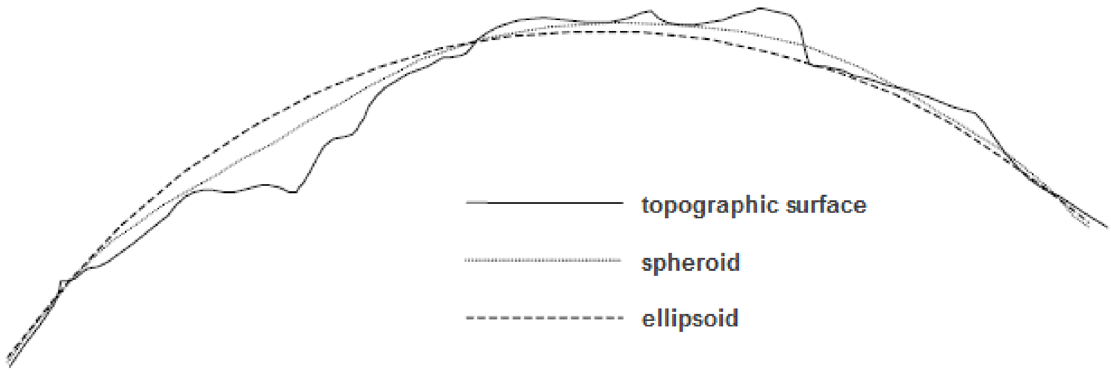
- The *Name* list contains a collection of ellipsoid models that cartographers have historically used in attempts to produce more accurate maps of different regions of the Earth. The *Name* list also contains the option to create a *<custom>* ellipsoid.
- The *Semimajor Axis* box can be edited to define the major axis.
- The *Semiminor Axis* box can be edited to define the minor axis.
- The *Inverse Flattening (1/f)* can be edited to define the flattening ratio.
- The *X Displacement, Y Displacement,* and *Z Displacement* are the axis displacements in meters.

Prime Meridian

The *Prime Meridian* section contains options for defining the prime meridian *Name* and *Prime Meridian Shift*. A meridian is a line of constant longitude running north-south on a map. The zero meridian or prime meridian is used as a reference line from which longitude east and west is measured. The prime meridian passes through Greenwich, England. The *Prime Meridian Shift* is the shift from the prime meridian, typically 0 degrees, in decimal degrees.

Understanding Local Datums

To understand how local datum transforms affect data, you need to understand the ellipsoid, the spheroid, and how both relate to the datum. The spheroid is the a perfect circular object, located at mean sea level. This is constant everywhere. Because of local variations in the gravitational field, the shape actually has local variations.



The image from Verhoogan, shows the variation in the geoid, ellipsoid, and actual topographic surface.

The ellipsoid is used to approximate the global differences. But additional differences exist because of the actual topographic surface. Local datums were created to locally account for these differences. The datum includes the ellipsoid, the prime meridian shift, and any offsets in the X or Y direction. Because local datums align the ellipsoid with a particular location on the earth's surface, local datums are not suitable for use outside the designed area.

In the Define Coordinate System dialog, you can choose how datums are converted from one datum to another. This is the *Conversion Method*. The *Molodensky* method is the most widely used method of datum conversion. It adjusts latitude and longitude coordinates by taking into account the displacement between two datum's ellipsoids on all three axes. It does not take into account any rotational differences between the two ellipsoids. The *Bursa-Wolfe* method is similar to the *Molodensky* method, but in some instances it produces more accurate results because it takes into account both displacement and rotational differences between two ellipsoids. **Surfer** supports the *Bursa-Wolfe* method for conversions from the WGS84 datum to the following datums: World Geodetic System 1972, DHDN-1, DHDN, Australian Geodetic 1984, ANS84, MRT - Everest Modified, Switzerland - CH1903, NTF France - Paris Meridian, and Pulkovo 1942 - Hungary.

Changing the datum incorrectly can cause maps to appear wrong or not appear at all. It is advised that changing the datum be done with caution and a basic understanding of the local datums is advised.

Types of Projections

Most forms of projection operate by projecting Earth coordinates onto a geometric shape that can be easily flattened to a two-dimensional image. This mathematical transformation is commonly referred to as a map projection. A map projection systematically projects locations from the surface of the spheroid to represent positions on the geometric shape.

Three geometric shapes are frequently used:

Type of Shape	**How it works**	**Characteristics of Projection**	**Examples of Projection**
Cylinder	Earth coordinates may be projected onto a cylinder. The cylinder is cut lengthwise and unrolled to make a two-dimensional map. This type of projection is called a cylindrical projection.	• Lines of longitude are parallel to each other. • Lines of latitude are parallel to each other. • Lines of longitude are at right angles to lines of latitude. • Regions near the equator or selected standard parallels are minimally distorted. • Regions near the poles are highly distorted.	Cassini, Equidistant Cylindrical, Hotine Oblique Mercator, Mercator, Miller Cylindrical, New Zealand Map Grid, Oblique Mercator, Transverse Mercator, and Universal Transverse Mercator

Cone	Earth coordinates may be projected onto a cone. The point of the cone is usually directly above the pole and the sides of the cone pass through the globe at two user-defined latitudes, called the Standard Parallels. At the standard parallels, there is no difference between the east-west and north-south scales. The cone is cut from tip to base and unrolled to make a two-dimensional map. This type of projection is called a conic projection.	• Lines of latitude form concentric arcs. • Lines of longitude are straight and radiate outward from the tip of the imaginary cone.	Albers Equal Area, Equidistant Conic, Lambert Conformal Conic, Polyconic, and Bonne
Plane	Earth coordinates may be projected directly onto a flat plane. This type of projection is called an azimuthal projection. Projections of this type are recommended for maps of polar regions because cylindrical and conic projections generally either have severe distortion in polar regions or are unable to project coordinates in polar regions.	• The side of the Earth that is facing away from the center of the projection is not visible.	Azimuthal Equidistant, Gnomonic, Orthographic, Stereographic, and Lambert Azimuthal Equal Area
Other	Projections in this category are pseudocylindrical, pseudoconic, or based on some other mathematical projection or mathematical tables.		Eckert IV, Eckert VI, Mollweide, Robinson, Robinson-Sterling, Sinusoidal, State Plane*, Unprojected Lat/Long, and Van der Grinten

* The State Plane Coordinate System uses Transverse Mercator, Lambert Conformal Conic, or the Hotline Oblique Mercator projection.

Characteristics of Projections

Some projections are imbued with characteristics that tell us if certain types of measurements (e.g. measurements of distance, area, etc.) are accurate on the projected map. Some of these characteristics include the following:

Type of Projection	Characteristic of Projection	Drawbacks	Example Projections
Equal Area	A *equal area* projection is when the area of any given part of the map is preserved. This means that the any object that covers the same area on the Earth as any other part of the map will be the same size. For example, if a one inch diameter circle on the map covers a 100 mile diameter circle on the Earth's surface, then we know that a one inch diameter circle anywhere else on the map is known to cover another 100 mile diameter circle on the Earth. In maps of smaller regions, shapes may not be obviously distorted.	In order for a projection to be equal area, however, consistency in the shapes, scales, and/or angles across the map must be sacrificed. Meridians and parallels may not intersect at right angles.	Albers Equal Area, Bonne, Eckert IV, Eckert VI, Lambert Azimuthal Equal Area, Mollweide, and Sinusoidal

Conformal	A *conformal* projection preserves local shapes. This means that when the local angles for points on the map are represented accurately. This means that the angles between any given point and any nearby points are accurate, but are not necessarily accurate for widely separated points on the map. A side effect is that conformal projections preserve the precise perpendicular intersections between parallels and meridians on the map. When mapping smaller areas, relative shape is preserved.	In order for a projection to be conformal, however, consistency in the surface areas, shapes, and/or scales across the map must be sacrificed. An area enclosed by a series of arcs may be greatly distorted.	Hotine Oblique Mercator, Lambert Conformal Conic, Mercator, Oblique Mercator, State Plane Coordinate System, Transverse Mercator, and Universal Transverse Mercator
Equidistant	A *equidistant* projection is when the scale between at least one specific origin point on the map with respect to every other point on the map is represented accurately.	In order for a projection to be equidistant, however, consistency in the surface areas, shapes, and/or angles across the map must be sacrificed.	Azimuthal Equidistant, Equidistant Cylindrical, Equidistant Conic, and Cassini
Azimuthal	A *azimuthal* projection is when the direction of (or angle to) all points on the map are accurate with respect to the center point of the projection.	In order for a projection to be azimuthal, areas, shapes, and angles may be sacrificed at areas not close to the center of the map.	Azimuthal Equidistant, Gnomonic, Lambert Azimuthal Equal Area, Orthographic, and Stereographic

Other	Some projections try to minimize the effects of all distortions and as a result do not minimize any one distortion in particular.		Polyconic, Robinson and Robinson-Sterling, Unprojected Lat/Long, and Van der Grinten

In addition to the characteristics described above, some projections have highly specialized characteristics that may be useful in certain applications. For example, on maps made with a Mercator projection, all lines of constant direction (rhumb lines) are known to be straight, thereby making such maps very desirable as navigational charts.

Predefined Coordinate Systems

The following sections describe the various predefined coordinate systems.

Geographic (lat/long) Coordinate Systems

In the Assign Coordinate System dialog, the *Geographic (lat/long)* category contains unprojected latitude and longitude coordinate systems. Coordinates are always in latitude and longitude. Each coordinate system has a different datum. Datums are defined for specific regions. Properties for each coordinate system and datum are listed in the dialog by clicking on the desired system.

Polar/Arctic/Antarctic Projection Systems

In the Assign Coordinate System dialog, the coordinate systems under *Predefined* are separated into different categories.

The *Polar/Arctic/Antarctic* category contains different projections that can be used near the north and south poles. Several different projections fit into this category: Lambert Conformal Conic, Orthographic, and Stereographic. Coordinates are always in latitude and longitude. The difference between the projections are in the definition of the false easting, northing, central longitude, central latitude, standard parallel or datum. Projections are defined for specific regions. Properties for each coordinate system and datum are listed in the dialog by clicking on the desired system.

Regional/National Projection Systems

In the Assign Coordinate System dialog, the coordinate systems under *Predefined* are separated into different categories.

The *Regional/National* category contains different projections that can be used for specific regions or countries of the world. Several different projections fit into this category: Albers Equal Area Conic, Azimuthal Equidistant, Cassini, Gnomonic, Hotine Oblique Mercator, Lambert Azimuthal Equal Area, Lambert Conformal Conic, Mercator, New Zealand Map Grid, Polyconic, Stereographic, Transverse Mercator, and UTM. Coordinates are in latitude and longitude, feet, meters, or occasionally other units. The difference between the projections are in the definition of the false easting, northing, central longitude, central latitude, standard parallel or datum. Projections are defined for specific countries or specific regions. Properties for each coordinate system and datum are listed in the dialog by clicking on the desired system.

State Plane Coordinate Systems

In the Assign Coordinate System dialog, the coordinate systems under *Predefined* are separated into different categories.

The *State Plane* category contains both 1927 and 1983 systems. Coordinates are in either feet or meters. The difference between the 1927 and 1983 systems are in the definition of the false easting, northing, central longitude, central latitude, standard parallel or datum. Settings are defined for specific regions of individual states. Properties for each coordinate system and datum are listed in the dialog by clicking on the desired system.

UTM Coordinate Systems

In the Assign Coordinate System dialog, the coordinate systems under *Predefined* are separated into different categories.

The *UTM* category contains different systems separated by country, region, or type. Coordinates are in meters. The difference between the systems are in the definition of the false easting, northing, central longitude, central latitude, standard parallel or datum. Settings are defined for specific regions or individual countries. Properties for each coordinate system and datum are listed in the dialog by clicking on the desired system.

World Projection Systems

In the Assign Coordinate System dialog, the coordinate systems under *Predefined* are separated into different categories.

The *World* category contains different projections that can be used to display the map in a particular way. Several different projections fit into this category: Bonne, Eckert IV, Eckert VI, EPSG 3395 – Mercator, EPSG 3785 – Popular Visualisation CRS / Mercator, Equidistant Conic , Equidistant Cylindrical, Miller, Mollweide, Orthographic Western Hemisphere, Robinson and Robinson-Sterling, Sinusoidal, and Van Der Grinten. Coordinates are in latitude and longitude. The difference between the projections are in the definition of the false easting, northing, central longitude, central latitude, standard parallel or datum. Projections can be used for multiple countries or specific regions, depending on the projection type. Properties for each coordinate system and datum are listed in the dialog by clicking on the desired system.

Templates

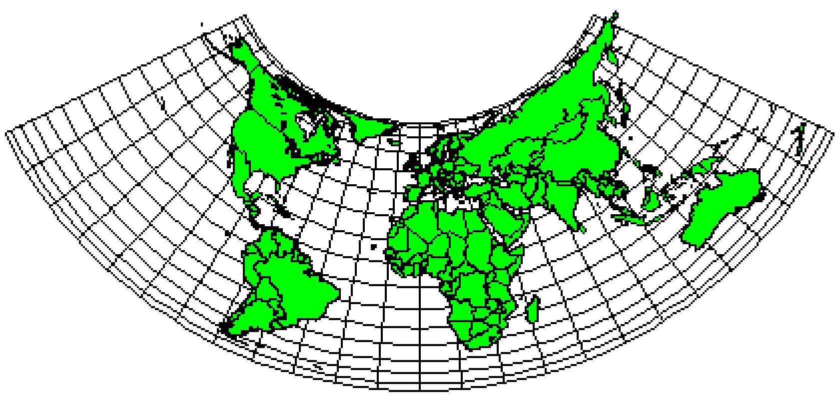
In the Assign Coordinate System dialog, the coordinate systems under *Predefined* are separated into different categories.

The *Templates* section is an easy way to modify one of the predefined systems without needing to find the system in one of the other *Predefined* lists. To modify a coordinate system, click on the desired template coordinate system and click the *Modify* button.

Supported Projections

The following projections are supported in Surfer.

Albers Equal Area Conic Projection



World Map
Albers Equal Area Conic Projection
Central Longitude: 0
Central Latitude: 0
Standard Parallel: 45
2nd Standard Parallel: 0

Projection Characteristics

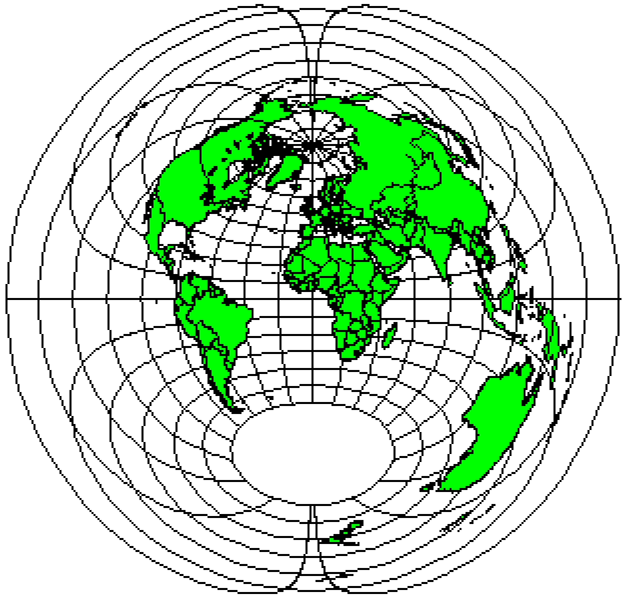
The Albers Equal Area Conic projection scale is constant along any given parallel and accurate along the two specified standard parallels. This projection is used in the National Atlas of the United States. The Albers Equal Area Conic projection is useful for equal area maps of low-aspect regions (regions that are wider than they are tall).

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to *meters*. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce. For example, the value -95.5 represents the geographic center of the United States, so the map of the U.S. is drawn upright.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value should be defined as the latitudinal center of the map you are going to produce. This value is only significant when you define *False Easting* and *False Northing* values.
Standard Parallel	Specifies the latitude of the first of two standard parallels, in degrees. The standard parallels typically are defined at approximately one-sixth of the distance inside the north and south limits of the map. For example, if your map latitude ranges from 30° to 36°, you could place your *Standard Parallels* at 31° and 35°. There are alternative methods for determining the best position of the standard parallels. Please see Snyder for more information.

2nd Standard Parallel	Specifies the latitude of the second of two standard parallels, in degrees. See above.

Azimuthal Equidistant Projection



World Map
Azimuthal Equidistant Projection
Standard Parallel: 0
Central Longitude: 0

Projection Characteristics

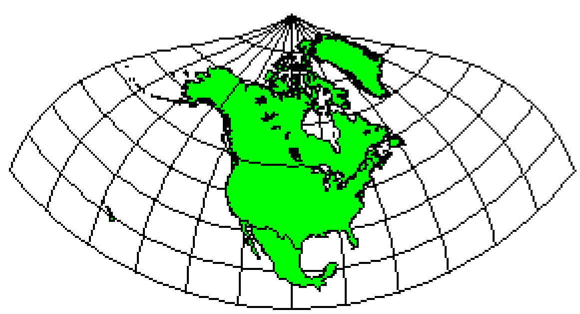
The *Azimuthal Equidistant* projection is a planar projection. Directions and scale are true from the center point of the map. Shapes are true at the center of the map, but are distorted the further you move from the center. When using a polar view of this projection, all meridians are straight lines. When using an equatorial view, the central longitude and equator are straight lines, otherwise, only the central longitude is a straight line. This projection is typically used in polar hemispheric maps.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to *meters*. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Standard Parallel	Specifies the central latitude of the projection in degrees. The *Standard Parallel* value typically should be defined as the latitudinal center of the map you are going to produce.

Bonne Projection



North America Map
Bonne Projection
Central Longitude: -100
Standard Parallel: 40

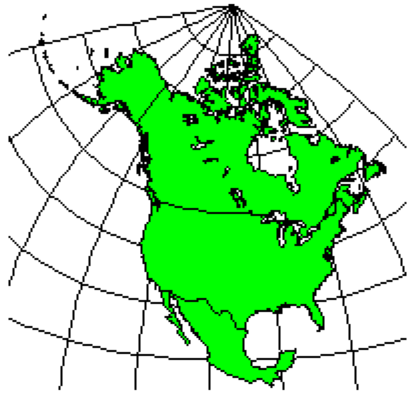
Projection Characteristics

The Bonne projection is a pseudo-conical, equal area projection. The scale is constant along any given parallel, and accurate along the specified standard parallels. The Bonne projection is distortion-free along the central longitude and the parallels. This projection is used for continental and topographic mapping.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Standard Parallel	Specifies the central latitude of the projection in degrees. The *Standard Parallel* value typically should be defined as the latitudinal center of the map you are going to produce.

Cassini Projection



Map of North America
Cassini Projection
Central Longitude = -100
Central Latitude = 40

Projection Characteristics

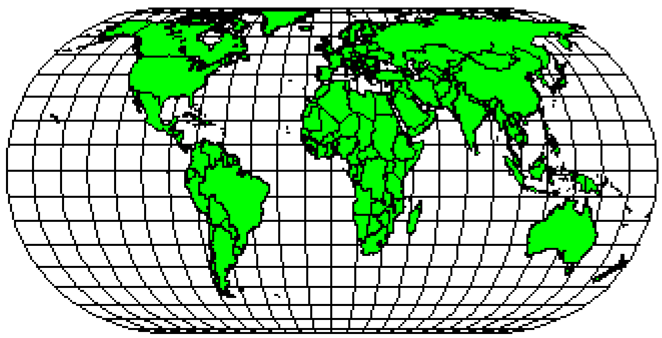
The Cassini projection is a cylindrical projection. The scale is accurate along the central longitude and along latitude lines perpendicular to the central longitude. This projection is useful for high-aspect regions (regions taller than they are wide).

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.

False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.

Eckert IV Projection



World Map
Eckert IV Projection
Central Longitude: 0

Projection Characteristics

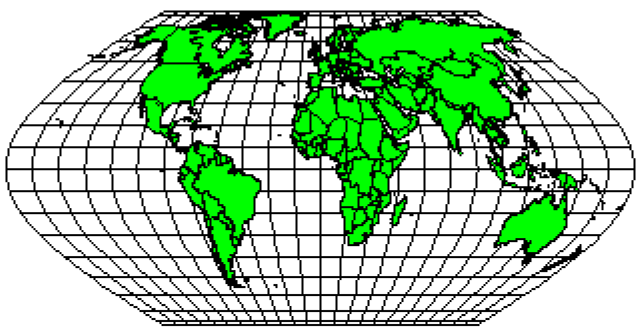
The scale is constant along any given parallel and accurate along the parallels 40°30' north and south in the Eckert IV projection. This is a pseudo-cylindrical, equal area projection with the central longitude half the length of the equator. The poles are represented by lines half the distance of the equator as well. The meridians are semi-ellipses. Eckert IV is designed to produce aesthetically pleasing world maps.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.

Eckert VI Projection



World Map
Eckert VI Projection
Central Longitude: 0

Projection Characteristics

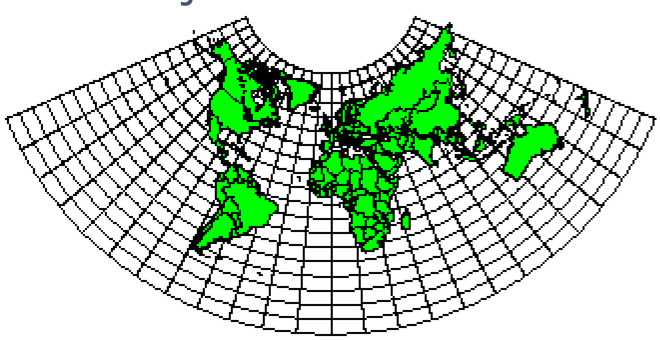
The scale is constant along any given parallel, and accurate along the parallels 49°16' north and south in the Eckert VI projection. This is a pseudo-cylindrical, equal area projection with the central longitude half the length of the equator. The poles are represented by lines half the distance of the equator as well. The meridians are sinusoidal. The Eckert VI projection is useful for world maps.

Projection Parameters

Parameter	**Description**

Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.

Equidistant Conic Projection



World Map
Equidistant Conic Projection
Central Longitude: 0
Central Latitude: 0
Standard Parallel: 45
2nd Standard Parallel: 0

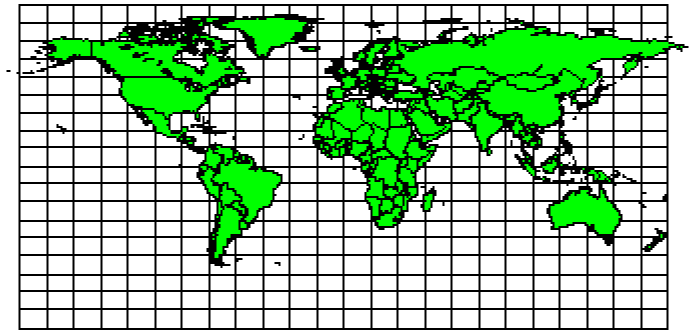
Projection Characteristics

There is no distortion in scale, shape, or area along the standard parallels in an Equidistant Conic projection. Scale is true along all longitudes and along the standard parallels. Direction is locally true along the standard parallels. This projection is useful for maps of low-aspect regions (regions that are wider than they are tall).

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce. For example, the value -95.5 represents the geographic center of the United States, so the map of the U.S. is drawn upright.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value should be defined as the latitudinal center of the map you are going to produce. This value is only significant when you define *False Easting* and *False Northing* values.
Standard Parallel	Specifies the latitude of the first of two standard parallels, in degrees. The standard parallels typically are defined at approximately one-sixth of the distance inside the north and south limits of the map. For example, if your map latitude ranges from 30° to 36°, you could place your *Standard Parallels* at 31° and 35°. There are alternative methods for determining the best position of the standard parallels. Please see Snyder for more information.
2nd Standard Parallel	Specifies the latitude of the second of two standard parallels, in degrees. See above.

Equidistant Cylindrical Projection



World Map
Equidistant Cylindrical Projection
Central Longitude: 0
Standard Parallel: 0

Projection Characteristics

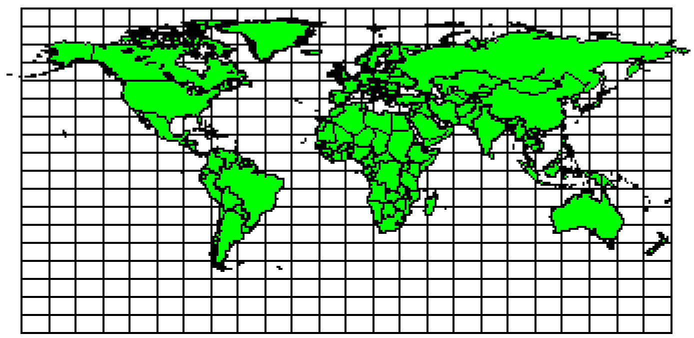
The coordinates are equidistant with respect to the center of the Equidistant Cylindrical projection. Distortion is minimal at the specified standard parallel and increases dramatically with distance north or south from this parallel. Typically, this projection is used with maps covering small areas.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.

Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce. This value is only significant when you define *False Easting* and *False Northing* values, and has no apparent effect on the map.
Standard Parallel	Specifies the central latitude of the projection in degrees. The *Standard Parallel* value typically should be defined as the latitudinal center of the map you are going to produce.

Geographic Coordinate System



World Map
Unprojected Lat/Long

Projection Characteristics

This is a simplistic cylindrical projection. If the coordinates in a **Surfer** map layer are stored in a geographic coordinate system (also known as Unprojected Lat/Long or latitude/longitude), the map is displayed on the screen by simply treating the longitudes as horizontal offsets and the latitudes as vertical offsets.

When plotting latitude/longitude coordinates, **Surfer** constructs the map coordinate system by first determining the latitude for the center of the map. Then the appropriate scale is determined for the east-west (longitude) relative to the north-south (latitude) dimension of the map. See latitude/longitude and Using Scaling to Minimize Distortion on Latitude/Longitude Maps for more information on relative scaling of latitude versus longitude.

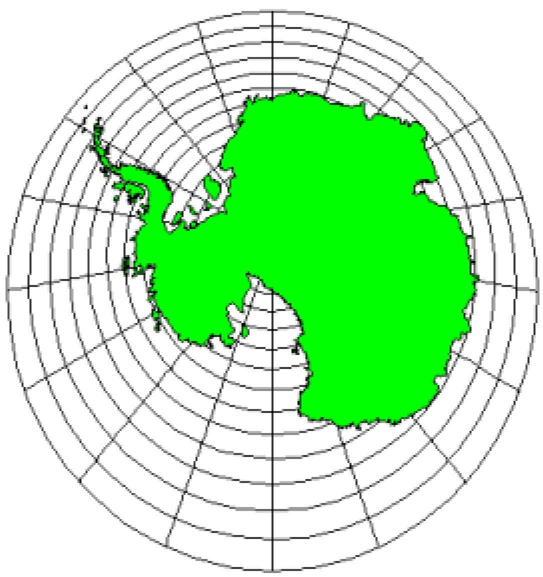
The features of a geographic coordinate system (Unprojected Lat/Long) map are:

- Meridians (lines of constant longitude) are equally spaced and are drawn perpendicular to the parallels.

- Parallels (lines of constant latitude) are equally spaced over the entire map and are drawn perpendicular to the meridians. The further your map area is from the equator, the further apart the parallels are spaced.
- The spacing between meridians is different than the spacing between parallels except when the equator is at the north-south center of the map.
- At the center of the map, the scale is accurate in both the north-south and east-west direction.
- Scale is accurate in the north-south direction over the extent of the map.
- East-west scale increases towards the poles, and decreases towards the equator.

There are no parameters for this coordinate system.

Gnomonic Projection



Antarctica
Gnomonic Projection
False Easting: 0
False Northing: 0
Central Longitude: 19.0
Standard Parallel: 2.4

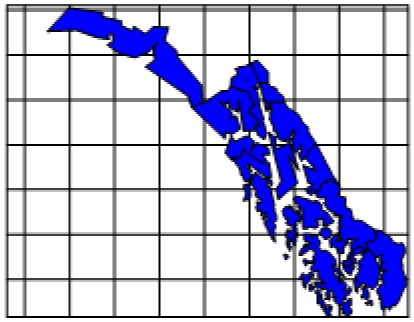
Projection Characteristics

The Gnomonic projection is an azimuthal projection. It is represented as a plane tangent to the globe. At this point of tangency, which is called the standard parallel, all major characteristics are retained. When you move away from the standard parallel in any direction the map is not conformal, not equal-area, and distances are not true to scale. Only areas of less than a hemisphere can be shown and distortion increases noticeably as you move further from the standard parallel.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce. This value is only significant when you define *False Easting* and *False Northing* values, and has no apparent effect on the map.
Standard Parallel	Specifies the central latitude of the projection in degrees. The *Standard Parallel* value typically should be defined as the latitudinal center of the map you are going to produce.

Hotine Oblique Mercator Projection



Alaska Panhandle
Hotine Oblique Mercator Projection
Scale = 1
False Easting = 818676.73440112
False Northing = 575097.68887519
Central Scale Factor = .9999
Azimuth (Alpha) = -36.8698976
Central Latitude = 57
1st Meridian = -133.6667
Rotate U/V to X/Y = True
Offset by U = True

Projection Characteristics

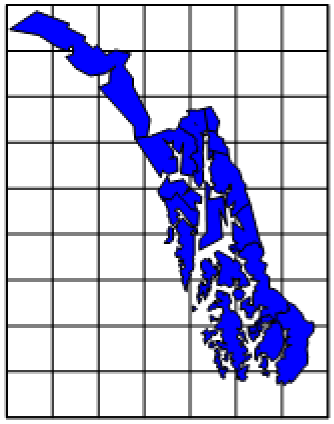
The Hotine Oblique Mercator projection is a cylindrical, conformal projection. The scale is accurate along the chosen central line by the longitude/latitude settings below. This projection is useful for oblique areas (areas that do not follow lines of latitude and longitude), such as the Alaska panhandle because the central line does not have to follow a line of constant longitude. This projection is typically used with areas that are state or province sized and it is not suitable for maps of the world. There are two forms of the Hotine projection equation. The *Hotine Oblique Mercator* projection is defined by a point and an azimuth that defines a line (Alternate B, Snyder).

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Scale Factor (KO)	Specifies the central scaling factor for the projection. This value is often set to 1.0, but may be set to another value for specific applications.
Azimuth (Alpha)	The angle in degrees in which to rotate the central line. Zero is north, and rotation is clockwise. Value must be greater than -360 and less than +360. Value cannot equal 0, 360, or -360. Value cannot equal 90, 270, -90, or -270 if *Offset by U* is set to *False*.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce, and should typically be defined as the center of the map.
1st Meridian	Defines one end of the X extent for the central line.
Rotate U/V to X/Y	u,v are unrectified coordinates that follow the central line of the projection. x,y are rectified rectangular coordinates. When True, u,v are rotated to x,y. See Snyder page 70.
Offset by U	When True, u coordinates are offset to remove the *Us* center component, to normalize the origin of the u axis. This is typically required for State Plane coordinate systems that use the Hotine Oblique Mercator projection method.

Hotine Oblique Mercator 2-Point Projection



Aleutian Islands, Alaska
Hotine Oblique Mercator Projection
Scale = 1
False Easting = 0 False Northing = 0
Central Scale Factor = 1 Central Latitude = 0
Standard Parallel = 56 2nd Standard Parallel = 58
1st Meridian = -133 2nd Meridian = -135
Rotate U/V to X/Y = True
Offset by U = True

Projection Characteristics

The Hotine Oblique Mercator 2-Point projection is a cylindrical, conformal projection. The scale is accurate along the chosen central line by the longitude/latitude settings below. This projection is useful for oblique areas (areas that do not follow lines of latitude and longitude), such as the Alaska panhandle because the central line does not have to follow a line of constant longitude. This projection is typically used with areas that are state or province sized and it is not suitable for maps of the world. There are two forms of the Hotine projection equation. The *Hotine Oblique Mercator 2-Point* projection is defined by selecting two points to form a line (Alternate A, Snyder).

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Scale Factor (KO)	Specifies the central scaling factor for the projection. This value is often set to 1.0, but may be set to another value for specific applications.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce, and should typically be defined as the center of the map.
Standard Parallel	Defines one end of the Y extent for the central line.
1st Meridian	Defines one end of the X extent for the central line.
2nd Standard Parallel	Defines the other end of the Y extent for the central line.
2nd Meridian	Defines the other end of the X extent for the central line.
Rotate U/V to X/Y	u,v are unrectified coordinates that follow the central line of the projection. x,y are rectified rectangular coordinates. When True, u,v are rotated to x,y. See Snyder page 70.
Offset by U	When True, u coordinates are offset to remove the *Us* center component, to normalize the origin of the u axis. This is typically required for State Plane coordinate systems that use the Hotine Oblique Mercator projection method.

Lambert Azimuthal Equal Area Projection

World Map
Lambert Azimuthal Equal Area Projection
Central Longitude: -90
Standard Parallel: 45

Projection Characteristics

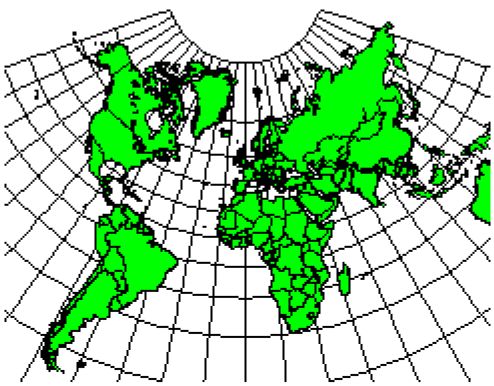
Scale on a Lambert Azimuthal Equal Area projected map is accurate only from the center to any other point on the map. Distortion is minimal near the center and increases with distance from the center. This projection is useful for continents, polar regions (hemispheres), or smaller regions. The Lambert Azimuthal Equal Area projection is not generally used for world maps due to extreme distortion outside the center of the map.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Standard Parallel	Specifies the central latitude of the projection in degrees. The *Standard Parallel* value typically should be defined as the latitudinal center of the map you are going to produce.

Lambert Conformal Conic Projection



World Map
Lambert Conformal Conic Projection
Central Longitude: 0
Central Latitude: 0
Standard Parallel: 45
2nd Standard Parallel: 0

Projection Characteristics

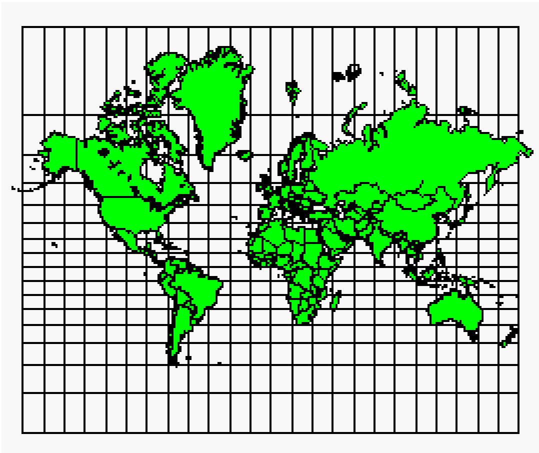
In a Lambert Conformal Conic projection, scale is constant along any given parallel and accurate along the specified standard parallels. Scale is the same in all directions at any given point. This projection is useful for equal area maps of low-aspect regions (regions that are wider than they are tall). The pole is a point in the hemisphere containing the standard parallels and the graticules stretch to infinity in the other hemisphere. The Lambert Conformal Conic projection is used for many of the zones in the State Plane Coordinate System.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce. For example, the value -95.5 represents the geographic center of the United States, so the map of the U.S. is drawn upright.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value should be defined as the latitudinal center of the map you are going to produce. This value is only significant when you define *False Easting* and *False Northing* values.

Standard Parallel	Specifies the latitude of the first of two standard parallels, in degrees. The *Standard Parallels* typically are defined at approximately one-sixth of the distance inside the north and south limits of the map. For example, if your map latitude ranges from 30° to 36°, you could place your *Standard Parallels* at 31° and 35°. There are alternative methods for determining the best position of the standard parallels. Please see Snyder for more information.
2nd Standard Parallel	*Specifies the latitude of the second of two standard parallels, in degrees. See above.*

Mercator Projection



World Map
Mercator Projection
Central Longitude: 0
Central Latitude: 0

Projection Characteristics

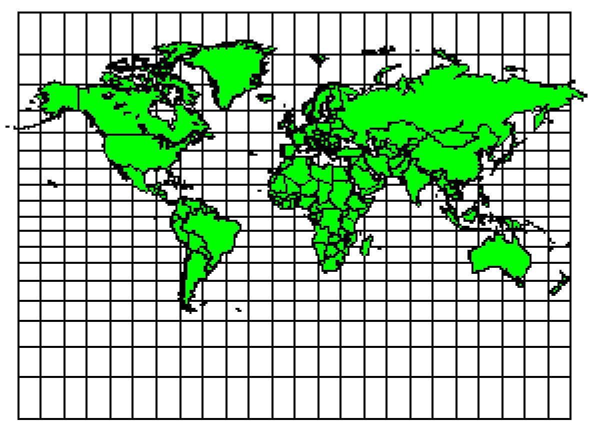
The Mercator projection is a cylindrical projection and it is conformal. In a Mercator projection, scale is constant along any given parallel and accurate along the specified center latitude. Scale is the same in all directions near any given point. Distortion is minimal near the center parallel, but becomes extreme toward the poles. All lines of constant direction (rhumb lines) are known to be straight, thereby making this projection very desirable for producing navigational charts. A limitation of this projection is that coordinates at or near the poles cannot be projected due to constraints of the mathematical formulas used.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.

Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce.

Miller Cylindrical Projection



World Map
Miller Cylindrical Projection
Central Longitude: 0

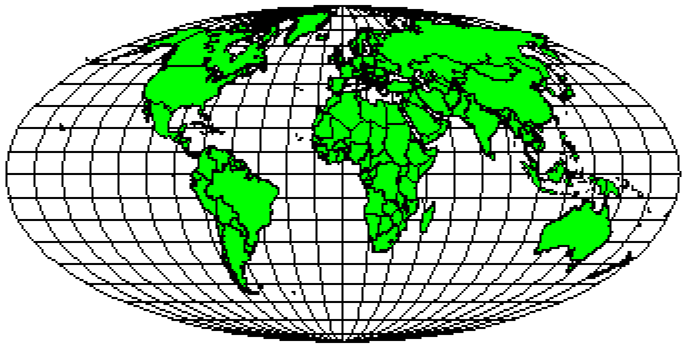
Projection Characteristics

Scale is constant along any given parallel and accurate along the equator in a Miller Cylindrical projection. Scale is the same in all directions near any given point. Miller Cylindrical projection maps use variable latitudinal scale as a way to minimize distortion as you move north or south from the equator. The method effectively corrects for the relative distances covered by one degree of longitude relative to one degree of latitude as you move away from the equator. Distortion is minimal near the equator, but becomes extreme toward the poles. Miller Cylindrical maps do not represent relative land areas accurately, but do approximate the relative shapes of individual land areas. Miller Cylindrical projection maps are useful for displaying the entire world.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. This value should be defined as the longitudinal center of the map you are going to produce. This value is only significant when you define *False Easting* and *False Northing* values and it has no apparent effect on the map.

Mollweide Projection



World Map

Mollweide Projection

Central Longitude: 0

Projection Characteristics

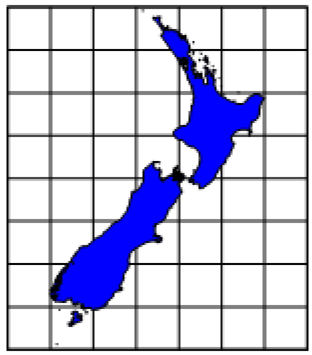
The Mollweide projection is a pseudo-cylindrical, equal area projection. Scale is constant along any given parallel, and true along 40°44' north and south. The central longitude is half the length of the equator. This projection was designed to produce aesthetically pleasing world maps.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.

False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.

New Zealand Map Grid



New Zealand
New Zealand Map Grid Projection
False Easting: 2510000
False Northing: 6023150
Scale = 1

Projection Characteristics

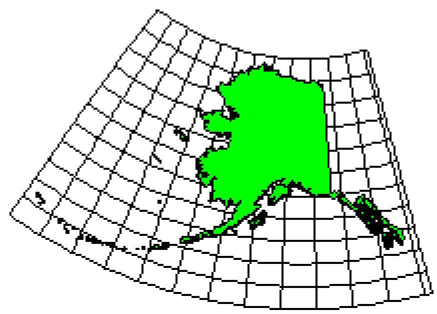
The New Zealand Map Grid projection is a modified cylindrical projection and is conformal. It is a sixth-order conformal modification of the Mercator projection using the International spheroid. Scale is constant along any given parallel and is highly accurate for New Zealand. Scale is the same in all directions near any given point. Distortion is minimal near 173° East, 41° South, and becomes more distorted the further from this location. This projection should only be used for large-scale maps of New Zealand and is not useful outside of New Zealand.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not change them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.

Oblique Mercator Projection



State of Alaska
Oblique Mercator Projection
Scale = 1
Central Scale Factor (KO) = 1
1st Meridian = -132.27
Standard Parallel = 55.95
2nd Meridian = -177.22
2nd Standard Parallel = 52.45

Projection Characteristics

The Oblique Mercator projection is a cylindrical, conformal projection. The scale is accurate along the chosen central line by the longitude and latitude settings below (*1st Point* and *2nd Point*). This projection is used for oblique areas which are areas that do not follow lines of latitude and longitude, such as the Alaska panhandle. This projection is usually used with areas that are state or province sized. This projection is not suitable for maps of the world.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
1st Meridian	Defines one end of the X extent for the central line.
Standard Parallel	Defines one end of the Y extent for the central line.
2nd Meridian	Defines the other end of the X extent for the central line.
2nd Standard Parallel	Defines the other end of the Y extent for the central line.
Central Scale Factor (KO)	Specifies the central scaling factor for the projection. This value is often set to 1.0, but may be set to another value for specific applications.

Orthographic Projection

World Map
Orthographic Projection
Central Longitude: -90
Central Latitude: 45

Projection Characteristics

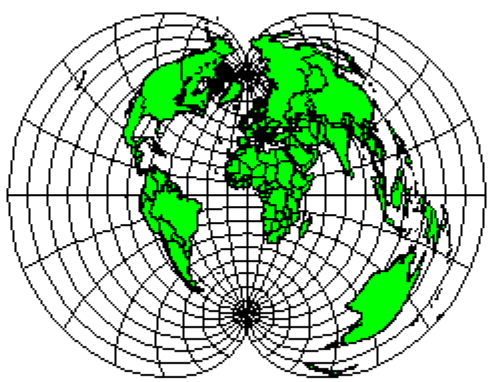
The Orthographic projection is an azimuthal projection. In an Orthographic projection, scale is accurate at the center and along any circle circumscribed around the center. Distortion is nil at the center, and increasingly extreme with increasing distance from the center. This projection is useful for "view of globe" or "view from space" pictures of the Earth. A limitation of this projection is that the hemisphere facing away from the center of the projection is not visible. Objects near the edge of the visible hemisphere may be clipped.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce.

Polyconic Projection



World Map
Polyconic Projection
Central Longitude: 0
Central Latitude: 0

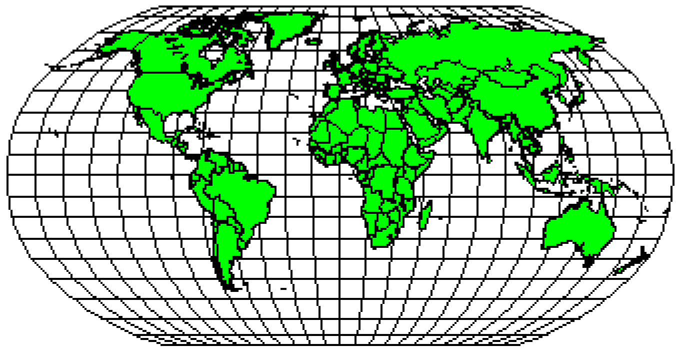
Projection Characteristics

The Polyconic projection is useful for maps of continental or smaller regions. Generally, this projection is not used for world maps due to extreme distortion at any significant distance from the center of the projection. Only the central meridian is distortion-free. Notice in the sample map shown above that Africa is relatively undistorted, but the rest of the world is barely recognizable. Scale is true along the central longitude and along each parallel.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce.

Robinson and Robinson-Sterling Projections



World Map
Robinson-Sterling Projection
Central Longitude: 0

Projection Characteristics

These projections are pseudo-cylindrical. Scale and area are always distorted by the Robinson and Robinson-Sterling projections. These projections are designed to produce aesthetically pleasing world maps.

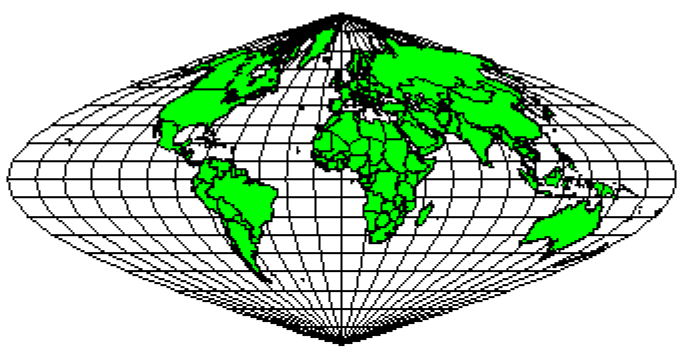
The Robinson and Robinson-Sterling projections produce visually similar results, but use two entirely different mathematical processes. As a result, the numeric values of the projected coordinates produced by these two methods are slightly different. For most applications, these differences are not significant.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce. This option is only available for the Robinson-Sterling projection.

Sinusoidal Projection



World Map
Sinusoidal Projection
Central Longitude: 0

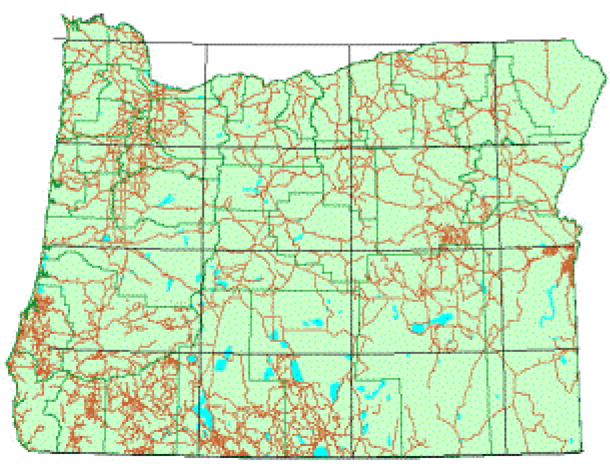
Projection Characteristics

The Sinusoidal projection is a pseudo-cylindrical, equal area projection. Scale is accurate along any given parallel and along the specified central longitude. This projection is useful for continental or world maps, particularly for high-aspect regions (regions taller than they are wide). To get good results with the Sinusoidal projection, the map must have coordinates between +/-180 degrees longitude and +/- 90 degrees latitude.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.

State Plane Coordinate System Projections



State of Oregon
State Plane Coordinate System of 1983
Projection for Oregon North Zone

Projection Characteristics

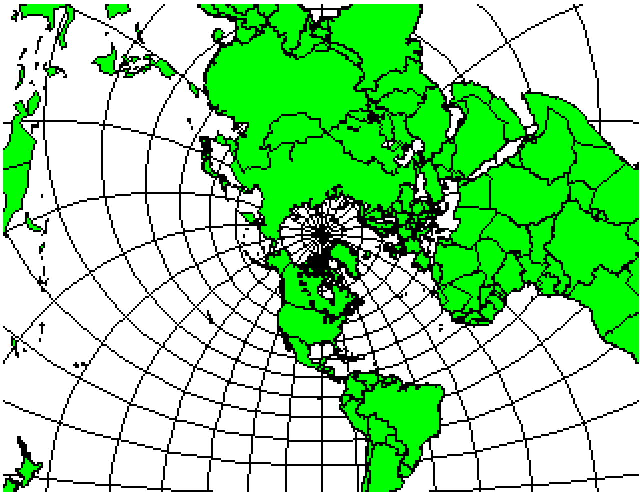
The State Plane Coordinate System (SPCS) divides the United States into a number of zones, and defines a different projection for each zone such that a suitable map of any given zone is plotted. SPCS is used mainly for intrastate views such as county or parish maps.

Unlike most forms of projection where the datum may be specified separately, the SPCS is tied to a specific datum. There are two State Plane Coordinate Systems commonly used. The State Plane Coordinate System of 1927 uses the North American Datum 1927 (NAD27), while the State Plane Coordinate System of 1983 uses the North American Datum 1983 (NAD83).

Projection Parameters

Parameter	Description
Zone	Specifies which one of the predefined zone projections to use for this coordinate system.
Feet or Meters	Most SPCS have both a meters and feet option available in the predefined list.

Stereographic Projection



North America
(Other continents included to show distortion of objects far away from projection center)
Stereographic Projection
Central Longitude: -90
Central Latitude: 45

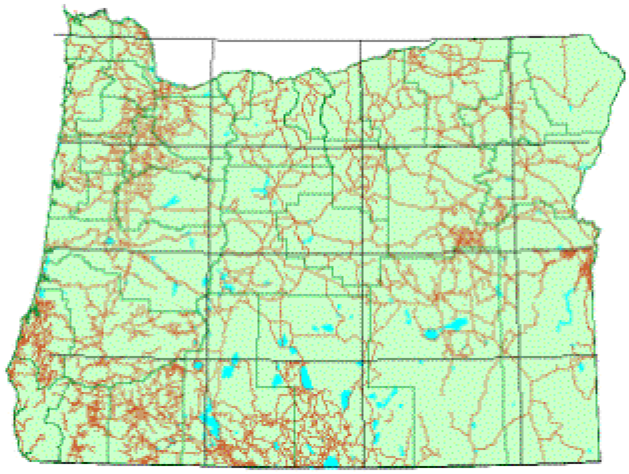
Projection Characteristics

The Stereographic projection is an azimuthal, conformal projection. In a Stereographic projection, scale is constant along any circle circumscribed around the center of the projection. Distortion is minimal at the center and becomes extreme with distance from the center. Generally, this projection is not used for regions larger than a continent or a hemisphere due to distortion effects. The Stereographic projection is often used for maps of the poles.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Scale Factor (KO)	Specifies the central scaling factor for the projection. This value is often set to 1.0, but may be set to another value for specific applications.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce.

Transverse Mercator Projection



State of Oregon
Transverse Mercator Projection
Central Latitude: 41.75
Central Longitude: -120.5

Projection Characteristics

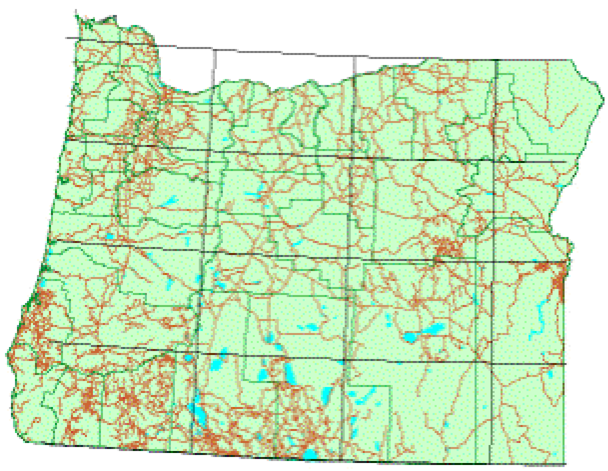
The Transverse Mercator projection is also known as the Gauss-Kruger projection or the Gauss Conformal projection. This projection is cylindrical and conformal. In this projection, scale is constant along any straight line that is parallel to the specified central meridian. Scale increases with distance from the central meridian. Distortion is minimal near the center of the projection and increases dramatically with distance from the center. Distortion is considerable when projecting coordinates that fall within a few degrees of the poles. This projection is useful primarily for mapping small regions no more than a few degrees across, particularly high-aspect regions (regions taller than they are wide). A limitation of this projection is that coordinates at or near the poles cannot be projected. An additional limitation is that regions larger than a quadrant (e.g. having greater than 90 degrees extent either vertically or horizontally) result in portions of the projected image folding over each other due to constraints of the mathematical formulas used. The projection is best used with areas that are no greater than 30 degrees wide or tall, and preferably with areas that are much smaller.

Projection Parameters

Parameter	Description
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.

False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.
Central Scale Factor (KO)	Specifies the central scaling factor for the projection. This value is often set to 1.0, but may be set to another value for specific applications.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value typically should be defined as the latitudinal center of the map you are going to produce.

Universal Transverse Mercator (UTM) Projections



State of Oregon
Universal Transverse Mercator Projection
Central Latitude: 41.75
Central Longitude: -120.5

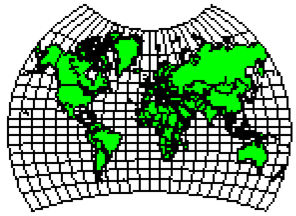
Projection Characteristics

The Universal Transverse Mercator system, commonly known as UTM, divides the Earth into sixty discrete zones, each representing a vertical slice of the globe spanning six degrees of longitude. A Transverse Mercator projection is applied to each zone with the central meridian of the projection at the center of the given zone and the central latitude of the projection at the equator. This coordinate system is the basis for many standardized regional maps, such as tract or neighborhood maps by the US Census Bureau and topographic quadrangles by the US Geological Survey. UTM is not generally used for coordinates outside the range of -80 to +84 degrees latitude due to the distortion inherent in Transverse Mercator projections near the poles.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
Central Scale Factor (KO)	Specifies the central scaling factor for the projection. This value is often set to 1.0, but may be set to another value for specific applications.
Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce. For example, the value -95.5 represents the geographic center of the United States, so the map of the U.S. is drawn upright.
Central Latitude	Specifies the central latitude of the projection in degrees. The *Central Latitude* value should be defined as the latitudinal center of the map you are going to produce.
Projection in Southern Hemisphere	Choose True if your map is in the southern hemisphere. Choose False if your map is in the northern hemisphere.

Van der Grinten Projection



World Map
Van der Grinten Projection
Central Longitude: 0

Projection Characteristics

This projection is typically used for maps of the world and the scale is accurate along the equator. This projection was used by the National Geographic for world maps. The central longitude and the equator are straight lines and the poles are greatly distorted.

To get good results with the Van der Grinten projection, the map must have coordinates between +/-180 degrees longitude and +/- 90 degrees latitude. **Surfer** does not wrap around +/-180 degrees longitude. The example map is using world-proj.gsb as the base map. If you are using world-scale maps, the central longitude generally cannot be set far from 0 degrees since there is no wrap around in **Surfer**.

Projection Parameters

Parameter	**Description**
Name	Specifies the units used in the map.
Scale	Specifies the unit scale of the projected map, relative to meters. For example, a scale of one means one unit in the projected map equals one meter; a scale of two means two units in the projected map equal one meter; etc.
False Easting	Specifies the false easting, or horizontal offset, of the projected coordinates, in meters. *False Eastings* and *False Northings* are added to the underlying "projected" coordinates as a way to arbitrarily offset their internal XY coordinates after the projection. Unless you have a reason for using these offset values, do not use them. These values do not affect the latitude/longitude coordinates for the map, only the internal coordinates used to plot the map on the screen. If you use *False Easting* and *False Northing* offsets for a map, any subsequent boundaries you append to the map must also use these same offsets if you want the imported boundaries to be drawn in the correct relative position to the existing boundaries.
False Northing	Specifies the false northing, or vertical offset, of the projected coordinates, in meters. See above.

Central Longitude	Specifies the central longitude of the projection in degrees. The *Central Longitude* value typically should be defined as the longitudinal center of the map you are going to produce.

Golden Software Reference Files

If you have looked in a folder that contains data, boundary files, or grid files created by **Surfer**, you may have noticed files with a .GSR2 extension in the folder. These Golden Software Reference files are created when you save a projected file from **Surfer**. For example, if you create a data file and use the Data | Coordinate System | Assign Coordinate System command, when you save the data file, you will see both the filename.dat and filename.dat.gsr2 in the directory.

The Golden Software Reference files contain the projection settings used to project the data in **Surfer**. Projection, datum, and georeference information are stored in the .GSR2 file. When you create a map from data or a grid that has an associated .GSR2 file into **Surfer**, the projection information is used when displaying the map. If the Golden Software Reference file is deleted, the boundary can be imported but it will not be projected properly.

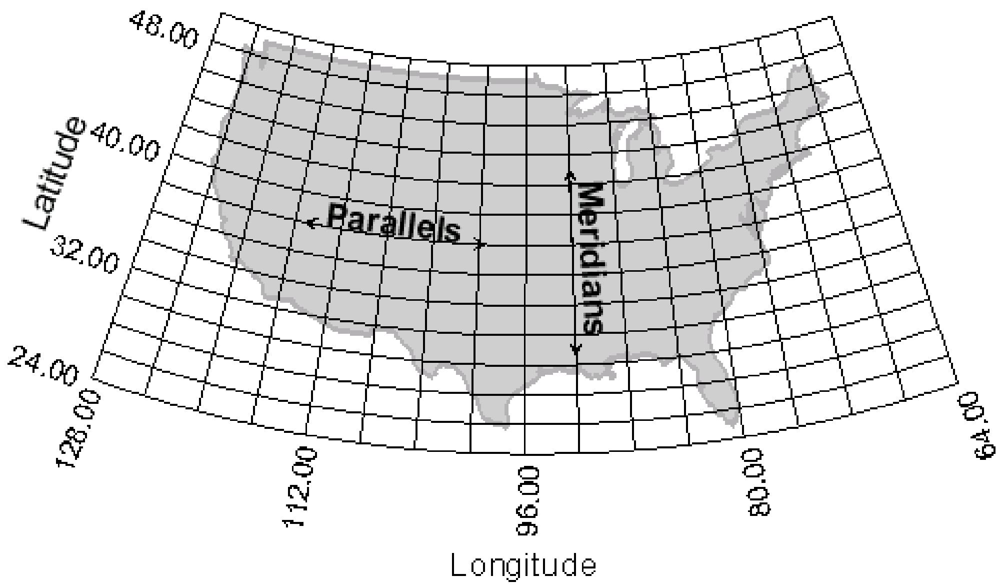
Surfer .SRF file format retains all of the information in a map, including projection information.

Latitude and Longitude Coordinates

In **Surfer** , worksheet data can be projected using a projection. The projected data can then be used to create a grid file and a grid based map.

Latitude and longitude are spherical coordinates used to locate a point on the earth. Many maps do not need to take the curvature of the earth into account. For maps covering relatively small land areas, such as a state or small group of states, the earth can be assumed to be flat. In these cases, the latitude/longitude coordinates can be plotted on a Cartesian coordinate system. Maps plotted in this way must use different scaling in the two dimensions to minimize distortion on the map.

Surferonly plots latitude and longitude coordinates in decimal degrees. You can see DMS to DD for information on converting DMS to decimal degrees with **Surfer** and Latitude and Longitude Coordinates in Decimal Degrees for information on converting degrees-minutes-seconds to decimal degrees manually.



Parallels define lines of constant latitude. Meridians define lines of constant longitude. This Albers projected map makes use of graticule lines to indicate the relationship.

Latitude

Latitude is the Y coordinate and defines north-south global position measured from the equator. Lines of constant latitude are called parallels because they define a series of rings parallel to the equator. Parallels run east-west, but define north-south position on the globe. Parallels are designated in degrees from 0° at the Equator to 90° at the poles. **Surfer** uses the convention that parallels are positive north of the equator (north latitudes), and negative south of the equator (south latitudes). Designations such as 45° indicate a position 45 degrees north of the equator, while -65° indicates a position 65 degrees south of the equator. At any position on the globe, the distance covered by a degree of latitude remains nearly constant.

Longitude

Longitude is the X coordinate and indicates east-west position on the globe. Lines of constant longitude are called meridians. Meridians lie at right angles to the parallels and are half-circles drawn from the North Pole to the South Pole. One meridian is designated as the prime meridian. The prime meridian most commonly in use in the United States runs through Greenwich, England, although there are several other prime meridians in use throughout the world. Longitude is measured 180° east and 180 degrees west from the prime meridian. In **Surfer**, longitude is positive east (east longitude) of the prime meridian, and negative west of the prime meridian (west longitude). A designation such as -105° is used to indicate a location 105 degrees west of the prime meridian. Meridians converge at the poles so the distance covered by one degree of longitude decreases as you move north or south from the equator.

Latitude and Longitude in Decimal Degrees

Latitude and Longitude coordinates are often presented in degrees, minutes, and second, such as 39°25'30" (39 degrees, 25 minutes, 30 seconds). However, **Surfer** can only plot values in decimal degrees. So, for example, 39°25'30" is referred to as 39.425 in **Surfer**.

Converting from degrees, minutes, and seconds is easy. There are 60 minutes in one degree and 3600 seconds in one degree. To convert minutes and seconds to decimal degrees, divide minutes by 60, divide seconds by 3600, and then add the results to obtain the decimal equivalent.

Convert columns of DMS values with the DMS to DD command.

Conversion Equation:

Decimal Degrees = Degrees + (Minutes / 60) + (Seconds / 3600)

Example

Consider the latitude value 39°25'30". This value needs to be converted to decimal degree in order to use it in **Surfer**.

To convert 39°25'30" to decimal degrees:

1. First, convert minutes (25') and seconds (30") to their degree equivalents and add the results.

   25'/60 = 0.4167

   30"/3600 = 0.0083

   0.4167 + 0.0083 = 0.425

2. Then, add this number to the number of degrees.

   39 + 0.425 = 39.425

3. The final result is the decimal degree value.

   39°25'30" = 39.425°

How to Convert from NAD27 to NAD83 Using NTv2

NTv2 is the Canadian government's officially sanctioned method of converting Canadian map data from the old NAD27 datum to the NAD83 datum. If you are converting from Lat/Long WGS84 (or any coordinate system with a datum other than NAD27) to NAD83, you will not need to use NTv2. If you are converting data with a datum of NAD83 back to NAD27, you will not need to use NTv2. Again, this is only a method of converting data from the NAD27 datum to the NAD83 datum.

To perform the conversion, you need an additional grid shift file in the **Surfer** installation directory. Golden Software cannot supply this file, but you can download it from the Natural Resources Canada website.

To convert NAD27 to NAD83 using NTv2, the first step is to download the conversion file.

1. Download the ZIP file from the Natural Resources Canada webpage. You can find more information about this NTv2 grid shift file from their site.

2. Extract the file *NTV2_0.GSB* to the **Surfer** installation directory (by default this is *C:\Program Files\Golden Software\Surfer*).

Now you can use the *NAD 1927 - Canada (NTV2)* datum as the source coordinate system in **Surfer** when converting to another system with the NAD83 datum. For example:

1. Select a map layer in the **Contents** window (i.e. *Contours*).

2. In the **Properties** window, click the **Coordinate System** tab and click the *Set* button.

3. Select the coordinate system for the layer (i.e. *Predefined | UTM | North America | North America NAD27 UTM zone 12N*)

4. Click *Modify*.

5. Change the datum to *NAD 1927 – Canada (NTV2).*

6. Click *OK*, then *OK* again. The souce coordinate system is specified to use the NTv2 datum transformation.

7. To convert the map to NAD83, select *Map* in the **Contents** window.

8. In the **Properties** window, click the **Coordinate System** tab and click the *Change* button.

9. Select the NAD83 coordinate system you want the map to be displayed in (i.e. *Predefined | UTM | North America | North America NAD83 UTM zone 12N*).

10. Click *OK* and the map is converted.

Projection References

Dent, Borden D., *Cartography, Thematic Map Design*, Wm. C. Brown Publishers, Dubuque, 1990.

Greenhood, David, *Mapping*, The University of Chicago Press, Chicago, 1964, pp. 113-171.

Robinson, A.H., et al., *Elements of Cartography*, Fifth Edition, John Wiley & Sons, New York, 1984, pp. 75-105.

Snyder, John P., *Map Projections - A Working Manual*, U.S. Geological Survey Professional Paper 1395, Washington D.C., Department of the Interior, 1987.

Verhoogan, John, Francis., J. Turner, Lionel E. Weiss, Clyde Wahrhaftig, William S. Fyfe, *The Earth: An Introduction To Physical Geology,* Holt, Rinehart and Winston, Inc., New York, 1970.

Coordinate Systems Overview, http://www.-colorado.edu/geography/gcraft/notes/coordsys/coordsys_f.html, July 2, 2001.

Map Projection Overview, http://www.-colorado.edu/geography/gcraft/notes/mapproj/mapproj_f.html, July 2, 2001.

Map Projection Home Page, http://www.geography.hunter.cuny.edu/mp/, June 16, 2004.

Datums, Ellipsoids, Grids, and Grid Reference Systems, Defense Mapping Agency Technical Manual 8358.1, August 2, 2004.

Matching the Map Projection to the Need, http://www.gis.psu.edu/projection/, June 16, 2004.

Chapter 28 - Creating and Editing Features

Features Tab Commands

The **Features** tab in the plot document has the following commands:

Text	Creates a text block
Polyline	Create a polyline
Polygon	Creates a polygon
Point	Creates a point
Spline Polyline	Creates a spline polyline
Range Ring	Creates a range ring
Rectangle	Creates a rectangle or square
Rounded Rectangle	Creates a rectangle with rounded corners
Ellipse	Creates an ellipse or circle
Group	Combine objects into a group
Ungroup	Split grouped objects
Change	Convert selected objects to other object types: Points to Polyline, Points to 3D Polyline, Polyline to Polygon, Polyline to Points, Polyline to 3D Polyline Polygon to Polyline, Polygon to 3D Polygon
Reshape	Moves, adds, or deletes points in the selected polyline, polygon, or spline polyline
Thin	Simplify polylines and polygons by removing additional points
Smooth	Smooth polylines and polygons by adding additional points
Crop	Crop an image or raster base layer
Connect Polylines	Connect two or more polylines
Break Polyline	Break a polyline at a specific point
Break at Intersections	Break polylines at intersections with polylines and polygons
Alpha Shape	Creates a polygon boundary around a set of three or more selected data points
Union of Polygons	Create a new polygon by combining polygons
Intersect Polygons	Create a new polygon from intersecting regions of selected polygons

Difference of Polygons	Create new polygons from the difference between selected polygons
Buffer	Creates a polygon at a specified distance around or within selected objects
Intersection Points	Creates points at intersections of two or more polyline or polygon objects
Triangulation	Creates a Delaunay triangulation
Thiessen Polygons	Creates Thiessen (Voroni) polygons
Split	Split complex polygons into multiple single ring polygons
Combine	Combine multiple polygons into a single complex polygon

Text

Click the **Home | Insert | Text**command, the  button, or the **Features | Insert | Text** command to create text in the plot document. The typeface, size, style, alignment, opacity, and color can be set for individual characters in a text blocks.

Drawing Text

To create text:

1. Click the **Home | Insert | Text** command.
2. The cursor changes to a cross hair. Left-click in the plot window where you want text to appear.
3. Enter text into the Text Editor dialog.
4. Highlight the text to be changed and set the properties by clicking the appropriate button or option in the **Text Editor** dialog. The math text instructions are not used in this dialog.
5. Click *OK* to return to the plot window.
6. Press the ESC key or click another toolbar button to end draw mode.

The text appears inside a movable, sizable text box. Click on the box and drag it to move the box to the desired location.

Editing Text Properties

The text can be edited in the **Text Editor**. To edit existing text content, double-click on the text to open the **Text Editor**, or you can edit text content in the **Properties** window:

1. Click on the text object in the plot window or the Contents window to select it.
2. In the Properties window, the text and font properties can be edited.

3. To change the font properties, click on the ⊞ next to *Font Properties*. Set any font properties for all of the text in the text block.
4. To change the text, click on the ⊞ next to *Text Properties*. Click in the box next to *Text* to edit the text. For a single line of text, highlight the characters and type the desired text. Math text instructions can be added directly to the *Text* line in the **Properties** window.
5. To include multiple lines of text or to edit individual characters in the text block, click the Σ button. Edit the text in the **Text Editor**.
6. When all changes have been made in the **Text Editor** , click *OK* to view the changes in the plot window.

Text Properties

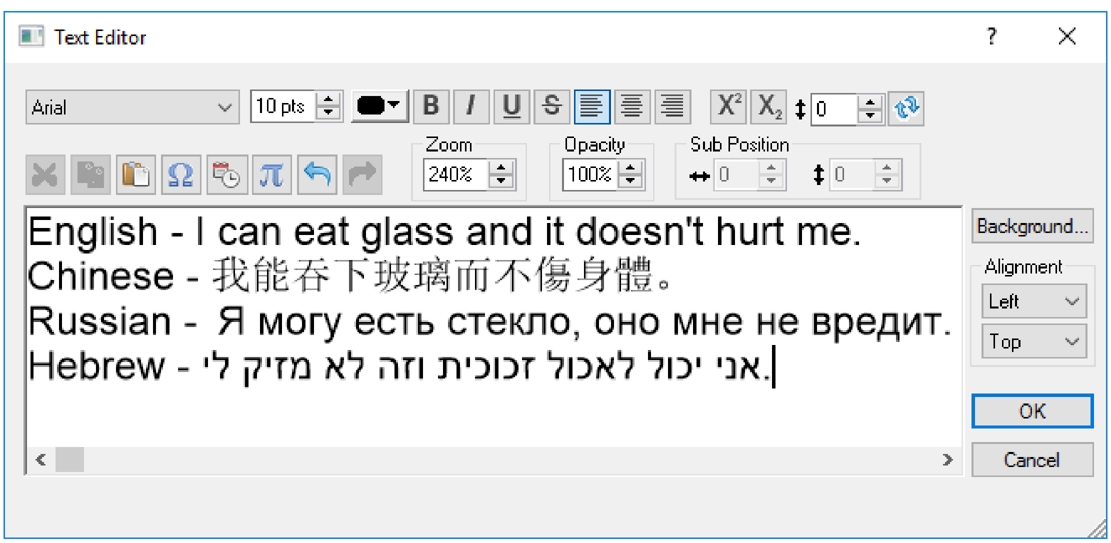
The *Font*, *Size*, *Color*, *Opacity*, and *Style* can be set for text blocks in the **Text Editor** dialog when typing text. These options can be set in the Properties window in the *Font Properties* section after the text is drawn. The Properties window also includes the Drop Shadow and Info pages.

Default Properties

Set the default font properties with the **File | Options** command. In the Default Properties section, click on *Font* to set the default font and text properties.

Text Editor

You can access the **Text Editor** by creating new text or by editing existing text. Type the desired text into the text editor, add templates and symbols, date/time stamps, and apply unique formatting. Add tabs in the **Text Editor** by pressing CTRL + TAB. When you click *OK*, the text in the text editor is parsed to math text instructions and the *Text* field is updated in the **Properties** window Text page.



*Type or edit text in the **Text Editor**.*

Text Appearance

The top row of tools and buttons change the appearance of currently selected text. Additionally, subsequent text will appear with the selected options.

- Select a typeface from the list in the upper left corner of the dialog.
- Set the size (in points) of the typeface in the box to the right of the typeface list.
- Click the colored button to the right of the size box to display the color palette and change the text color. Click the *Custom* button at the bottom of the color palette to use a custom color.
- Several styles (including bold, italic, underline, and strikethrough) can be applied to the text. Note that some typefaces, such as Symbol, do not support bold or italicized text.
- You can left justify, center, or right justify the text in the bounding box. These options only make a difference with multiple lines of text.

Superscripts and Subscripts

When working with superscripts and subscripts, you can type the character, highlight it, and then click the superscript or subscript buttons. Alternatively, you can click the superscript or subscript button and then type the characters. Click the superscript or subscript button a second time to return to the normal size font and placement.

After clicking the or button, the superscript or subscript will be sized relative to the text immediately preceding it until the **Text Editor** is closed. Superscripts and subscripts are 60% the size of normal text by default. To maintain the relative sizing, select both the normal text and the superscript or subscript and then change the font size. For example, if 'x2' is typed in 10pt font and the '2' is superscripted, the font size for the '2' will be 6pt font, x^2. Next if x^2 is selected and font size is increased to 15pt, the superscript will be size 9pt font, x^2. Once the **Text Editor** is closed, the superscripts and/or subscripts must be sized independently in the **Text Editor**. However, the *Size (points)* field in the **Text Properties** window page can be used to relatively increase the size of the normal, superscripted, and subscripted text all at same time.

If the default superscript or subscript placement is not sufficient, you can highlight the text and enter a number in the character position box to raise or lower text from the midpoint of the existing line. You can also change the character position by clicking the buttons. Click the reset button to reset the highlighted character's position to the default position.

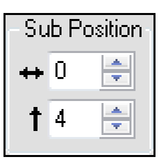
You can click in any field in a template and use the boxes below *Sub Position* to modify the position of the field in a template. The values are offsets from the main character's zero position and are in pixels. How far each value moves the template field is determined in part by the font size. Positive values move the field to the right and up.

Editing Tools

Text can be edited using the following tools:

- You can cut, copy, or paste selected text, or paste objects from the clipboard into the **Text Editor**.
- Click the *Insert Symbol* button to open the Symbol Properties dialog and add a symbol to the text block.
- Click the *Insert Date/Time* button to open the Date/Time Format Builder dialog and enter a date and/or time format. The current date and/or time will be updated and displayed in the project every time the **Surfer** project is redrawn.
- Click *Insert Template* button to open the Template Library dialog and enter equations based on a template.
- You can undo and redo actions.
- You can magnify text in the **Text Editor** by entering a new number in the *Zoom* box. By default, the text is zoomed to a reasonable level.
- You can adjust the opacity of the selected text by entering a new number in the *Opacity* box. Enter a value between zero (no opacity, full transparency) and 100% (full opacity, no transparency).

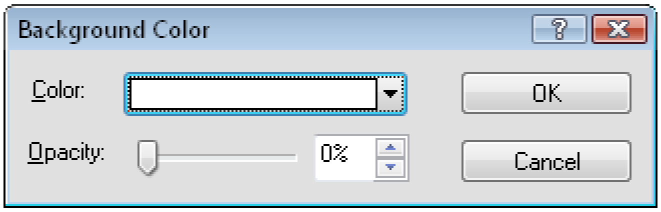
Sub Position



You can click in any field in a template and use the boxes below *Sub Position* to modify the position of the field in a template. The values are offsets from the main character's zero position and are in pixels. How far each value moves the template field is determined in part by the font size. Positive values move the field to the right and up. The *Sub Position* values are only available when editing text in a template field.

Background

Click the *Background* button to open the **Background Color** dialog to set the background fill properties for the text object. To edit the background fill for text that already exists, change the *Background* and *Background Opacity* options in the **Properties** window on the Text tab.



Set the background color and opacity in the ***Back-ground Color****dialog.*

Text Box Alignment

A reference point is the point clicked on in the view window after clicking the **Features | Insert | Text** command. The text box is horizontally and vertically aligned relative to the reference point. The default position is that the reference point is at the upper left corner of the bounding box (left, top).

- *Right* horizontally aligns the text box so that the reference point is to the right of the text box.
- *Left* horizontally aligns the text box so that the reference point is to the left of the text box.
- *Center* horizontally centers the text box on the reference point.
- *Bottom* vertically aligns the text box so that the reference point is below the text box.
- *Top* vertically aligns the text box so that the reference point is above the text box.
- *Baseline* vertically aligns the text box so that the reference point is located at the base of the text. The baseline is the imaginary line along which characters are positioned as they are drawn. Descenders on characters are drawn below the baseline.

To edit the alignment for text that already exists, change the *Horizontal Align-ment* and *Vertical Alignment* options in the **Properties** window on the Text tab.

Resize the Text Editor

To make the **Text Editor** larger or smaller, click and drag on the lower left corner of the dialog. When the dialog is the desired size, release the mouse but-ton.

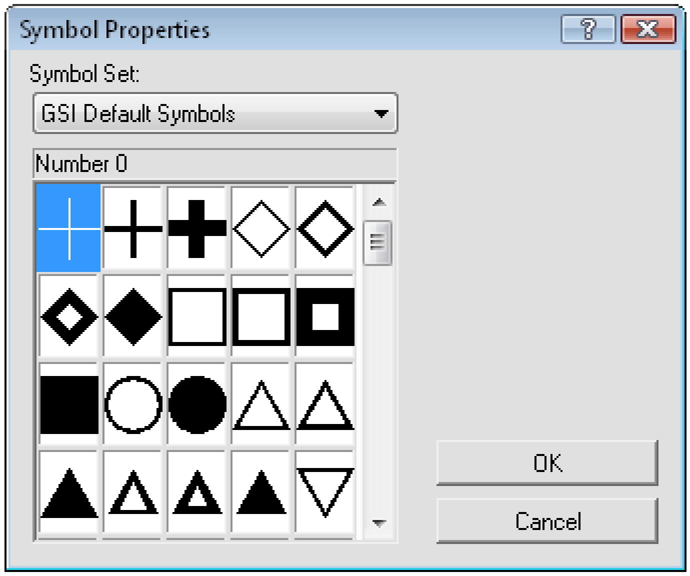
OK or Cancel

Click the *OK* button to save your changes and close the **Text Editor**. Click the *Cancel* button to exit the **Text Editor** without saving your changes.

Symbol Properties Dialog

The **Symbol Properties** dialog can be used to insert symbols from any font into a text block in the Text Editor or change for selected objects in the text template.

You can set default symbol properties through the Options command. In the **Options** dialog, scroll down to the *Symbol* section to access these defaults. Changes made in the **Options** dialog affect all subsequent documents. Custom symbols can be created using a third party TrueType font editing software.



Specify a Symbol Set *and* Symbol *in the* ***Symbol Properties*** *dialog.*

Symbol Set

The *Symbol Set* displays all the fonts installed on the computer. Click on the symbol set name and then you can choose a new font from the list.

Symbol

Click on the displayed symbol to choose a *Symbol* from the symbol palette. The number of the selected symbol is indicated in the title bar above the palette and adjacent to the symbol in the **Properties** window. Add 32 to the value to use the symbol in a script or when using a *Symbol* column with a post map.

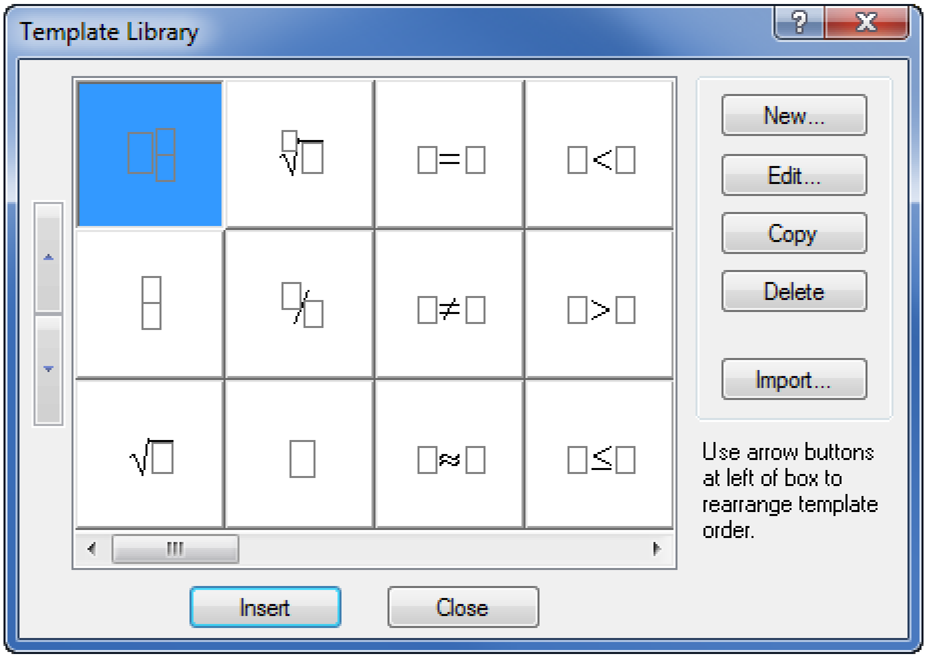
The symbol index is the symbol or glyph number as it appears in the title bar above the palette and adjacent to the symbol in the **Properties** window.

Text Editor Template Library

You can use text templates from the **Template Library** to add equation formats into the Text Editor.

Template Library Dialog

Click the button in the **Text Editor** dialog to open the **Template Library** dialog.



*Use the **Template Library** to insert templates that can be used to input equations into the **Text Editor** .*

Up/Down

Use the up and down buttons at the left side of the screen to reposition a selected template in the library.

New

Click the *New* button to create a new template. The Symbol Properties dialog opens. Select a base symbol, click *OK*, and the **Create\Edit Template** dialog appears.

Edit

Click the *Edit* button to edit the selected template in the **Create\Edit Template** dialog.

Copy

Click the *Copy* button to duplicate the selected template. The copied template is automatically pasted at the end of the template library.

Delete

Click the *Delete* button to delete a template.

Import

Click the *Import* button to use a different Golden Software template library file [.LBT].

Insert

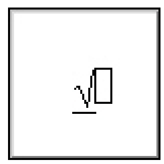
Click the *Insert* button to insert a template into the **Text Editor**.

Close

Click the *Close* button to close the template library without inserting a template into the **Text Editor**.

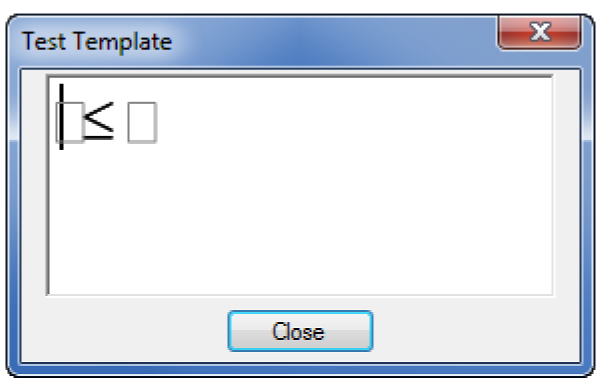
Example

For example, to use the **Template Library** to create an image of the square root of a number:

1. Click the **Home | Insert | Text** command.
2. Click on the view window where you want the text to be displayed.
3. In the **Text Editor** dialog, click the button.
4. In the **Template Library** dialog, select the  template and click the *Insert* button.
5. In the **Text Editor**, the template is inserted as . Enter the numeric value, for example the number nine, and click *OK* to display the  in the view window.

Text Editor Test Template

Click *Test* in the Create/Edit Template dialog to see what the final template will look like without the string and line symbols.

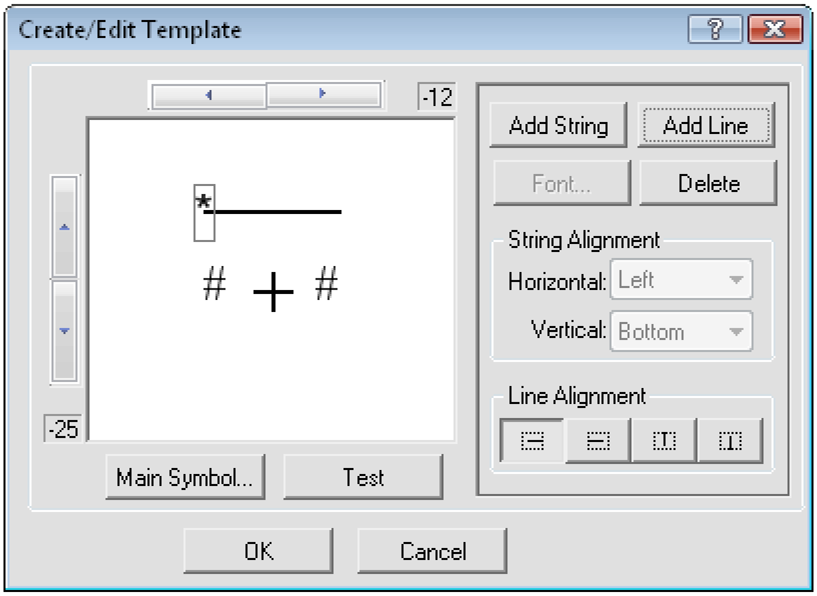


*The **Test Template** dialog displays the final template.*

Text Editor Create/Edit Template

To add custom templates to the Text Editor template library, click the *New* button in the Template Library dialog. To edit an existing template, select the template and click the *Edit* button.

If you are creating a new template select a symbol and click the *OK* button before the **Create/Edit Template** dialog opens.



*Use the **Create/Edit Template** dialog to create new templates or edit existing templates from the template library.*

Arrow Buttons

Click the arrow buttons on the left and top sides of the dialog to position a string or a line. The numbers at the edge of the arrow buttons show the string location.

Add String

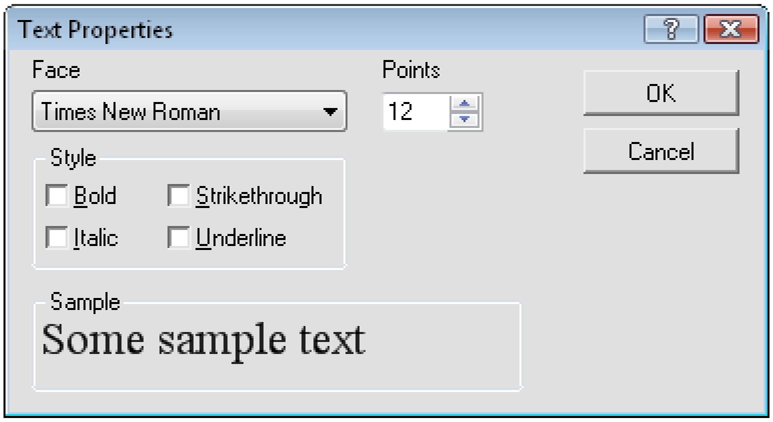
Click the *Add String* button to add a text string box. Text string boxes in the templates allow numbers or letters to be entered into the template in the **Text Editor**.

Add Line

Click the *Add Line* button to add a line to the template.

Font

Select a text string (#) and click the *Font* button to open the **Text Properties** dialog and set the properties of the text string.



*Set the properties of the text string in the **Text Properties** dialog.*

Delete

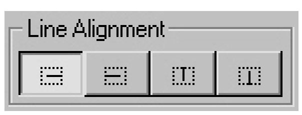
Select a text string or a line, and then click the *Delete* button to remove it from the template.

String Alignment

Once a text string is created (#), you can set the *String Alignment* of the entered text. Text is entered after the template has been inserted into the text editor. Refer to the *Text Box Alignment* section of the Text Editor topic for more information on alignment.

Line Alignment

You can set the *Line Alignment* to extend to the right, left, top, or bottom of the marker by selecting a line and clicking one of the *Line Alignment* buttons. The line length depends on the bounding box size.



Use the Line Alignment *buttons to determine how lines are drawn.*

Main Symbol

Click the *Main Symbol* button open the [Symbol Properties] dialog and change the main symbol appearing in the template.

To delete the main symbol, click the *Main Symbol* button to open the **Symbol Properties** dialog. Choose an empty symbol box, such as symbol number zero.

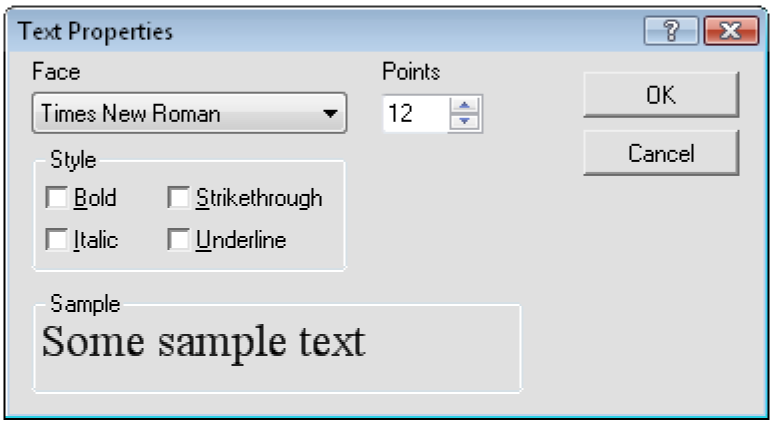
Test

Click the Test button to open the Test Template dialog and see how the template will appear in the **Text Editor**.

1. Use the *Symbol Set* menu to select a set of symbols or text.
2. Highlight the desired symbol and click *OK*. The **Create/Edit Template** dialog opens.
3. Use the following instructions with the **Create/Edit Template** dialog.

Text Properties

In the Create/Edit Template dialog, click the *Font* button to open the **Text Properties** dialog. Set the font attributes for the string.



The ***Text Properties*** *dialog allows you to set the text attributes for the template text string.*

Text Editor Template Editor Example

When entering complex equations in a text box, it is sometimes necessary to create one or more templates to create the equation. When the templates are created they can be inserted into the text box. Templates can be imbedded within other templates. The following example contains two templates: a square root template and a division template.

Example 1 - Square Root Template

This example creates a square root template that will be used in the second example.

1. In the plot window, click the **Home | Insert | Text** command.

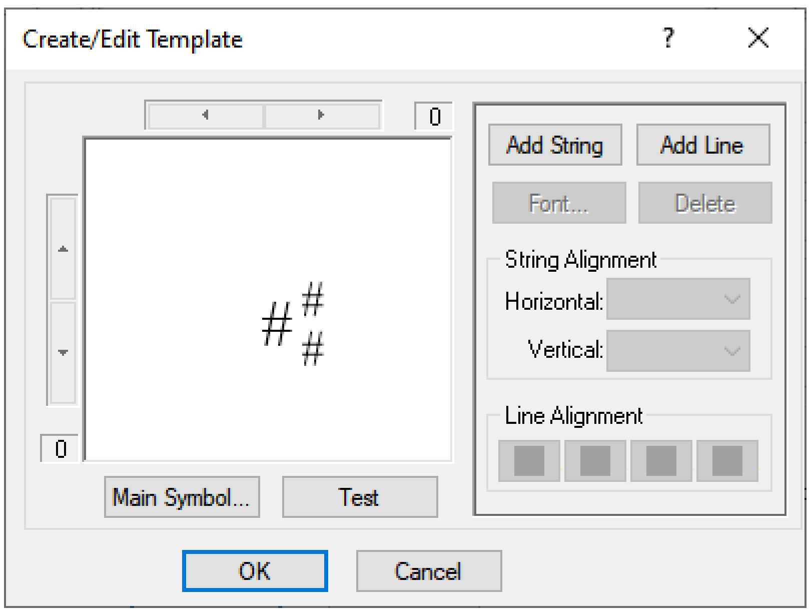
2. Click on the screen where the text should be located and the Text Editor opens.

3. Click the button in the **Text Editor** to open the Template Library dialog.

4. Click *New* in the **Template Library** dialog. The Symbol Properties dialog opens.

5. In the **Symbol Properties** dialog, scroll down and select the square root symbol and click *OK*. (The square root symbol is number 182 near the bottom of the *Symbol* set.)

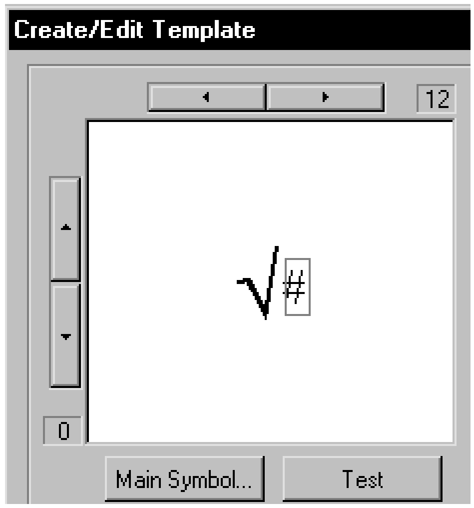
6. The Create/Edit Template dialog opens with the square root symbol in the center of the main window. Click *Add String* in the **Create/Edit Template** dialog and a pound symbol appears.



Clicking the Add String *button creates a pound symbol.*

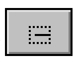
7. Set the *Horizontal String Alignment* to *Left* so the top line will grow with the text entered into the box.

8. Use the arrow buttons on the left and top sides of the dialog to position the string to the right of the square root sign.

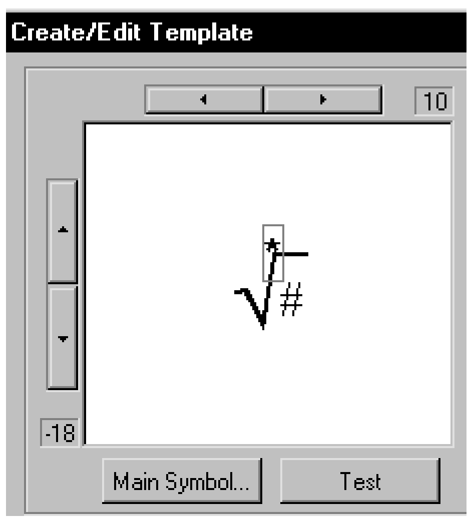


Use the up/down and left/right arrows to move the pound sign to the right of the square root symbol.

9. Click *Add Line*.

10. Click *Extend Right*  in the *Line Alignment* group box.

11. Use the up/down and left/right arrow buttons to position the line above the string and to the upper right side of the square root sign.



Position the line so it is next to the square root symbol.

12. Click *Test* to view the final template. Click *Close* in the Test Template dialog.

13. Click *OK* to add this template to the template library. Leave the **Template Library** dialog open for the next example.

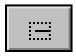
Example 2 - Division Template

This example uses the square root template created in the first example.

1. Click *New* in the **Template Library** dialog.

2. In the **Symbol Properties** dialog, Select empty symbol for the main symbol and click *OK* . The empty symbol is the first symbol (number zero) in the *Symbol* set.

3. Click *Add Line* in the **Create/Edit Template** dialog.

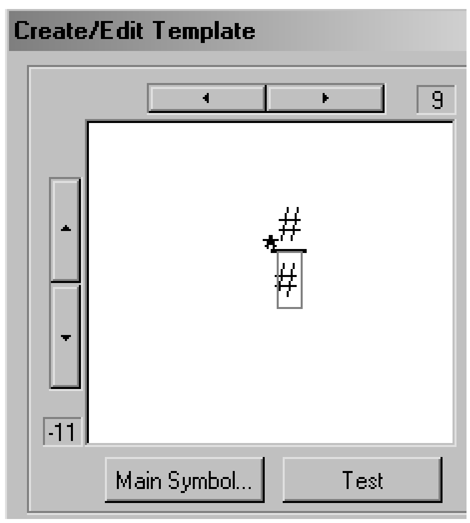
4. Click *Extend Right*  in the *Line Alignment* group box.

5. Click *Add String* .

6. Use the up/down and left/right arrow buttons to position the string above the line, and to the right of the line symbol.

7. Click *Add String* again.

8. Use the up/down and left/right arrow buttons to position the second string below and to the right of the line symbol.



The two string symbols appear to the upper right and lower right of the line symbol.

9. Click *Test* to view the template. Click *Close* in the **Test Template** dialog.

10. Click *OK* to add this template to the template library. Leave the **Template Library** dialog open for the next example.

Using Multiple Templates

Once these templates have been created, they can be inserted into the **Text Editor**. Templates can be inserted into other templates.

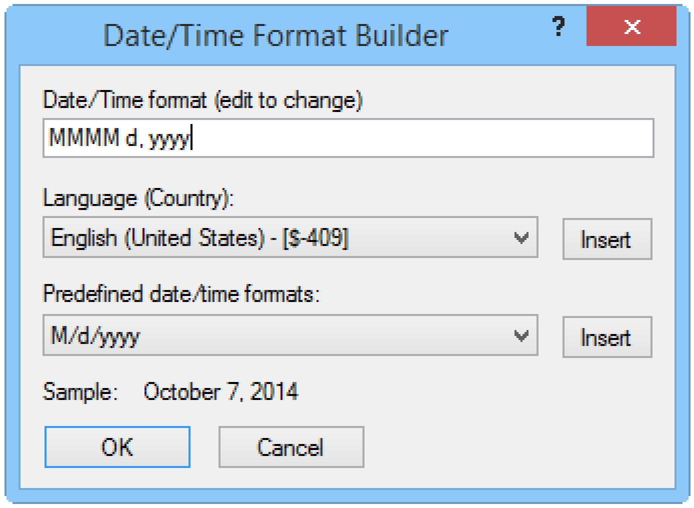
To create the dividing/square root equation:

1. If the template library is not open, in the **Text Editor** click the button to open the **Template Library**.
2. Highlight the new square root template in the **Template Library**, and click *Insert*.
3. You will be asked to save the changes in the template library. Click *Yes.* The square root symbol appears in the **Text Editor**.
4. You may want to make the symbol larger. Adjust the *Zoom* of the image with the zoom up arrow.
5. Click inside the text box underneath the square root sign.
6. Click to open the **Template Library**.
7. Click the newly created division template in the **Template Library** and click *Insert*.
8. The division template is inserted under the square root sign.
9. Click in the numerator box and type the letter X.
10. Click in the denominator box and type the number 2.

Once the text is entered into the templates the substrings (added lines or added symbols) can be moved around. For example, the top line on the square root symbol may not quite touch the square root. Select the line by clicking just to the left of the line where the pound sign (#) would be in theCreate/Edit Tem-platedialog. Use theSub Positionboxes in theText Editorto move the line around.

Date/Time Format Builder Dialog

In the Text Editor dialog, click the button to open the **Date/Time Format Builder** dialog. From the worksheet, click the ... button in the Format Cells dia-log Number page. The **Date/Time Format Builder**dialog is also accessed by clicking the ... button in the *Date/Time Format* field in the Label Properties sec-tion of the **Properties** window when the label format *Type* is *Date/time*. When the **Date/Time Format Builder** is used to insert date/time math text instruc-tion with the Text Editor, the date/time will update every time the project updates.



*The **Date/Time Format Builder** dialog is used to insert or create date/time formats for worksheet cells or text objects and labels.*

Date/Time Format

Type a Date/Time Format into the *Date/Time format (edit to change)* field to set the date/time format. You can also use the *Language (Country)* and *Predefined date/time formats* lists to insert multiple date/time formats and languages.

Language (Country)

By default, the program will use the computer's default language settings for displaying the date/time options in the worksheet. The computer default is controlled by the Windows Control Panel. Refer to your Windows documentation for information about setting the locale. The *Language (Country)* uses the same codes to override the display. For instance, if the date/time values should always be displayed in English, regardless of locale, you could select *English (United States) - [$-409]* and click the *Insert* button. Insert the locale setting first in the *Date/Time format* box. Any cells with the specified language will appear in that language. In addition, the options in the *Predefined date/time formats* will change to show the common formats for that locale. Locale IDs are input as [$-####] in the *Date/Time format* field, where the #### is the locale identifier.

Note: The *Insert* button must be clicked after selecting the *Language (Country)* option. Simply selecting the *Language (Country)* does not change the *Date/Time format* . The *Date/Time format* does not change until *Insert* is clicked.

Predefined Date/Time Formats

The *Predefined date/time formats* list contains the common formats for the selected *Language (Country)* option or for your Windows locale. Available formats are made of combinations of year, month, day, hours, minutes, seconds, and AM/PM designation. Years are shown as yy or yyyy. Months are shown as M, MM, MMM, MMMM, or MMMMM. Days are shown as d, dd, ddd, or dddd. Hours are shown as h, hh, H, HH, or [h]. Minutes are shown as m, mm, or [mm]. Seconds are shown as ss, ss.0, ss.00, ss.000, ss.0000, or [ss]. AM/PM designation is shown as tt or TT. BC/AD designation is shown as gg or GG. BCE/CE designation is shown as g, G, ggg, or GGG. Refer to formats for information about each specific option.

Note: The *Insert* button must be clicked after selecting the *Predefined date/time formats* option. Simply selecting the *Predefined date/time formats* does not change the *Date/Time format*. The *Date/Time format* does not change until *Insert* is clicked.

Sample

The *Sample* text updates to show a sample of the current entry in the *Date/Time format (edit to change)* field.

Polyline

A polyline is a collection of one or more connected line segments. Click the **Home | Insert | Polyline**command, the button, or the **Features | Insert | Polyline** command to draw a polyline. Polylines can be converted to 3D polylines with the addition of Z coordinates to vertices.

To draw a polyline:

1. Click the **Home | Insert | Polyline** command to begin drawing a polyline.
2. The cursor changes to a cross hair cursor to indicate drawing mode.
3. Move the cursor over the location for the start of the polyline and click the left mouse button.
4. Move the cursor to the next position along the line and click again.
5. Continue this procedure until you click at the final point for the line. Press the ENTER key.
6. To end drawing mode, click on another tool button or press the ESC key on your keyboard.

Drawing Tips

- Click the endpoints of the line to draw a straight line, or click several points to create an irregularly shaped line.
- Click the right mouse button to remove the last drawn point. This can be done repeatedly.
- Click and hold the left mouse button to create a continuous stream of points.
- If the CTRL key is pressed while clicking points, the points are constrained to 45-degree angles.
- Hold the SHIFT key to snap the next vertex to the nearest vertex on another polyline or polygon.
- Double-click the left mouse button or press the ENTER key to end the line.

- To cancel drawing the line, press the ESC key before ending the line.
- Edit the shape of the line using Reshape.
- Edit the line style and color by clicking on the line in the plot window or in the Contents window.
- Set default *line* properties with the **File | Options** command. On the Default Properties page, click the *Line* button to set the default line properties.

Polyline Properties

Polyline properties are edited in the **Properties** window. Click on the polyline in the **Contents** window or plot window to select it. The properties automatically appear in the **Properties** window:

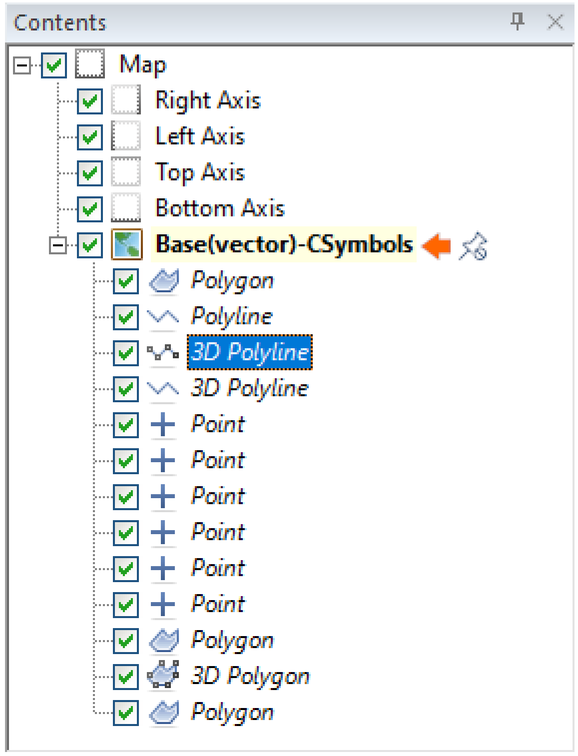
Line
Drop Shadow
Coordinates
Info

3D Polyline

A 3D polyline is a collection of one or more connected line segments using a Z coordinate at each vertex. 3D polyline objects appear as polylines in the plot window and as 3D polylines in the 3D view window. 3D polyline objects, regardless of whether the object has Z coordinates, are shown in the hierarchical list of objects in the **Contents** window. The icon for 3D polylines with Z values will have nodes on the icon's line; 3D polylines that do not have any Z coordinates will not have any nodes on the icon's line.

The Z coordinates, if any, are displayed with the X and Y coordinates on the Coordinates page in the **Properties** window of the base layer. Coordinates may only be changed from the plot window.

Note: Z values are not exported in BLN or DXF file types.



Surfer's iconography distinguishes 3D polylines that have Z values with nodes on their icon's line. 3D polylines that do not have Z values do not have nodes on their icon's line.

Add 3D Polylines to a Map

Polylines are added to maps by importing files that contain 3D polylines or creating 3D polylines in the plot window of Surfer. Very large widths are constrained in the 3D View window and 3D objects will be clipped at the map extents.

Import 3D Polylines

Files that contain polylines with Z values at XY vertices will be imported as 3D polyline objects.

Create 3D Polylines in the Plot Window

3D polylines cannot be created in the 3D view window. Draw a new 3D polyline from the plot window to create a polyline. Convert it to a 3D polyline by clicking on the Features | Change | Change Polyline to 3D Polyline command.

The polyline's name should change to the default "3D Polyline". Use Rename Object to change the object name.

Convert 3D Polylines to Polylines

Convert 3D polylines to polylines by clicking on the object in the Contents window and use the **Features | Change | Change 3D Polyline to Polyline** command. The column for Z values on the **Coordinates** page will be removed.

The object name should change to the default "3DPolyline". Use Rename Object to change the object name.

Editing 3D Polylines

Use Surfer **Features** tab commands to reshape, connect and break 3D polylines. Edit 3D polyline properties in the **Properties** window.

Reshape 3D Polylines

Use the Reshape command to move, add, and delete vertices within a selected 3D polyline in the plot window. The X and Y values are automatically populated on the **Coordinates** page in the **Properties** window but Z coordinates must be changed manually.

Connect 3D Polylines

3D polylines can be connected to other 3D polylines, but cannot be connected to 2D polylines or spline polylines. The Z values for the ends of new 3D polyline remain the same as the original 3D polylines. The Z value for the connection point is interpolated based on the merged vertices from the original 3D polylines. Use the Connect Polylines command to connect selected 3D polylines.

Break 3D Polylines

Use the Break Polyline command to break the selected 3D polyline into multiple 3D polylines. The Z value for the new end nodes will be interpolated from the nodes on either side of the break point. A new object with the name *3D Polyline* will be added in the **Contents** window.

3D Polyline Properties

3D polyline properties are edited in the **Properties** window. Click on the polyline in the **Contents** window or plot window to select it. The properties automatically appear in the **Properties** window:

Line
Drop Shadow
Coordinates
Info

Polygon

Click the **Home | Insert | Polygon** command, the  button, or the **Features | Insert | Polygon** command to draw an irregularly shaped area. Polygons must have at least three vertices (points). Polygons can be converted to 3D polygons with the addition of Z coordinates to vertices.

To draw a polygon:

1. Click the **Home | Insert | Polygon** command to begin drawing a polygon.
2. The cursor changes to a cross hair cursor to indicate drawing mode.
3. Move the cursor over the location for the start of the polygon and click the left mouse button.
4. Move the cursor to the next position along the line and click again.
5. Continue this procedure until you click the final point. Then press the ENTER key.

The first and last points are automatically connected and the new polygon is drawn. To end draw polygon mode, click on another toolbar button or press the ESC key on your keyboard.

Drawing Tips

- Click points on the page to draw a polygon, or click and hold the left mouse button and drag the cursor to draw a continuous stream of points.
- Click the right mouse button to remove the last drawn point. This can be done repeatedly.
- If the CTRL key is pressed while clicking points, the points are constrained to 45-degree angles.
- Hold the SHIFT key to snap the next vertex to the nearest vertex on another polyline or polygon.
- Double-click the left mouse button or press the ENTER key to close the polygon.
- To cancel drawing a polygon, press the ESC key before closing the polygon.
- Edit the polygon shape by using Features | Edit Features | Reshape.
- Edit polygons by clicking on the polygon in the plot window or in the Contents window.
- Set default *line* and *fill* properties with the **File | Options** command. On the Default Properties page, click the *Line* or *Fill* button to set the default line or fill properties.

Polygon Properties

To edit a polygon, click once on the polygon to select it. In the Properties window, the properties are listed. The polygon properties contain the following pages:

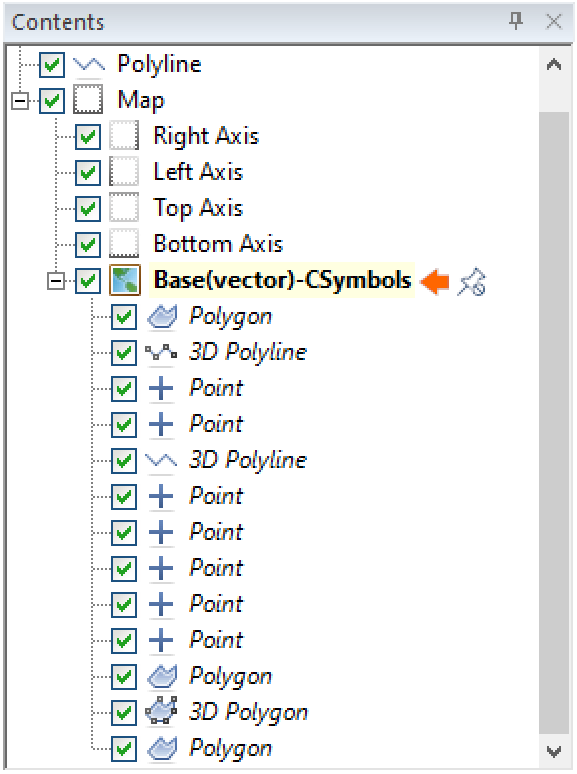
Fill
Line
Drop Shadow
Coordinates
Info

3D Polygon

A 3D polygon is a polygon with the addition of a Z coordinate at each vertex. 3D polygon objects appear as polygons in the plot window and as 3D polygons in the 3D view window. Properties of 3D polygons may be changed in the plot window.

Click on the 3D Polygon in the Contents window to see the Z coordinates displayed with the X and Y coordinates on the Coordinates page in the Properties window of the base layer. Coordinates for 3D polygons may only be changed from the plot window.

Note: Z values are not exported in BLN or DXF file types.



Click on 3D polygon objects in the plot window to change their properties.

Add 3D Polygons to a Map

Polygons are added to maps by importing files that contain 3D polygons or by creating 3D polygons in the plot window of Surfer. Very large widths are constrained in the 3D View window and 3D objects will be clipped at the map extents.

Import 3D Polygons

Files that contain polygons with Z values at XY vertices will be imported as 3D polygon objects. 3D polygons can be identified as "3D Polygon" by default in the **Contents** window.

Create 3D Polygons in the Plot Window

3D polygons cannot be created in the 3D view window. Draw a new 3D polygon from the plot window create a polygon. Convert it to a 3D polygon by clicking on the Features | Change | Change Polygon to 3D Polygon command.

The polygon's name should change to the default "3D Polygon". Use Rename Object to change the object name.

3D Polygon Properties

To edit a 3D polygon, click once on the 3D polygon in the plot window to select it. In the **Properties** window, the properties are listed. The 3D polygon properties contain the following pages:

Fill
Line
Coordinates
Info

Point

Points are markers that use symbols to indicate *2D* or *3D* point positions. Any TrueType font can be used as a symbol, including several custom fonts provided with **Surfer**. You can use the **Home | Insert | Point** , the + button, or the **Features | Insert | Point** command to place symbols in the plot document.

The *Symbol Set, Size, Fill color, Fill opacity,* and *Symbol* can be customized in the Symbol Properties section of the **Properties** window.

To draw a point:

1. Click the **Home | Insert | Point** command.
2. The cursor changes to a cross hair cursor to indicate drawing mode.
3. Click on a location in the plot window to create a point.
4. Press the ESC key or click another tool button to end drawing mode.

Drawing Tips

- Edit a symbol style, color, and size by clicking on the point in the plot window or in the Contents window.
- Set default *symbol* properties with the **File | Options** command. On the Default Properties page, click the *Symbol* button to set the default symbol properties.

Symbol Properties

Symbol properties are edited in the Properties window. Click on the symbol in the **Contents** window or plot window to select it. The properties automatically appear in the **Properties** window.

Symbol
Drop Shadow
Coordinates
Info

Spline Polyline

A spline polyline is a smooth, flowing polyline with no sharp or distinct angles.

Click the **Home | Insert | Spline Polyline**, the button, or the **Features | Insert | Spline Polyline** command to draw a spline polyline. You can change the line style, color, opacity, width, and end styles for the spline polyline display.

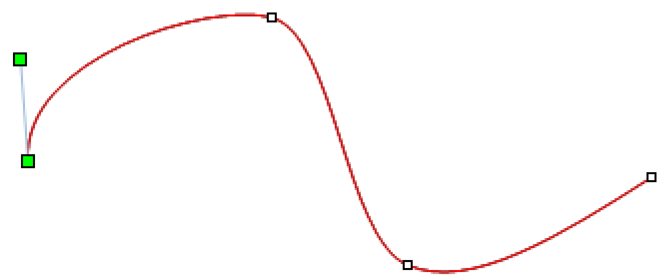
To draw a Spline Polyline

1. Click the **Home | Insert | Spline Polyline** command to enter drawing mode.
2. Move the cross hair pointer over the location for the start of the spline polyline and click the left mouse button.
3. Move the mouse to the next position along the line and click again. Generate the spline polyline by clicking on the anchor points during the polyline creation. The anchor points identify a change in the spline polyline's shape and direction. Notice that the spline polyline shape is visible and that you can change the curvature of the line by moving the mouse in different directions.
4. Continue clicking on the anchor points until you click the last point.
5. Press the ENTER key on the keyboard or double-click the left mouse button to end drawing mode. The new spline polyline is drawn.
6. Press the ESC key on the keyboard or click the button to exit drawing mode.

To edit a Spline Polyline

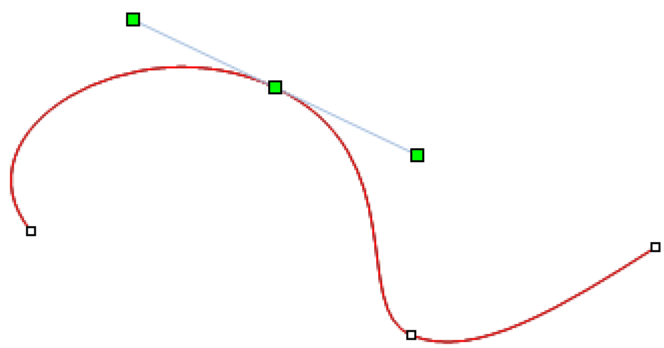
1. Select the spline polyline in the **Contents** window or the plot window by clicking on it.

2. Click the **Features | Edit Features | Reshape** command to enter editing mode.
3. Anchor points appear as white boxes on the spline polyline. Click on an anchor point to select it.
4. For anchor points that are the first or last point on the spline polyline, two control points will be active. The control point that is located at the exact anchor point controls the position of the anchor point. Click and drag the green control point to a new location to move the line. The control point that is connected by a line to the anchor point controls the degree of curvature of the line. Click and drag the green control point to make the line more or less curvy. The closer the control point is to the line, the straighter the line will be.



The end anchor point is being edited. The top green control point controls the curvature of the spline polyline. The bottom green control point controls the location of the anchor point.

5. For anchor points that are not the first or last point on the spline polyline, three control points will be active for each anchor point. The three control points are connected by a light blue line. The middle control point determines the location of the anchor point. The two outer control points determine the curvature of the line. Click the outer control points and drag to a new location to make the line more or less curvy. The line connecting the control points will always be tangent to the anchor point on the spline polyline. The shorter the connecting line, the sharper the angle at the anchor point. The longer the connecting line, the smoother the curve at the anchor point.



A middle anchor point is being edited. The left and right green control points control the curvature of the spline polyline. The center green control point determines the location of the anchor point.

Drawing Tips

- Click points on the page to draw a spline polyline.
- If the CTRL key is pressed while clicking points, the points are constrained to 45-degree angles.
- Right-click to remove the last drawn point. Repeated clicking of the right mouse button removes all points in reverse order.
- Double-click the left mouse button or press the ENTER key to end drawing mode.
- Press the ESC key to cancel drawing the spline polyline before ending the line.
- Use the **Features | Edit Features | Reshape** command to change the spline polyline's shape.
- Edit the spline polyline properties in the **Properties** window.
- Change the properties for a group of selected spline polylines in the **Properties** window.
- Use the **Options** command Default Properties page to set the default line properties for the spline polyline.

Spline Polyline Properties

The spline polyline properties includes the following pages:

Line
Drop Shadow
Info

Range Ring

Click the **Home | Insert | Range Ring**, the button, or the **Features | Insert | Range Ring** command to draw a range ring or set of rings around a point. By default a symbol is displayed at the center of the range rings. A range ring with multiple rings is drawn as concentric circles with the largest ring on the bottom and the smallest ring on the top. Therefore range ring fill opacities are additive. Multiple rings can be used with a semi-transparent fill to create a bulls-eye effect.

To draw a range ring:

1. Click the **Home | Insert | Range Ring** or **Features | Insert | Range Ring** command.
2. The cursor changes to a cross hair cursor to indicate drawing mode.
3. Click on a location in the plot window to create a point and range ring. The center of the range ring is drawn at the clicked location.
4. Press the ESC key or click another tool button to end drawing mode.

Drawing Tips

- Edit the range ring properties by clicking on the range ring in the plot window or in the Contents window.
- The point in the center of the range ring uses the default symbol properties. Set default symbol properties with the **File | Options** command. On the Default Properties page, click *Symbol* on the left side of the dialog to set the default symbol properties.
- The range ring uses the default line and fill properties. Set default line and fill properties with the **File | Options** command. On the Default Properties page, click *Line* on the left side of the dialog to set the default line properties or *Fill* on the left side of the dialog to set the default fill properties.

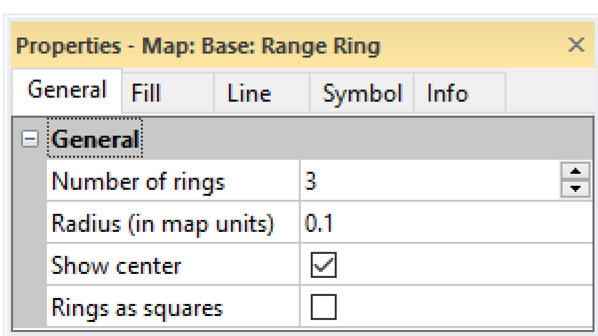
Range Ring Properties

Range ring properties are edited in the Properties window. Click on the range ring in the **Contents** window or plot window to select it. The properties automatically appear in the **Properties** window.

General
Fill Note: All rings use the same fill properties when *Number of rings* on the **General** page is greater than 1. If you wish to use different fill properties for the rings, consider creating a buffer around a point.
Line Note: All rings use the same line properties when *Number of rings* on the **General** page is greater than 1. If you wish to use different line properties for the rings, consider creating a buffer around a point.
Symbol
Drop Shadow
Info

Range Ring General Properties

The **General** page in the Properties window controls the general appearance options for a range ring. To edit the properties for a range ring, click a range ring in the plot window or Contents window.



*Edit the general range ring properties on the **General** page of the **Properties** window.*

Number of Rings

The *Number of rings* property sets the number of range rings drawn extending from the center point. One ring is created by default. Change the number of range rings by typing a value between 1-100 in the *Number of rings* field or clicking the buttons. Range rings are drawn as concentric stacked circles with a symbol in the center. Each additional range ring increases in size by the *Radius* value. For example, the first ring has a radius of *r*, the second ring has a radius of *2r*, the third ring has a radius of *3r*, etc.

Radius

The *Radius* property specifies the size of the range ring or rings. The first range ring has the radius specified by the *Radius* property. When the *Number of rings* value is greater than 1, range rings are drawn as concentric stacked circles with a symbol in the center. Each additional range ring increases in size by the *Radius* value. For example, the first ring has a radius of *r*, the second ring has a radius of *2r*, the third ring has a radius of *3r*, etc.

The *Radius* value is in page units (inches or centimeters) when the range ring is drawn in the plot document. The value is followed by *in.* or *cm.* to indicate the current page units setting. The radius value is in map units when the range ring is drawn in a base layer or empty base layer. The property name is displayed as *Radius (in map units)* when the range ring is in a base layer.

Show Center

The *Show center* property determines if a symbol indicates the center of the range ring. The *Show center* check box is checked by default. Clear the *Show center* check box to hide the symbol in the center of the range ring. Check the *Show center* check box to display a symbol at the center of the range ring.

Rings as Squares

The *Rings as squares* property changes the display from rings to boxes, i.e. from circles to squares. The range rings are displayed as circles by default. Check the *Rings as squares* check box to display the range rings as concentric squares. When *Rings as squares* is checked, the *Radius* property indicates one half of the square side length. This is also the minimum center to edge distance. This means the area is increased when checking the *Rings as squares* check box.

If you wish to show rings as squares where the square covers less area than the circle, i.e. center to vertex distance is equal to the original circle's radius, change the *Radius* value to your original value divided by the square root of two, 1.414214.

Rectangle

You can use the **Home | Insert | Rectangle** command, the button, or the **Features | Insert | Rectangle** command to create a rectangle or square in the plot document. The feature is created as a polygon and the object ID is set to *Rectangle*.

To draw a rectangle:

1. Click the **Home | Insert | Rectangle** command. The cursor changes into a cross hair to indicate drawing mode.
2. Press and hold the left mouse button at one corner of the rectangle.
3. Drag the mouse to the opposite corner of the rectangle. The size of the rectangle appears in the status bar as it is drawn. You can draw a rectangle out from the center rather than corner to corner by holding down the SHIFT key.
4. Release the left mouse button when the rectangle is the preferred size and shape.
5. Press the ESC key or click another tool button to end draw mode.

To draw a square:

- Hold down the CTRL key while dragging the mouse to draw a square rather than a rectangle.
- Hold down the SHIFT and CTRL keys while dragging the mouse to draw the square from a center point rather than from end to end.

Drawing Tips

- Edit the rectangle or square properties by clicking on it in the plot window or in the Contents window.
- Set default *line* and *fill* properties with the **File | Options** command. On the Default Properties page, click the *Line* or *Fill* button to set the default line or fill properties.

Rectangle Properties

Rectangle properties are edited in the **Properties** window. Click on the rectangle to select it. The rectangle properties contain the following pages:

Fill
Line
Drop Shadow
Coordinates
Info

Rounded Rectangle

You can use the **Home | Insert | Rounded Rectangle** command, the button, or the **Features | Insert | Rounded Rectangle** command to create a rounded rectangle or square in the plot document. The feature is created as a polygon and the object ID is set to *Rectangle*.

To draw a rounded rectangle:

1. Click the **Home | Insert | Rounded Rectangle** command.
2. The cursor changes to a cross hair to indicate drawing mode.
3. Press and hold the left mouse button at one corner of the rounded rectangle.
4. Drag the mouse to the opposite corner of the rounded rectangle. The size of the rounded rectangle appears in the status bar as it is drawn.
5. Release the left mouse button when the rounded rectangle is the preferred size and shape.
6. Press the ESC key or click another tool button to end draw mode.

Drawing Tips

- Hold down the CTRL key while dragging the mouse to draw a rounded square rather than a rounded rectangle.
- Hold down the SHIFT key while dragging the mouse to draw the rounded rectangle out from the center rather than from the corner.
- Hold down the SHIFT and CTRL keys while dragging from end to end.
- Edit the rounded rectangle or square properties by clicking on it in the plot window or in the Contents window.
- Set default *line* and *fill* properties with the **File | Options** command. On the Default Properties page, click the *Line* or *Fill* button to set the default line or fill properties.

Rounded Rectangle Properties

Rounded rectangle properties are edited in the **Properties** window. Click on the rounded rectangle to select it. The rounded rectangle properties contain the following pages:

Fill
Line
Drop Shadow
Coordinates
Info

Ellipse

You can use the **Home | Insert | Ellipse** command, the button, or the **Features | Insert | Ellipse** command to create an ellipse or circle in the plot document. The feature is created as a polygon and the object ID is set to *Ellipse*.

To draw an ellipse:

1. Click the **Home | Insert | Ellipse** command.
2. The cursor changes to a cross hair to indicate drawing mode.
3. Press and hold the left mouse button at one corner of the bounding box of the ellipse.
4. Drag the mouse to the opposite corner of the ellipse. The size of the ellipse's bounding box appears in the status bar as it is drawn.
5. Release the left mouse button when the ellipse is the preferred size and shape.
6. Press the ESC key to end draw mode.

Drawing Tips

- Hold down the CTRL key while dragging the mouse to draw a circle rather than an ellipse.
- Hold down the SHIFT and CTRL keys while dragging the mouse to draw the circle from a center point rather than from end to end.
- Edit the ellipse or circle properties by clicking on the object in the plot window or in the Contents window.
- Set default *line* and *fill* properties with the **File | Options** command. On the Default Properties page, click the *Line* or *Fill* button to set the default line or fill properties.

Ellipse Properties

Ellipse properties are edited in the **Properties** window. Click on the ellipse to select it. The ellipse properties contain the following pages:

Fill
Line
Drop Shadow
Coordinates

Create a North Arrow

A North arrow can be added to a map multiple ways. By drawing a symbol, the north arrow is a page object and cannot be layered with a map. By creating a post map or base map symbol, the map can be layered with other maps.

To Draw a North Arrow Symbol

This method allows you to draw a North arrow as a page object.

1. Create a new point object using the Home | Insert | Point command.
2. Left-click on the plot window where you want the north arrow to be displayed. The default symbol is drawn. Press the ESC key to exit the draw mode.
3. After the point is drawn, click once on the point to select it. The symbol properties are shown in the Properties window.
4. Change the *Symbol Set* to either *GSI Default Symbols* or *GSI North Arrows*.
5. Select a North arrow *Symbol* (i.e. *GSI Default Symbols Number 64* - 71, or *GSI North Arrows Number 1 - 25)*.
6. Change the *Size* to make your north arrow larger to fit your needs. Change the *Fill color* , *Line color, Fill opacity* , and *Line opacity* to fit your needs. The north arrow is displayed with the specified properties.
7. You can rotate the North arrow to the desired angle to make sure it lines up correctly with the North orientation. Select the symbol and use the **Layout | Layout | Arrange | Free Rotate | Rotate** command.
8. In the **Rotate** dialog, enter a value in the *Counterclockwise rotation in Degrees* box. Click *OK*. The North arrow is rotated the specified angle.

This North arrow is rotated to 25 degrees. The GSI North Arrows symbol set Number 9 is used.

To Add a North Arrow as a Post Map

This method allows you to specify exact X, Y map coordinates to display a North arrow.

1. Create a data file in the worksheet with the X, Y map coordinates where you want the North arrow to appear on the map. You can get the coordinates manually from your map, or by right-clicking on the map and selecting the Digitize option.

2. Click the **Home | New Map | Post** command, specify the .DAT file name, and click *Open* to display a new post map with default symbol at the coordinate location specified in the data file.

3. Click on the post map to select it. The properties are displayed in the **Properties** window.

4. On the **Symbol** page, open the *Symbol* section. Open the *Symbol Properties* section. Change any of the symbol properties.

5. Change the *Symbol Set* to either *GSI Default Symbols* or *GSI North Arrows*.

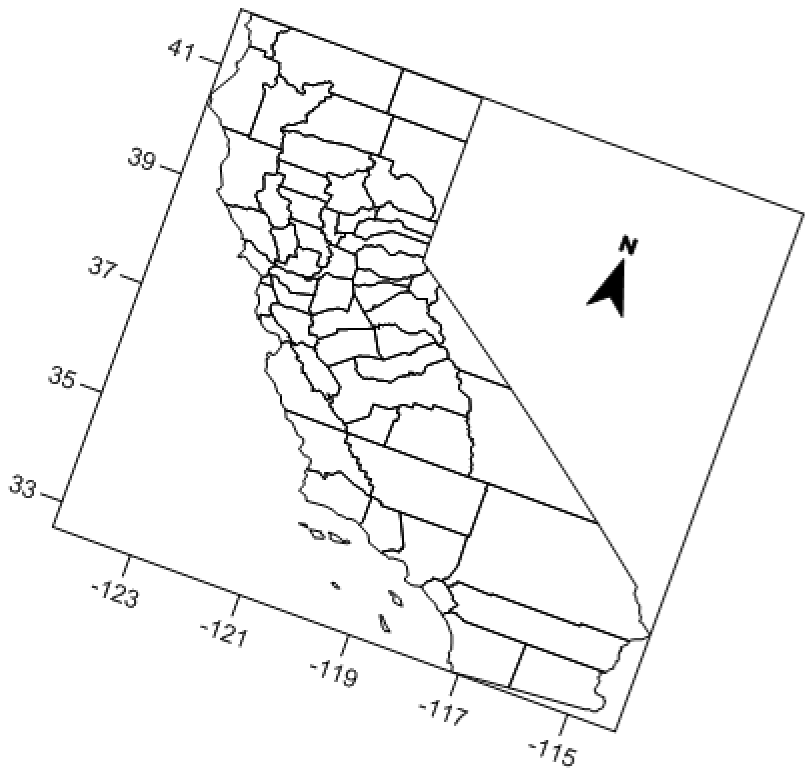
6. Select a North arrow *Symbol* (i.e. *GSI Default Symbols Number 64* - 71, or *GSI North Arrows Number 1 - 25)*.

7. Change the *Fill color*, *Line color,Fill opacity*, and *Line opacity* to fit your needs. The north arrow is displayed with the specified properties.

8. In the *Symbol Size* section, set the *Sizing method* to *Fixed size* and change the *Symbol size* to make your north arrow larger to fit your needs.

9. You can rotate the North arrow to the desired angle to make sure it lines up correctly with the North orientation. In the *Symbol Angle* section, change the *Default angle (degrees)* until the north arrow is aligned correctly.

10. Overlay the north arrow post map with other maps if desired.



This North arrow is a post map layer on a base map of California. The GSI North Arrows *symbol set* Number 6 *is used. The map is rotated to -20 degrees.*

To Add a North Arrow as a Base Map

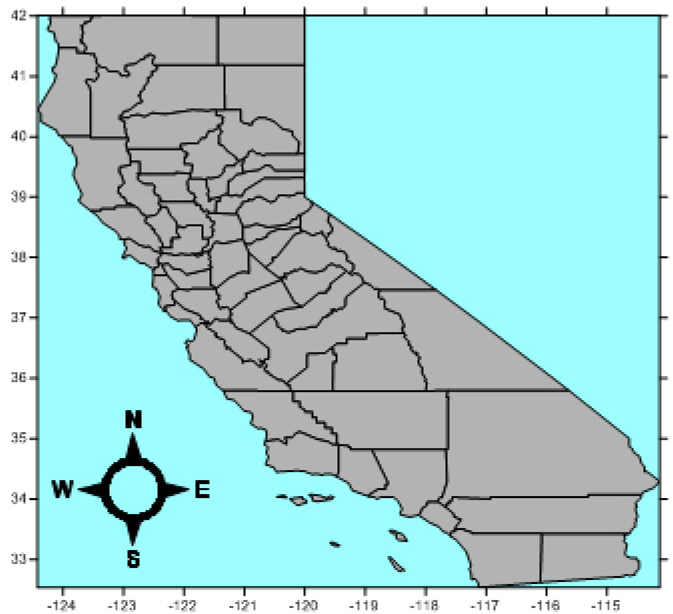
You can draw a North arrow symbol that will be included in the base map coordinates. The North arrow symbol will be a part of the base map and able to be layered and scaled.

1. Create a base map with the **Home | New Map | Base | Empty Basemap** command.
2. Create a new point object using the Home | Insert | Point command.
3. Left-click in the base map boundaries where you want the north arrow to be displayed. The default symbol is drawn. Press the ESC key to exit the draw mode.
4. After the point is drawn, click on the symbol to open the symbol properties in the **Properties** window.
5. Change the *Symbol Set* to either *GSI Default Symbols* or *GSI North Arrows*.

6. Select a North arrow *Symbol* (i.e. *GSI Default Symbols Number 64* - 71, or *GSI North Arrows Number 1 - 25)*.

7. Change the *Size* to make your north arrow larger to fit your needs. Change the *Fill color*, *Line color,Fill opacity*, and *Line opacity* to fit your needs. The north arrow is displayed with the specified properties.

8. If needed, rotate the North arrow to the desired angle to make sure it lines up correctly with the North orientation. Select the symbol and use the **Layout | Arrange | Free Rotate | Rotate** command.



The base map polygons are filled in gray. The base map background is blue.

Editing Objects

The following sections describe commands for editing drawn objects and features in base map layers.

Polyline to Polygon

Click the **Features | Change | Change To | Polyline to Polygon** command or right-click on a polyline and click **Polyline to Polygon** to convert one or more selected polylines or spline polylines into polygons. Each selected polyline is converted to a new separate polygon. To close the polygon, the first point and last point in the polyline are connected with a new line.

The new polygon uses the default fill properties and retains the line properties from the polyline. If the object name has not been changed from the default *Polyline*, the object is renamed *Polygon*. If the object name had been altered, the custom object name is retained.

Spline Polyline to Polygon

One way to get a smoothed polygon in **Surfer** is to create a spline polyline with the **Features | Insert | Spline Polyline** command. Click on the screen to create the polyline. Make sure that the last point is near the first point. Press ESC on the keyboard to end drawing mode. Click on the spline polyline to select it. Click the **Features | Change | Change To | Polyline to Polygon** command. The spline polyline is converted to a smooth polygon with enough nodes in the polygon to mimic the spline polyline appearance.

Polygon to Polyline

Click the **Features | Change | Change To | Polygon to Polyline** command or right-click on a polygon and click **Polygon to Polyline** to convert one or more selected polygons into polylines. Each selected polygon is converted to a new separate polyline.

The new polyline retains the line properties from the polygon. If the object name has not been changed from the default *Polygon*, the object is renamed *Polyline*. If the object name had been altered, the custom object name is retained.

Polyline to Points

The **Features | Change | Change To | Polyline to Points** command converts a polyline or spline polyline into points. Each vertex in the polyline is converted to a point when the **Polyline to Points** command is used.

The new points use the polyline's color and opacity Line properties to set the Symbol*Fill color*, *Fill opacity*, *Line color*, and *Line opacity* properties. The remaining symbol properties are set by the default symbol properties. If the object name has not been changed from the default *Polyline*, the resulting point objects are renamed *Point*. If the object name had been altered, the custom name is retained. The attributes from the polyline are also applied to the new points.

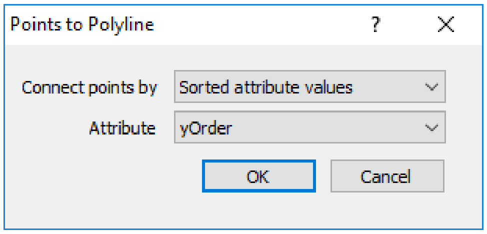
Points to Polyline

The **Features | Change | Change To | Points to Polyline** command converts two or more points to a polyline object. The points become vertices in a single polyline ordered by attribute or draw order in the Contents window.

The new polyline uses the color and opacity from the Symbol *Line color* and *Line opacity* properties. The remaining line properties are the default line properties. If the object name has not been changed from the default *Point*, the object is renamed *Polyline*. If the object name had been altered, the custom name is from the last point is retained. The attributes from the last point are also applied to the new polyline.

Points to Polyline Dialog

The **Points to Polyline** dialog is displayed when the **Features | Change | Change To | Points to Polyline** command is clicked. The **Points to Polyline** dialog specifies the sort criteria for connecting the points to create a polyline.



*Set the vertex sort order in the **Points to Polyline** dialog.*

Connect Points By

Select *List order* or *Sorted attribute values* in the *Connect points by* list.

- *List order* uses the draw order of the points to form the polyline. The draw order is indicated by the order in the **Contents** window from bottom to top.
- *Sorted attribute values* uses the values from the specified *Attribute* column to sort the points.

Attribute

The *Attribute* list is enabled when *Sorted attribute values* is selected in the *Connect points by* option. The *Attribute* list displays all attributes listed in the Info page or Attribute Table. Select the attribute you wish to use for sorting the points by clicking the current option and selecting an attribute from the list.

When all of the selected points attribute values are numbers, the points are ordered by attribute value numerically from smallest to largest. The selected points are sorted alphabetically by attribute value when the attribute values are strings or mixed strings and numbers.

OK and Cancel

Click *OK* to convert the points to a polyline. Click *Cancel* to close the dialog without converting the points.

Note about Base Maps

If objects should be edited in a base map, click on the object in the *Base* layer to select it. Click the **Features | Group | Start Editing** command to enter editing mode. Then, click the **Features | Change Type | Points to Polyline** command. The points are changed to a polyline. Click the **Features | Group | Stop Editing** to end object editing mode in the base map.

Reshape

Click the **Features | Edit Features | Reshape** command or the  button to move, add, and delete vertices within a selected polyline, polygon, 3D polyline, 3D polygon, or spline polyline. Objects such as metafiles, composites, base maps, polygons, polylines, 3D polylines and 3D polygons can be edited with **Reshape**.

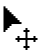
Entering Reshape Mode

Click the **Features | Edit Features | Reshape** command to enter the reshape mode. Alternatively, you can right-click on the object and click **Reshape**. After selecting **Reshape**, the cursor will change to  to indicate reshape mode. When you select an object that can be reshaped (a polyline, spline polyline, or polygon), all the vertices in the selected object are shown with hollow squares. Reshape mode is persistent and you can reshape multiple items until you exit the reshape mode. After you reshape an object, select another object to reshape, or exit reshape mode.

Exiting Reshape Mode

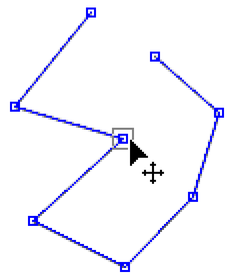
To exit reshape mode, press the ESC key on the keyboard. Alternatively, select another command mode.

Select Vertices

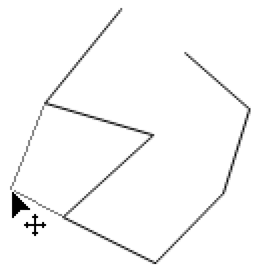
- Left-click on a vertex to select it. The selected vertex is indicated by a solid green square.
- To select the first vertex, press the HOME key. To select the last vertex, press the END key.
- To shift the selected vertices forward by one position, press the TAB key. To shift the selected vertices backward by one position, hold the SHIFT key and press the TAB key.
- To select multiple vertices, hold down the SHIFT key and left-click additional vertices or left-click and drag the cursor to make a rectangular block selection. Vertices can be added or removed from the block selected vertices by holding down the SHIFT key.
- Hovering the mouse over an unselected vertex will display a grey highlight around the vertex indicating it may be selected or dragged.
- The cursor will change to a  when it is over a vertex to indicate the vertex may be selected or dragged.
- If you have multiple polyline or polygon objects in the plot window, you can edit multiple objects while in the reshape mode. Vertices can only be edited for the selected object.

Move Vertices

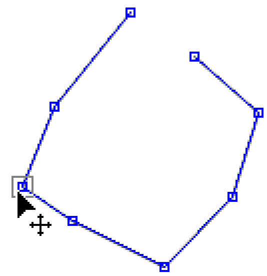
Once a vertex or vertices are selected and the cursor display is a , hold the left mouse button and drag the vertex to a new location. Release the left mouse button to place the vertex in the new location.



Enter reshape mode and select a vertex.



Drag the vertex to a new location. The original line is displayed in addition to the new line that will connect to the new vertex position.



Release the mouse button and the vertex is moved.

Alternatively, place the cursor over the vertex, hold the SPACEBAR, and use the ARROW keys on the keyboard to move the vertex to a new location.

Deselect Vertices

- A selected vertex can be deselected by holding down the SHIFT key and left-clicking the vertex.
- All vertices can be deselected by clicking in an unused space.
- Pressing ESC while dragging will cancel the drag. Pressing ESC while NOT dragging will exit the reshape tool.
- Each individual edit can be undone using the Undo command.

Add Vertices

To enter insert mode, hold down the CTRL key. The cursor will change to . Left-click anywhere in the plot window, or on the existing object and a new node will be added at the closest point on the existing object.

Remove Vertices

Select a vertex or multiple vertices with one of the methods outlined above. Press DELETE to remove the selected vertex/vertices, and the next vertex in the object is selected. You can hold DELETE to remove contiguous vertices as the selection moves throughout the object. Degenerate polylines (one-point polylines) and degenerate polygons (one- or two-point polygons) can be created by removing vertices. The objected can be deleted by removing all of the vertices. However, spline polylines are deleted when there are fewer than two knot vertices.

Status Bar

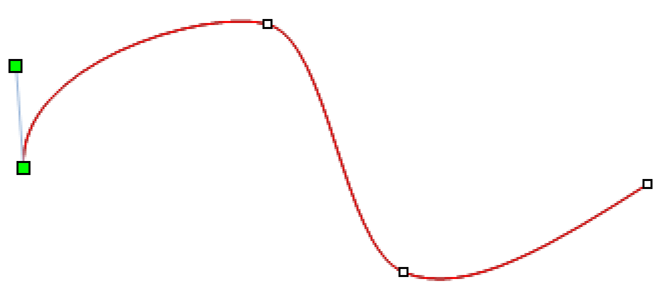
When the reshape tool is active, the status bar updates with current position in world coordinates and map coordinates (if available).

To edit a polyline, polygon, 3D polyline or 3D polygon

1. Select the object.
2. Click the **Features | Edit Features | Reshape** command.
3. The arrow pointer turns into an arrowhead pointer and all the vertices appear as small hollow squares.
4. To move a vertex, left-click on the vertex with the mouse and drag it to a new location. To add a vertex, hold down the CTRL key and click the area on the polygon or polyline. To delete a vertex, select it and then press the DELETE key.
5. After reshaping the object, press the ESC key to exit edit mode.

To edit a spline polyline

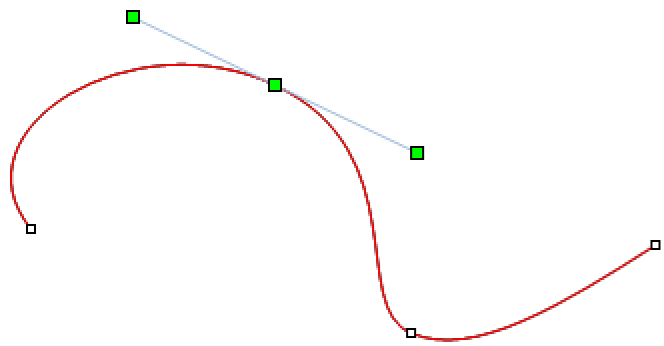
1. Select the spline polyline in the **Contents** window or the plot window by clicking on it.
2. Click the **Features | Edit Features | Reshape** command to enter editing mode.
3. Anchor points appear as white boxes on the spline polyline. Click on an anchor point to select it.
4. For anchor points that are the first or last point on the spline polyline, two control points will be active. The control point that is located at the exact anchor point controls the position of the anchor point. Click and drag the green control point to a new location to move the line. The control point that is connected by a line to the anchor point controls the degree of curvature of the line. Click and drag the green control point to make the line more or less curvy. The closer the control point is to the line, the straighter the line will be.



The end anchor point is being edited. The top green control point controls the curvature of the spline polyline. The bottom green control point controls the location of the anchor point.

5. For anchor points that are not the first or last point on the spline polyline, three control points will be active for each anchor point. The three control

points are connected by a light blue line. The middle control point determines the location of the anchor point. The two outer control points determine the curvature of the line. Click the outer control points and drag to a new location to make the line more or less curvy. The line connecting the control points will always be tangent to the anchor point on the spline polyline. The shorter the connecting line, the sharper the angle at the anchor point. The longer the connecting line, the smoother the curve at the anchor point.



A middle anchor point is being edited. The left and right green control points control the curvature of the spline polyline. The center green control point determines the location of the anchor point.

6. To add points to the spline polyline, hold down the CTRL key and click on the line. A new control point is added.
7. To delete points, click on an existing point and press the DELETE key on the keyboard.

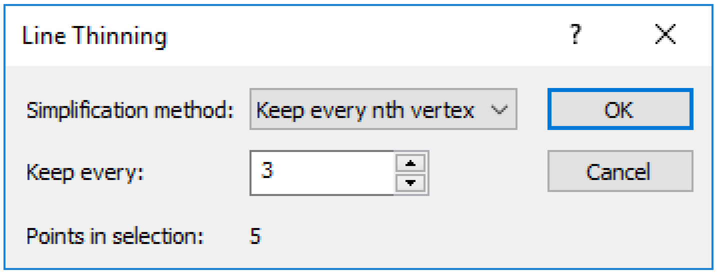
The contents of all map layers are always constrained within the map limits. If an object exceeds the map limits, the object is clipped. Change the map limits to expand the map limits to include reshaped contents. Check the *Use data limits* box to resize the map limits to fit your edited object.

Thin

Click on a polyline or polygon and click the **Features | Edit Features | Thin** command or the button or right-click on the polyline or polygon and click **Thin** to open the **Line Thinning** dialog. The **Line Thinning** dialog removes or thins unnecessary points in a selected polyline or polygon from the line. The selected polyline or polygon automatically updates as options are changed in the **Line Thinning** dialog. Multiple polylines or polygons can be selected before selecting the command.

Line Thinning Dialog

The **Line Thinning** dialog controls the simplification method and properties for thinning polylines and polygons.



*Set the options in the **Line Thinning** dialog to remove points from a selected polyline or polygon.*

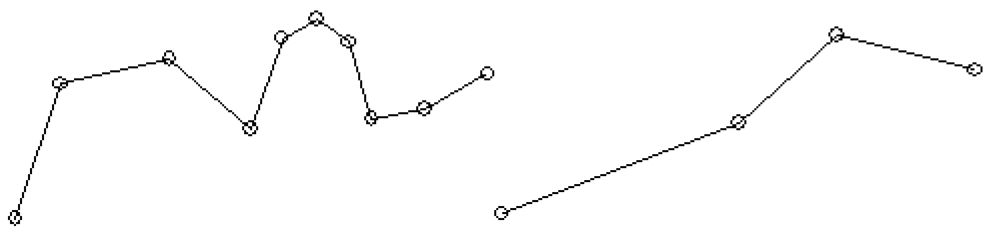
Simplification Method

The *Simplification method* controls how the points are selected to be removed from the polyline or polygon. Available options are *Keep every nth vertex*, *Vertex averaging*, and *Deviation distance*. To select the method, click on the existing option and select the desired method from the list. Depending on the selected *Simplification method*, a *Keep every*, *Average every*, or *Deviation Distance* option will appear.

Keep Every nth Vertex

The *Keep every nth vertex* option keeps every nth vertex point from the polyline or polygon, and then reconnects the polyline segments with only the new points. The value of "n" is entered into the *Keep every* field. For example, if the *Keep every* is set to 3, the first point is kept, the next two vertices are removed, the forth point is kept, and so on.

Set the *Keep every* value based on the number of points to remove. To change the *Keep every* value, highlight the current value and type a new value. Alternatively, click the  to increase or decrease the value.

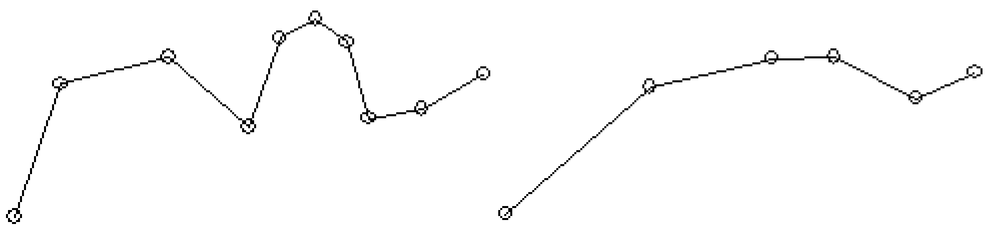


The original polyline is on the left. The polyline after a Keep every of 3 was applied is shown on the right.

Vertex Averaging

The *Vertex averaging* option preserves the first and last point in a polyline, but averages the vertices along the polyline based on the number set in the *Average every* field. For example, a polyline that has 10 vertices when averaged using a rate of 3 yields a polyline with 6 vertices. The new vertices are located based on the averaged locations of the previous surrounding vertices.

Set the *Average every* value based on the number of points to average to create new points in the polyline. To change the *Average every* value, highlight the current value and type a new value. Alternatively, click the  to increase or decrease the value.

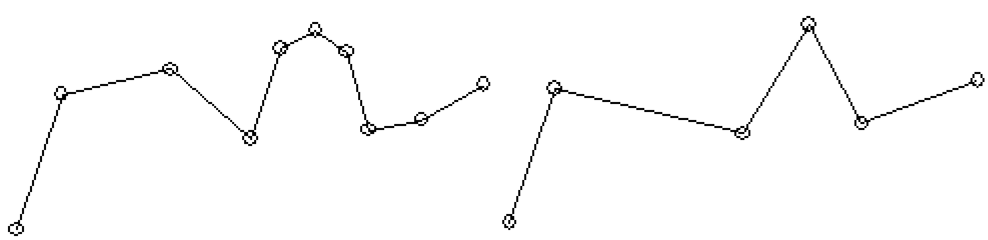


The original polyline is on the left. The polyline after an Average every *value of 3 was applied is shown on the right.*

Deviation Distance

The *Deviation distance* option controls how many points are removed by the thinning process. Points closer than the *Deviation distance* from the general trend of the polyline are removed. A value of 0 removes any duplicate or collinear points. A value of .1 removes all points that are .1 coordinates or closer to the general trend of the polyline. The larger the *Deviation distance*, the more points are removed from the polyline. The *Deviation distance* is based on the Douglas Peucker algorithm.

Set the *Deviation Distance* to the distance away from the general trend line to remove points. To change the *Deviation Distance* value, highlight the existing value and type a new value. The *Deviation Distance* is in the local units for the objects. Therefore, if multiple objects are selected, the objects should all use the same local units.



The original polyline is on the left. The polyline after applying a Deviation distance *of 0.50 is shown on the right.*

OK or Cancel

Click *OK* to make simplify the lines and return to the plot window. Click *Cancel* to return to the plot window without simplifying the lines. The lines retain their original shape.

Preview

As the *Simplification method* and applicable options are changed in the **Line Thinning** dialog, the selected objects update automatically in the plot window. This is a preview, and the changes are not saved unless *OK* is clicked in the **Line Thinning** dialog.

References

Douglas, David H and Peucker, Thomas K., Algorithms for the Reduction of the Number of Points Required to Represent a Digitized Line or its Caricature, *The Canadian Cartographer*, Vol 10, No 2, p 112-122.

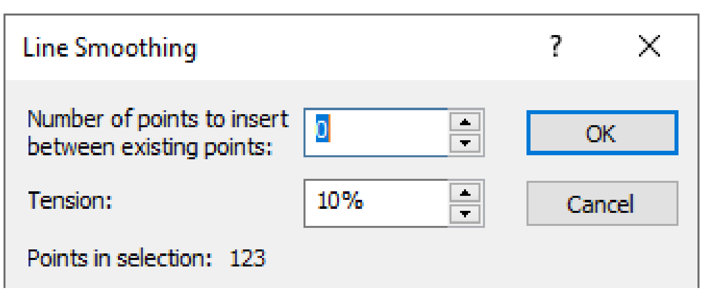
Smooth

Click on a polyline or polygon and click the **Features | Edit Features | Smooth** command or the button or right-click on the polyline or polygon and click **Smooth** to open the **Line Smoothing** dialog. The **Line Smoothing** dialog adds points to a selected polyline or polygon to smooth the line. Smoothing removes or lessens jagged sections in polylines. The selected polyline or polygon automatically updates as options are changed in the **Line Smoothing** dialog. Multiple polylines or polygons can be selected before selecting the command.

The **Features | Edit Features | Smooth** command is a good way to create a spline polygon. Create the polygon using the Polygon command. Then, click the **Features | Edit Features | Smooth** command to add additional points between the drawn points on the polygon. A smooth polygon is created.

Line Smoothing Dialog

The **Line Smoothing** dialog controls the number of points and tension when smoothing polylines or polygons.



*Set the options in the **Line Smoothing** dialog to add points and control the tension for smoothing a selected polyline or polygon.*

Number of Points to Add

Smoothing produces a uniform polyline that passes through all of the data points, regardless of the spacing of the data points or the tension factor applied to the smoothing. Set the *Number of points to insert between existing points* value to the desired number of points to add between existing point. If set to 3, 3 points are added between existing points. For example when *Number of points to insert between existing points* is 3 and a line contains 10 points initially, the smoothed line will contain 37 points. To change the *Number of points to insert between existing points* value, highlight the existing value and type a new value. Alternatively, click the to increase or decrease the value.

Tension

The *Tension* can range from 0 to 100%. Higher tension factors result in straighter polylines between the data points and lower tension factors result in more curvature. To change the *Tension* value, highlight the existing value and type a new value. Alternatively, click the to increase or decrease the value.

OK or Cancel

Click *OK* to make the lines smoother and return to the plot window. Click *Cancel* to return to the plot window without smoothing the lines. The lines retain their original shape.

Preview

As the line smoothing options are changed in the **Line Smoothing** dialog, the selected objects update automatically in the plot window. This is a preview, and the changes are not saved unless *OK* is clicked in the **Line Smoothing** dialog.

References

For additional information on the spline smoothing method, refer to either of these resources.

- Renka, R. J. Interpolatory tension splines with automatic selection of tension factors.*SIAM J. Sci. Stat. Comput. 8* (1987), pp. 393-415.
- Renka, Robert J. TSPACK: Tension Spline Curve Fitting Package. 05/27/91

Crop Image

Click the **Features | Image | Crop** command or the button to interactively crop an image. When **Crop** is enabled, the image extents can be adjusted by clicking and dragging the sides or corners of the image limits rectangle or by drawing a new limits rectangle while holding CTRL. The image limits rectangle can be clicked and dragged to move. An image or Base(raster) layer must be selected to enable the **Features | Image | Crop** command.

The plot window appearance changes when the **Crop** command is clicked. The areas outside the image limits are shaded. Regions of the plot window within the image limits are displayed normally. The image limits are represented by a blue rectangle with yellow selection handles. Additionally, the **Crop** command button is highlighted when **Crop** is active.

Crop Image Tips

The following keyboard and mouse actions control the **Crop** command mode.

- Click and drag the yellow selection handles to move the map limits.
- Press and hold CTRL, then click and drag to draw a rectangle to specify the map limits. The cursor will change to a crosshair to indicate draw mode.
- Double-click in the plot window or press ENTER to update the map with new map limits and exit **Crop** mode. Alternatively, right-click in the plot window and click **Save** to update the image and exit **Crop** mode.
- Press ESC to exit **Crop** mode without making any changes to the map. Alternatively, right-click in the plot window and click **Cancel** to exit **Crop** mode without making changes.
- Use the mouse wheel or scroll bars to zoom and pan the plot window.
- The Undo command will undo changes to the image limits after you press ENTER or double-click the plot window. However, using the **Undo** command while **Crop** is activated will undo actions made prior to activating **Crop** mode. If you wish to make changes to the image limits while still in **Crop** mode, click and drag the selection handles or press CTRL and draw a new rectangle.
- The **Features | Image | Crop** command is disabled for tilted images/maps, rotated images/maps, Base(raster) layers with non-affine warps, and Base (raster) layers that are geotransformed. Crop the image as needed before applying one of these changes.

Connect Polylines

Click the **Features | Edit Polylines | Connect Polylines** command or the button or right-click on the selection and click *Connect Polylines* to connect selected polylines and spline polylines. The polylines are connected at the closest endpoints on each line. For the command to be available, two or more polylines, 3D polylines or spline polylines must be selected. 3D polylines cannot be connected to polylines or spline polylines.

The new polyline or 3D polyline retains the line properties from the polyline at the bottom of the Contents window. If the object name has not been changed from the default *Polyline*, the object is renamed *Polyline*. If the object name had been altered, the custom object name from the bottom polyline is retained.

Break Polyline

Click the **Features | Edit Polylines | Break Polyline** command or the button or right-click a polyline or 3D polyline and click **Break Polyline** to break the selected polyline, 3D polyline or spline polyline into multiple polylines. The cursor changes to to indicate break mode. Click on the polyline, 3D polyline or spline polyline in the location where the line should be broken. For the command to be available, a single polyline, 3D polyline, or spline polyline must be selected. Spline polylines are converted to regular polylines after being broken apart. 3D polylines continue as 3D polylines.

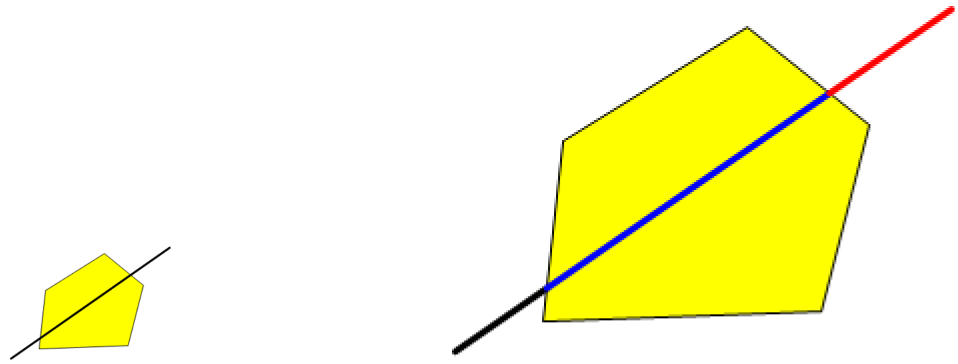
The new polylines retain the line properties from the original polyline. If the object name has not been changed from the default *Polyline*, *3D Polyline*, or *Spline Polyline*, the new objects are named *Polyline*. If the object name had been altered, the custom object name from the original polyline is retained for both new polylines.

Break Polyline at Intersections

The **Features | Edit Polylines | Break at Intersections** command breaks a selected polyline, spline polyline, or combination of polylines and spline polylines at every intersection with another object, such as polygons, polylines, spline polylines, rectangles, rounded rectangles, and ellipses. The original line properties are applied to the created polylines. Only the polylines or spline polylines to be broken at intersections need to be selected. After a polyline is broken, multiple polyline objects are created based on the boundary intersections with the original polyline. When a spline polyline is broken, the resulting lines are converted to polylines.

To break a polyline:

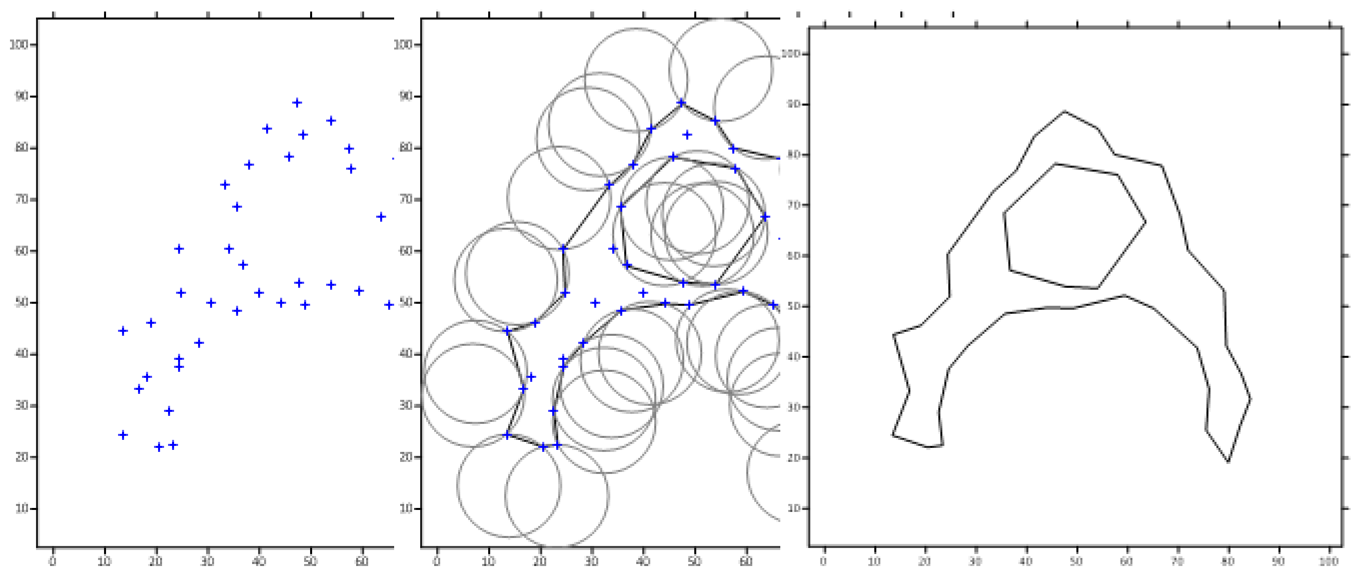
1. Select only the polyline that should be broken at its intersections with other objects. It is not necessary to select the intersecting objects.
2. Click the **Features | Edit Polylines | Break at Intersections** command or the button. Alternatively, right-click on the selected polyline, and click **Break at Intersections** in the context menu.



In this example, the polyline in the image on the left is broken into three polylines in the image on the right since the polyline crosses the polygon twice. The line color was changed to show the three created polylines.

Alpha Shape

The **Features | New Features | Alpha Shape** command creates a polygon boundary around a set of three or more selected data points in a base layer or a base layer created from a data file. The resulting polygon is an alpha shape, which is a subset type of Delaunay triangulation. Holes will be created within the alpha shape where there is no data. Edges of the alpha shape are trimmed so a closed boundary is formed that conforms to the number of the alpha value.



Alpha shapes can be created from drawn point features or points within a base layer.

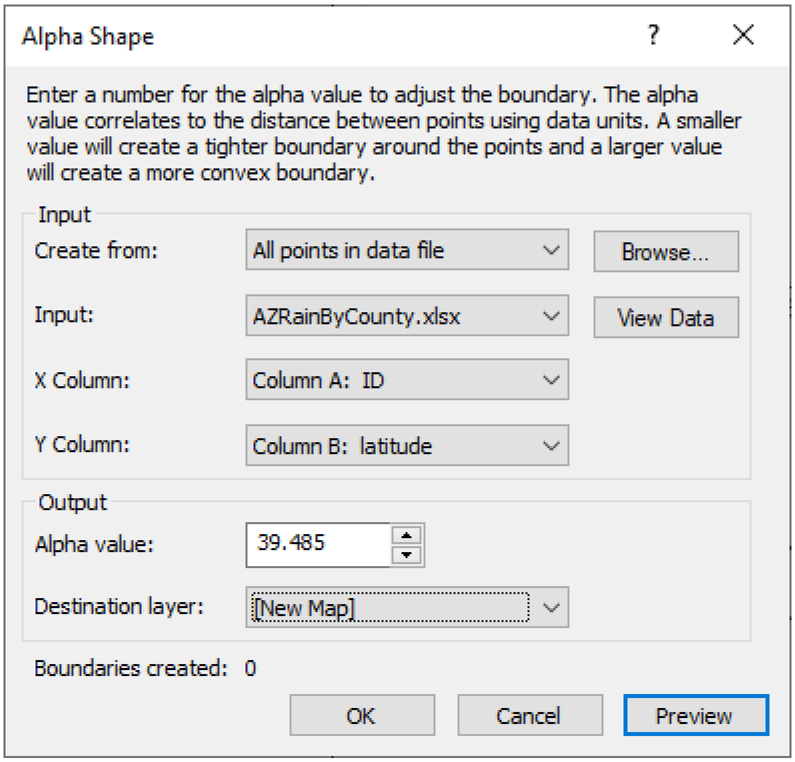
The alpha value determines the radius of the alpha circles. The edges are drawn based on Delaunay Triangulation.

The edges are trimmed so the boundary of the alpha shape is complete. Where there is no data, a hole is drawn.

Creating an Alpha Shape

The following steps describe how to create an alpha shape.

1. Click the **Features | New Features | Alpha Shape** command or the  button to display the **Alpha Shape** dialog.

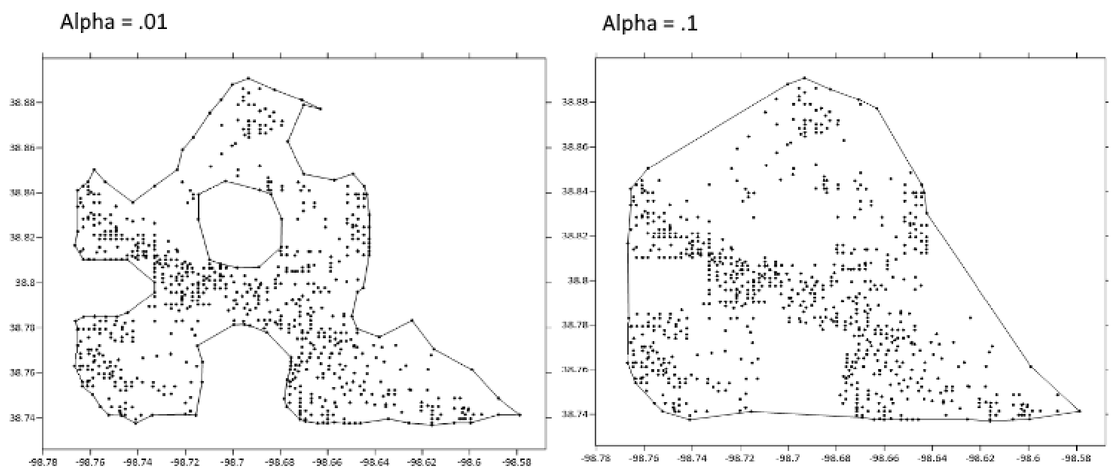


***Alpha Shape** creates a polygon based on the alpha value.*

2. In the **Alpha Shape** dialog, select a *Create from* value. Depending on the *Create from* value selected, the **Alpha Shape** dialog will add or remove options.
    a. If the selected *Create from* value is *All points in a data file*, click the *Browse* button to select and add a data file to the *Input* list. Use the *X Column* and *Y Column* lists to select the XY data from the selected input file. Click the *View Data* button for a quick view of the data file.
    b. If the selected *Create from* value is *All points on a layer*, select the layer from the *Input* list.
    c. If the selected *Create from* value is *Selected points only*, *Input* will be grayed out.
3. Select the desired *Alpha value*. The *Alpha value* defines the radius of the circles that are created by the edges of the triangulation of the points. The units are in the units of the layer.
    - The alpha shape depends on both data density and the alpha value. The alpha shape, despite the alpha value, will not extend past the points.

- Larger alpha values create a more convex boundary and smaller alpha numbers create a tighter, concave boundary.
- With small alpha values, the alpha radii can pass through the data set, and the alpha shape can form several smaller shapes.
- When working in latitude and longitude, use alpha values greater than 0 and less than 10.
- When working in meters, use alpha values of 10 or greater.
- If the selected alpha value is too small or too large, during its preview calculations, the **Alpha Shape** dialog will show a message to try a different alpha value.

4. Select the desired *Destination layer*. The values in this list depend on the value selected in the *Create from* list.
    - If *All points on layer* is selected from the *Create from* list, and the input layer is a post or classed post map, the values in the *Destination Layer* list will be *New Layer* and *New Map*. *New Layer* will add the alpha shape to the same map as the input layer, and *New Map* will create a new map that includes the alpha shape. If all *All points on layer* is selected from the *Create from* list, and the input layer is a base map, the values in the *Destination Layer* list will be *New Layer*, *New Map*, and the name of the input layer. *New Layer* will add the shape to a new layer, *New Map* will create a new map that includes the alpha shape, and selecting the input layer will add the shape to that layer.
    - If *All points in data file* is selected from the *Create from* list from an empty plot window, the only *Destination Layer* value will be *New Map*. If *All points in data file* is selected from the *Create from* list with an existing map in the plot window, and *New Layer* is selected from the *Destination Layer* list, the layer will be added to the bottom map in the **Contents** window.
    - If *Selected points only* is selected from the *Create from* list, all three *Destination layer* values will be available. *New Layer* will add the shape to the same layer as the input layer, *New Map* will create a new map that includes the alpha shape, and selecting the input layer will add the shape to that layer.
5. After defining the alpha shape, click the *Preview* button to generate a preview of the alpha shape and display the number of *Boundaries created*. The number of boundaries created is dependent on the *Alpha value*. Preview calculations may display a message about adjusting the alpha value.
6. After previewing the alpha shape, either reconfigure the alpha shape settings or click *OK* to close the **Alpha Shape** dialog and add the shape to the selected layer.



Example with alpha = .01 and alpha = .1

References

Gardiner, James, Julia Behnsen and Charlotte A. Brassey, *Alpha shapes: determining 3D shape complexity across morphologically diverse structures*, BMC Evolutionary Biology, 18, Article number: 184, 2018.

Edelsbrunner, Herbert, and Ernst P Mucke, *Three-dimensional Alpha Shapes*, University of Illinois at Urbana-Champaign, https://arxiv.org/pdf/math/9410208.pdf.

Union Polygons

The **Features | New Features | Union of Polygons** command automatically creates a new polygon by combining multiple objects. This command can be used with any combination of existing polygons, rectangles, rounded rectangles, and/or ellipses. The original objects are retained or removed based on the *Keep original objects* selection in the Union Polygons dialog.

Selections including objects that do not touch or intersect result in a complex polygon, similar to the Combine Island/Lakes command when combining islands.

Sometimes selections made up of many contiguous objects, for example states or provinces in a country, have objects that do not completely touch or intersect. If objects do not touch or intersect along the entire border, the **Union Polygons** command results in a complex polygon with small lakes. To remove the lakes, use the Split Islands/Lakes command, and then delete the interior polygons. Alternatively, use the Reshape command to delete the vertices for the lakes.

To use the **Union Polygons** command

1. Select all the objects that you wish to combine into a single polygon.
2. Click the **Features | New Features | Union of Polygons** command or the  button.

3. Select whether or not to keep the original objects in the Union Polygons dialog.
4. Click *OK* in the **Union Polygons** dialog, and the selected objects are combined into a single polygon. The new polygon is selected when the process is completed.

*With **Union Polygons**, you do not have to preserve the original objects. In this example, four new polygons are based on groups of counties. When creating the example on the left, Keep original objects was not checked. Keep original objects was checked when creating the example on the right.*

Difference between Combine Islands/Lakes and Union Polygons

This command differs from the Combine Islands/Lakes command. With the **Union Polygons** command, a new polygon is created from the selected objects. With **Combine Islands/Lakes**, the selection is combined into a single complex polygon. The original objects become islands and lakes, and a new polygon is not created.

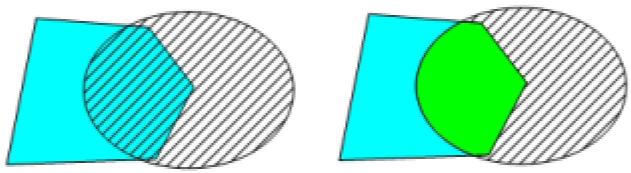
- Contiguous objects are part of the created polygon with the **Union Polygons** command. With the **Combine Islands/Lakes** command, contiguous objects are separate islands in a single complex polygon.
- Intersecting areas of the selected objects are treated as part of the new polygon when the **Union Polygons** command is used. When the **Combine Islands/Lakes** command is used, intersecting portions of the selection are treated as lakes, or holes, in the complex polygon.
- The **Combine Islands/Lakes** and **Union Polygons** commands behave similarly only when objects in the selection do NOT touch or intersect. However, the **Combine Islands/Lakes** command combines the selection into a complex polygon. The **Union Polygons** command creates a new complex polygon, and the original objects can be removed or kept.

For example, if you want to indicate an outer boundary with a heavy line and preserve the inner boundaries with thinner lines as in the example above, you can accomplish this with the **Union Polygons** command, not with the **Combine Islands/Lakes** command.

Intersect Polygons

The **Features | New Features | Intersect Polygons** command creates a new polygon from two or more intersecting objects. The **Intersect Polygons** command can be used with combinations of polygons, rectangles, rounded rectangles, and ellipses. The new polygon is created from the overlapping area of the selected objects (see image below). The overlapping area must include portions of each selected object for a new polygon to be created. If the objects overlap in more than one place, a complex polygon is created. Click the Split Islands/Lakes command to separate the complex polygon into multiple polygons.

Clicking the **Intersect Polygons** command opens the Intersect Polygons dialog, where you can decide whether or not to keep the original selected objects. If the original objects are kept, you can emphasize the intersecting polygon by applying a unique line and/or fill style to the new polygon, for example. If you only want to use the intersection polygon, the *Keep original objects* option can be unchecked.



*Use the **Intersect Polygons** command to create a new polygon (green on right) from intersecting objects (left).*

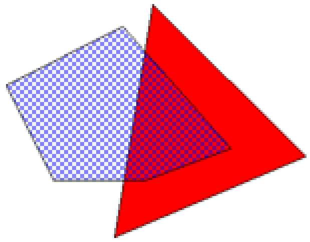
To use the **Intersect Polygons** command:

1. Select two or more overlapping objects.
2. Click the **Features | New Features | Intersect Polygons** command or the  button, or right-click the selection and click **Edit Boundaries | Intersect Polygons** in the context menu.
3. Select whether or not to keep the original objects Intersect Polygons dialog.
4. Click *OK* in the **Intersect Polygons** dialog and the intersecting polygon is created.

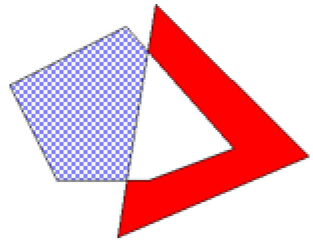
Difference of Polygons

The **Features | New Features | Difference of Polygons** command creates new polygons that do not contain overlapping portions of the selected objects. Polygons, rectangles, rounded rectangles, and ellipses can be used with the **Difference of Polygons** command. When the selected objects do not overlap, the created polygons duplicate the selected objects.

Clicking the **Difference of Polygons** command or right-clicking the selection and clicking **Edit Boundaries | Difference of Polygons** in the context menu opens the Difference of Polygons dialog. The original objects are removed unless the *Keep original objects* option is checked in the **Difference of Polygons** dialog. When the *Keep original objects* option is checked, the new polygons are created without overlapping areas, and the original objects remain unchanged.



These two polygons are selected.



*After **Difference of Polygons**, two new polygons are created without the intersecting area.*

Creating a Difference of Polygons:

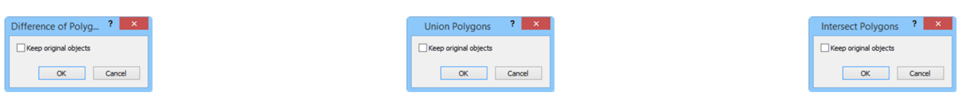
To create a difference of polygons:

1. Select two or more polygons.
2. Click the **Features | New Features | Difference of Polygons** command or the button, or right-click the polygons and click **Edit Boundaries | Difference of Polygons** in the context menu.
3. Select whether to keep the original objects in the Difference of Polygons dialog.
4. Click *OK* in the **Difference of Polygons** dialog to create new polygons without the overlapped areas.

You may wish to keep the original objects if the overlapping areas need to be represented by a polygon. For example, using the **Difference of Polygons** command with *Keep original objects* checked, then using the Intersect Polygons command with *Keep original objects* unchecked results in separate polygons representing overlapping and non-overlapping portions of the original objects.

Difference of Polygons, Union Polygons, and Intersect Polygons Dialogs

The **Difference of Polygons** dialog is displayed after clicking the Difference of Polygons command. The **Union Polygons** dialog is displayed after clicking the Union Polygons command. The **Intersect Polygons** dialog is displayed after clicking the Intersect Polygons command.



*Keep or discard original objects with the **Difference of Polygons** , **Union Polygons** , and **Intersect Polygons** dialogs.*

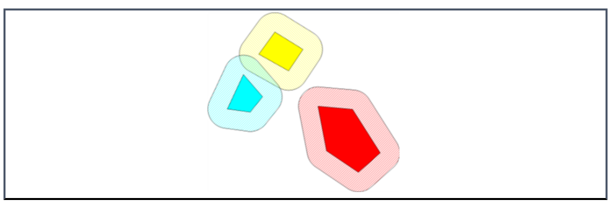
Keep Original Objects

When *Keep original objects* is checked, a new polygon is created while leaving the original selected objects unchanged. The new polygon is created, and the original objects are deleted when *Keep original objects* is unchecked.

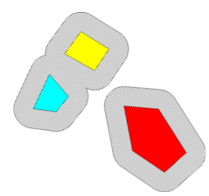
Buffer

The **Features | New Features | Buffer** command creates a polygon around or within an object or group of objects at a specified distance. Any number or combination of points, polylines, spline polylines, polygons, rectangles, rounded rectangles, and ellipses can be selected before clicking the **Buffer** command. A polygon will be created around each selected object.

If *Combine overlapping buffer polygons* is checked in the **Buffer** dialog, the newly created buffer polygons are combined as with the Union Polygons command. For detailed explanation of the combination operation, see the *Difference between Combine Islands/Lakes and Union Polygons* section of the Union Polygons help page. The following images provide a visual example of the difference between using the **Buffer** command with and without the *Combine overlapping buffer polygons* option checked.



*The Combine overlapping buffer polygons option was not checked in this example. Notice the overlapping region between the yellow and blue buffer polygons. Three new polygons were added to the plot window and **Contents** window.*



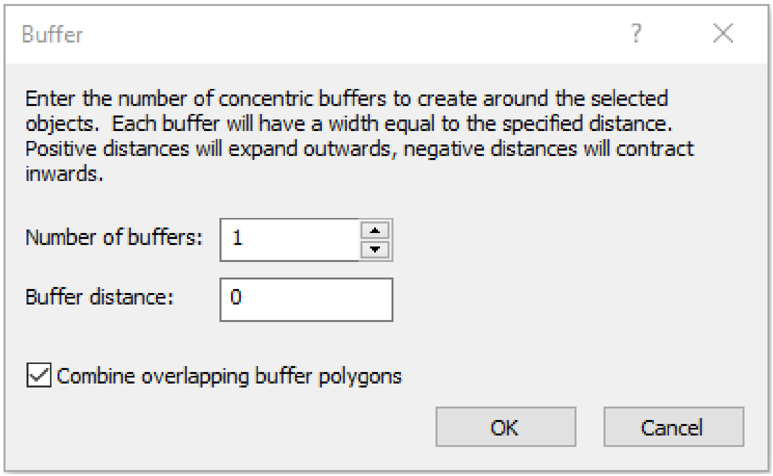
*The Combine overlapping buffer polygons option was checked for this example. Notice the buffer polygons created from the yellow and blue polygons have been combined. Here the buffer polygons are a single complex object consisting of two sub-polygons. Only one polygon was added to the **Contents** window.*

Creating a buffer polygon

1. Select the object around which to create a buffer polygon by clicking the object in the plot window or Contents window.
2. Click the **Features | New Features | Buffer** command or the button.
3. Specify the Number of buffers and Buffer distance and select whether to Combine overlapping buffer polygons in the **Buffer** dialog.
4. Click *OK*, and the buffer polygon is created.

Buffer Dialog

The options for the **Buffer** command are located in the **Buffer** dialog. Open the **Buffer** dialog by selecting one or more objects and clicking the **Features | New Features | Buffer** command.



*The number of buffers and buffer distance is specified in the **Buffer** dialog.*

Number of Buffers

Specify the number of buffers you wish to create around each object in the *Number of buffers* field. Type a number into the field or click the buttons to change the value. The first buffer is created at the *Buffer distance* from the object, and subsequent buffer polygons are created at the *Buffer distance* from the previous buffer polygon.

Buffer Distance

Specify the distance to draw the buffer polygon from the selected object or group of objects in the *Buffer distance* field. Positive distances expand the buffer polygon. Negative distances contract the buffer polygon. A buffer polygon with a negative *Buffer distance* is not created for points or polylines.

The *Buffer distance* is specified in the source coordinate system distance units, e.g. degrees, feet, meters, or miles. For objects that are not part of a base layer, the *Buffer distance* is in page units (inches or centimeters).

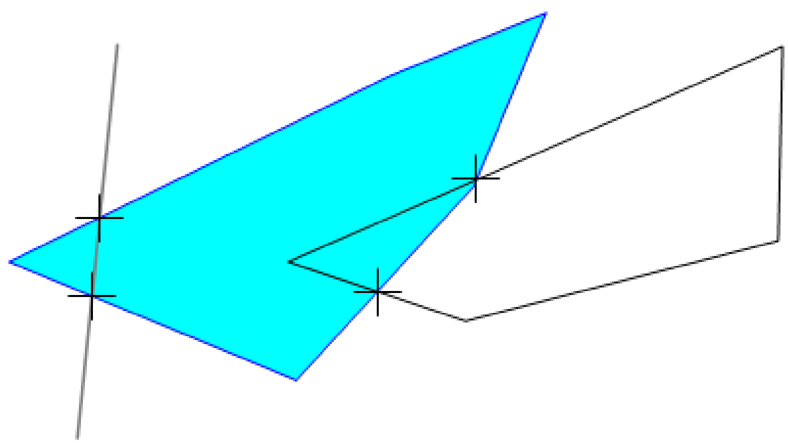
Combining Buffer Polygons

Click the *Combine overlapping buffer polygons* check box to combine the buffer polygons after they are created. Overlapping buffer polygons will combine to form a single polygon. When the buffer polygons do not overlap, the buffer polygons are combined into a single complex polygon.

The buffer polygons are combined as with the Union Polygons command. For detailed explanation of the combination operation, see the *Difference between Combine Islands/Lakes and Union Polygons* section of the Union Polygons help page.

Create Intersection Points

Click the **Features | New Features | Intersection Points** command to create a point at each place selected objects intersect with one another. Points are created at all object intersections, and the original objects remain unmodified. The **Intersection Points** command can be used with any combination of polylines, spline polylines, polygons, rectangles, rounded rectangles, and ellipses.



***Intersection Points** creates points at each place selected objects intersect.*

Creating Intersection Points

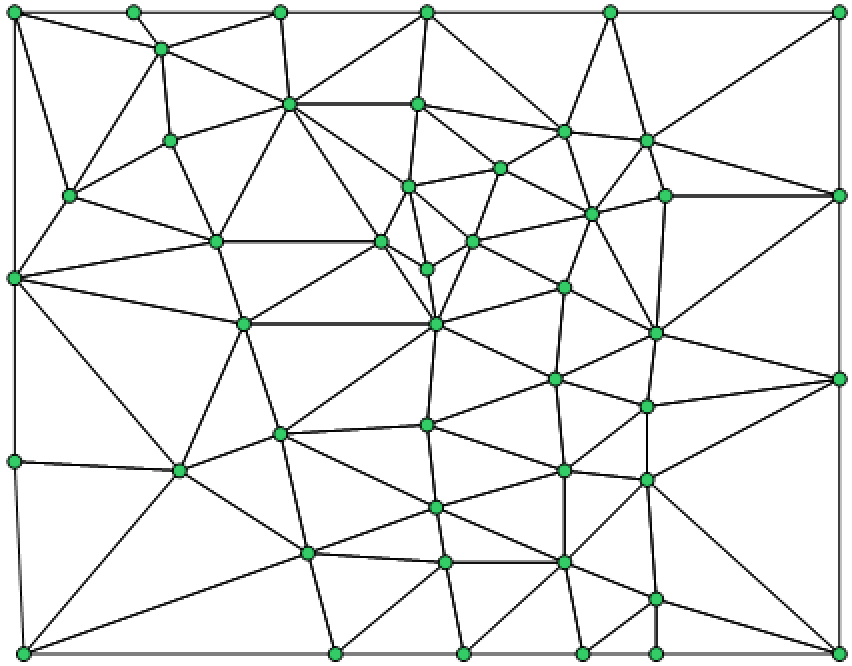
To create points at intersections:

1. Select two or more intersecting objects.
2. Click the **Features | New Features | Intersection Points** command or the button.
3. Points are created at every intersection.

Triangulation

The **Features | New Features | Triangulation** command performs a Delaunay triangulation on the selected points. A Delaunay triangulation for a set of points is a triangulation such that no point is within any triangle. The triangles can be polylines or polygons. None of the triangles are intersected by other triangles.

A triangulation diagram can be created from drawn point features or points within a base layer.



***Triangulation** creates triangles from selected points.*

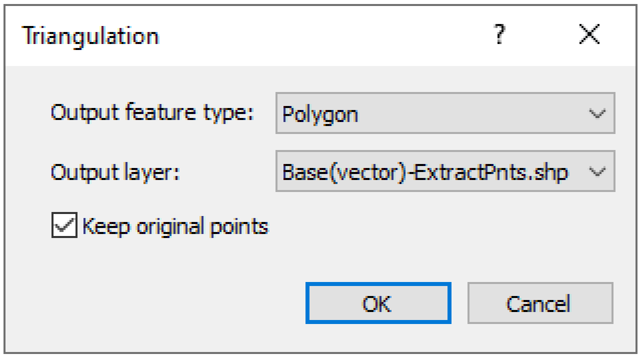
Creating a Triangulation Diagram

The following steps describe how to create a triangulation diagram:

1. Select three or more points from which to create the diagram.
2. Click the **Features | New Features | Triangulation** command or the button.
3. Select the desired options in the **Triangulation** dialog.
4. Click *OK* in the **Triangulation** dialog.

Triangulation Dialog

The **Triangulation** dialog is displayed after clicking the **Features | New Features | Triangulation** command.



*Select the output feature type, output layer, and whether or not to keep the original points in the **Triangulation** dialog.*

Output Feature Type

The triangulation diagram can consist of polygons or polylines. Select *Polygon* or *Polyline* in the *Output feature type* list.

Output Layer

The *Output layer* option specifies the layer where the triangulation polygons or polylines will be created when the selected points are in a base layer. The *Output layer* list includes the base layers in the map, as well as the option to create a the triangulation in a new base layer. Select *[New Base Layer]* to create the triangulation in a new layer. When a new base layer is created, the base layer is assigned the same coordinate system as the map frame. Select a base layer to add the triangulation to an existing layer. The *Output layer* option is not available when the selected points are not in a base layer.

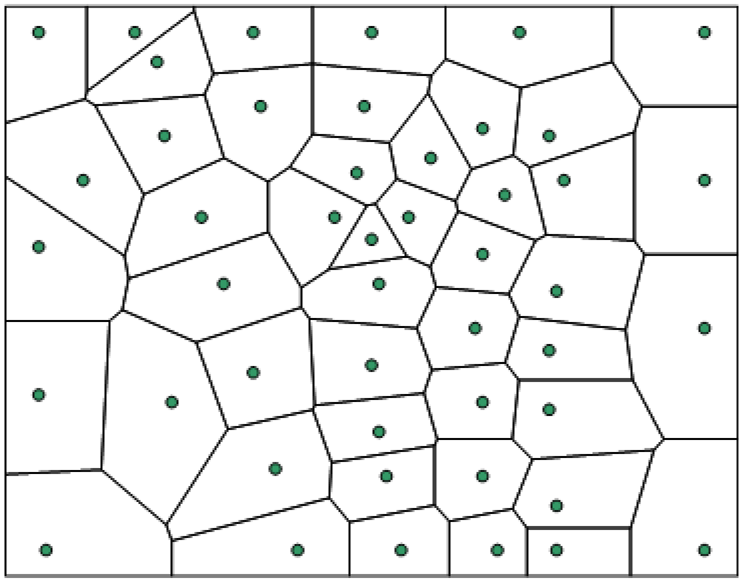
Keep Original Objects

Select *Keep original points* to create the triangulation while leaving the original points unchanged. Clear the *Keep original points* check box to remove the points after the triangulation is created.

Thiessen Polygons

The **Features | New Features | Thiessen Polygons** command creates polygons from selected points. The diagram can consist of polygons or polylines. In a Thiessen polygon diagram, also known as a Voronoi diagram, a region is drawn around each point so that for each point every position in the region around that point is closer to that point than to any of the other points.

Thiessen polygons can be created from drawn point features or points within a base layer.



Thiessen Polygons *creates a Thiessen polygon diagram from selected points.*

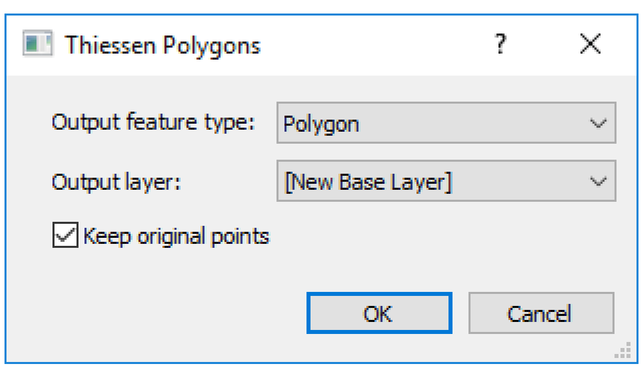
Creating a Thiessen Polygon Diagram

The following steps describe how to create a Thiessen polygon (Voronoi) diagram:

1. Select three or more points from which to create the diagram.
2. Click the **Features | New Features | Thiessen Polygons** command or the  button.
3. Select the desired options in the **Thiessen Polygons** dialog.
4. Click *OK* in the **Thiessen Polygons** dialog.

Thiessen Polygons Dialog

The **Thiessen Polygons** dialog is displayed after clicking the **Features | New Features | Thiessen Polygons** command.



Select the output feature type, output layer, and whether or not to keep the original points in the ***Thiessen Polygons*** *dialog.*

Output Feature Type

The Thiessen polygons diagram can consist of polygons or polylines. Select *Polygon* or *Polyline* in the *Output feature type* list. When *Polygon* is selected, any input point attributes are assigned to the resultant Thiessen polygons.

Output Layer

The *Output layer* option specifies the layer where the Thiessen polygons or polylines will be created when the selected points are in a base layer. The *Output layer* list includes the base layers in the map, as well as the option to create a the Thiessen polygons in a new base layer. Select *[New Base Layer]* to create the Thiessen polygons in a new layer. When a new base layer is created, the base layer is assigned the same coordinate system as the map frame. Select a base layer to add the Thiessen polygons to an existing layer. The *Output layer* option is not available when the selected points are not in a base layer.

Keep Original Objects

Select *Keep original points* to create the Thiessen polygons while leaving the original points unchanged. Clear the *Keep original points* check box to remove the points after the Thiessen polygons are created.

Split

Click the **Features | New Features | Split** command or right-click on a selected polygon and click **Split Islands/Lakes** to break a single complex polygon into multiple polygons. A new polygon is created for each of the polygons contained in the complex polygon.

The new polygons use the line and fill properties from the original complex polygon. The object name is also retained from the original complex polygon.

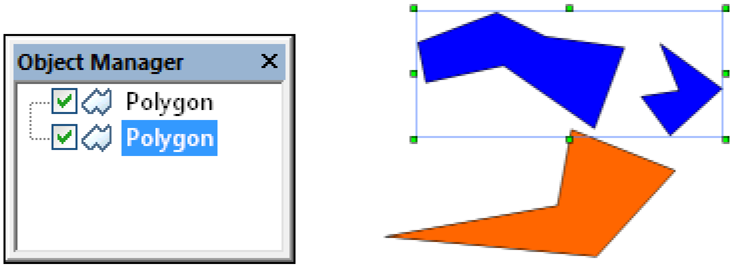
Complex polygons can contain polygons that are touching, intersecting, completely separate, or contained completely within other polygons. Each individual polygon in the group is called an island. An example of islands that can form a complex area is the Hawaiian islands. When a polygon is completely contained inside another polygon, a lake is formed. Lakes are the holes in a complex polygon. When the islands and lakes are split, each island and lake becomes a new polygon.

Note about Base Maps

If objects should be edited in a base map, click on the object in the *Base* layer to select it. Click the **Features | Group | Start Editing** command to enter editing mode. Select the polygon to be split. Then, click the **Features | Islands/Lakes | Split** command. The new polygons are created from the single complex polygon. Click the **Features | Group | Stop Editing** to end object editing mode in the base map.

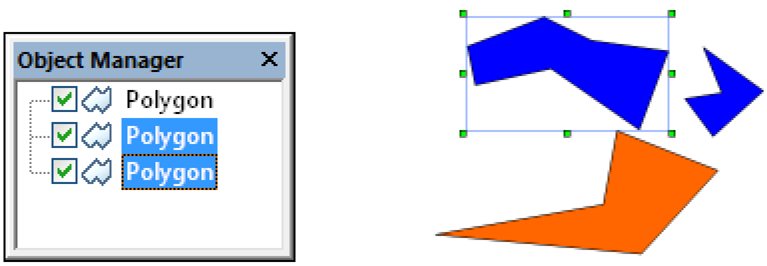
Splitting a Complex Polygon into Multiple Polygons

1. Click on a complex polygon in the **Contents** window. The polygon can be imported with the **Home | Insert | Graphic** command or can be created by combining multiple polygons into a single complex polygon using the Combine command.



*Click on the complex polygon in the **Contents** window or in the plot window.*

2. Click the **Features | New Features | Split** command or the button. Multiple polygons are created from the single complex polygon. Both polygons maintain the properties of the original polygon.



The polygon is split into multiple polygons. Each polygon can be individually selected and edited.

Combine Islands/Lakes

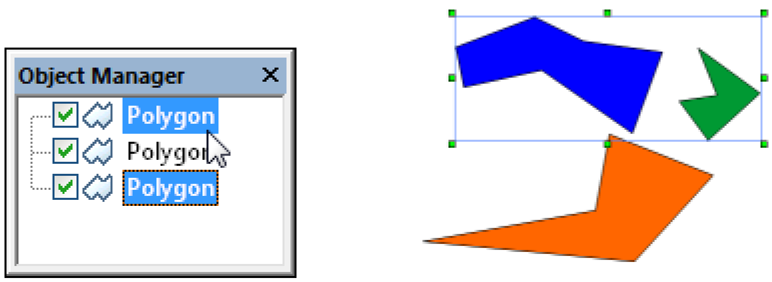
Click the **Features | New Features | Combine** command or right-click on multiple selected polygons and click **Edit Boundaries | Combine Islands/Lakes** to combine multiple polygons into a single complex polygon. A new complex polygon containing all of the selected polygons is created.

The new polygon uses the line and fill properties from the lowest selected polygon in the Contents window. The object name is also retained from the lowest selected polygon in the **Contents** window.

Complex polygons can be created from polygons that are touching, polygons that are completely separate, or polygons that are contained completely within other polygons. Each individual polygon in the group is called an island. An example of islands that can form a complex area is the Hawaiian islands. When a polygon is completely contained inside another polygon, a lake is formed. Lakes are the holes in a complex polygon.

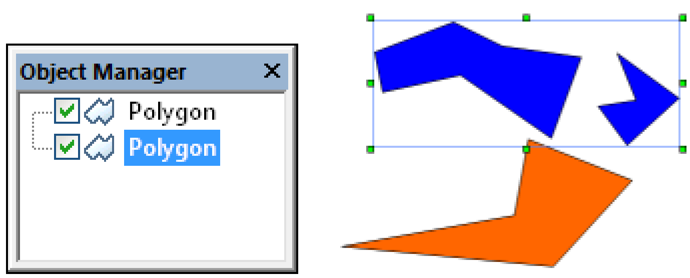
Combining Multiple Drawn Polygons

1. Click the Polygon command.
2. On the plot window, click several points. Double-click the last point to close the polygon.
3. Repeat step 2 a few more times so that several polygons are created.
4. Press ESC on the keyboard to end drawing mode.
5. In the **Contents** window, click on the first polygon that you want to combine.
6. Hold down the CTRL key on the keyboard.
7. In the **Contents** window, click on the each additional polygon that you want to combine.



*In this example, the top and bottom polygons are selected in the **Contents** window and in the plot window.*

8. Click the **Features | New Features | Combine**command or the button. A single polygon is created from the selected polygons. Both polygons have the properties of the bottom-most polygon in the **Contents** window.



The two polygons become a single complex polygon.

Converting to 3D Objects

The following sections describe commands for converting to the following: Points to 3D Polyline, Polyline to 3D Polyline, Polygon to 3D Polygon.

Points to 3D Polyline

The **Features | Change | Change To | Points to 3D Polyline** command converts two or more points to a 3D polyline object. Points can be 3D points. The points become vertices in a single 3D polyline ordered by attribute or draw order in the Contents window.

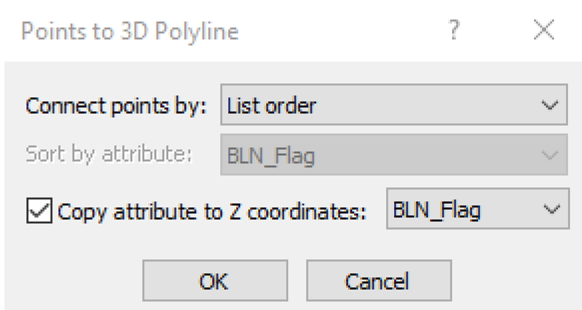
The new 3D polyline uses the color and opacity from the Symbol *Line color* and *Line opacity* properties. The remaining line properties are the default line properties. If the object name has not been changed from the default *Point*, the object is renamed *3D Polyline*. If the object name had been altered, the custom name is from the last point is retained. The attributes from the last point are also applied to the new polyline.

Points to 3D Polyline

Select the map layer that contains the points and click the Home | Selection | Select All command to select all the points to create the 3D polyline. To select some of the points, press SHIFT and click the points to include in the new 3D polyline.

Points to 3D Polyline Dialog

The **Points to 3D Polyline** dialog is displayed when the **Features | Change | Change To | Points to 3D Polyline** command is clicked. The **Points to 3D Polyline** dialog specifies the sort criteria for connecting the points to create a 3D polyline. If there are no numeric attributes, then this dialog will not appear.



*Set the vertex sort order in the **Points to 3D Polyline** dialog.*

Connect Points By

Select *List order* or *Sorted attribute values* in the *Connect points by* list.

- *List order* uses the draw order of the points to form the polyline. The draw order is indicated by the order in the **Contents** window from bottom to top.
- *Sorted attribute values* uses the values from the specified *Attribute* column to sort the points.

Attribute

The *Attribute* list is enabled when *Sorted attribute values* is selected in the *Connect points by* option. The *Attribute* list displays all attributes listed in the Info page or Attribute Table. Select the attribute you wish to use for sorting the points by clicking the current option and selecting an attribute from the list.

When all of the selected points attribute values are numbers, the points are ordered by attribute value numerically from smallest to largest. The selected points are sorted alphabetically by attribute value when the attribute values are strings or mixed strings and numbers.

Copy Attribute to Z Coordinates

Copy attribute to Z coordinates: is checked by default. The *Attribute* list displays all attributes listed in the **Info** page or **Attribute Table**. Select the attribute you wish to use for the Z coordinates.

OK and Cancel

Click *OK* to convert the points to a 3D polyline. Click *Cancel* to close the dialog without converting the points.

Polyline to 3D Polyline

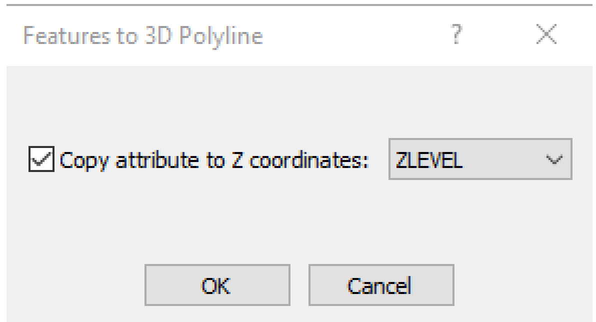
Click the **Features | Change | Change To | Polyline to 3D Polyline** command or right-click on a polyline and click **Polyline to 3D Polyline** to convert one or more selected polylines or spline polylines into 3D polylines. Each selected polyline is converted to a new separate 3D polyline.

The new 3D polyline uses the default line properties from the polyline. If the object name has not been changed from the default *Polyline*, the object is renamed *3DPolyline*. If the object name had been altered, the custom object name is retained.

X, Y, and Z coordinates for each vertex can be changed on the Coordinates page in the Properties window.

Features to 3D Polyline Dialog

The **Features to 3D Polyline** dialog is displayed when the **Features | Change | Change To | Polyline to 3D Polyline** command is clicked and there are numeric values for the Z coordinates. The **Features to 3D Polyline** dialog specifies the attribute for the Z coordinates.



Specify the attribute for the Z coordinates.

Copy Attribute to Z Coordinates

Copy attribute to Z coordinates: is checked by default. The *Attribute* list displays all attributes listed in the Info page or Attribute Table. Select the attribute you wish to use for the Z coordinates.

OK and Cancel

Click *OK* to convert the points to a 3D polyline. Click *Cancel* to close the dialog without converting the points.

3D Polyline to Polyline

To change a 3D polyline back to a polyline, click the **Features | Change | Change To | 3D Polyline to Polyline** command. Note that the Z column on the **Coordinates** page will be removed.

Polygon to 3D Polygon

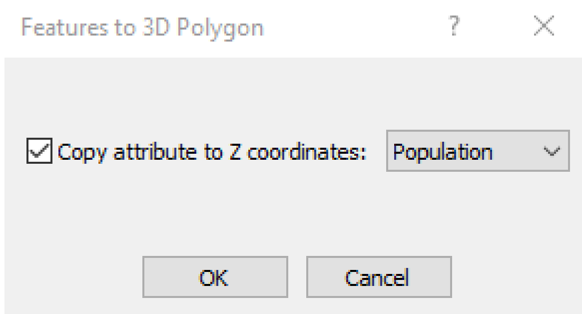
Click the **Features | Change | Change To | Polygon to 3D Polygon** command or right-click on a polygon and click **Polygon to 3D Polygon** to convert one or more selected polygons into 3D polygons. Each selected polygon is converted to a new separate 3D polygon.

The new 3D polygon uses the default fill and line properties from the polygon. If the object name has not been changed from the default *Polygon*, the object is renamed *3DPolygon*. If the object name had been altered, the custom object name is retained.

X, Y, and Z coordinates for each vertex can be changed on the Coordinates page in the Properties window.

Features to 3D Polygon Dialog

The **Features to 3D Polygon** dialog is displayed when the **Features | Change | Change To | Polygon to 3D Polygon** command is clicked and there are numeric values for the Z coordinates. The **Features to 3D Polygon** dialog specifies the attribute for the Z coordinates.



Specify the attribute for the Z coordinates.

Copy Attribute to Z Coordinates

Copy attribute to Z coordinates: is checked by default. The *Attribute* list displays all attributes listed in the Info page or Attribute Table. Select the attribute you wish to use for the Z coordinates.

OK and Cancel

Click *OK* to convert the points to a 3D polyline. Click *Cancel* to close the dialog without converting the points.

3D Polygon to Polygon

To change a new 3D polyline back to a polyline, click the **Features | Change | Change To | 3D Polygon to Polygon** command. Note that the Z column on the **Coordinates** page will be removed.

Undo

Click the **Home | Undo | Undo** command the button or press CTRL+Z on the keyboard to reverse the last operation performed. If the last operation cannot be reversed, the **Undo** command is grayed. After you close a file, the undo operations are cleared, so you cannot undo any operations performed before the file was closed. After you have undone an operation, the Redo command becomes available, allowing you to reverse the last completed **Undo** command.

Hover the cursor over the **Undo** command to see the last action. For example, after drawing a polyline the **Undo** command tooltip displays *Undo Creation* .

Zoom and pan operations are not included in the undo list. Consider using a view command or keyboard command to quickly move to a different view.

Grid Editor

Click the **Grid Editor | Undo | Undo** command or press CTRL+Z to reverse the last operation performed when in the grid editor view. Undo operations are only performed on ribbon commands in the **Grid Editor** . Property changes in the **Grid Editor** cannot be undone with the **Undo** command.

Undo Levels

Up to 100 undo levels can be set though **Options** dialog General page. If the **Undo** command is not available, check the *Undo levels* . If the value is set to zero, undo will not be available.

Redo

Click the **Home | Undo | Redo** command or the button, or press CTRL+Y on the keyboard to reverse the last Undo command.

After you have used the **Home | Undo | Undo** command, the **Redo** command indicates the operation that you have undone. Hover the cursor over the **Redo** command to see the last undone action. For example, after deleting an object and then clicking **Undo** , the **Redo** command tooltip displays *Redo Deletion*.

Grid Editor

Click the **Grid Editor | Undo | Redo** command or press CTRL+Y to reverse the last operation performed when in the grid editor view.

Paste

Click the **Home | Clipboard | Paste** command or the button, or press CTRL+V on the keyboard to paste the clipboard contents into the current document. The objects to be pasted must first be placed in the clipboard using the Cut or Copy commands of **Surfer** or some other application. The clipboard contents remain on the clipboard until something new is cut or copied to the clipboard.

Worksheet

In the worksheet, the upper left corner of the pasted data is placed in the active cell. Any cells in the existing worksheet that lie to the right of and below the active cell will be overwritten with the contents of the pasted data. The following rules are used to paste into the worksheet:

- Only the TAB character is recognized as a column separator. Spaces, commas, semi-colons, etc. are included in a text cell.
- The RETURN character is recognized as the row separator.
- Numbers paste as number values. The period can always be used as a decimal separator, and the system locale determines any other decimal separator. For example if the system locale uses a comma as the decimal

separator, then both 123.456 and 123,456 are pasted as the number 123,456.

- Mixed text and numbers paste as text.
- Dates and/or times will paste as date values in a wide variety of date/time formats. Ambiguous dates are determined by the system locale setting. If dates are not pasted correctly, consider using Paste Special and the *Locale* settings in the **Data Import Options** dialog.

Pasting a single cell's contents across multiple cells

A single cell's contents can be pasted into multiple cells by coping the cell, selecting a block of cells, and using the **Paste** command. Each cell in the selection is populated with the clipboard contents when this operation is performed. If more than one cell is copied, then the copied cells are only pasted once.

The multiple paste operation will not be performed for an entire row, entire column, or across multiple selections. When an entire row or column is selected, only the first cell in the row or column will receive the pasted content. If you attempt to paste in multiple selections an error message will be displayed.

Plot

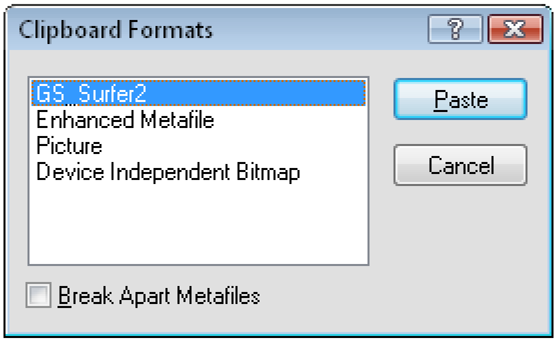
In the plot window, the clipboard contents are pasted in the center of the window. The pasted contents are automatically selected, and can be dragged to a new location. Alternatively, change the position or size of the selected object with the Position/Size group.

Paste Special - Plot Document

The **Home | Clipboard | Paste | Paste Special** command pastes objects from the clipboard into a plot window. With the **Home | Clipboard | Paste | Paste Special** command, you have the option of specifying the format to use when pasting the object into .

Clipboard Formats Dialog

Use the **Paste Special** command to open the **Clipboard Formats** dialog and select one of the clipboard formats.



*Select a paste special format in the **Clipboard Formats** dialog.*

GS_Surfer2

The *GS_Surfer2* format lets you cut or copy an entire map or drawing to the clipboard. It is stored in the clipboard as a native object. You can paste this map or drawing into another plot window in . The map is identical to the original and is a complete map that can be edited. This is an easy way to place several complete drawings on a single page.

Enhanced Metafile and Picture

The *Enhanced Metafile* and *Picture* formats are Windows metafile formats, which store objects as a series of Windows drawing commands. Metafiles can be resized without distorting the image. These formats are supported by most Windows applications.

Break Apart Metafiles

When you select an *Enhanced Metafile* or *Picture* format, you can check the *Break Apart Metafiles* box at the bottom of the **Clipboard Formats** dialog. This option converts the metafile into a series of native objects and groups them as a single group object. Group objects can be broken apart using the Features | Group | Ungroup command.

For example, you could paste a contour map in the picture format, and check the *Break Apart Metafiles* check box. In this case, the contour map is pasted as a group of polylines and text. You can then break apart the group object and modify the individual polylines (such as the contour lines).

Device Independent Bitmap

The *Device Independent Bitmap* format stores objects as images. Bitmap images are difficult to resize without distorting the image, and their colors are limited. This format is relatively common, and is supported by most other Windows applications.

Text

The *Text* format is used to import text. Imported text can contain any number of lines and uses the default text properties. If you want to paste both text and other objects, and retain the text, you must copy and paste the text separately from the other objects.

Copy

Click the **Home | Clipboard | Copy** command or the button, or press CTRL+C on the keyboard to copy the selected objects to the clipboard. The original objects remain in the window. Use this command to duplicate objects in a different location in the same window, or copy the objects into a different window or application. The copied objects can later be pasted with the Paste or Paste Special commands.

Only one set of data may be placed in the clipboard at a time. The next Cut or **Copy** command replaces the contents of the clipboard.

Cut

Click the **Home | Clipboard | Cut** command or the button, or press CTRL+X on the keyboard to move the selected objects to the clipboard. This deletes the selected objects from the file after copying them to the clipboard. Cut objects can later be pasted with the Paste or Paste Special commands.

Only one set of data may be placed in the clipboard at a time. The next **Cut** or Copy command replaces the contents of the clipboard.

Delete

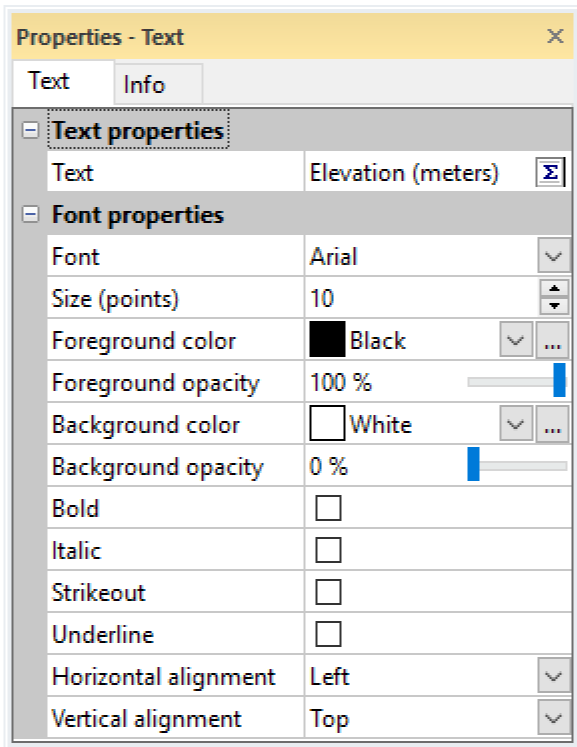
The **Home | Clipboard | Delete** command or the button removes an object from the document. An object can sometimes be restored with the Undo command.

To delete an object, select the object in the plot window or Contents window and press the DELETE key or click the **Delete** command.

Chapter 29 - Common Properties

Text and Font Properties

Click the **Home | Insert | Text** command to create new text. The Text Editor is used to initially create text. After the text is created, the text and font properties are displayed in the Properties window. Use the following options to change the text and font properties.



Edit text in the Text Properties *section and font properties in the* Font Properties *section of the* ***Properties*** *window.*

Text Properties

The text can be edited in the Properties window or in the **Text Editor**. Click the ⊞ next to *Text properties* to open the *Text properties* section. You can set the text that appears by highlighting the existing text next to *Text* and typing the desired text for simple text strings.

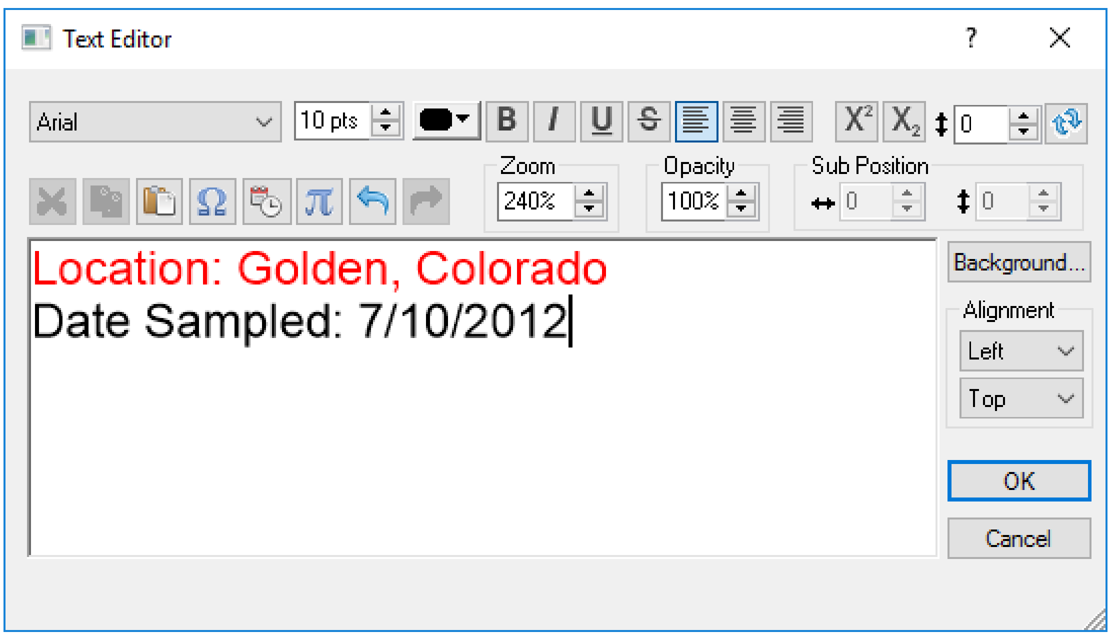
For complex text or multiple lines of text, click the Σ button to open the Text Editor. Enter text, edit text properties for individual characters, and add templates, symbols and timestamps with the **Text Editor**. The text you enter into the **Text Editor** is converted into math text instructions and written in the *Text* field. You can also type math text instructions directly into the *Text* field, if desired.

Font Properties

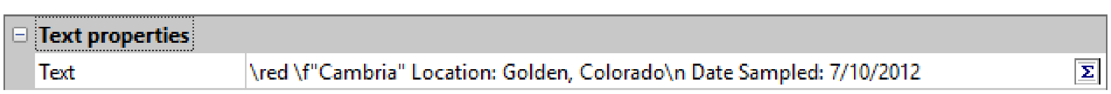
Text properties for the entire text block can be edited in the **Properties** window in the *Font Properties* section. Some options may not be available for all text, depending on the type of object selected. For example, when editing axis labels, the *Alignment* options are not available.

The changes made in the *Font Properties* section are considered the baseline text properties and are applied to each line of a text block. Changes made here override "normal" font options in the **Text Editor**. To have all text the same, click the Σ button. Highlight all the text and set all the properties in the **Text Editor** , as desired. Then, the changes in the *Font Properties* section will apply to all text in the text object.

For example, suppose you type in the following in the **Text Editor**. After typing the text, you have highlighted the first line and made the color red and the font *Cambria*.



After clicking *OK* in the **Text Editor**, the following appears in the *Text Properties* section of the **Properties** window.



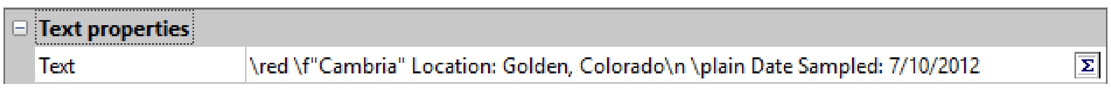
The two lines of text appear in the plot window as:

Location: Golden, Colorado
Date Sampled: 7/10/2012

Now, in the **Properties** window in the *Font Properties* section, set the *Font* to "Times New Roman", the *Foreground Color* to Blue, and check the boxes next to *Bold* and *Italic*. Because the first line contains a font name and color, the font and color will not change for this line. But, the red text will be bold and italic. The text now appears as:

Location: Golden, Colorado
Date Sampled: 7/10/2012

If the second line of text should remain without any text embellishments, such as bold or italics, in the **Properties** window, click before the word *Date* and add a *\plain* math text option. This does not control the font name and color, so they will still change. The **Properties** window would now appear as:



And, the text would appear in the plot window as:

Location: Golden, Colorado
Date Sampled: 7/10/2012

Font

The *Font* is the font that is used for the text. To change the *Font*, click on the current font name. Select the desired font name from the *Font* list. Click the arrow button or select a font and use the ARROW keys on the keyboard to scroll through the *Font* list. The selected font is displayed in the **Text Editor**. The font files that are installed on your computer are displayed in the *Font* list. **Surfer** supports true type fonts. All text in a text block uses the same *Font*, unless a math text operation is applied or unless the font for some of the text has been specifically edited in the **Text Editor**.

Size (points)

Set the text size in the *Size (points)* field. Highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Or, click the buttons to increase or decrease the size. A *Size (points)* value between zero and 720 can be specified.

Foreground and Background Color

Change the *Foreground color* and *Background color* of the text by selecting a new color from the color palette. The *Foreground color* controls the color of the text. The *Background color* controls the area behind the text. Create new colors by clicking the ... button to the right of the color name.

Foreground and Background Opacity

Change the *Foreground opacity* and *Background opacity* of the text by entering a value from 0% (completely transparent) to 100% (completely opaque). To change the opacity, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Or, click and drag the slider to change the opacity percentage.

Style

Check the boxes next to *Bold*, *Italic*, *Strikeout*, or *Underline* to apply a style to the text. Note that some typefaces, such as *Symbol*, do not support bold or italicized text.

- *Bold* will increase the thickness of the text (i.e. **example**).
- *Italic* will create obliqueslanted, sloped text (i.e. *example*).
- *Strikeout* will add a horizontal line through the center of the text (i.e. ~~example~~).
- *Underline* will add a horizontal line under the text (i.e. example).

Alignment

The *Alignment* controls the location of the text relative to the reference point. A reference point is the point clicked in the plot window when the crosshair cursor is placing the text on the screen. The text box is horizontally and vertically aligned relative to the reference point. The default position is that the reference point is at the upper left corner of the bounding box (left, top).

- *Left* horizontally aligns the text box so that the reference point is to the left of the text box.
- *Center* horizontally centers the text box on the reference point.
- *Right* horizontally aligns the text box so that the reference point is to the right of the text box.
- *Top* vertically aligns the text box so that the reference point is above the text box.

- *Baseline* vertically aligns the text box so that the reference point is located at the base of the text. The baseline is the imaginary line along which characters are positioned as they are drawn. Descenders on characters are drawn below the baseline.

- *Bottom* vertically aligns the text box so that the reference point is below the text box.

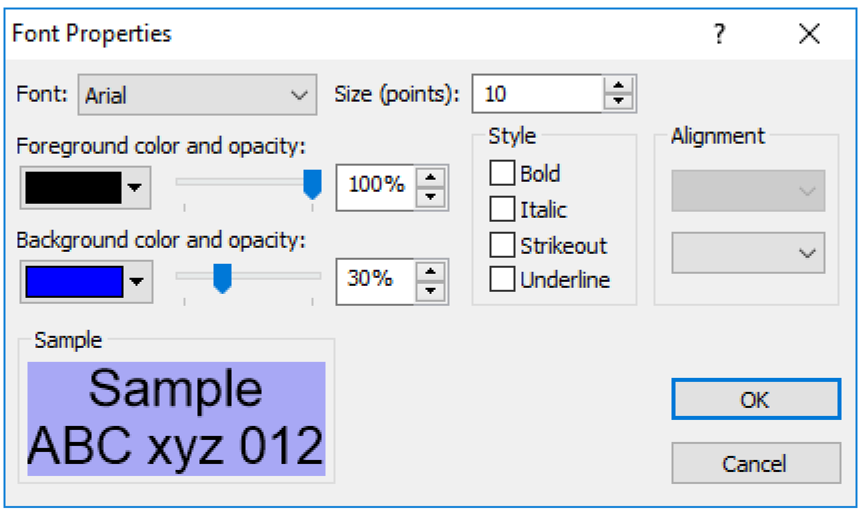
- *Center* vertically centers the text box on the reference point.

Default Settings

Set default text properties through **File | Options**. In the Default Properties section, click *Font* to specify default properties.

Font Properties Dialog

When changing the advanced label font property settings for a contour map, a **Font Properties** dialog will appear. The options are the same as described above in the **Properties** window. In addition, the **Font Properties** dialog has a *Sample* section where the font can be previewed.



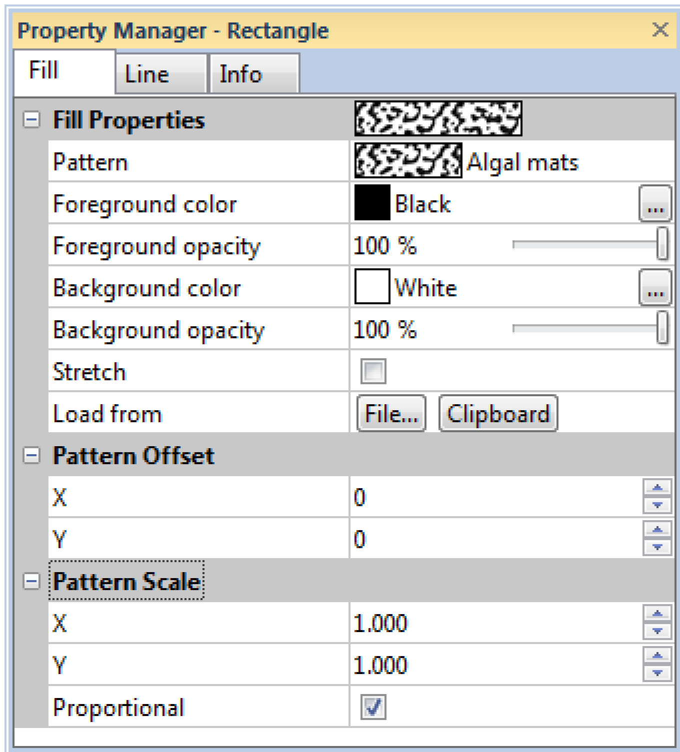
*Set the font properties for contour labels and grid node editor labels in the **Font Properties** dialog.*

Fill Properties

Use the **Fill Properties** to change fill properties for selected objects in the document. You can set default line properties through File | Options | Defaults by clicking on *Fill*.

Most fill properties are edited in the Properties window in a *Fill Properties* section. When changing fill properties for a selected object, the *Fill Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Occasionally, objects will open a Fill Properties dialog to access the fill properties.



Edit the Fill Properties *for a selected object in the* ***Properties*** *window.*

Sample

The sample shown next to the *Fill Properties* line shows the selected pattern, foreground color, and background color.

Fill Pattern

Change the *Pattern* by selecting a pattern from the fill pattern palette. Open the pattern palette by clicking the pattern button. Click on a new pattern in the list to select it.

Foreground Color

Foreground color is the color of the pattern lines or pixels. Select a new color by clicking on the color in the color palette. Only the foreground color can be applied to solid colors. The foreground colors can be applied to any stock hatch pattern or grayscale image pattern. They cannot be applied to the *None* pattern or non-grayscale image patterns. Click the button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Foreground Opacity

The *Foreground opacity* is the amount of transparency of the fill. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change.

Alternatively, click and drag the to change the opacity percentage. Opacity is disabled if it does not apply to the current pattern. Note that for true color image fill patterns the *Foreground opacity* applies to the image in its entirety.

Background Color

Background color is the color behind the pattern. All patterns must have a background color. If you do not wish to see the background color, change the *Background opacity* to 0%. Click on the button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Background Opacity

Change the *Background opacity* by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the slider to change the opacity percentage. Opacity is disabled if it does not apply to the current pattern.

Stretch

Check the *Stretch* option to stretch image fills to completely fill the geometry. If *Stretch* is not selected, the image will be repeated to fill the geometry.

Load a Fill Pattern File

To load a fill pattern from a raster image file, click the *File* or *Clipboard* button in the *Load from* field. The *Clipboard* button is only active when there is an image on the clipboard suitable to use for a fill pattern. If you select *File,* an **Import** dialog will appear. Click on the image file and click *Open* to load the image as the fill pattern.

Pattern Offset

The *Pattern Offset* can be changed for image patterns. The offset controls the location of the pattern within the geometry. Change the *X* and *Y* values separately to move the image in the desired direction.

Pattern Scale

The *Pattern Scale* can be changed for image patterns. The scale controls the density of the pattern. In the **Properties** window, check the *Proportional* box to connect the *X* and *Y* scale values.

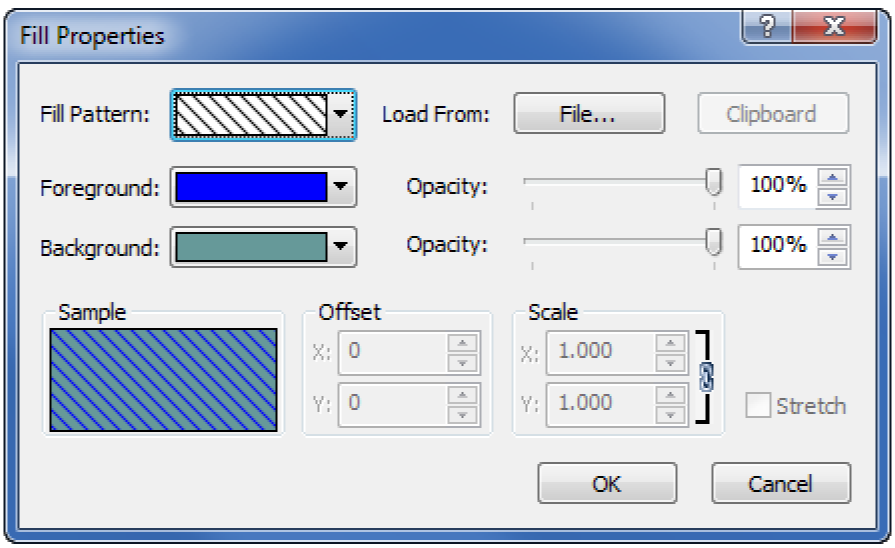
When the scale is proportional, the current aspect ratio is maintained. Changing the X or Y scale will cause the other scale to be adjusted proportionally to maintain the aspect ratio at the time the scale was set to proportional.

Fill Properties Dialog

Use the **Fill Properties** dialog to change fill properties for selected objects in the document. You can set default line properties through File | Options | Defaults by clicking on *Fill*.

Most fill properties are edited in the Properties window in a *Fill Properties* section. When changing fill properties for a selected object, the *Fill Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Occasionally, objects will open a **Fill Properties** dialog to access the fill properties. This dialog is accessed differently depending on the type of object created.



*Specify fill properties in the **Fill Properties** dialog.*

Fill Pattern

Change the *Fill Pattern* by selecting a pattern from the fill pattern palette. Open the pattern palette by clicking the pattern button. Click on a new pattern in the list to select it.

Load a Fill Pattern File

To load a fill pattern from a raster image file, click the *File* or *Clipboard* button next to *Load from*. The *Clipboard* button is only active when there is an image on the clipboard suitable to use for a fill pattern. If you select *File,* an **Import** dialog will appear. Click on the image file and click *Open* to load the image as the fill pattern.

Foreground Color

Foreground color is the color of the pattern lines or pixels. Select a new color by clicking on the color in the color palette. Only the foreground color can be applied to solid colors. The foreground colors can be applied to any stock hatch pattern or grayscale image pattern. They cannot be applied to the *None* pattern or non-grayscale image patterns. To create a custom color, click on the *Custom* button at the bottom of the palette. This opens the Colors dialog.

Foreground Opacity

Change the Foreground *Opacity* by entering a value from 0% (completely transparent) to 100% (completely opaque), using the arrow buttons to the right of the box, or dragging the slider to change the opacity percentage. Opacity is disabled if it does not apply to the current pattern. Note that for true color image fill patterns the Foreground *Opacity* applies to the image in its entirety.

Background Color

Background color is the color behind the pattern. All patterns must have a background color. If you do not wish to see the background color, change the Background *Opacity* to 0%. To create a custom color, click on the *Custom* button at the bottom of the palette. This opens the Colors dialog.

Background Opacity

Change the Background *Opacity* by entering a value from 0% (completely transparent) to 100% (completely opaque), using the arrow buttons to the right of the box, or dragging the slider to change the opacity percentage. Opacity is disabled if it does not apply to the current pattern.

Offset

The *Offset* can be changed for image patterns. The offset controls the location of the pattern within the geometry. Change the *X* and *Y* values separately to move the image in the desired direction.

Scale

The *Scale* can be changed for image patterns. The scale controls the density of the pattern. Click on the  button to connect or disconnect the X and Y scale for proportional or non-proportional scaling.

When the scale is proportional, the current aspect ratio is maintained. Changing the X or Y scale will cause the other scale to be adjusted proportionally to maintain the aspect ratio at the time the scale was set to proportional.

Stretch

Check the *Stretch* option to stretch image fills to completely fill the geometry. If *Stretch* is not selected, the image will be repeated to fill the geometry.

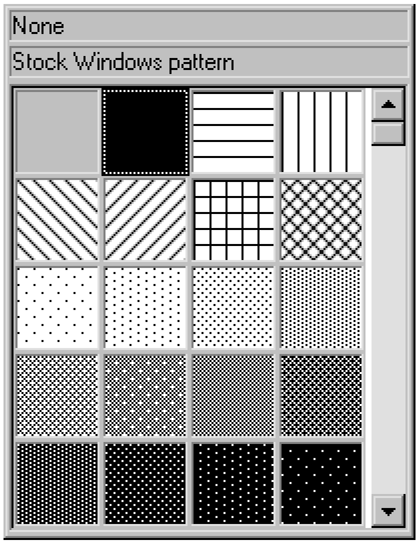
Sample

The *Sample* box shows the selected pattern, foreground color, and background color.

Fill Palette

The fill pattern palette is opened by clicking on the fill pattern in the Fill Properties section.

- The pattern name and type appears at the top of the palette.
- Select a pattern by clicking on it.
- Hold the cursor over a pattern to display a tool tip of the pattern name.
- Use the scroll bar to see all of the available fill patterns.



Select a fill pattern from the fill pattern palette.

Fill Patterns

There are 315 predefined fill patterns. There are stock patterns and high resolution image fill patterns. Stock and image patterns can have a transparent background. Only image patterns can be scaled and offset in the Fill Properties.

Select a pattern from the fill palette.

Stock Windows Patterns

None	80 Percent	Small Confetti
Solid	90 Percent	Large Confetti
Horizontal	Light Downward Diagonal	ZigZag
Vertical	Light Upward Diagonal	Wave
Forward Slash	Dark Downward Diagonal	Diagonal Brick
Backward Slash	Dark Upward Diagonal	Horizontal Brick
Crosshatch	Wide Downward Diagonal	Weave
Diagonal Cross	Wide Upward Diagonal	Plaid
5 Percent	Light Verticle	Divot
10 Percent	Light Horizontal	Dotted Grid
20 Percent	Narrow Vertical	Dotted Diamond
25 Percent	Narrow Hortizontal	Shingle
30 Percent	Dark Vertical	Trellis
40 Percent	Dark Horizontal	Spheres
50 Percent	Dashed Downward Diagonal	Small Grid
60 Percent	Dashed Upward Diagonal	Small Checkerboard
70 Percent	Dashed Horizontal	Checkerboard
75 Percent	Dashed Vertical	Outlined Diamond
		Solid Diamond

Image Patterns

6.25% Black	Inorganic silt 4	Chalk
12.5% Black	USGS 207	Limestone USGS
25.0% Black	USGS 214	Clastic limestone
50.0% Black	USGS 215	Fossiliferous limestone
75.0% Black	USGS 216	Nodular bedded lime-stone
87.5% Black	USGS 217	Limestone w/ sac-charoidal
Sand	USGS 218	Crossbedded limestone
Swamp	USGS 219	Cherty crossbedded lime-stone
Forest	USGS 226	Cherty clastic limestone
Orchard	USGS 228	Oolitic limestone
Water	USGS 229	Sandy limestone
Limestone	USGS 230	Silty limestone
Tilted Limestone	USGS 231	Argillaceous shaley lime-stone
Dolomite	Intrusive igneous 1	Cherty limestone 1
Dolomite2	Intrusive igneous 2	Cherty limestone 2
Shale	Intrusive igneous 3	Dolomitic limestone
Granite	Intrusive igneous 4	Dolomite USGS
Igneous	Basalt	Crossbedded dolomite
Schist	Basalt 3	Oolitic dolomite
Gravel	Basalt 4	Sandy dolomite
Iron	Greenschist	Silty dolomite
Spheres	Greenschist 2	Argillaceous dolomite
Solid Dash	Diorite	Cherty dolomite
Thatch	Granodiorite	Bedded chert 1
Andesite 1	Granodiorite 2	
Andesite 2	Granodiorite 3	

Andesite 3	Hornfels	Bedded chert 2
Basalt 1	Hornfels 2	Fossiliferous bedded chert
Basalt 2	Carbonatite	Fossiliferous rock
Conglomerate	Kimberlite	Diatomaceous rock
Dolomite	Dendrites	Subgraywacke
Fish Fossil	Calc-silicate	Crosbedded sub-graywacke
Gneiss 1	Oolite	Ripple-bedded gray-wacke
Gneiss 2	Med unconsolidated sed-iment	Peat
Granite1	Fine unconsolidated sed-iment	Bony coal
Granite 2	Jointed/Fractured rock	Underclay
Gravel	Gneiss USGS 2	Flint clay
Limestone - Fossil 1	Gneiss USGS	Bentonite
Limestone - Fossil 2	Clastic dike	Glaconite
Limestone - Oolitic	Glacial-1	Limonite
Limestone - Stomatolitic	Glacial-2	Siderite
Limetsone	Rhyolite USGS	Phosphatic rock
Mudstone	Rhyolite USGS 2	Gypsum
Quartzite	Dacite	Salt 3
Rhyolite	Salt 1	Interbedded sandstone
Rocks	Salt 2	Interbedded sandstone 2
Sand 1	Volcanic A	Ripple-bedded sand-stone
Sand 2	Volcanic B	Interbedded shale
Sandstone 1	Bedded tilted sandstone	Interbedded shale 2
Sandstone 2	Sandy gravel	
Schist	Volcanic tuff	
Slate		

Stones	Rhyolitic tuff	Interbedded shale 3
White Marble	Algal mats	Interbedded calcareous shale
BIF	Lignite	Interbedded silty lime-stone
Black Marble	Fine, poorly-graded sand 2	Interbedded limestone 1
Gneiss 3	claystone	Interbedded limestone 2
Granite 3	Marsh	Interbedded limestone 3
Granite 4	Periglacial-1	Interbedded limestone 4
Granite 5	Periglacial-2	Metamorphism
Green Marble	Periglacial-3	Quartzite USGS
Orbicular Granite	Periglacial-4	Slate USGS
Oxidized Zone	Periglacial-5	Schistonse
Pahoehoe	Gravel or conglomerate 1	Schist USGS
Pink Marble	Gravel or conglomerate 2	Contorted schist
Pumice	Crosbedded gravel	Schist and gneiss
Ruby Zoisite	Till or diamicton	Gneiss
Scoria	Breccia 1	Contorted gneiss
Zebra Dolomite	Breccia 2	Soapstone
Sand and Gravel	Massive sand	Tuffaceous rock
Coarse sand	Bedded sand	Crystal tuff
Well graded coarse sand	Crossbedded sand	Devitrified tuff
Coarse, medium-graded sand	Crossbedded sand 2	Volcanic breccia and tuff
Coarse, well-graded sand	Ripple-bedded sand	Volcanic breccia
Coarse, well-graded sand 2	Argillaceous sandstone	Zeolitic rock
Fine, well-graded sand	Calcareous sandstone	Basaltic flows
		Banded igneous rock

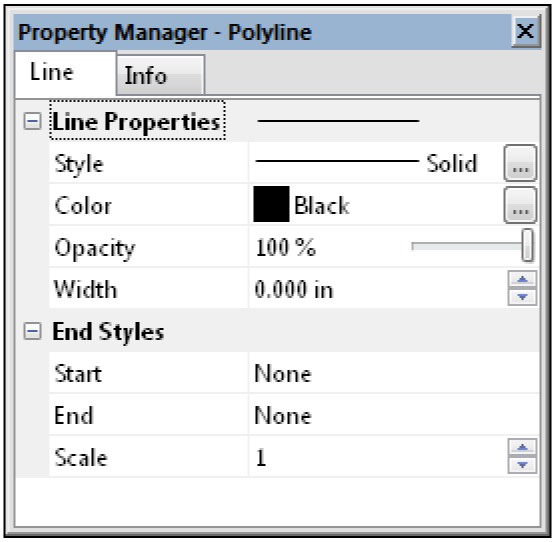
Fine, poorly-graded sand	Dolomitic sandstone	Granite 2 USGS
Medium, poorly-graded sand	Loess	Igneous rock 1
Coarse, crossbedded sand	Silty shale	Igneous rock 2
Medium, poorly-graded sand 2	Calcareous siltstone	Igneous rock 3
Loose, medium-graded gravel	Dolomitic siltstone	Igneous rock 4
Medium graded sand	Silty shale 2	Igneous rock 5
Coarse crossbedded gravel	Clay shale	Igneous rock 6
Medium, well-sorted sand	Cherty shale	Igneous rock 7
Inorganic silt 1	Dolomitic shale	Igneous rock 8
Inorganic silt 2	Calcareous shale or marl	Pophyritic rock 1
Inorganic silt 3	Carbonaceous shale	Porphyritic rock 2
	Oil shale	Vitrophyre
		Quartz
		Ore

Line Properties

Use the **Line** page in the Properties window to change line properties for selected lines in the document. You can set default line properties in the **Options** dialog Defaults page by clicking on *Line*.

Most line properties are edited in the **Properties** window in a *Line Properties* section. When changing line properties for a selected object, the *Line Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Occasionally, objects will open a Line Properties dialog to access the line properties.



Specify individual line properties in the ***Properties*** *window in a* Line Properties *section.*

Sample

The sample of the line is displayed next to *Line Properties*. The sample shows the line style, color, opacity, and width options.

If the *Line Properties* section is closed, click the ⊞ next to *Line Properties* to open the section.

Style

Click the line next *Style* to open the line style palette. Click on a style to use it for the selected line. The line style sample updates to show the new selection. Click on the ... button at the right of the line style to open the **Custom Line** dialog, where you can specify a custom line style.

Complex line styles can be selected for most object types. Wireframe map layers and surface mesh lines do not support complex line styles. When a complex line style is selected, the line *Width* automatically increased to 0.03125 inches (0.079 cm). Most complex line styles require lines with the *Width* set to something larger than 0.03125 inches to fully distinguish the line style.

Color

Click the color next to *Color* to open the color palette. Click on a color to use it for the selected line. The color box and the sample line update to show the new selection. Click on the ... button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Opacity

The *Opacity* is the amount of transparency of the line. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard or click anywhere else in the **Properties** window to make the change. Alternatively, click and drag the to change the opacity percentage.

Width

The *Width* controls the thickness of the line in page units. The value can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide. To change the *Width*, highlight the existing number and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the width.

End Styles

The *End Styles* section is unique to polylines. The ends of the polylines can have arrowheads on them as defined in the *End Styles* section. To open the *End Styles* section, click the next to *End Styles*.

The *Start* style is placed at the first vertex of the polyline. The *End* style is placed at the last vertex of the polyline. To change the *Start* or *End* style, click on the current option and select the desired option from the list.

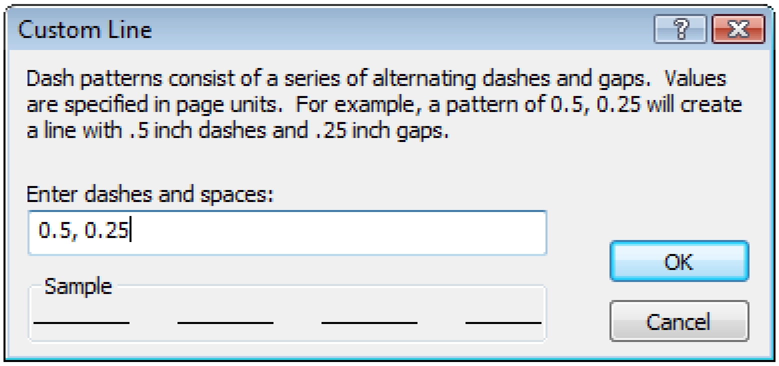
The *Scale* determines the scale factor of the arrowhead. To change the size, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the values. A value of 1 makes the arrow the default size.

Custom Line Style

Click the button to the right of the line style in the Properties window to create new line styles. You can add line styles to the palette, remove line styles from the palette, or replace existing line styles in the palette.

Custom Line Dialog

Click the button to the right of the line style to open the **Custom Line** dialog.



Use the Custom Line dialog to create custom line styles.

Enter Dashes and Spaces

The *Enter dashes and spaces* box determines the pattern of the custom line.

As an example, refer to the custom dash line style shown above. A pattern of 0.5, 0.25 will create a line with .5 inch dashes and .25 inch gaps. The pattern repeats the dash pattern and is displayed in the *Sample* box.

Sample

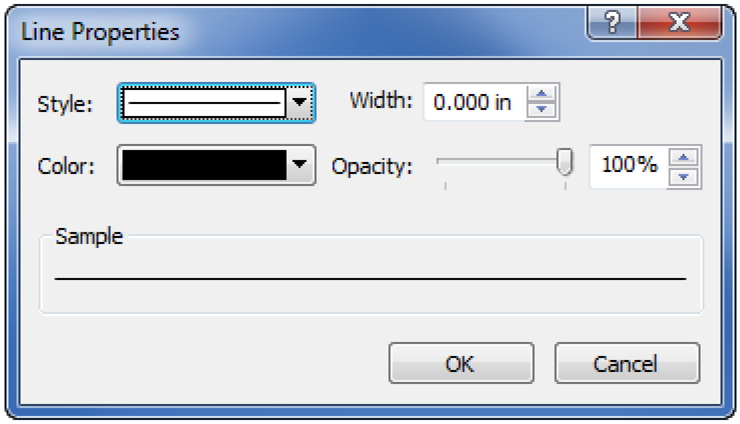
The *Sample* box shows the new line style.

Line Properties Dialog

Use the **Line Properties** dialog to change line properties for selected lines in the document. You can set default line properties in the **Options** dialog Defaults page by clicking on the *Line option*.

Most line properties are edited in the Properties window in a Line Properties section. When changing line properties for a selected object, the *Line Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Some objects use a **Line Properties** dialog. This dialog is accessed differently, depending on the type of object selected. For example, with the *Level method* set to *Advanced* for a contour map, the *Line Properties* dialog appears when you double-click on an individual line in the **Levels for Map** dialog.



*Specify individual line properties in the **Line Properties** dialog.*

Style

Click the button next *Style* to open the line style palette. Click on a style to use it for the selected line. The line style sample updates to show the new selection. Click on the *Custom* button at the bottom of the line style palette to specify a custom line style.

Complex line styles can be selected for most object types. Wireframe map layers and surface mesh lines do not support complex line styles. When a complex line style is selected, the line *Width* automatically increased to 0.03125 inches (0.079 cm). Most complex line styles require lines with the *Width* set to something larger than 0.03125 inches to fully distinguish the line style.

Color

Click the button next to *Color* to open the color palette. Click on a color to use it for the selected line. The color box and the sample line update to show the new selection. Click on the *Custom* button at the bottom of the color palette to choose a custom color.

Width

Change the line *Width* by typing a new number into the box or by using the buttons to the right of the box to increase or decrease the value. The line width can be 0 to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide.

Opacity

The *Opacity* is the amount of transparency of the line. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value, use the buttons to the right of the box to increase or decrease the value, or click and drag the to change the opacity percentage.

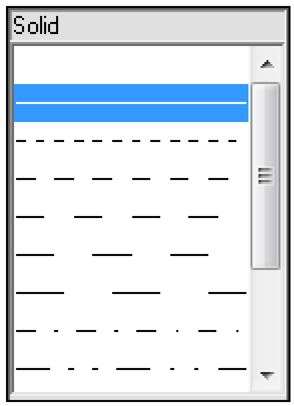
Sample

The sample of the line is displayed in the *Sample* section. The sample shows the line style, color, opacity, and width options.

Line Palette

The line palette is opened by clicking the arrow to the right of the line sample.

- The name of the line appears at the top of the palette.
- Select a line from the palette by clicking on a line.
- Create a custom line style by clicking the ... button to the right of the selected line style.



Select a predefined or custom color from the line palette.

Line Styles

Select a line style from the line palette.

Line Styles

Invisible	
Solid	
.1 in. Dash	
.2 in. Dash	
.3 in. Dash	
.4 in. Dash	
.5 in. Dash	
Dash Dot	
Dash Dot Dot	
Dash Dot Dot	
Dash Dash Dot	
Dash Dash Dot	
Dash Dash Dot	

Complex Line Styles

Solid - Hollow squares	
Solid - Hollow half squares, right	
Solid - Ticks	
Solid - Half ticks, alternating	
Solid - Double ticks	
Solid - Double half ticks, alternating	
Solid - Half ticks, right	
Solid - Double half ticks, right	
Solid - Circles	
Solid - Squares 1	
Solid - Squares 2	
Solid - Half squares, left	
Solid - Half squares, right	
Solid - Diamonds	
Solid - Triangles	
Solid - Half ovals	
Solid - Dashes	
Solid - Arrow, left	
Solid - Back arrow, left	
Solid thin - Half ticks, right	
Double solid	
Double solid - Ticks 1	
Double solid - Ticks 2	
Double solid - Double half ticks, right	
Double solid - Interior ticks	
Double solid - Circles	
Double solid - Squares	
Double solid - Diamonds	
Double solid - Triangles	
Double solid - Dashes	
Double solid, thin	
Double solid, thin - Ticks	
Double solid, thin - Interior ticks	
Triple solid	
Triple solid - Interior ticks	
Triple solid, thin	
Long dash - Ticks	
Long dash - Double ticks	
Long dash - Pluses	
Double long dash	
Medium dash - Half ticks, alternating	

Medium dash - Half ticks, right	
Medium dash - Half ticks, left	
Medium dash - Double half ticks, right	
Medium dash - Double half ticks, left	
Medium dash - Double half ticks, alternating	
Medium dash - Circles	
Medium dash - Diamonds	
Medium dash - Triangles	
Medium dash - Ovals	
Medium dash - Squares	
Long barbell	
Medium barbell	
No line - Ticks	
No line - Chevrons	
No line - Chevrons, reversed	
No line - Circles	
No line - Triangles	
No line - Diamonds	
No line - Squares	
Medium dash, wide	
Short dash, wide	
Dash/dot, wide	
Dash/dot/dot, wide	
Scallop	
Wave	
Square wave	
Wavy dash	
Double wavy dash	
Wavy dash, queried	
Warm front	
Cold front	
Stationary front	
Occluded front	
Fault - Approximately located, queried	
Fault - Inferred, queried	
Fault - Concealed, queried	
Fault - Upthrown/Downthrown	
Strike-slip fault, left-lateral offset - Certain	
Strike-slip fault, right-lateral offset - Certain	
Thrust fault, 1st generation - Certain	
Thrust fault, 1st generation - Approximately located	

Thrust fault, 1st generation - Approximately located, queried	
Thrust fault, 1st generation - Inferred	
Thrust fault, 1st generation - Inferred, queried	
Thrust fault, 1st generation - Concealed	
Thrust fault, 1st generation - Concealed, queried	
Thrust fault, 2nd generation - Certain	
Thrust fault, 2nd generation - Approximately located	
Thrust fault, 2nd generation - Approximately located, queried	
Thrust fault, 2nd generation - Inferred	
Thrust fault, 2nd generation - Inferred, queried	
Thrust fault, 2nd generation - Concealed	
Thrust fault, 2nd generation - Concealed, queried	
Detachment fault, type 1, 1st generation - Certain	
Detachment fault, type 1, 1st generation - Approximately located	
Detachment fault, type 1, 1st generation - Approximately located, queried	
Detachment fault, type 1, 1st generation - Inferred	
Detachment fault, type 1, 1st generation - Inferred, queried	
Detachment fault, type 1, 1st generation - Concealed	
Detachment fault, type 1, 1st generation - Concealed, queried	
Anticline - Certain	
Anticline - Approximately located	
Anticline - Approximately located, queried	
Anticline - Inferred	
Anticline - Inferred, queried	
Anticline - Concealed	
Anticline - Concealed, queried	
Overturned anticline - Certain	
Overturned anticline - Approximately located	
Overturned anticline - Approximately located, queried	

Overturned anticline - Inferred	
Overturned anticline - Inferred, queried	
Overturned anticline - Concealed	
Overturned anticline - Concealed, queried	
Inverted anticline - Certain	
Inverted anticline - Approximately located	
Inverted anticline - Approximately located, queried	
Inverted anticline - Inferred	
Inverted anticline - Inferred, queried	
Inverted anticline - Concealed	
Inverted anticline - Concealed, queried	
Syncline - Certain	
Syncline - Approximately located	
Syncline - Approximately located, queried	
Syncline - Inferred	
Syncline - Inferred, queried	
Syncline - Concealed	
Syncline - Concealed, queried	
Overturned syncline - Certain	
Overturned syncline - Approximately located	
Overturned syncline - Approximately located, queried	
Overturned syncline - Inferred	
Overturned syncline - Inferred, queried	
Overturned syncline - Concealed	
Overturned syncline - Concealed, queried	
Inverted syncline - Certain	
Inverted syncline - Approximately located	
Inverted syncline - Approximately located, queried	
Inverted syncline - Inferred	
Inverted syncline - Inferred, queried	
Inverted syncline - Concealed	
Inverted syncline - Concealed, queried	
Dipping Bed - Certain	
Dipping Bed - Approximately located	
Dipping Bed - Approximately located, queried	
Dipping Bed - Inferred	
Dipping Bed - Inferred, queried	
Dipping Bed - Concealed	
Dipping Bed - Concealed, queried	
Overturned Bed - Certain	

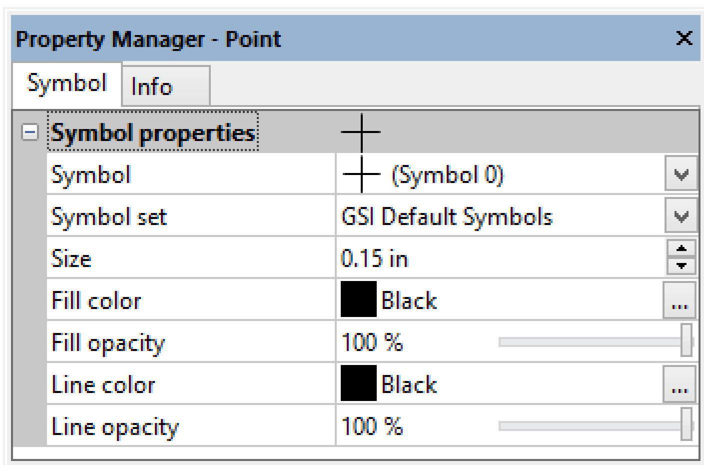
Symbol Properties

Use the **Symbol Properties** in the Properties window to change or set symbol properties. You can set default symbol properties in the Options dialog in the *Default Properties* section by clicking on *Symbol*.

Symbol Properties

Most symbol properties are edited in the **Properties** window in a *Symbol Properties* section. When changing symbol properties for a selected object, the *Symbol Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Occasionally, objects will open a Symbol Properties dialog to access the symbol properties.



*Specify symbol properties in the **Properties** window.*

Symbol

Choose the *Symbol* by clicking the existing symbol and selecting a new symbol from the symbol palette. The index number of the selected symbol is indicated in the title bar above the palette. This number is useful when automating the application with **Scripter** or when using a symbol column with a post map.

The symbol index is the symbol or glyph number as it appears next to the *Symbol* option. This is the 0-based offset of the symbol within the symbol set. To make the symbol the same as its ASCII code, add 32 to the value. You can also use the Window’s Character Map to determine the ASCII code for font symbols.

Symbol Set

Select the *Symbol set* from the list. The *Symbol set* can be any TrueType font installed on your system. To change the *Symbol set*, click on the existing font name. Select the new font from the list.

Size

Change the *Size* of the symbol by highlighting the existing value and typing a new number into the box. Alternatively, click the buttons to increase or decrease the symbol size. Symbols can be from 0 to 4 inches (0 to 10.160 centimeters) in size. Sizes are reported in page units.

Fill Color

The *Fill color* is the inside color of a solid symbol. Change the *Fill color* of the symbol by selecting a new color from the color palette. Create new colors by clicking the ... button to the right of the selected color.

Fill Opacity

Change the *Fill opacity* of the symbol by highlighting the existing value and typing a new value or by clicking and dragging the to change the opacity percentage. Percentages range from 0% (completely transparent) to 100% (completely opaque).

Line Color

The *Line color* is the outside edge color of the symbol. Change the *Line color* of the symbol by selecting a new color from the color palette. Create new colors by clicking the ... button to the right of the selected color.

Line Opacity

Change the *Line opacity* of the symbol by highlighting the existing value and typing a new value or by clicking and dragging the to change the opacity percentage. Percentages range from 0% (completely transparent) to 100% (completely opaque).

Custom Symbols

Custom symbols can be created using a third party TrueType font editing software.

Info Page

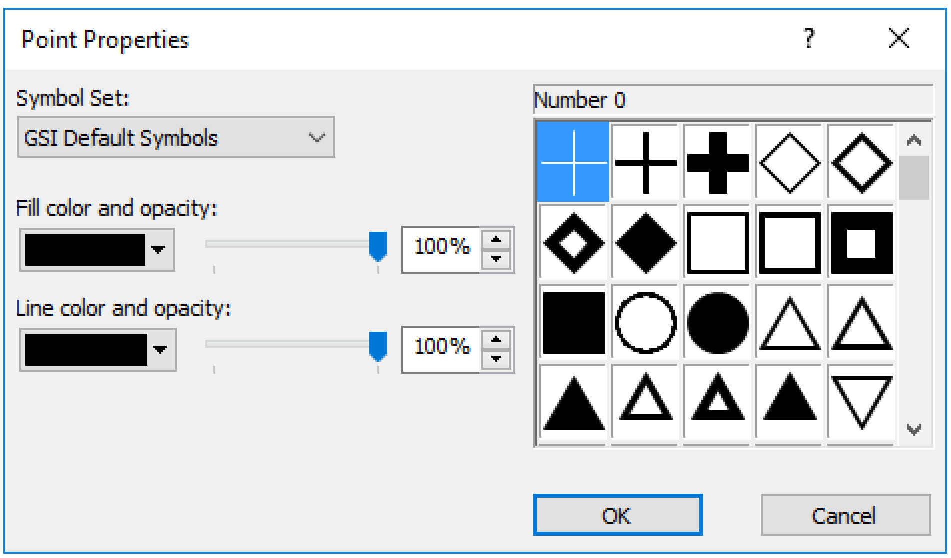
The Info page in the **Symbol Properties** dialog displays the *Position* and any attributes associated with the point.

Point Properties Dialog

Use the **Point Properties** in the Properties window to change or set symbol properties. You can set default symbol properties in the Options dialog in the *Default Properties* section by clicking on *Symbol*.

Most symbol properties are edited in the **Properties** window in a *Symbol Properties* or section. When changing symbol properties for a selected object, the *Symbol Properties* section will appear in a different location in the **Properties** window, depending on the type of object selected.

Occasionally, objects will open a **Point Properties** dialog to access the symbol properties.



*Specify symbol properties in **Point Properties** dialog.*

Symbol Set

Select the *Symbol Set* from the list.

Symbol

Choose the *Symbol* from the symbol palette. The index number of the selected symbol is indicated in the title bar above the palette. This number is useful when automating the application with **Scripter** or when using a *Symbol* column with a post map.

The symbol index is the symbol or glyph number as it appears next to the *Symbol* option. This is the 0-based offset of the symbol within the symbol set. To make the symbol the same as its ASCII code, add 32 to the value. You can also use the Window's Character Map to determine the ASCII code for font symbols.

Fill Color and Opacity

The *Fill color* is the inside color of a solid symbol. Change the *Fill color* of the symbol by selecting a new color from the color palette. Create new colors by clicking *Custom* at the bottom of the color list.

Change the *Fill opacity* of the symbol by highlighting the existing value and typing a new value or by clicking and dragging the to change the opacity percentage. Percentages range from 0% (completely transparent) to 100% (completely opaque).

Line Color and Opacity

The *Line color* is the outside edge color of the symbol. Change the *Line color* of the symbol by selecting a new color from the color palette. Create new colors by clicking *Custom* at the bottom of the color list.

Change the *Line opacity* of the symbol by highlighting the existing value and typing a new value or by clicking and dragging the to change the opacity percentage. Percentages range from 0% (completely transparent) to 100% (completely opaque).

Size

Change the *Size* of the symbol by typing a new number into the box. The *Size* property is not displayed when the **Point Properties** dialog is opened from the Symbology dialog.

Custom Symbols

Custom symbols can be created using a third party TrueType font editing software.

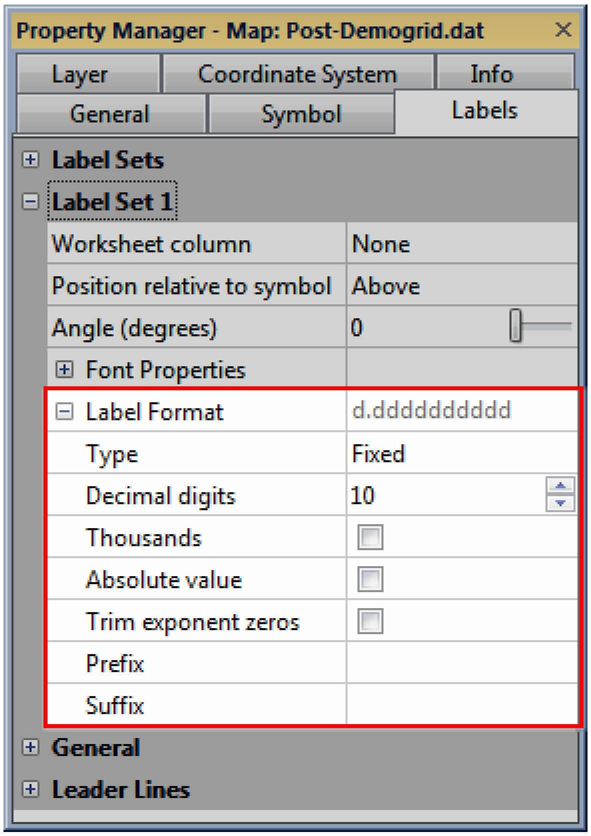
Info Page

The Info page in the **Point Properties** dialog displays the *Position* in terms of X, Y.

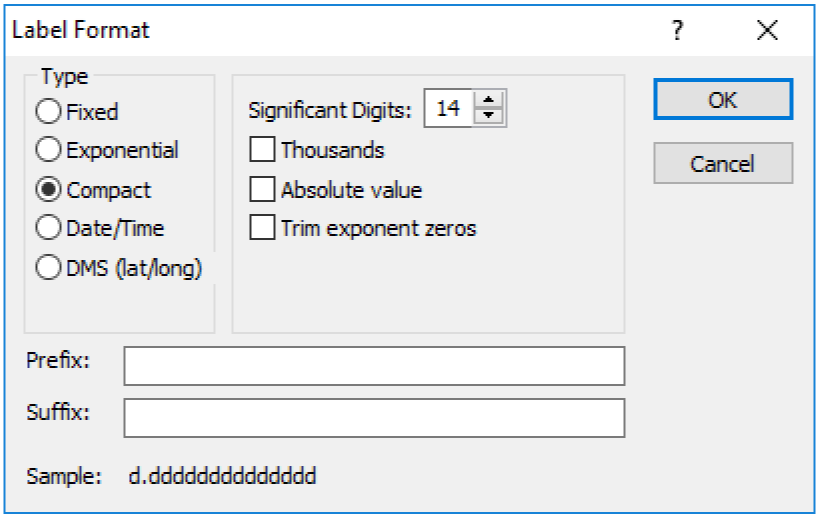
Label Properties

The label format used by various objects may be modified using the *Label Format* section in the Properties window or the **Label Format** dialog. The label *type*, *length*, *prefix*, and *suffix* may be set using the label format options.

The *Label Format* section in the **Properties** window is typically accessed by clicking the **Labels** tab and opening the *Label Format* section. The dialog is displayed for a contour map that uses the *Advanced Levels* options. The options in both the **Properties** window and the dialog work in the same manner.



The Label Format section for a post map allows customizing the format for post map labels.



Contour maps with Advanced label options display in the ***Label Format*** *dialog.*

Type

The *Type* option changes how numbers are displayed.

Setting *Type* to *Fixed* displays numbers as dd.dd. The numbers to the right of the decimal are set in theDecimal digitsbox. For example, if the numeric format is set toFixedwith three digits after the decimal point, the number 1998 displays as 1998.000.

The *Exponential* option displays numbers as d.ddE+dd. The first d represents a single digit before the decimal. The dd after the decimal represents one or more digits, and the number of digits displayed here is controlled by theDecimal digit-sproperty.The +dd represents a sign and at least two digits for the exponent. At least two digits are displayed unless theTrim exponent zerosoption is selec-ted.For example, if the numeric format is set toExponentialwith twoDecimal digits, then 1998 displays as 1.99E+03.

The *Compact* option displays the labels in either fixed or exponential fashion, whichever requires fewer digits. Enter the number of digits to display in the *Sig-nificant digits* box. For example, if the numeric format is set to *Compact* with two total digits, the year 1998 displays as 1.9E+03.

The *Date/time* option displays the labels as a combination of date and time formats. When the *Type* is set to *Date/Time*, the *Date/Time Format* option becomes available. Type a date/time format string in the *Date/Time Format* field to set the date/time format. Alternatively click the ... button to create the format in the Date/Time Format Builder dialog.

The *DMS (Lat/long)* option displays the labels in Degrees, Minutes, Seconds format. The properties in the *Label Format* section change to *DMS (Lat/long)* specific properties when the *Type*is set to *DMS (Lat/long).*

The *Text* option displays the labels as unformatted text. The *Text* option is available for base map labels.

Decimal Digits

The numbers to the right of the decimal are set in the *Decimal digits* box when *Type* is set to either *Fixed*, *Exponential* , or *DMS (Lat/long)* labels. When the *Type* is *DMS (Lat/long)* , the *Decimal digits* property is only displayed for applicable *DMS Format* selections.

Significant Digits

The number of significant digits is set in the *Significant digits* box when *Type* is set to *Compact* labels. *Significant digits* include numbers before and after the decimal. Scientific notation is used when the number of digits before the decimal exceeds the number of significant digits. For example when *Significant digits* is 3, the value 1001 is displayed as 1e+03 and the value 1005 is displayed as 1.01e+03.

Thousands

If the *Thousands* box is checked, a comma appears every three digits to the left of the decimal point.

Absolute Value

Check the *Absolute value* check box to display the absolute value of the numbers. Negative numbers are displayed without negative signs.

Trim Exponent Zeros

Check the *Trim exponent zeros* box to remove leading zeros in exponential numbers. For example, 1.9E+03 becomes 1.9E+3 if the *Trim exponent zeros* box is checked.

Date/Time Format

When the *Type* is set to *Date/time*, the *Date/Time Format* option becomes available. Type the desired date format into the *Date/Time Format* field, or click the  button to open the Date/Time Format Builder dialog to select and edit predefined date/time formats.

DMS Format

The *DMS format* property specifies the format of Degrees, Minutes, Seconds display. Click the current *DMS format* and select the desired format from the list to change the DMS style.

The following examples use the longitude -105.2250°.

DMS Format	**Sample**
ddmmss E	105 13 30 W
ddmmss e	105 13 30 w
ddmmss.ss E	105 13 30.0 W
ddmmss.ss e	105 13 30.0 w
ddmm.mm E	105 13.5 W
ddmm.mm e	105 13.5 w
d.dd	-105.2
d.dd E	105.2 W
d.dd e	105.2 w

Use Symbols

When the *Use symbols* check box is checked, the degrees symbol, °, follows the degrees, the minutes symbol, ', follows the minutes, and the seconds symbol, ", follows the seconds. The symbols are disabled when the *Use symbols* check box is unchecked.

Embed Space

Check the *Embed space* check box to add spaces after each of the DMS symbols and before the hemisphere suffix. There are no spaces between the symbols and numbers and no space between the hemisphere suffix and final symbol/number when *Embed space* is unchecked.

Suppress Leading Zeros

Check the *Suppress leading zeros* check box to remove leading zeros from the Degrees, Minutes, and Seconds values. When *Suppress leading zeros* is unchecked, the degrees value will always display three digits, and the minutes and seconds values will always two digits before the decimal.

Suppress Trailing Zeros

Check the *Suppress trailing zeros* check box to remove trailing zeros after the decimal point. When *Suppress trailing zeros* is unchecked, the number of decimal digits specified by the *Decimal digits* property will always be displayed.

Prefix

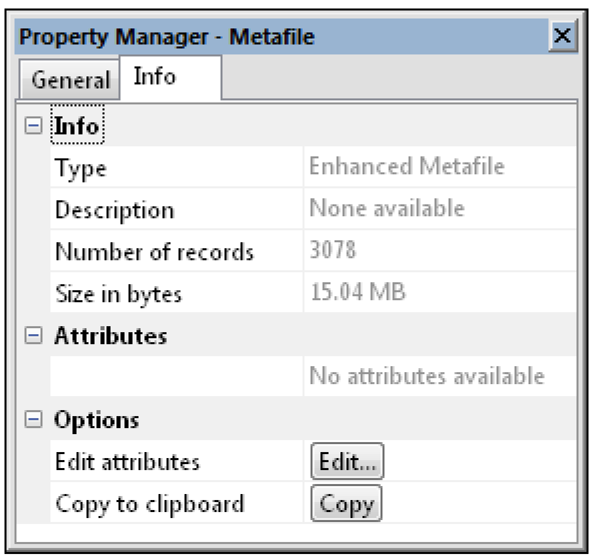
You can add a text string before each label using the *Prefix* box. For example, a "$" could be used as a prefix. Type the text exactly as you want it to appear in the *Prefix* box.

Suffix

You can add a text string after each label using the *Suffix* field. For example, a unit of measure " ppm" could be used as a suffix. Type the text exactly as you want it to appear.

Metafile Properties

A metafile is a collection of Windows graphic commands that create text and images. Metafiles are usually transferred through the clipboard or by file. Metafiles can be imported with the **Home | Insert | Graphic** command. Metafiles can also be used to create a base map with the **Home | New Map | Base** command.



*The **Properties** window has information and opacity control for a metafile.*

General Page

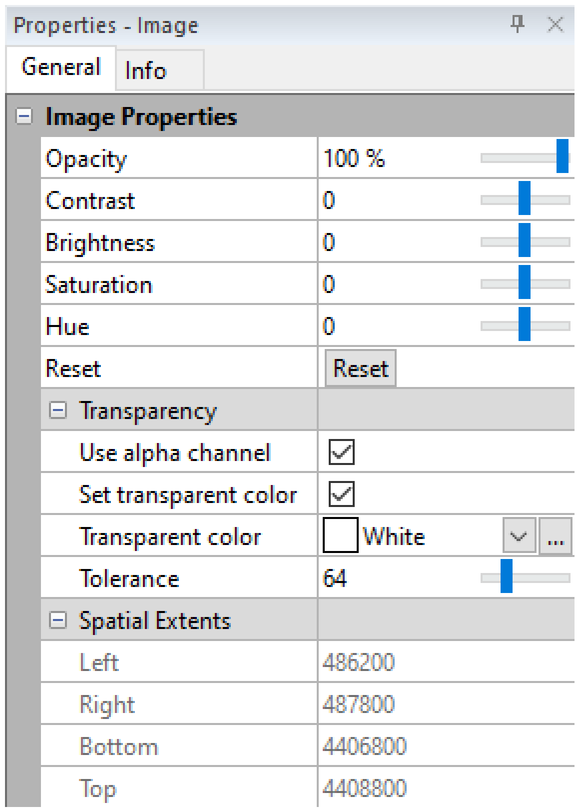
The **General** page controls the metafile opacity. Change the *Opacity* of the metafile by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the slider to change the opacity percentage.

Info Page

The Info page for a metafile displays the *type* of metafile, *description, number of records,* and *size in bytes*.

Image General Properties

Image files can be loaded into **Surfer** with the Home | Insert | Graphic command. Click on the *Image* object in the Contents window to open the image properties in the Properties window. If you wish to use the image in a base (raster) layer, and georeference the image, use the Home | New Map | Base or **Home | Add to Map | Layer | Base** command instead.



*The **General** page controls the image appearance and shows the* Spatial Extents.

Image Properties

The first five properties on the **General** page transform the image appearance: *Opacity*, *Contrast*, *Brightness*, *Saturation*, and *Hue*. These image transformations are applied to the display of the image or base layer, not to the image file. However, the image transformations are saved to the exported file when exporting the image or map.

Opacity

Change the *Opacity* of an image by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the to change the opacity percentage.

Contrast

Contrast is the difference in luminance or color of the objects in the image. Light and dark objects appear to stand out from one another when contrast is high. The difference in luminance between objects is difficult to see when contrast is low. The *Contrast* value is relative in **Surfer**. The original image is given a *Contrast* value of 0. Positive values increase the image contrast. Negative values decrease the image contrast. Type a value between -100 and 100 in the *Contrast* field or click and drag the to adjust the image contrast.

Brightness

Brightness is the perception of emitted or reflected light. Brightness can also be considered the average of the Red, Green, and Blue values for the pixels in the image. As brightness increases objects become lighter until they begin to "wash out" or turn white. Objects in the image get darker until they become black as brightness decreases. The *Brightness* value is relative in **Surfer**. The original image is given a *Brightness* value of 0. Positive values increase the image brightness. Negative values decrease the image brightness. Type a value between -100 and 100 in the *Brightness* field or click and drag the to adjust the image brightness.

Saturation

Saturation is a combination of light intensity and distribution across different wavelengths. High saturation relates to vivid, bright colors, i.e. high intensity across a narrow wavelength band or even single wavelength. This can also be considered very pure color. Low saturation leads to muted or gray colors, i.e. low intensity across a wide band of wavelengths. No saturation transforms the image to grayscale. The *Saturation* value is relative in **Surfer**. The original image is given a *Saturation* value of 0. Positive values increase the image saturation. Negative values decrease the image saturation. Type a value between -100 and 100 in the *Saturation* field or click and drag the to adjust the image saturation.

Hue

Hue describes the colors in the image. The *Hue* property shifts the colors in the image around the color wheel. The *Hue* value is relative in **Surfer**. The original image is given a *Hue* value of 0. Positive values shift the colors in the red to orange, then yellow, green, blue and finally magenta direction. Negative values shift colors in the opposite direction, from red to magenta then blue, green, yellow and finally orange. Type a value between -180 and 180 in the *Hue* field or click and drag the to adjust the image hue.

Reset

Click *Reset* to return the image to its original appearance. This sets *Opacity* to 100% and *Contrast*, *Brightness*, *Saturation*, and *Hue* to 0.

Transparency

The *Transparency* section includes properties for defining a color to make transparent in the image.

Use Alpha Channel

Select *Use alpha channel* to use the fourth channel of data as alpha for transparency. This setting is independent of *Set transparent color* and *Opacity* on the Layer page.

Set Transparent Color

Select *Set transparent color* to specify a color to make transparent in the image. This can be useful when the image includes a background or some other solid color element you wish to remove from the display.

Transparent Color

Specify the color to make transparent in the *Transparent color* field. Select a color from the color palette or click ... to set a custom color in the Colors dialog.

Tolerance

Specify a tolerance to include colors near the *Transparent color* to also make transparent. Select a tolerance between 0 - 255. A *Tolerance* of 0 means the color in the image must match the *Transparent color* exactly. As *Tolerance* increases, more colors are included.

Click *Reset* to return the image to its original appearance. This sets *Opacity* to 100% and *Contrast*, *Brightness*, *Saturation*, and *Hue* to 0.

Spatial Extents

The *Left, Right, Bottom,* and *Top* values in the *Spatial Extents* group displays the current extents of the image. These values are read-only. Click the *Georefence image* button to change the spatial extents.

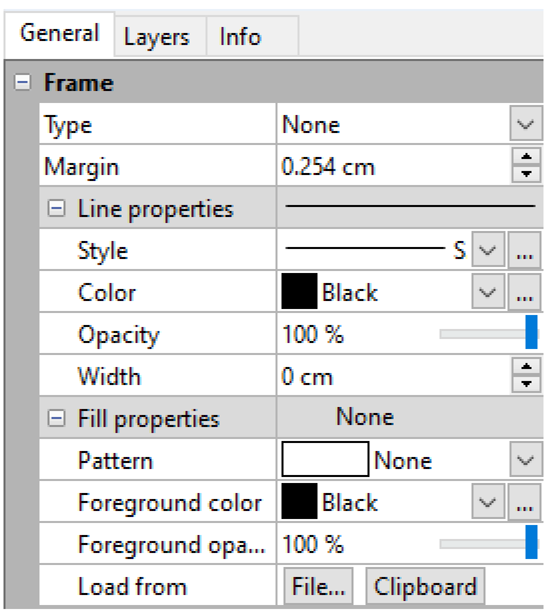
Georeference

When an image is included in a base (vector) layer, the *Georeference* property is available. Click *Georeference image* in the *Georeference* field to apply control points to the image and reference it to your map's coordinate system. Clicking *Georeference image* opens the Georeference Image window. The image must be at least 2x2 pixels to be georeferenced.

The **Georeference Image** command is only available in the image properties when the image is part of a base (vector) layer. An image is only part of a base (vector) layer when the base map was created from a file format that includes both vector and raster data. A base (raster) layer is created when creating a base map from an image file, and the image properties are found in the General page for the base (raster) layer.

Frame Properties

The *Frame* section in the **General** tab specifies the properties to use for the border around different objects to which a frame can be applied, such as legends, color scale bars, and map scale bars.



*Change Frame properties in the **General** tab*

Frame Type

The *Type* list specifies the type of border to use for the frame: *None*, *Square*, or *Rounded*. Setting this value to *None* shows no border. Fill properties cannot be applied to a frame when the *Type* is set to *None*. Setting the *Type* to *Square* creates a rectangle at the edge of the object at the *Margin* distance. Setting *Type* to

Rounded creates a rounded rectangle at the edge of the object at the *Margin* distance.

Margin

Margin specifies the distance between the edge of the frame and the text or symbols within the framed object. Values are in inches or centimeters, determined by the page units.

Line

Click the ⊞ next to *Line Properties* to open the Line Properties section. Set the *Style*, *Color, Opacity*, and *Width* of the line to use for the frame.

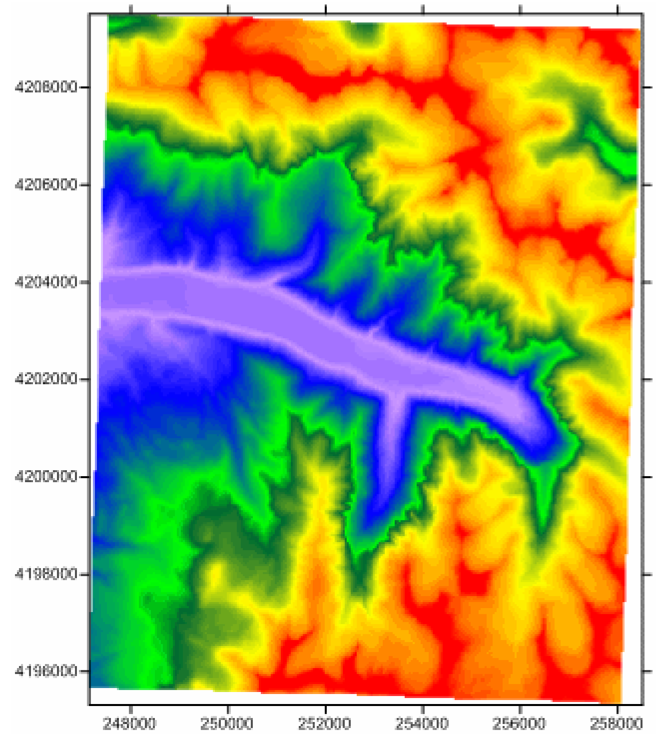
Fill

Click the ⊞ next to *Fill Properties* to open the Fill Properties section. Set the *Pattern, Color,* and other fill properties to use for the foreground and background of the framed area.

Introduction to Color Spectrums

Color fill can be added to several map types and objects in **Surfer**. Contour maps can be line drawings or they can have color fill. The vector symbols on a vector map can be colored. You can choose a preset color spectrum for color relief maps, contour maps, and 3D surface maps or you can create and save your own color spectrum.

Colors are assigned through the Fill, Fill Properties, or Colormap properties, depending on the type of object. Color spectrums can be saved for future use as color spectrum files .CLR or as level files .LVL.



Color spectrums, such as the variation of Rainbow.CLR used in the color relief map above, can be saved for future use in other maps.

Opening the Colormap Editor

The Colormap Editor is opened slightly differently for each map type.

Contour Maps

Contour maps have two different ways of setting the contour fill color, depending on if you are using the simple, logarithmic, or advanced *Level Method*.

To set the contour map fill color:

1. Click on the contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Check the *Fill contours* option. This fills the contours with the default color scheme.
4. If the *Level method* is set to *Simple* or *Logarithmic:*
   a. Click the color bar next to *Fill colors* and select a new color scale.
   b. To open the **Colormap Editor**, click the ... button to the right of the color.
5. If the *Level method* is set to *Advanced*:
   a. Click the *Edit Levels* button next to *Contour levels*.
   b. Click the *Fill* button.

c. In the Fill dialog, click the *Foreground Color* button. The **Colormap Editor** opens.

Contour map lines also use the **Colormap Editor** if the gradational line color option is selected. To set the gradational line option and set the line color:

1. Click on the contour map to select it.
2. In the **Properties** window, click the **Levels** tab.
3. Set the *Level method* to *Advanced.*
4. Click the *Edit Levels* button next to *Contour levels*.
5. In the dialog, click the *Line* button.
6. In the Line dialog, select *Gradational*.
7. Click the *Color* button. The **Colormap Editor** opens.

Color Relief Maps

To open the **Colormap Editor** for a color relief map,

1. Click on the color relief map to select it.
2. In the **Properties** window, click on the **General** tab.
3. Open the *General* section and click the *Colors* button to select a new colormap from the list.
4. To open the **Colormap Editor**, click the ... button to the right of the colormap.

Vector Maps

In a vector map, the symbols can use a colormap to gradationally color the vector symbols. To set the gradational symbol option:

1. Click on the vector map to select it.
2. In the **Properties** window, click the Symbol page.
3. Open the *Color Scaling* section.
4. Change the *Scaling method* to either *By magnitude* or *By grid file*.
5. Click the color bar next to *Vector colors* and select a new color scale.
6. To open the **Colormap Editor**, click the ... button to the right of the colormap.

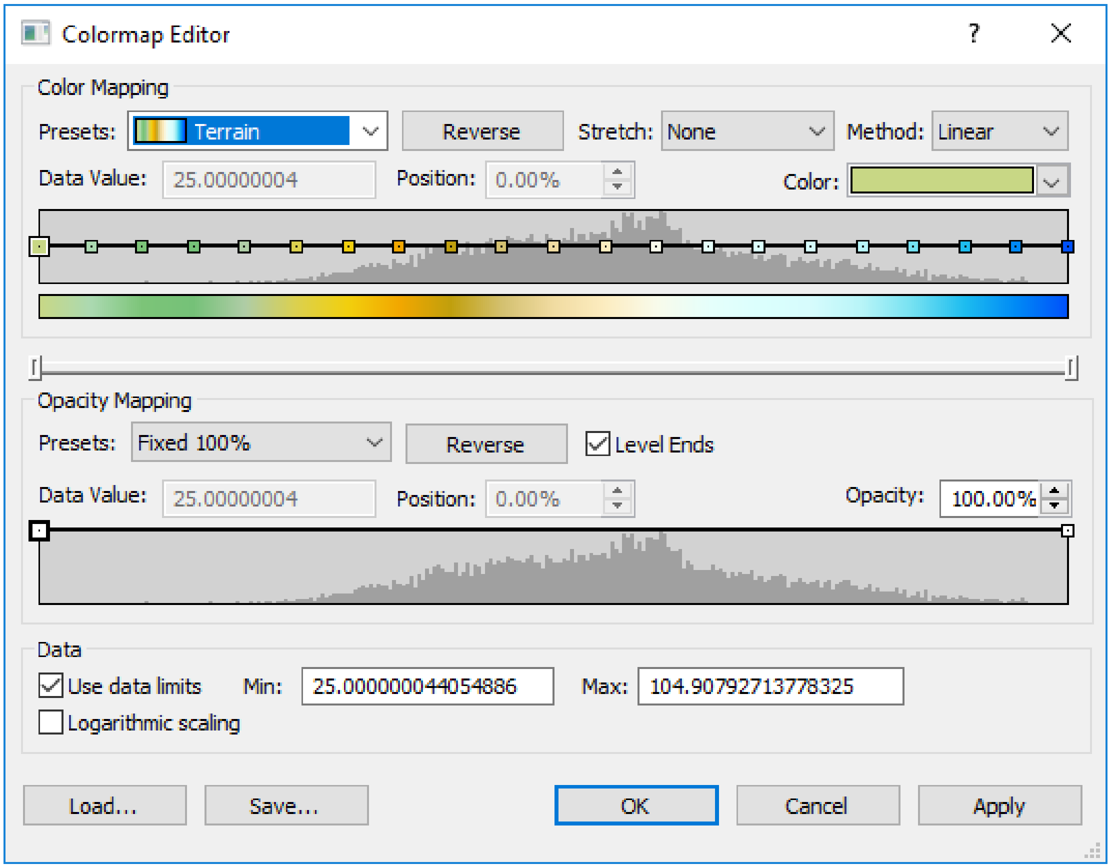
3D Surface Maps

To open the **Colormap Editor** for a surface map:

1. Click on the surface map to select it.
2. In the **Properties** window, click on the **General** tab.
3. Open the *Material Color* section.
4. Click the color bar next to *Upper* to select a new color scale.
5. To open the **Colormap Editor**, click the ... button to the right of the colormap.

Colormap Editor

The **Colormap Editor** is used to assign a gradation of colors. The color spectrum has specific colors assigned to anchor nodes along the spectrum. Surfer automatically blends colors to produce a smooth color gradation over the color fill. After you create a color spectrum, you can save custom spectrums for later use with other plots, plot fills, or object fills. Several predefined color spectrum .CLR files are available in the \Surfer\Color Scales\ directory.



*Use the **Colormap Editor** to assign gradient colors to a fill.*

Presets

The *Presets* section includes the list of presets and commands for modifying the preset color gradient list. Select a predefined .CLR file from the *Presets* list. When a custom colormap is displayed in the **Colormap Editor**, click *Add* to add the custom colormap to the *Presets* list. Click *Edit* to edit the *Presets* list in the Edit Presets List dialog.

Color Mapping

The *Color Mapping* section includes options for mapping specific colors to data values, distributing color anchor nodes, and interpolation methods.

Reverse

The *Reverse* button flips the order of colors in the colormap so that colors currently associated with low data values will be mapped to high values and colors currently associated with high values will be mapped to low values. The sample colormap updates to show the new color order.

Stretch

The *Stretch* field includes options to distribute colors (anchor nodes) across the colormap based on data values.

- By default, the *Stretch* is set to *None*. When *None* is selected, each anchor node can be positioned independently.
- Select *Evenly Distribute* to evenly distribute the anchor nodes between the data minimum and maximum. The nodes are automatically redistributed evenly when nodes are added or removed or when the data *Min* or *Max* is changed.
- Select *Equalize* to distribute the nodes to improve the contrast in the map via histogram equalization. The nodes are distributed so that each histogram "bin" between nodes contain an equal number of data points. The nodes are automatically redistributed evenly when nodes are added or removed. The *Equalize* option is not available when a histogram is not displayed, e.g. for watershed maps.

The *Stretch* option must be set to *None* before anchor nodes can be manually positioned.

Interpolation Method

The *Method* option determines which interpolation method is used to specify the colors between anchor nodes.

- *Linear* interpolation uses weighted averages to interpolate colors between nodes. This linear interpolation between each color node results in a gradual transition and smooth appearance.
- *Reverse* interpolation uses weighted averages to interpolate colors between nodes like the *Linear* interpolation. However, the colors are reversed between nodes. This reverse linear interpolation between each color node results in a segmented appearance.
- *Cosine* interpolation uses 180 degree segments of the cosine wave. The rate of change of the color is faster than the *Linear* interpolation. This interpolation results in a smooth appearance.
- *Flat Start* applies the color from the left node to the area between nodes. This interpolation results in a sharp transition between colors.
- *Flat Middle* applies the color from the midpoint linear interpolation between the left and right nodes to the area between nodes. This interpolation results in a sharp transition between colors.
- *Flat End* applies the color from the right node to the area between nodes. This interpolation results in a sharp transition between colors.

- *HSL CW* interpolates the colors between nodes from the Hue-Saturation-Luminosity color wheel in a clockwise direction.
- *HSL CCW* interpolates the colors between nodes from the Hue-Saturation-Luminosity color wheel in a counterclockwise direction.
- *HSL Shortest* interpolates the colors between nodes from the shortest distance (clockwise or counterclockwise)on the Hue-Saturation-Luminosity color wheel between each node pair.
- *HSL Longest* interpolates the colors between nodes from the longest distance (clockwise or counterclockwise)on the Hue-Saturation-Luminosity color wheel between each node pair.

Data Value and Position

The *Data Value* displays the data value of the selected node. The *Position* field displays the node's relative position in the colormap where the colormap minimum *Position* is 0% and the maximum is 100%. The selected node may be accurately repositioned by entering a new value in the *Data Value* or percentage in the Position field. The first and last nodes cannot be changed and this control is disabled when an end node is selected. To change the first and last nodes, change the *Min* and *Max* values in the *Data* section.

The *Data Value* option applies to any map divided into classes, such as maps with classed symbols or classed colors. To determine the color in the colormap associated with each class, the following rules are used:

- The first class uses the *Minimum* value color. This is the node on the far left side of the colormap.
- Subtract the *Minimum* from the *Maximum* value and divide by the *Number of classes*. This is the separation value. Add the separation value to the *Minimum* value. This will be the value color associated with the next class.
- Add the separation value to the previous value. This is the value color associated with the next class.
- Repeat adding the separation value to the previous value until only the last class remains. The last class uses the *Maximum* value color. This is the node on the far right side of the colormap.
- Surfer automatically interprets the colormap to determine the color value associated with each class. Any number of nodes can exist on the colormap. The nodes and the *Value* in the colormap are not associated with the classes.

Color

Click the *Color* box to specify the color of the currently selected anchor node. The color palette opens.

Data Histogram

A data histogram is displayed in both the *Color Mapping* and *Opacity Mapping* sections. The histogram consists of equal width bins between the data minimum and data maximum. The number of bins is the number of data points or 256, whichever is fewer.

- For most grid-based maps, the grid Z values are used to create the histogram.
- For base map symbology, the selected *Attribute field* data is used to create the histogram.
- For vector maps with magnitude scaling, the vector magnitude data is used to create the histogram. The magnitude will be calculated in the case of 1-grid vector and Cartesian 2-grid vector maps. The magnitude is used from the magnitude grid directly in the case of a polar 2-grid vector map. For vector maps with grid scaling, the Z values from the selected grid are used to create the histogram.
- For point cloud maps, the data selected in the *View by* property is used to create the histogram.

When the *Logarithmic scaling* option is selected in the *Data* section, the histogram bins are calculated from the log of the data values. This ensures the histogram remains aligned with the colormap when *Logarithmic scaling* is selected. This scales the X axis (bins).

You can also logarithmically scale the Y axis (frequency) values. This option is useful when only a few bins contain most of the data values. Logarithmic scaling the Y axis will show more detail in the histogram. To apply logarithmic scaling to the Y axis, right-click either of the data histograms and select **Logarithmic Scale** in the context menu. To return to linear scaling on the Y axis, right-click either of the data histograms and select **Linear Scale**.

In the *Color Mapping* section, the colormap anchor nodes are displayed above the data histogram. An anchor node's horizontal position corresponds to its *Data Value* and *Position*.

Colormap Example

The sample of the generated colormap is display directly below the histogram and anchor nodes.

Anchor Nodes

Anchor nodes are displayed directly above the data histogram. Anchor nodes specify a color and position in the *Color Mapping* section. Anchor nodes specify an opacity and position in the *Opacity Mapping* section.

You can add additional anchor nodes at any position along the colormap. To create a new anchor node, double-click on the data histogram where you want the new node added. In the *Color Mapping* section, the new anchor is automatically assigned a *Color* based on the previous colormap, and the *Data Value* is assigned based on where you clicked. In the *Opacity Mapping* section the new anchor is assigned the *Opacity* and *Data Value* corresponding to the exact location you clicked. You can add as many anchor nodes as you want. This lets you blend colors and opacities in many different ways on the colormap.

Left-click on an anchor node and press the DELETE key on the keyboard to delete a node. The first (far left) and last (far right) anchor nodes cannot be deleted.

Scroll Control

The scroll control appears as a horizontal bar with end handles. Drag a handle left or right to zoom the colormap and histogram in or out. Drag the center section to scroll the visible portion left or right. Double-click the center section to return it to the fully visible state. The scroll control changes the view of both the *Color Mapping* and *Opacity Mapping* sections.

Use the scroll control to zoom the colormap in or out.

Opacity Mapping

The *Opacity Mapping* section contains the opacity mapping controls. These controls specify how a range of data values are mapped to opacity in the final output. Note that opacity is the opposite of transparency (0.0 is completely transparent, 1.0 is completely opaque).

Presets

The options in the *Presets* list may be used to preset the opacity settings which can then be modified as desired.

- *Fixed 100%* sets the entire data range to a fixed opacity of 100% (fully opaque).
- *Fixed 20%* sets the entire range of data to a fixed opacity of 20%.
- *Ramp 0 to 100%* sets the data *Min* value to an opacity of 0% and the data *Max* value to an opacity of 100%. The opacity increases linearly from 0% to 100% throughout the data range.
- *Ramp 0 to 20%* sets the data *Min* value to an opacity of 0% and the data *Max* value to an opacity of 20%. The opacity increases linearly from 0% to 20% throughout the data range.
- *Middle Ramp* sets the lowest (fourth) quartile to 0% opacity, followed by a linear ramp over the third and second quartiles, followed by a fixed opacity of 100% for the greatest (first) quartile.
- *Custom* is displayed automatically when a node is selected, created, removed, or the *Opacity* value or *Data Value* options are adjusted.

Data Value

The *Data Value* displays the data value of the selected node. The *Position* field displays the node's relative position in the colormap where the colormap minimum *Position* is 0% and the maximum is 100%. The selected node may be accurately repositioned by entering a new value in the *Data Value* or percentage in the *Position* field. The first and last nodes cannot be changed and this control is disabled when an end node is selected. To change the first and last nodes, change the *Min* and *Max* values in the *Data* section.

Level Ends

Check the *Level Ends* box to level the first and last segments of the opacity graph. When the nodes at one end of a level segment are dragged up or down, the other end is also dragged.

Opacity

Specify the opacity for the selected node in the *Opacity* field. The *Opacity* value ranges from 0.00% (completely transparent) to 100.00% (completely opaque). To change the opacity, click on one of the nodes. Then, highlight the existing value and type a new value or click the button to increase or decrease the opacity for the selected node.

Data Histogram

The *Opacity Mapping* section includes a data histogram. See the *Color Mapping* section above for the histogram description. Opacity anchor nodes are displayed on the data histogram. An anchor node's vertical position corresponds to its *Opacity*. An anchor node's horizontal position corresponds to its *Data Value* and *Position*.

Data

The *Data* section contains options for setting the minimum and maximum data values and scaling to use in the colormap.

Use Data Limits

Check the *Use data limits* check box to use the data minimum and maximum values. This option is not available for classed post maps.

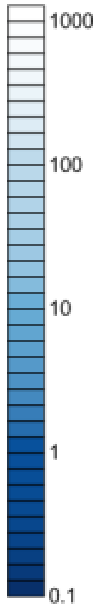
Colormap Data Minimum and Maximum

If you would prefer to set the colormap minimum and maximum values, enter the new numbers into the *Min* and *Max* boxes. This is useful when you are mapping different data sets in a similar range and would like to have the same data values represented by the same colors in each map. If a data value within the map falls outside this range, it is assigned the minimum or maximum color, whichever is closest.

The minimum and maximum values cannot be set for the classed post map. For the classed post map, the far left *Min* node is always the median value for the first class. The far right *Max* node is always the median value for the last class. This cannot be changed.

Logarithmic Scaling

Check the box next to the *Logarithmic scaling* to set the intervals between the nodes to a log(10) scale. The *Min* and *Max* values are the same. The nodes between the minimum and maximum and the color definitions adjust to fit the log(10) scale. On a log(10) scale, there is as much distance on the colormap sample between 1 and 10 as between 10 and 100 or 100 and 1000. In the example below, the nodes are displayed at 0.1, 1, 10, 100, and 1000. To use a regular linear scale on the colormap, clear the box next to *Logarithmic scaling.*



The associated color scale bar is shown with the major log intervals displayed.

Loading a Colormap

The *Load* button opens an existing colormap .CLR file. When you click the *Load* button, the **Open** dialog is displayed with a list of colormap files. Click the file you want to use and click *Open*. The colormap is updated to show the colormap file settings.

Colormap files can be used with maps of varying Z ranges since the anchor nodes are stored as percentages rather than as data values. To use the exact same colors in the exact same data locations when the Z ranges vary slightly, override the default data limits and assign custom values in the *Data to Color Mapping* group.

Saving a Colormap

Click the *Save* button in the **Colormap Editor** to create a colormap .CLR file based on the current colormap settings. When you click the *Save* button, the **Save As** dialog is displayed. Type the *File name* for the colormap file and click *Save*. The file is saved for use with other files.

Anchor Nodes

The color spectrum is defined by anchor nodes at user-defined points along a color spectrum in the Colormap Editor. Colors and opacities are automatically blended between adjacent anchors. Anchor nodes are represented by a slider button along a line on the *Color Mapping* and *Opacity Mapping* data histograms.

Anchor Node Value

The data value for the node is displayed in the *Data Value* box. Except for the first and last nodes, any node can be assigned to a specific value by typing a new number into the *Data Value* box. The first and last node data values are controlled by the *Min* and *Max* in the *Data* section.

Add Anchor Node

You can add additional anchor nodes at any position along the colormap. To create a new anchor node, double-click on the data histogram where you want the new node added. In the *Color Mapping* section, the new anchor is assigned a *Color* and *Data Value* based on where you clicked. In the *Opacity Mapping* section, the new anchor is assigned an *Opacity* and *Data Value* corresponding to the exact location you clicked. You can add as many anchor nodes as you want. This lets you blend colors and opacities in many different ways on the colormap.

Delete Anchor Node

Left-click on an anchor node and press the DELETE key on the keyboard to delete a node. The first (far left) and last (far right) anchor nodes cannot be deleted.

Positioning a Color Anchor Node

To position an anchor node:

1. Move the mouse cursor over the slider button.
2. Click and hold the left mouse button.
3. Drag the slider button to the desired position and release the left mouse button. As you move the slider button left or right, the data value is indicated in the *Data Value* box. Alternatively, you can set the anchor to a specific data value by entering a number into the *Data Value* box. Note that the beginning and ending anchors cannot be moved or deleted.
4. Specify the *Min* and *Max* values in the *Data* section to change the range, and therefore, the position of the starting and ending nodes.

Selecting a Color for an Anchor

To select a color to associate with the anchor node in the *Color Mapping* section:

1. Click on the anchor you wish to modify.
2. Click on the desired color in the *Color* palette. The color spectrum is updated to indicate the change.

Alternatively, use a preset .CLR color file by selecting one of the options from the *Presets* list or click the *Load* button to load a custom .CLR file.

Positioning an Opacity Anchor Node

To position an anchor node:

1. Move the mouse cursor over the slider button.
2. Click and hold the left mouse button.
3. Drag the slider button to the desired position and release the left mouse button. As you move the slider button left or right, the data value is indicated in the *Data Value* box. Alternatively, you can set the anchor to a specific data value by entering a number into the *Data Value* box. As you move the slider button up or down, the *Opacity* value is increased or decreased. Alternatively, you can set the anchor to a specific value by typing a number in the *Opacity* box. Note that the beginning and ending anchors cannot be moved or deleted.
4. Specify the *Min* and *Max* values in the *Data* section to change the range, and therefore, the position of the starting and ending nodes.

Select an Opacity for an Anchor

To select an opacity to associate with the anchor node in the *Opacity Mapping* section:

1. Click on the anchor you wish to modify.
2. Type the desired value in the *Opacity* field. The color spectrum is updated to indicate the change.

Alternatively, click and drag the anchor node up or down, use a preset .CLR color file by selecting one of the options from the *Presets* list, or click the *Load* button to load a custom .CLR file.

Using Color Spectrums in Map Series

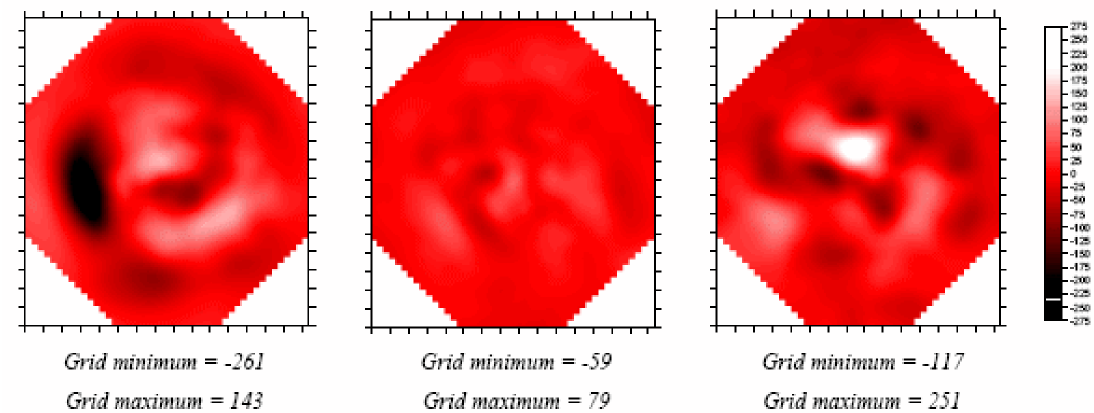
Sometimes it is necessary to use the same range of colors on different maps. By default, color spectrums are assigned to the minimum and maximum data values for a given data set. If the maps have different data ranges, the colors will not map to the same data values and it will be difficult to compare the maps. In this case, you may want to override the default data limits and assign custom values in the Colormap Editor*Data* group.

Example

The following example contains steps on how to create maps in a series that contain the same color spectrums although the data minimums and maximums differ. The example color relief maps show the same gridded area at three different points in time. To accurately compare the maps, the color spectrum needs to be the same through time even when the range of values fluctuates between grids.

Know Your Data

The first step in producing maps with the same color series is to know your data. Find the minimum and maximum data values for all of your data sets before you begin. This will help you to decide the minimum and maximum values to use in the color spectrum for the series of maps. In this example, the maximum Z value is +251 in one of the grids, and the minimum Z value is -261 in another grid. The largest and smallest Z values do not appear in the same grid.



The three maps shown above all use the same color spectrum range, even though the grid minimums and maximums differ.

The three maps above show the same phenomena over a period of time. The grid ranges are not the same, but since the maps are to be compared, the colors need to map to the same data values for all maps. Using the same color spectrum .CLR file and changing the minimum and maximum spectrum data values allows you to quickly compare the maps visually. The color scale bar to the right is used for all three maps. The map on the left contains a dark black area because it has some extreme negative values, but it does not have any pure white because it does not have extreme positive values. The map on the right is opposite, it contains extreme positive values so there is an area of pure white, but the negative values are not extreme so there is no pure black. The map in the center does not contain extremes in either direction so it is mostly red. Note that these maps have "stair step" edges due to NoData regions in the corners. A smaller grid spacing or interpolating the pixels would smooth the edges.

Creating a Color Spectrum File

After you know the series minimum and maximum and have decided on a spectrum minimum and maximum, you can create a color spectrum .CLR file to use with all the data. In this example, -275 is the minimum and +275 is the maximum.

To create a color spectrum .CLR file:

1. Create a map using the **Home | New Map | Color Relief** command. A color relief map is used throughout this example, though you can use contour maps or any other map type that uses a colormap in this procedure.

2. Select one of the sample .GRD files and click *Open*.

3. Click once on the color relief layer to select it.

4. In the **Properties** window, click on the **General** tab.

5. In the *General* section, click the ... button next to the currently selected colormap displayed in the *Colors* option to open the Colormap Editor.

6. In the *Data* section, change the *Minimum* to -275 and the *Maximum* to +275.

7. In the color spectrum bar, left-click in the middle of the color spectrum to add an anchor node. While the node is still selected, change the *Value* to -200 or slide the node to the -200 position (watch the numbers in the *Value* box). The minimum color (at -275) is black. Select black for the -200 node as well.

8. Repeat step 6 to add nodes at zero (gray) and +200 (white).

9. Click the *Save* button to save the color spectrum .CLR file.

10. In the **Save As** dialog, type a *File name* and click *Save*.

11. Click *OK* to make the color change to existing map.

Using the Color Spectrum File in Other Maps

After you have created the color spectrum file as above, you can use it in the remainder of the maps in a series. To use a color spectrum file in another map:

1. Create the map by clicking the **Home | New Map | Color Relief** command.

2. In this example, a new color relief map is created with the next grid file. Select the next grid file and click *Open*.

3. Click once the color relief layer to select it.

4. In the **Properties** window, click on the **General** tab.

5. In the *General* section, click the ... button next to the currently selected colormap displayed in the *Colors* option.

6. Click the *Load* button in the **Colormap Editor** and open the color spectrum .CLR file you just created.

7. In the *Data* group, set the *Minimum* and *Maximum* to the pre-determined values. In this example, the minimum is set to -275 and the maximum to +275.

8. Click *OK*to make the color change to this map.

9. Repeat this process for the rest of the maps in the series.

Level Files

.LVL Level files contain information needed to set the contour interval and basic line and fill information for contour and wireframe maps. Label and hachures information is also retained in the .LVL file. .LVL files can be created from the contour map, wireframe map, or in the worksheet.

Level files can be used in contour maps, 3D wireframe maps, and the grid editor.

Level Files

.LVL Level files contain information needed to set the contour interval and basic line and fill information for contour and wireframe maps. Label and hachures information is also retained in the .LVL file. .LVL files can be created from the contour map, wireframe map, or in the worksheet.

Level files can be used in contour maps, 3D wireframe maps, and the grid editor.

Level Files - Contour

For contour maps, .LVL level files contain information about contour levels, including the line properties, color fill, label frequency, and hachure information. The level file contains all the information contained on the **Levels** page in the advanced contour map Levels for Map dialog. After defining custom contour levels and colors on a map, you can save the level and color information in a level file. A level file can be recalled for any other contour map or 3D wireframe. Not all of the information in a level file can be used in a 3D wireframe. If you use a level file created in the contour map properties dialog in a wireframe, the color fill, contour label, and hachure information are ignored.

Creating Level Files

The easiest way to create a level file is from the advanced contour map properties or 3D wireframe properties dialogs. You can also create a level file in the worksheet or in an ASCII editor, using the .LVL file format.

To create a .LVL level file containing contour level information in the advanced contour map level properties:

1. Click once on a contour map to select it. The contour map properties will be shown in the Properties window.
2. Click on the **Levels** tab. Change the *Level method* to Advanced.
3. Click the *Edit Levels* button next to *Contour levels*. The Levels for Map dialog opens.
4. Change any of the contour parameters, including the contour levels, line properties, fill properties, contour labels, and hachure information.
5. After changing the settings, click *Save*.
6. In the **Save As** dialog, type the name of the level file and click the *Save* button. The level file is saved with a .LVL extension.
7. Click *OK* to close the dialog.

Using Level Files

To use an existing .LVL file with any contour map:

1. Click once on a contour map to select it. The contour map properties will be shown in the **Properties** window.
2. Click on the **Levels** tab. Change the *Level method* to Advanced.
3. Click the *Edit Levels* button next to *Contour levels*. The **Levels for Map** dialog opens.
4. Click *Load*.
5. Select the .LVL file and click *Open*.
6. Click OK and the contour map updates.

Note: All contour maps using the level file must have comparable Z data ranges, otherwise contour lines and fill will not appear on the map.

Wireframe Level Files

For 3D wireframes, .LVL level files contain information for the Z level line properties and for the color zone line properties. After defining custom level and line properties on the Z Levels page or the Color Zones page, save the level and color information in a level file. This level file can be recalled for any other 3D wireframe or contour maps.

Level files created from the wireframe properties dialog contain only information for the Z level line properties. Color fill, contour label, or hachure information is not written to the level file if the level file is created from the wireframe. However, a level file created from the advanced contour map level properties dialog may contain such additional information. A level file created from the wireframe properties dialog can be used for a contour map and vice versa. Any additional information not used is ignored.

Creating Level Files

To create a .LVL level file containing level and line property information from the wireframe properties:

1. Click once on a wireframe map to select it. The wireframe map properties will be shown in the **Properties** window.
2. Click on the **Z Levels** tab or the **Color Zones** tab.
3. Click the *Edit Levels* button. The **Properties** dialog opens.
4. Change any of the parameters, including the contour levels or line properties.
5. After changing the settings, click *Save.*
6. In the **Save As** dialog, type the name of the level file and click the *Save* button. The level file is saved with a .LVL extension.
7. Click *OK* to close the dialog. The wireframe map updates.

Using Level Files

To use an existing .LVL file with any wireframe map:

1. Click once on a wireframe map to select it. The wireframe map properties will be shown in the **Properties** window.
2. Click on the **Z Levels** tab or the **Color Zones** tab.
3. Click the *Edit Levels* button. The **Properties** dialog opens.
4. Click *Load*.
5. Select the .LVL file and click *Open.*
6. Click *OK* to close the dialog. The wireframe map updates.

Note, all wireframe maps using the level file must have comparable Z data ranges, otherwise lines will not appear on the map.

Creating Level Files in the Worksheet

You can create simple level files in the **Surfer** worksheet or an ASCII editor. The minimum amount of information a level file contains is the elevation data telling **Surfer** which contour lines to place on a contour map. For more information on the level file format, see the level file format.

Creating Level Files from the Worksheet

1. Enter level values into column A.
2. Add any other information you wish to include for the levels (i.e. line color) according to the level file format.
3. After entering the level information, click **File | Save As**.
4. In the *File name* field, enter the level file name and .LVL extension enclosed in quotes ("colors.lvl").
5. Choose *DAT Data (*.dat)* from the *Save as type* field and click *Save*.
6. In the **Data Export Options** dialog, choose *Space* as the *Delimiter* and *Double Quote* as the *Text Qualifier*. Choosing any other delimiter in the **Data Export Options** dialog results in an invalid level file since the file must be space delimited.
7. Click *OK* to create the level file.

Color Palette

The color palette is opened by clicking on an existing *color* button in the Colors list.

- The name of the color appears at the top of the palette.
- Select a color from the palette by clicking on a color.
- Create a custom color by clicking the ... button to the right of the selected color.

Select a predefined or custom color from the color palette.

Colors Dialog

Click the ... button to the right of any color in the Properties window to open the **Colors** dialog. Use the **Colors** dialog to load standard colors or create custom colors. There is a *Custom* button at the bottom of all palettes.

Standard Page

The standard colors appear on the **Standard** page in a standard palette spectrum.

Colors

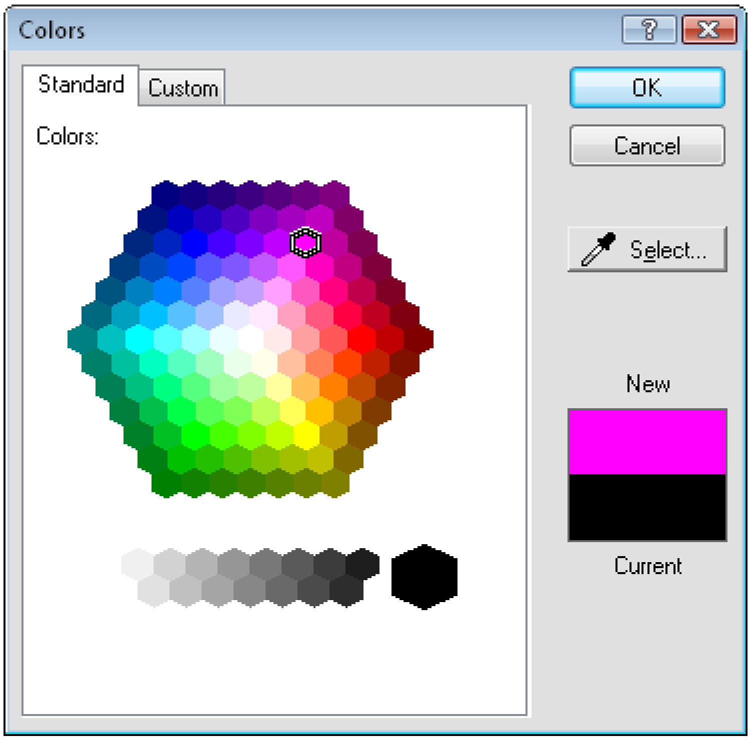
Click a color in the standard palette spectrum.

New

A preview of the selected color appears under *New* on the right side of the dialog. Click the *OK* button to accept the new color. The **Colors** dialog closes.

Select

Click the *Select* button to color match to any color on the screen. The cursor changes to an eyedropper. Move the cursor around the screen and the color under *New* changes accordingly. Click the mouse when you find the color you want, and that color appears under *new*. Click the *OK* button to accept the new color. The **Colors** dialog closes.



*Use the **Colors** dialog to load standard colors or create custom colors.*

Custom Page

You can create custom colors on the **Custom** page. New colors are created by mixing red, green, and blue.

Colors

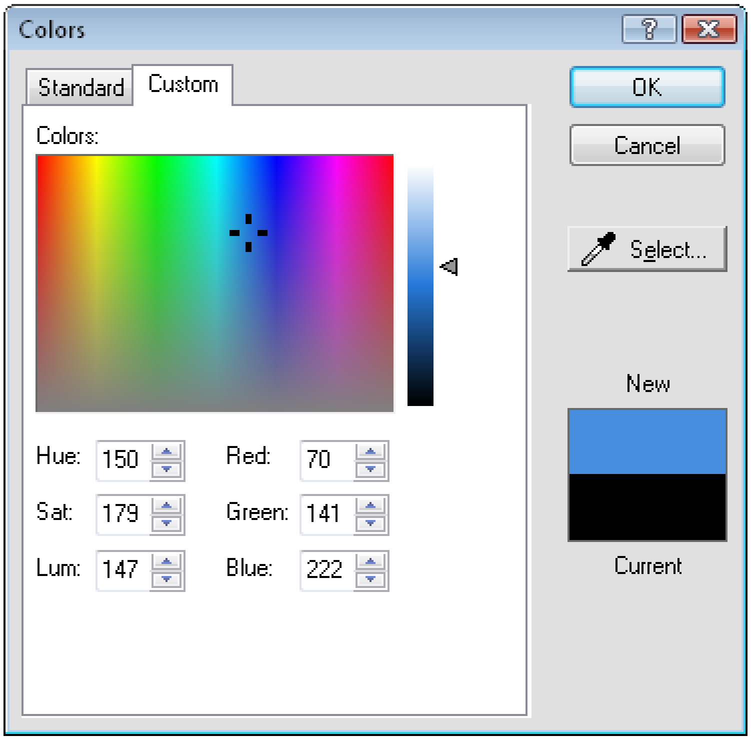
Click anywhere in the color spectrum to choose a new color, or enter new values in the *Hue, Sat, Lum, Red, Green,* and *Blue* boxes. You can drag the slider next to the color spectrum to adjust the new color's intensity.

New

A preview of the new color appears under *New* on the right side of the dialog. Click *OK* to accept the new color. The **Colors** dialog closes.

Select

Click *Select* to color match to any color on the screen. The cursor changes to an eyedropper. Move the cursor around the screen and the color under *New* changes accordingly. Click the mouse when you find the color you want, and that color appears under *New*. Click *OK* to accept the new color. The **Color** dialog closes.



*Use the **Colors** dialog to load standard colors or create custom colors.*

Cancel

While in the **Colors** dialog, click *Cancel* to close the dialog without making any color changes.

Color List

Colors can be referenced by RGB or RGBA values for advanced options. The syntax is `Rxxx Gyyy Bzzz Aaaa` where xxx, yyy, and zzz and aaa specify a red, green, blue color and transparency component respectively. Each color component can range from 0 to 255.

Automation does not require color names because there are explicit enumerations for the colors. Refer to srfColor values or wksColor values for a list of color enumerations. Also, refer to the specific automation topic for specifying colors in automation.

Select a color from the color palette.

Example

`R0 G0 B0`	All components are 0, resulting in black
`R0 G255 B0`	Pure green
`R255 G255 B255`	All components are at full intensity, resulting in white

Color Names

Black=0 0 0 255	Moss Green=51 102 102 255	Walnut=102 51 0 255
90% Black=25 25 25 255	Dark Green=0 51 51 255	Ruby Red=153 0 0 255
80% Black=51 51 51 255	Forest Green=0 102 51 255	Brick Red=204 51 0 255
70% Black=77 77 77 255	Grass Green=0 153 51 255	Tropical Pink=255 102 102 255
60% Black=102 102 102 255	Wild Willow=181 204 97 255	Soft Pink=255 153 153 255
50% Black=128 128 128 255	Kentucky Green=51 153 102 255	Faded Pink=255 204 204 255
40% Black=153 153 153 255	Light Green=51 204 102 255	Dark Red=128 0 0 255
30% Black=179 179 179 255	Spring Green=51 204 51 255	Crimson=153 51 102 255
20% Black=204 204 204 255	Turquoise=102 255 204 255	Regal Red=204 51 102 255
10% Black=230 230 230 255	Sea Green=51 204 153 255	Deep Rose=204 51 153 255
White=255 255 255 255	Faded Green=153 204 153 255	Neon Red=255 0 102 255
Blue=0 0 255 255	Ghost Green=204 255 204 255	Deep Pink=255 102 153 255
Cyan=0 255 255 255	Mint Green=153 255 153 255	Hot Pink=255 51 153 255
Green=0 255 0 255	Army Green=102 153 102 255	Dusty Rose=204 102 153 255
Yellow=255 255 0 255	Avocado Green=102 153 51 255	Plum=102 0 102 255
Red=255 0 0 255	Martian Green=153 204 51 255	Deep Violet=153 0 153 255
Magenta=255 0 255 255	Dull Green=153 204 102 255	Light Violet=255 153 255 255
Purple=153 0 204 255		Violet=204 102 204 255
Orange=255 102 0 255		
Pink=255 153 204 255		
Dark Brown=102 51 51 255		

Powder Blue=204 204 255 255

Pastel Blue=153 153 255 255

Baby Blue=102 153 255 255

Electric Blue=102 102 255 255

Twilight Blue=102 102 204 255

Navy Blue=0 51 153 255

Deep Navy Blue=0 0 102 255

Desert Blue=51 102 153 255

Dodger Blue=19 137 255 255

Sky Blue=0 204 255 255

Ice Blue=153 255 255 255

Smalt Blue=0 104 208 255

Light BlueGreen=153 204 204 255

Ocean Green=102 153 153 255

Chartreuse=153 255 0 255

Moon Green=204 255 102 255

Murky Green=51 51 0 255

Olive Drab=102 102 51 255

Khaki=153 153 102 255

Olive=153 153 51 255

Banana Yellow=204 204 51 255

Light Yellow=255 255 102 255

Chalk=255 255 153 255

Pale Yellow=255 255 204 255

Brown=153 102 51 255

Red Brown=204 102 51 255

Gold=204 153 51 255

Autumn Orange=255 102 51 255

Light Orange=255 153 51 255

Peach=255 153 102 255

Deep Yellow=255 204 0 255

Sand=255 204 153

Dusty Plum=153 102 153 255

Pale Purple=204 153 204 255

Majestic Purple=153 51 204 255

Neon Purple=204 51 255 255

Light Purple=204 102 255 255

Twilight Violet=153 102 204 255

Easter Purple=204 153 255 255

Deep Purple=51 0 102 255

Grape=102 51 153 255

Blue Violet=153 102 255 255

Blue Purple=153 0 255 255

Deep River=102 0 204 255

Deep Azure=102 51 255 255

Storm Blue=51 0 153 255

Deep Blue=51 0 204 255

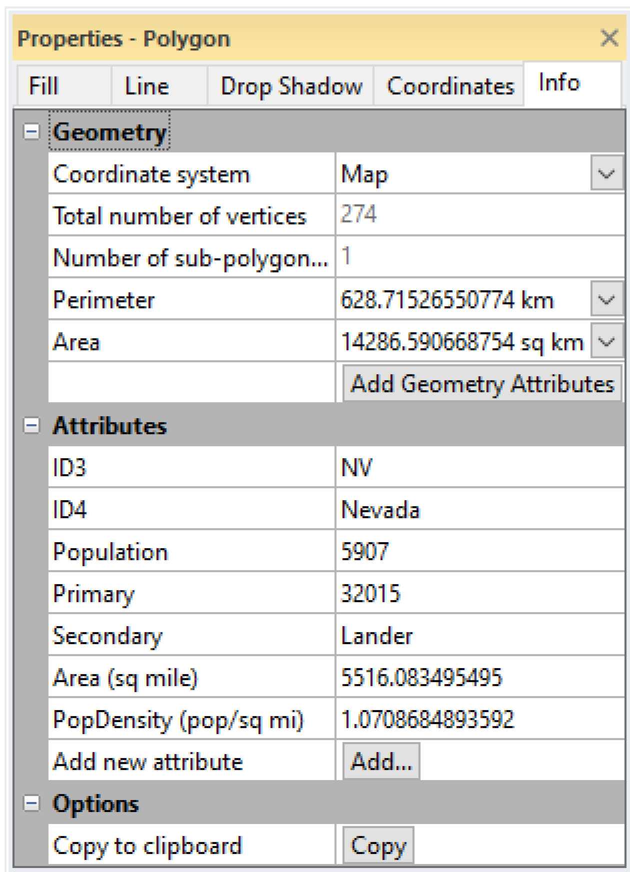
Dark Blue=0 0 128 255

Info Properties - Objects

The **Info** properties page contains information about the selected object. **Info** properties are available with drawn objects, such as polylines, polygons, points, grouped objects, images, and metafiles, with map objects, axes, and map layers, such as contours or base layers. The information displayed is dependent on the type of object, and the image below is just one example of an **Info** properties page.

The **Info** page is located in the Properties window for the selected object. A single object must be selected to display the **Info** page.

Note: Grid layers have their own Info properties page.



*The **Info** page displays information for the selected object.*

Geometry Section

The **Geometry** section includes properties that specify the *Coordinate system* for calculating the geometry values and display the perimeter, length, position, or area of the selected object. Click on the ⊞ next to *Geometry* to open the section. This section is only available for polylines, polygons, symbols, rectangles, rounded rectangles, ellipses, or spline polylines.

Select the units for the units for the *Perimeter*, *Area*, *Length*, or *Position* values by clicking the value and selecting the desired units from the list. The selected units are moved to the top of the list, otherwise the units are displayed alphabetically.

Coordinate System

Specify a coordinate system from the *Coordinate system* list to be used for the information displayed for the object.

- *Page* uses the page coordinate system (inches or centimeters depending on the setting in the **Options** dialog General page).
- *Local* uses the units from the layer in which the geometry was originally specified. For example, if the geometry was loaded from a file, this is the original file units. If the geometry was created interactively, this is an internal coordinate system with 0.0 in the center of the page (in inches).
- *Map* displays information in map units if the geometry is part of a map. If the object is not part of a map, this option is not available.

The default *Coordinate system* property is *Map* for map objects and *Page* for objects outside a map. Changes to the *Coordinate system* selection are applied to all objects. For example, if the property is changed to *Local* for one object, the *Local* information will be displayed for the next selected object. If the selected *Coordinate system* is a geographic coordinate system, i.e. uses spherical coordinates, some geometry information cannot be displayed.

Position

This displays the XY position of a point, for example in a base (vector) map.

Total Number of Vertices

The *Total number of vertices* displays how many vertices are used in the selected object. The Reshape command can be used to see the vertex locations for polygons and polylines.

Number of Sub-Polygons

If the object contains a complex polygon, the *Number of sub-polygons (rings)* displays how many polygons are included in the complex polygon. For example, the sample file ca2010.gsb has polygons of California counties. Some of the counties are complex polygons (i.e. Santa Barbara) that include sub-polygon islands.

Number of Curves

The *Number of curves* is displayed for a spline polyline. The *Number of curves* is the number of inflection points along the spline polyline. This is one less than the total number of vertices.

Perimeter/Length

Depending on the object selected, either *Perimeter* or *Length* will be displayed.

- The *Perimeter* displays the calculated perimeter of the selected closed object (i.e. polygon).
- The *Length* displays the calculated length of the selected open object (i.e. polyline). The units displayed are dependent on what is selected for the *Coordinate system*.

The *Perimeter* or *Length* is not displayed in spherical coordinates (i.e. degrees for *Unprojected Lat/Long* coordinate systems). If your map is in a geographic coordinate system, change the Target Coordinate System to a projected coordinate system to view the *Perimeter* or *Length*. Then specify *Map* in the *Coordinate system* field on the **Info** page.

Area

The *Area* displays the calculated area of the selected object. The units displayed are dependent on what is selected for the *Coordinate system*.

The *Area* is not displayed in spherical coordinates (i.e. degrees squared for *Unprojected Lat/Long* coordinate systems). If your map is in a geographic coordinate system, change the Target Coordinate System to a projected coordinate system to view the *Area*. Then specify *Map* in the *Coordinate system* list on the **Info** page.

Add Geometry Attributes

Click the *Add Geometry Attributes* command to add the geometry values displayed above to the object's attributes. If you wish to add the geometry attributes to all objects in the base layer, use the Attribute Table.

Info Section

The **Info** section allows you to view the *Type, Description, Number of records, size in bytes, Objects in group* and image information for the selected object. Click on the ⊞ next to *Info* to open the section. This section is available for text, images, metafiles, grouped objects, and point cloud layers.

Type

The *Type* displays the type of metafile being displayed.

Description

The *Description* displays any information about the metafile.

Number of Records

The *Number of records* displays the number of objects in the metafile.

Pixel Format

The *Pixel format* option displays the type of image imported, including the number of bits per pixel included in the image.

Size in Bytes

The *Size in bytes* or the *Size (bytes)* option displays the file size of the image or metafile in bytes.

Size in Pixels

The *Size (pixels)* option displays the number of pixels in the image.

Image Source

The *Image source* displays the name of the file imported for an image.

Number of Points

The *Number of points* displays the number of points in the point cloud layer.

Extents

The *xMin*, *xMax* , *yMin* , *yMax*, *Zmin*, and *Zmax* values display the extents of the point cloud layer.

Attributes Section

The *Attributes* section contains any information that is available about the selected object. To open the *Attributes* section, click the ⊞ next to *Attributes*.

Attributes can be image properties, such as *TIFF_Compression*, or can be information about a specific object that was imported from a file, such as a .DEM, .DXF, or .SHP file. Each object (such as polylines in a base map) can have its own attribute information. The *Attributes* can also contain user generated information.

Adding Attributes

To add an attribute to the Attribute Table,

1. Click on the object to select it.
2. In the **Properties** window, click on the **Info** page.
3. Open the *Attributes* section by clicking the ⊞ next to *Attributes*.

4. Click on the *Add* button next to *Add new attribute*.
5. Type the new attribute name, such as *Surveyor's Name*, in the *Attribute name* field of the New Attribute Name dialog.
6. Click *OK* in the **New Attribute Name** dialog. The new attribute is added to the object displayed on the **Info** page.
7. Click in the value column next to the new attribute name. Type the desired description, such as *Thomas Denver*.
8. Now the new attribute and its value has been added to the object.

To rename or remove an attribute, use the **Attribute Table**.

Edit Attributes

Edit attribute values by typing the new value into the field next to the attribute name in the **Properties** window. Edit values for multiple attributes in a base layer with the Attribute Table.

Options Section

The *Options* section contains the option to copy the attribute data for a selected object. To open the *Options* section, click the ⊞ next to *Options*.

Copy Attributes

Click the *Copy* button next to *Copy to clipboard* to copy all of the information on the **Info** page for the selected object to the clipboard. The text can be pasted into **Surfer** or any other program using the Paste command. All information on the **Info** page is copied, including the information in the *Geometry* and *Info* sections. If the *Copy* button is not available, no attribute information is available for the selected item.

Exporting Attributes

Attribute information is exported for polygons, polylines, and points to file formats that support metadata. The file type will determine how many attributes are exported. Refer to the specific file type pages for specific information.

Displaying Attributes

Attributes can be added to base maps as labels. Click on the *Base* map layer object in the **Contents** window to select the base layer. In the **Properties** window, click on the Labels tab to turn label display on. Any attribute can be used as a label for polylines, polygons, or symbols on a base map layer.

Information Displayed for Objects

All objects display an *Attributes* and *Options* section. The information in the *Geometry* section changes depending on the object selected.

Polyline

The **Info** properties for Polylines displays the *Coordinate system, Number of vertices,* and *Length.*

Polygon

The **Info** properties for Polygons displays the *Coordinate system, Total number of vertices, Number of sub-polygons (rings), Perimeter,* and *Area.*

Point

The **Info** properties for Symbols displays the *Coordinate system* and *Position* in X and Y units.

Rectangle

The **Info** properties for Rectangles displays the *Coordinate system*, *Perimeter* and *Area.*

Rounded Rectangle

The **Info** properties for Rounded Rectangles displays the *Coordinate system, Perimeter* and *Area.*

Ellipse

The **Info** properties for Ellipses displays the *Coordinate system*, *Perimeter* and *Area.*

Spline Polyline

The **Info** properties for Spline Polylines displays the *Coordinate system*, *Number of curves* and *Length*.

Range Ring

The **Info** properties for Range Rings displays the *Coordinate system*, *Perimeter*, and *Area*. Note that when *Number of rings* is greater than one, the *Perimeter* and *Area* values are for the largest ring.

Group Objects

The **Info** page for Group Properties displays the number of *Objects in group*. Each group object can be selected in the **Contents** window to display the object info.

Metafile

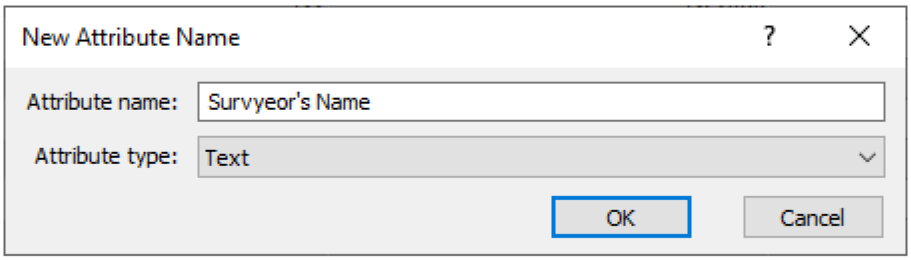
The **Info** properties for Metafiles displays the *Type of metafile*, *Description*, *Number of records*, and *Size in bytes*.

Image

The **Info** properties for Images displays the *Pixel format*, *Size (pixels)*, *Size (bytes)*, and *Image source*.

New Attribute Name Dialog

The **New Attribute Name** dialog is opened after the *Add* button is clicked in the *Attributes* section of **Properties** window Info Page or the button in the Attribute Table.



Enter the new attribute name in the ***New Attribute Name*** *dialog.*

Attribute Name

Type the name for the new attribute in the *Attribute name* field.

Attribute Type

Specify the data type for the attribute in the *Attribute type* field. Select *Text* or *Numeric*.

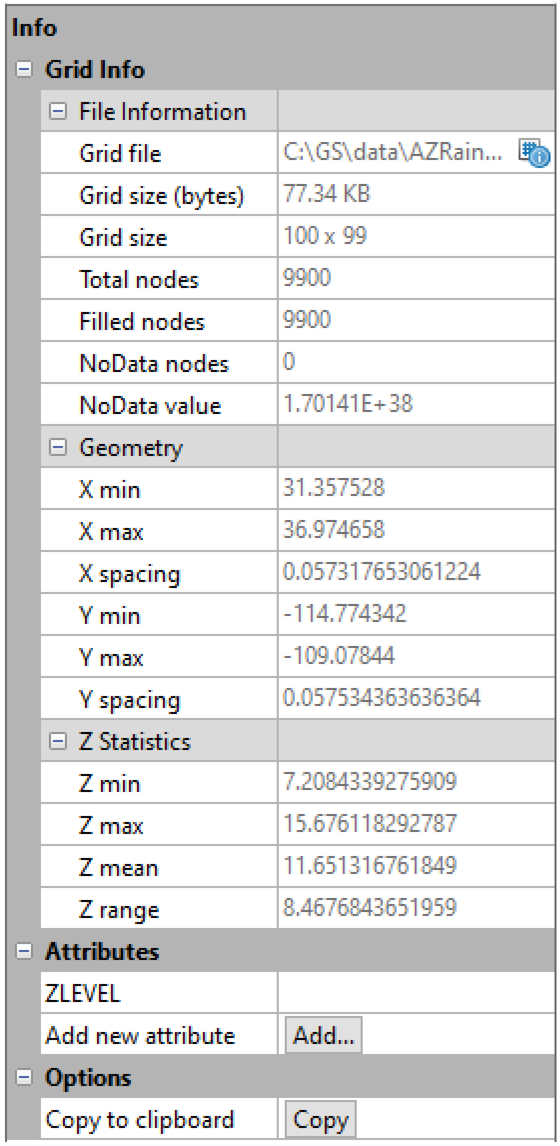
OK or Cancel

Click *OK* and the new attribute is added to the object. Click *Cancel* and the **New Attribute Name** dialog is closed without adding a new attribute.

Info Properties - Grids

For grid-based layers, the **Info** properties page contains information on the grid, such as grid resolution and descriptive statistics. The page also has buttons to add attributes and to copy the grid information.

Note: 2-grid vector layers do not have and **Info** properties page with grid data.



Example Info properties of the grid for the selected layer

In addition to the information-only data in the **Info** page, that page contains two buttons that perform specific actions.

Add

Click the **Add** button to define an attribute in the **New Attribute Name** dialog. Give the attribute a name and select if it is a **Text** or **Numeric** data type. After clicking **OK**, the attribute will be added to the *Attributes* section. You can add more attributes to that section and assign values to the attributes.

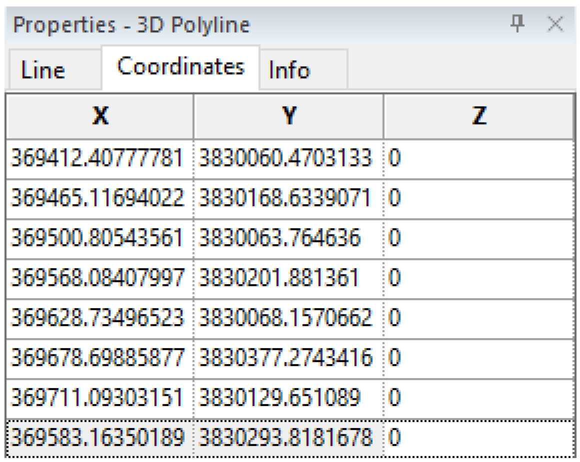
Copy

Select the *Copy* button to copy the grid data to the clipboard. You can then paste that information into another application.

Coordinates Properties

The **Coordinates** page in the **Properties** window displays the X, Y and Z vertex coordinates for the selected point, polyline, polygon, 3D polyline or 3D polygon.

The selected vertex on the **Coordinates** page is indicated by a blue diamond in the plot window. Move the point by typing the desired coordinates in the **Coordinates** page.



X	Y	Z
369412.40777781	3830060.4703133	0
369465.11694022	3830168.6339071	0
369500.80543561	3830063.764636	0
369568.08407997	3830201.881361	0
369628.73496523	3830068.1570662	0
369678.69885877	3830377.2743416	0
369711.09303151	3830129.651089	0
369583.16350189	3830293.8181678	0

*View and edit object coordinates in the **Coordinates** page.*

Z Values

Z values will only be updated on the **Coordinates** page for points, not polylines or polygons.

Points and Z Values

Changes made to points' Z coordinates on the **Coordinates** page will automatically update the corresponding ZLEVEL value in the attribute table. Conversely, changes made to Z values in the ZLEVEL field of the attribute table will be present on the **Coordinates** page. **Surfer** will add a ZLEVEL to the table if one does not exist. Surfer only recognizes Z values that are numeric in the attribute table in the ZLEVEL field.

The **Points to 3D Polyline** dialog specifies the sort criteria and Z coordinates for connecting the points to create a 3D polyline. If a layer has points, polylines and polygons that are being converted to 3D, then the **Points to 3D Polyline** dialog will be followed by the **Features to 3D Polyline** dialog or **Features to 3D Polygon** dialog.

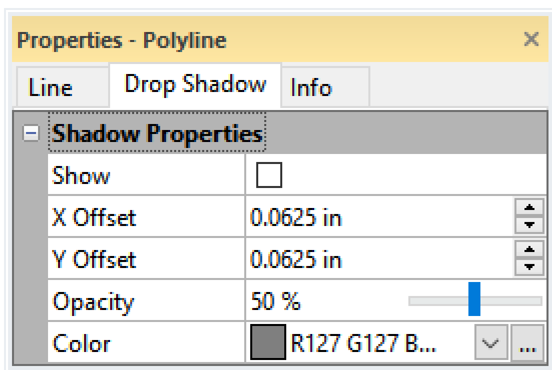
Polylines, Polygons and Z Values

Polylines and polygons have different Z values at each vertex, thus the Z values in the attribute table and on the **Coordinates** page may not be the same. Therefore, changes to Z values in the attribute table will not result in updates to **Coordinates** page and vice versa.

However, when converting a polyline to a 3D polyline, the **Features to 3D Polyline** dialog will specify the attribute for the Z coordinate. Similarly, when converting a polygon to a 3D polygon, the **Features to 3D Polygon** dialog will specify the attribute for the Z coordinate. Convert polylines and polygons separately if they have different attributes.

Drop Shadow Properties

The **Drop Shadow** properties in the Properties window add a drop shadow to drawn objects, including text, polylines, polygons, points, spline polylines, range rings, rectangles, rounded rectangles, ellipses, 3D polylines and 3D polygons.



Add a drop shadow to drawn objects.

Show

Select the *Show* check box to add a drop shadow to the selected feature. Clear the *Show* property to hide the drop shadow.

X Offset

Specify the horizontal offset for the drop shadow in the *X Offset* field.

Y Offset

Specify the vertical offset for the drop shadow in the *X Offset* field.

Opacity

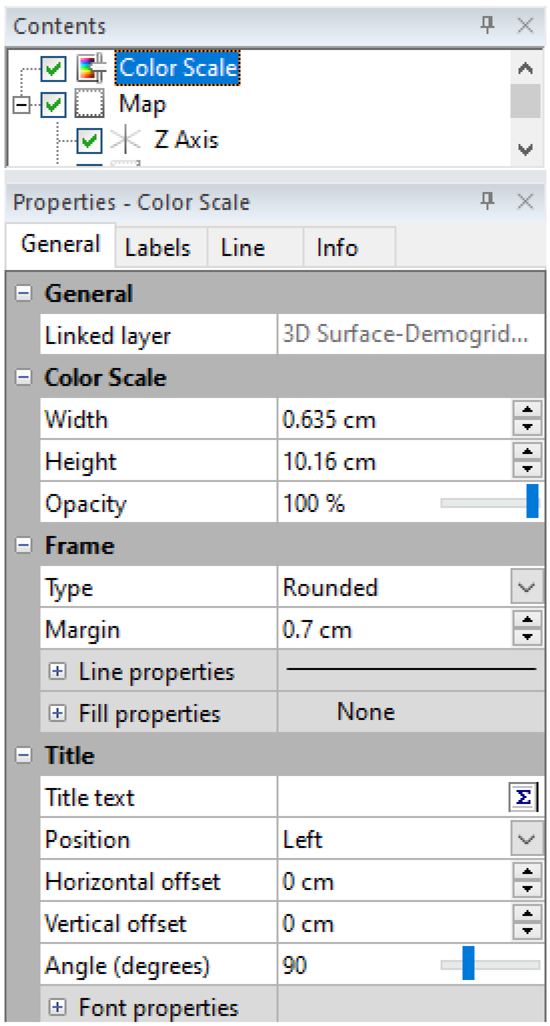
Set the drop shadow opacity by adjusting the *Opacity* slider or typing a value between 0% (completely transparent) and 100% (completely opaque).

Color

Select the color for the drop shadow in the *Color* field. Select a new color by clicking on the color in the color palette. Click the ... button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

Color Scale Bar General Properties

The color scale bar **General** page controls the color scale size, opacity, frame, and title properties of the color scale bar in the 2D view.



*Change color scale properties in the **Properties** window when a Color Scale is selected.*

General Section

The *Linked layer* property is an information-only property that shows the name of the layer that is associated with the selected color scale.

Color Scale Section

Expand the *Color Scale* section to change the width, height, and opacity properties.

Width

The *Width* property sets the width of the color scale bar in page units. Set the width to a value between 0 and 4 inches (0 and 10.16 centimeters).

Height

The *Height* property sets the height of the color scale bar in page units. Set the height to value between 0 and 100 inches (0 and 254 centimeters).

Opacity

Change the *Opacity* of the color scale by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the slider to change the opacity percentage. This is the opacity for the filled color portion of the color scale.

Frame Section

The *Frame* section specifies the properties to use for the border around the color scale bar. See the Frame Properties help topic for information on these common properties.

Title Section

The *Title* section includes the title text, position, offset, angle, and font properties.

Title Text

Add a title to the color scale by entering text into the *Title text* field. Click in the *Title text* field, and type a title. Press ENTER or click elsewhere, and the title will be added or updated. You can format the title in the *Title text* field using math text instructions or by using the Text Editor. Click the button to open the **Text Editor**.

Title Position

Change the title position relative to the color scale with the *Position* selection. Click the current selection in the *Position* field, and select *Left*, *Right*, *Top*, or *Bottom* from the list.

Title Offset

Adjust the title offset from the scale bar by entering a value in the *Horizontal offset* or *Vertical offset* field. To change the offset, type a number from -10 to 10 inches (-25.4 to 25.4 centimeters) in the *Horizontal offset* or *Vertical offset* field, or click the buttons. For the *Horizontal offset*, positive values move the title to the right, and negative values move the title to the left. For the *Vertical offset*, positive values move the title up, and negative values move the title down.

Title Angle

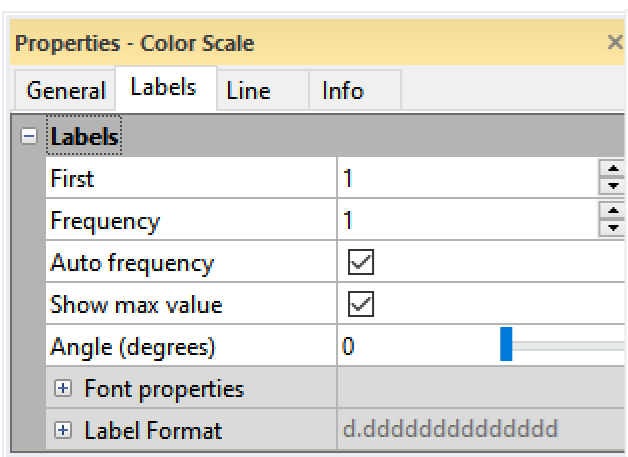
Change the title angle by entering a value from 0 to 360 in the *Angle (degrees)* field or dragging the slider. By default, *Left* and *Right* titles are rotated 90 degrees.

Title Font Properties

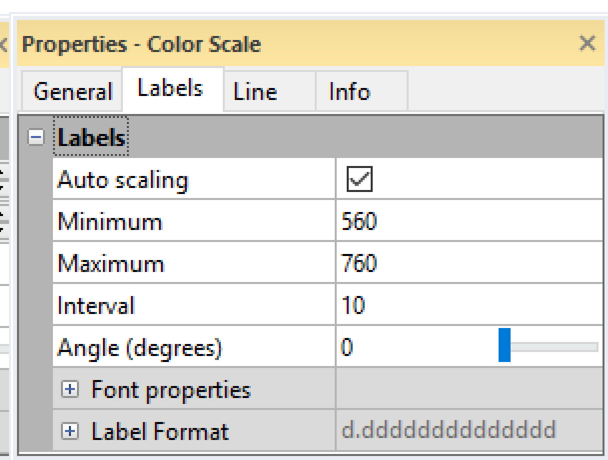
Click the button next to *Font properties* to expand and edit the title [Font Properties](#).

Color Scale Bar Labels Properties

The color scale bar **Labels** page controls the label properties for the color scale bar labels in the 2D view. The label properties are different for discrete (contour and wireframe) and continuous (color relief, surface, vector, and point cloud) color scale bars. Discrete color scale bars are labeled by specifying which levels include a label. Continuous color scale bars are labeled at a specific interval within the specified range.



Discrete color scale bar labels are displayed by frequency.



Continuous color scale labels are displayed by interval

Discrete Color Scale Bar Labels

The *Labels* section for contour and wireframe color scale bars includes the label frequency options, angle, font properties, and format properties.

First Label

Set the first label value in the *First* box. A one here means to set the first line in the color scale bar with a label. To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the values.

Label Frequency

Define the frequency of labels in the *Frequency* box. One draws every label, two draws every other label, etc. To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the values.

Automatic Frequency

Check the box next to *Auto frequency* to allow **Surfer** to automatically determine a reasonable frequency for labels on the contour scale. The *Auto frequency* sets the label frequency so labels do not overlap. If there are too many labels to fit into the space next to the color scale bar, some labels are dropped to make the labels legible.

Show Maximum Value

Select the *Show max value* option to display a label at the top of the color scale bar. Which value is displayed depends on the level method:

- Simple - displays the value that is the lowest multiple of the contour interval plus the last contour level that is greater than the data maximum.
- Logarithmic - displays a value that is the lowest multiple of the logarithmic contour interval plus the last contour level that is greater than the data maximum.
- Equal-area - displays the maximum data value.
- Advanced - displays the maximum data value when the data maximum is greater than the last contour level or displays the last contour level plus the last contour interval when the data maximum is less than the last contour level.

Label Angle

Choose the angle at which the labels are drawn in the *Angle (degrees)* box. Positive values rotate the labels counterclockwise. To change the *Angle (degrees)*, highlight the existing value and type in a new value. Or, drag the to change the rotation. The labels automatically rotate to the new position.

Label Font

Click the ⊞ next to Font Properties to set the label text properties.

Label Format

Click the ⊞ next to Label Format to choose the numeric label format.

When the *Type* is set to *Date/time*, if *Invalid date* text is displayed instead of the actual date/time format, the value for the label is outside the defined date/time range.

Continuous Color Scale Bar Labels

The *Labels* section for color relief, surface, vector, and point cloud color scale bars includes the label range and interval options, angle, font properties, and format properties.

Automatic Scaling

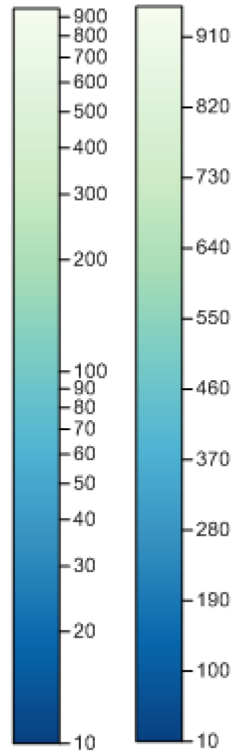
The *Auto Scaling* check box sets the label frequency so labels do not overlap. If there are too many labels to fit into the space next to the color scale bar, some labels are dropped to make the labels legible. When the *Auto Scaling* is not checked, set the *Minimum, Maximum,* and *Interval* values.

Minimum Label

Set the first label value in the *Minimum* box. The *Minimum* value is always labeled. This is a Z value in Z coordinates. To change the *Maximum* label, highlight the existing value and type a new value. When the color scale is displaying a logarithmic scale, the *Maximum* value must be greater than 0.

Maximum Label

Set the last label value in the *Maximum* box. This is the last possible value that will be labeled. This may not be actually labeled, depending on the starting *Minimum* value and the *Interval* specified. This is a Z value in Z coordinates. To change the *Minimum* label, highlight the existing value and type a new value. When the color scale is displaying a logarithmic scale, the *Minimum* value must be greater than 0.



The color scale on the left uses logarithmic scaling. The color scale on the right uses linear scaling. Any colormap can be used for linear or logarithmic scaled intervals.

Label Interval

The *Interval* option is the actual Z interval between ticks and labels on the color scale bar. This is in Z coordinates. To change the *Interval*, highlight the existing value and type a new value. This option is only available for linear scaled color scale bars.

Labels per Decade

The *Labels per decade* option is the number of labels between major decade label marks. Available options are *1; 1,3; 1,5; 1,2,5; 1,2,3,5; 1,2,4,6,8;* and *All*. *All* displays labels at 1, 2, 3, 4, 5, 6, 7, 8, and 9. To change the *Labels per decade*, click on the existing option and select the desired option from the list. This option is only available for logarithmic scaled color scale bars.

Label Angle

Choose the angle at which the labels are drawn in the *Angle (degrees)* box. Positive values rotate the labels counterclockwise. To change the *Angle (degrees)*, highlight the existing value and type in a new value. Or, drag the  to change the rotation. The labels automatically rotate to the new position.

Label Font

Click the ⊞ next to Font Properties to set the label text properties.

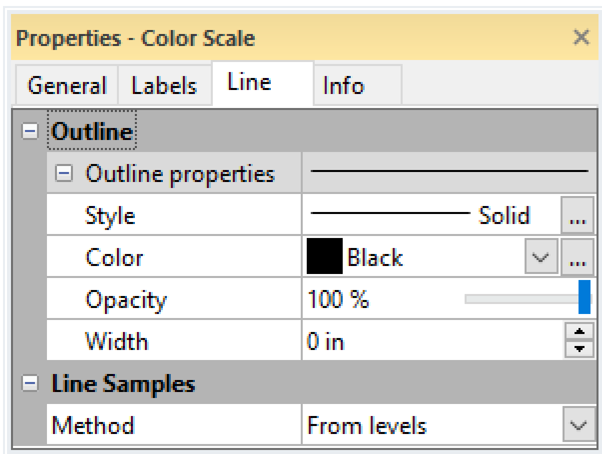
Label Format

Click the ⊞ next to Label Format to choose the numeric label format.

When the *Type* is set to *Date/time*, if *Invalid date* text is displayed instead of the actual date/time format, the value for the label is outside the defined date/time range.

Color Scale Bar Line Properties

The color scale bar **Line** page controls the line properties for the color scale bar outline in the 2D view. The line properties are also displayed for the levels in discrete (contour and wireframe) color scale bars.



Set the line properties for the color scale bar outline and levels.

Outline Properties

The *Outline* section includes the line properties for the color scale bar border. This line property is used for the outside of the color scale bar box. You can set *Style, Color, Opacity*, and *Width* of the line. The *Outline properties* can also be applied to the line samples by setting the *Method* to *Same as outline*.

Line Samples

The *Line Samples* section includes the method and line properties for the level line samples in discrete (contour and wireframe) color scale bars. The *Line Samples* section is only displayed for contour layers when the contour layer is color filled.

Method

The *Method* property sets the source for the line sample properties: *Same as outline*, *Uniform*, or *From levels*.

- *Same as outline* sets the line sample properties source to the *Outline properties*. Set the line properties for the outline and the line samples in the *Outline properties* section. Wireframe color scale bars use the *Same as outline* method by default.
- *Uniform* makes the line samples uniform regardless of the individual level line properties in the map. Set the level line sample line properties in the *Line properties* section displayed below the *Method* property.
- *From levels* sets the line sample properties source to the line properties for the levels in the map. Contour map color scale bars use the *From levels* method by default.

When a contour map is not color filled, the color scale bar uses the *From levels* method and the *Line Samples* section is not displayed.

Line Properties

When the *Method* is set to *Uniform*, a *Line properties* section is displayed. These properties specify the line properties for the color scale bar line samples.

Chapter 30 - Selecting and Arranging Objects

Selecting Objects

There are several ways to select objects in **Surfer**. An object is selected if there is a bounding box with selection handles surrounding the object. The name of the selected object appears in the status bar. You can set some selection options in File | Options.

Editing base layers and groups is done by pinning them. Objects within groups can be individually or selected in groups once the pin at the higher level in the hierarchy has been established.

Selecting Objects from the Contents Window

- Objects may be selected using the Contents window. To display the **Contents** window, use the **View | Show/Hide | Contents** command. Once the list is displayed, click on the layer or group you wish to select. To the right of the layer or group a button with a left arrow with a broken pin will appear and the group or layer is selected in the plot window. Click on the pin to toggle it to the pinned position . Objects contained within the hierarchy that is pinned may be edited without affecting objects in other groups or layers. Objects outside the hierarchy that is pinned cannot be edited. The object's properties are displayed in the Properties window.

- Hold down the CTRL key while clicking on objects to select multiple objects in the **Contents** window, or hold down the SHIFT key to select adjacent objects.

Selecting Objects from the Plot Window

Several procedures are available to select objects in a plot window:

- After the list of objects is displayed and the layer or group has been pinned in the contents window, objects within that hierarchy can be selected in the plot window.

- To select a single object in the plot window, move the cursor over the object using the mouse or the arrow keys. When the cursor is over the desired object, click the left mouse button or press the SPACEBAR on the keyboard. Eight rectangular selection handles appear, indicating that the object is selected.

- If an object other than the one you want is selected, hold down the CTRL key and continue clicking with the mouse or SPACEBAR until the desired object is selected. Any objects that were previously selected become deselected. (The bounding boxes for the objects are overlapping.)

- To select two or more objects in the plot window, hold down the SHIFT key while making your selections. This retains previously selected objects and includes the newly selected objects. You can hold down both the CTRL and SHIFT keys to select several overlapping objects in the plot window.

- The block select allows you to select one or more objects contained in a user-defined rectangle. Press and hold the left mouse button on an empty portion of the plot window. Drag the mouse to form a rectangle around the group of objects you wish to select. Alternatively, you can use the arrow keys to position the cursor on an empty portion of the plot window, press and hold the SPACEBAR, and move the cursor with the ARROW keys. If the *Rectangle must fully surround* check box is activated in the **Options** dialog Selection page, then only objects fully surrounded by the selection rectangle are selected. If the check box is not activated, then all objects with any portion of their bounding boxes within the block select rectangle are selected.

- The Home | Selection | Select All command is used to select all the objects in the plot window. Pressing the CTRL+A keys performs the same command.

- The Home | Selection | Select All | Invert command selects all unselected objects and deselects all selected objects. This command is useful for selecting a large number of objects and leaving a few isolated objects unselected. Select the objects you do not want to select and use the **Invert Selection** command.

- The TAB key can be used to cycle through all objects, selecting each one at a time.

When you select a nested object, the main object is also selected. For example, if you select the contours map layer, the map object is also selected. Only one nested object can be selected at a time. For example, it is not currently possible to select two axes at once.

Select Tool

Click the **Home | Selection | Select** command or the  button to click with the mouse on objects to select them. When the cursor is in any other tool mode, for example draw or reshape, you can also return to select mode by pressing ESC.

Block Select

Block select items by dragging a rectangle around them. To avoid accidentally moving an object when selecting objects, use the **Home | Selection | Select All | Block Select** command or the button. If an object is properly selected a bounding box will surround the object. The block select options are set in Selection page of the **Options** dialog.

You can use the block select feature by:

- Clicking the **Home | Selection | Select All | Block Select** command and dragging a rectangle around the objects to select.
- Use the mouse to drag a block selection.

To select an object with block select:

1. Click the **Home | Selection | Select All | Block Select** command.
2. Left-click outside one corner of the object.
3. Hold the left mouse button down.
4. Drag the mouse to the opposite corner of the object and release the mouse button.

There are many ways to select objects in **Surfer**. Two of the most common include the following:

- Select one object in the plot window and hold the SHIFT key, and select additional objects to add to the block selection.
- Select one object in the Contents window, hold the CTRL key, and select additional objects to add to the block selection. Hold SHIFT while selecting objects in the **Contents** window to select a contiguous group of objects. Select groups or layers and pin them in order to edit the objects in their hierarchy.

Select All

Use the **Home | Selection | Select All** command or the button , or press CTRL+A on the keyboard to select all objects in the document. A bounding box surrounds all selected objects.

Deselect All

Use the **Home | Selection | Select All | Deselect All** command or the button, or press CTRL+SHIFT+A on the keyboard, to deselect all selected objects. This command is useful when zoomed in on objects. Alternatively, you can deselect objects by clicking in the white space.

Invert Selection

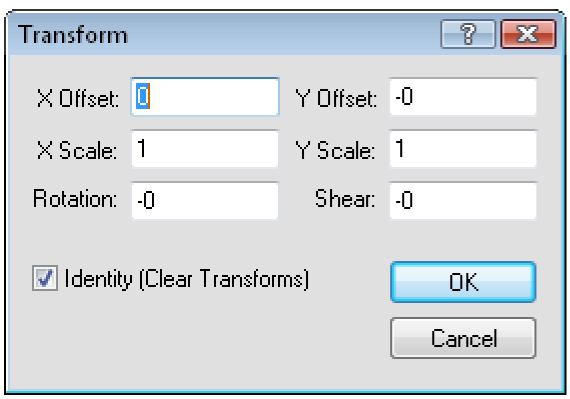
The **Home | Selection | Select All | Invert** command or the button reverses the selected and deselected objects. All previously selected objects become deselected. All previously deselected objects become selected. A bounding box surrounds all selected objects.

Transform

When you select and drag an object, or resize it via the sizing handles, you are transforming the object. The Align Objects, Rotate, and Free Rotate commands also transform the selected object. The accumulated effects of all transformations applied to an object are recorded as a set of page transformation parameters.

Transform Dialog

The **Home | Selection | Transform** command or the button open the **Transform** dialog. This dialog allows you to read, set, and reset the six page transformation parameters associated with the selected object. These six parameters allow for four distinct transformations: scale, shear, rotation, and offset. The coordinate origin for these transformations is the center of the page. Thus, in the untransformed state, the center of an object coincides with the center of the printed page.



*Use the **Transform** dialog to change the transform parameters for the selected object.*

X Offset

The *X Offset* specifies the horizontal translation (in page units). A positive *X Offset* shifts the object to the right, a negative *X Offset* shifts the object to the left.

Y Offset

The Y Offset specifies the vertical translation (in page units). A positive Y Offset shifts the object up, a negative Y Offset shifts the object down.

X Scale

The *X Scale* parameter modifies the width of the object. A value of 2.0 doubles the width, a value of 0.5 cuts the width in half, while a value of 1.0 leaves the width unchanged.

Y Scale

The *Y Scale* parameter modifies the height of the object. A value of 2.0 doubles the height, a value of 0.5 cuts the height in half, while a value of 1.0 leaves the height unchanged.

Rotation

The Rotation specifies a counterclockwise rotation about the origin. The rotation angle is given in degrees.

Shear

The *Shear* changes the X coordinate as the Y coordinate changes. For example, a shear of 1.0 shifts the X coordinate 1 inch to the right for every inch along the Y axis.

Identity (Clear Transform)

When you check the Identity (Clear Transform) box, the transformation is reset back to the identity; that is, the selected object is untransformed.

The order in which the transformations are applied makes a difference (these transformations are not commutative). The four transformations are applied in the following order: scale, shear, rotation, and translation.

References

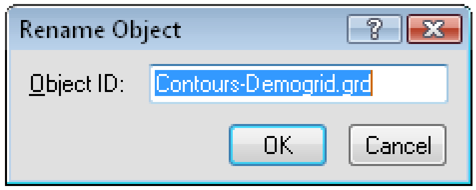
Foley, James D. and Andries van Dam (1990),Computer Graphics: Principles and Practice, Addison-Wesley, Reading, MA, 1174 pp.

Hearn, D. and M.P. Baker (1997),Computer Graphics C, 2nd edition, Prentice Hall, Upper Saddle River, NJ, 652 pp., ISBN 0-13-530924-7.

Rename Object

The **Home | Selection | Rename** command lets you assign a name to any type of object, including maps and map axes. After selecting a single object, click the **Home | Selection | Rename** command or the  button to display the

Rename Object dialog. Enter a new name in the *Object ID* box, click *OK*, and the name appears in the status bar when that object is selected. The name also appears in the **Contents** window list and in the **Properties** window.



Update the Object ID name in the Rename Object dialog.

Object IDs can also be edited through the Contents window. To edit the text ID associated with an object in theContentswindow,

- Click the object and press F2. The **Rename Object** dialog opens.
- Right-click on the object and select **Rename Object**. The **Rename Object** dialog opens.
- Click the object and then click again on the selected item (two slow clicks). You must allow enough time between the two clicks so it is not interpreted as a double-click. Enter the new name into the box that appears.

Layout Tab Commands

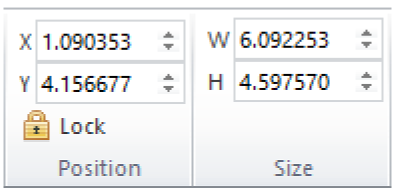
The **Layout** tab in the ribbon has the following commands:

Margins	Select a predefined margin size, or set custom margins in the Page Setup dialog.
Orientation	Set the page to portrait or landscape orientation
Size	Set the page size to Letter, Legal, Tabloid/Ledger, or A4, or select a different predefined or custom page size in the Page Setup dialog.
Header	Set header text and alignment.
Footer	Set footer text and alignment.
Page Outline	Turn the page outline display on or off.
Margins	Turn the page margins display on or off.
Units	Change the page units setting.
Size and Position	Specify an object's size or position in page units, or lock an object's position.
Bring to Front	Moves the selected object or objects to the front of the plot document.
Bring Forward	Moves the selected object one position forward in the plot document.

Send to Back	Moves the selected object or objects to the back of the plot document.
Send Backward	Moves the selected object one position backward in the plot document.
Size Both	Size one or more objects to a reference object's **Width**, **Height**, or **Both**.
Align	Aligns objects to the left, center, right, top, middle, or bottom. Distributes objects horizontally or vertically with equal spacing (or overlap). Align or distribute to the bounding box or page margins.
Free Rotate	Rotates an object with the mouse
Rotate	Rotates an object by specific degrees

Positioning and Sizing Objects

The **Size** and **Position** groups on the **Layout** tab contain options to position and size objects. Objects can also be positioned and resized using the mouse and keyboard. Objects can also be repositioned by dragging them to a new location with the mouse or with the keyboard.



*Change the size or position of any object in the **Size** or **Position** group on the **Layout** tab.*

Horizontal and Vertical Position

Use the horizontal position (*X*) and the vertical position (*Y*) to set the X, Y position on the page for most objects. To change the location, highlight the existing value and type the desired value. Or, click the buttons to increase or decrease the position. Press ENTER on the keyboard to make the change.

The X, Y location of the cursor is displayed in the status bar. This can be a good source of reference.

Width and Height

Use the *W* and *H* controls to set the width and height of the selected object. To change the size, highlight the existing value and type the desired value. Or, click the buttons to increase or decrease the size. Press ENTER on the keyboard to make the change.

Stretching a map with the **Layout | Size | W** and **Layout | Size | H** fields does NOT maintain the internal scale of the map. Use the Scale page in the **Properties** window to change the map size. Fix the internal scale of a stretched map by selecting the map and using the *Identity (Clear transforms)* option in the Transform command.

Resize Objects

You can resize objects graphically with the mouse or keyboard. Selected objects appear with selection handles at the corners and sides of the bounding box for the object. The size of a selected object is displayed in the status bar. The cursor changes to a two-headed arrow when it is moved over one of the selection handles. Resize a single selected object or several selected objects using the following methods. You can resize a single selected object or several selected objects using the selection handles. Side selection handles will move with the zoom level. The selection handles will always be visible along the side of the bounding box regardless of the zoom location or view.

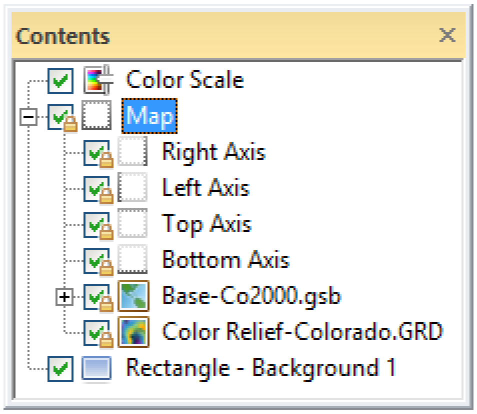
- To drag a handle with the mouse, move the cursor over the handle and then press and hold the left mouse button. Move the cursor to a new position. Release the left mouse button and the object is resized.
- To drag a handle with the keyboard, move the cursor over the handle, press and hold the SPACEBAR, and use the ARROW keys to move the cursor to a new position. Release the SPACEBAR and the object is resized.
- Drag one of the four corner handles to size the object proportionally.
- Drag one of the side handles to stretch or compress the object in one dimension only.
- Press and hold the ALT key while dragging a corner handle with the mouse to free size an object.
- To resize a map, use the map properties Scale page to size the map. STRETCHING THE MAP DOES NOT PRESERVE THE INTERNAL MAP SCALE. Use Transform to undo the effects of stretching a map.
- Use the *W* and *H* fields in the Position and Size groups on the **Layout** tab to change the width and height of an object.
- Use the Layout | Arrange | Size Both command to size one or more objects to a reference object's width and/or height.

To return an object to the original size, select the object. Click Home | Selection | Transform and check the *Identity (Clear Transforms)* check box . The object is returned to the original size and location.

Lock Position

To lock an object's position, click on the object to select it. Click the **Layout | Position | Lock** command or right-click on the object and select **Lock Position**. When an object is locked it cannot be moved either through the Position group or by dragging it with the mouse. Objects that are locked can be resized, edited, and rotated.

Repeat the above instructions to unlock a locked object. Locked objects and layers are indicated in the Contents window with a small lock icon.



The lock icon indicates the Map object is locked in the example above.

Locked Map Layers

When a map layer is locked, only the objects in that layer cannot be moved. If the map or any portion of the map is selected and moved, the locked layer will also be moved. To prevent map layers from moving on the page, lock the *Map* object.

Locked Objects in a Base Layer

When objects in a base layer are individually locked, that object cannot be moved individually. To move the object, unlock the object and move the object to the desired position. The entire base layer and map object can be moved when a sub-object is locked. The sub-object stays in the same relative location as the objects in the base layer or map.

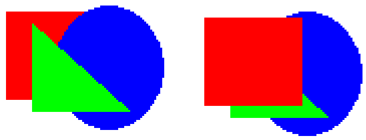
Bring to Front

The **Layout | Arrange | Bring to Front** command or the  button moves the selected object or objects to the front of the plot document. The object will appear on top of the other objects. Alternatively, right-click the object and select **Order Objects | Move to Front**.

Objects can also be moved by dragging them to a new position in the Contents window.

The default keyboard shortcut for the **Layout | Arrange | Bring to Front** command is SHIFT + PAGE UP. The keyboard shortcuts can be customized with the Customize command.

Example



In the left drawing, the red square is located behind all of the other objects. Use **Bring to Front** to move the square to the front layer so that it appears in front of all the other objects (right drawing). The Bring Forward command moves the square forward one layer at a time.

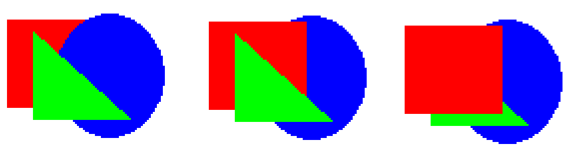
Bring Forward

The **Layout | Arrange | Bring to Front | Bring Forward** command or the moves the selected objects forward one position. Alternatively, right-click the object and select **Order Objects | Bring Forward**.

Objects can also be moved by dragging them to a new position in the Contents window.

The default keyboard shortcut for the **Layout | Arrange | Bring to Front | Bring Forward** command is CTRL + PAGE UP. The keyboard shortcuts can be customized with the Customize command.

Example



In the left drawing, the red square is located behind all the other objects. Use **Bring Forward** to move the square forward one layer so that it appears between the circle and the triangle (middle drawing). Clicking **Bring Forward** again places the square on top of the other objects (right drawing). Bring to Front places the square on top of the other objects.

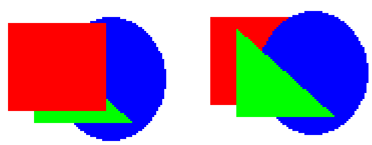
Send to Back

The **Layout | Arrange | Send to Back** command or the  button move the selected object or objects to the back of the plot document. The object will appear behind the other objects. Alternatively, right-click the object and select **Order Objects | Move to Back**.

Objects can also be moved by dragging them to a new position in the Contents window.

The default keyboard shortcut for the **Layout | Arrange | Send to Back** command is SHIFT + PAGE DOWN. The keyboard shortcuts can be customized with the Customize command.

Example



In the left drawing, the red square is located in front of all the other objects. Use **Send to Back** to move the square to the back layer so that it appears behind all of the other objects (right drawing). Send Backward moves the square backward one layer at a time.

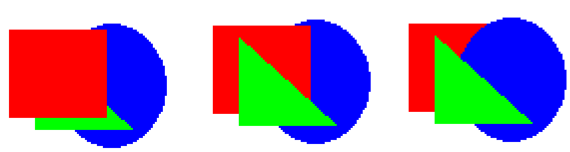
Send Backward

The **Layout | Arrange | Send to Back | Send Backward** command or the  button move the selected object backward one position. Alternatively, right-click the object and select **Order Objects | Move Backward**.

Objects can also be moved by dragging them to a new position in the Contents window.

The default keyboard shortcut for the **Layout | Arrange | Send to Back | Send Backward** command is CTRL + PAGE DOWN. The keyboard shortcuts can be customized with the Customize command.

Example



In the left drawing, the red square is located in front of all the other objects. Use **Send Backward** to move the square back one layer so that it appears between the circle and the triangle (middle drawing). Clicking **Send Backward** again places the square on behind all of the other objects (right drawing). Send to Back places the square on behind the other objects.

Size Objects

Click one of the **Layout | Arrange | Size Both** commands or the button to size one or more objects to a reference object's width, height, or both. At least two objects must be selected to use the **Layout | Arrange | Size Both** commands. The reference object's size does not change. All other objects are sized equally to the reference object. The reference object is the highest selected object in the Contents window, i.e. the front object in the plot window.

Width

Click **Layout | Arrange | Size Both | Width** or the button to set the width off all selected objects equal to the width of the reference object.

Height

Click **Layout | Arrange | Size Both | Height** or the button to set the height of all selected objects equal to the height of the reference object

Both

Click **Layout | Arrange | Size Both** to set both the width and height of all selected objects equal to the width and height of the reference object.

Changing the Reference Object

The quickest way to make sure you have the correct reference object is to select the object you wish to use as the reference object and click the Layout | Arrange | Bring to Front | Bring to Front command. The object is moved to the top of the **Contents** window and front of the plot window. It will be the reference object when the **Size** commands are used with any other objects. However, you may not wish to have your reference object at the front of the plot window. When this is the case, click and drag the desired reference object in the **Contents** window above the objects you wish to re-size. You can also use the Layout | Arrange | Bring to Front | Bring Forward command to move an object forward one place at a time.

Align Objects

The **Layout |Arrange | Align**commands are used to align selected objects relative to the bounding box surrounding the selected objects. The objects can be aligned both vertically and horizontally. Objects can also be distributed horizontally or distributed vertically. Enable the Align to Margins option to align objects to the page margins rather than the bounding box.

Two or more objects must be selected to enable the **Align** commands when **Align to Margins** is not active. One or more objects must be selected to enable the **Align** commands when **Align to Margins** is active.

Align Left

Left aligns all selected objects along the left side of the bounding box when **Align to Margins** is not checked. **Left** aligns the selected object or objects to the left margin when **Align to Margins** is checked.

Align Center

Center centers all selected objects between the left and right sides of the bounding box when **Align to Margins** is not checked. **Center** aligns the selected object or objects to the center of the page (between the left and right margins) when **Align to Margins** is checked.

Align Right

Right aligns all selected objects along the right side of the bounding box when **Align to Margins** is not checked. **Right** aligns the selected object or objects to the right margin when **Align to Margins** is checked.

Align Top

Top aligns all selected objects along the top of the bounding box when **Align to Margins** is not checked. **Top** aligns the selected object or objects to the top margin when **Align to Margins** is checked.

Align Middle

Middle centers all selected objects between the top and bottom sides of the bounding box when **Align to Margins** is not checked. **Middle** aligns the selected object or objects to the middle of the page (between the top and bottom margins) when **Align to Margins** is checked.

Align Bottom

Bottom aligns all selected objects at the bottom of the bounding box when **Align to Margins** is not checked. **Bottom** aligns the selected object or objects to the bottom margin when **Align to Margins** is checked.

Distribute Horizontally

Click the **Layout |Arrange | Align | Distribute Horizontally** command or the button to distribute three or more objects from left to right with equal spacing in their bounding box. The interior object or objects are aligned horizontally between the outermost objects. This means the bounding box size does not change, and the left and right objects do not move, while the objects are arranged. Three or more objects must be selected to enable the **Layout | Arrange | Align | Distribute Horizontally** command when Align to Margins is not active. If the objects must overlap to fit inside the bounding box, the objects are distributed with equal overlap. You can also right-click the selection and click **Align Objects | Distribute Horizontally** in the context menu to distribute the objects.

Distribute with Align to Margins

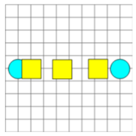
Click the **Distribute Horizontally** command when Align to Margins is active to distribute two or more objects from left to right with equal spacing between the page margins. All objects are aligned horizontally between the page margins. This means the bounding box size may change, and the left and right objects may move, while the objects are arranged. Two or more objects must be selected to enable the **Layout | Arrange | Align | Distribute Horizontally** command when Align to Margins is active. If the objects must overlap to fit inside the page margins, the objects are distributed with equal overlap.

Example

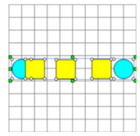
The following steps demonstrate how to distribute objects horizontally:

1. Move the outside objects to the desired locations.
2. Select the objects that are to be distributed, including the furthest left and furthest right objects.
3. Click the **Layout | Arrange | Align | Distribute Horizontally** command.

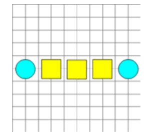
The interior objects are arranged between the outer most objects with equal spacing.



*Five objects are unevenly spaced. The circles are moved to the desired locations. The **Layout |Arrange | Align | Middle** command was used to align the objects in a row.*



*All five objects are selected before the **DistributeHorizontally** command is used.*



*The **Layout | Arrange | Align | Distribute Horizontally** command is clicked and the objects are now equally spaced from left to right.*

Distribute Vertically

Click the **Layout | Arrange | Align | Distribute Vertically** command or the  button to distribute three or more objects from top to bottom with equal spacing. The interior object or objects are aligned vertically between the outermost objects. This means the bounding box size does not change, and the top and bottom objects do not move, while the objects are arranged. Three or more objects must be selected to enable the **Layout | Arrange | Align | Distribute Vertically** command. If the objects must overlap to fit inside the bounding box, the objects are distributed with equal overlap. You can also right-click the selection and click **Align Objects | Distribute Vertically** in the context menu to distribute the objects.

Distribute with Align to Margins

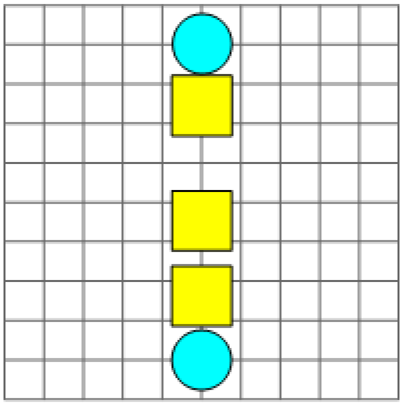
Click the **Distribute Vertically** command when Align to Margins is active to distribute two or more objects from top to bottom with equal spacing between the page margins. All objects are aligned vertically between the page margins. This means the bounding box size may change, and the top and bottom objects may move, while the objects are arranged. Two or more objects must be selected to enable the **Layout | Arrange | Align | Distribute Vertically** command when Align to Margins is active. If the objects must overlap to fit inside the page margins, the objects are distributed with equal overlap.

Example

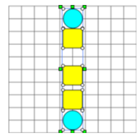
The following steps demonstrate how to distribute objects vertically:

1. Move the outside objects to the desired locations.
2. Select the objects that are to be distributed, including the top and bottom objects.
3. Click the **Layout | Arrange | Align | Distribute Vertically** command.

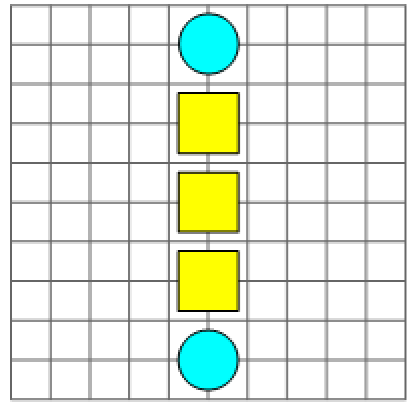
The interior objects are arranged between the outer most objects with equal spacing.



*Five objects are unevenly spaced. The circles are moved to the desired locations. The **Layout | Arrange | Align | Center** command was used to align the objects in a column.*



*All five objects are selected before the **DistributeVertically** command is used.*



*The **Layout | Arrange | Align | Distribute Vertically** command is clicked and the objects are now equally spaced from top to bottom.*

Align to Margins

The **Align to Margins** option determines whether objects are aligned and/or distributed relative to the bounding box or relative to the page margins. Click the **Layout | Align | Align to Margins** command or the button to turn the **Align to Margins** mode on or off. When **Align to Margins** is turned on, the Align commands are enabled when one or more objects is selected and the Distribute Horizontally and Distribute Vertically commands are enabled when two or more objects are selected.

When **Align to Margins** is turned off, two or more objects must be selected to enable the **Align Objects** commands, and three or more objects must be selected to enable the **Distribute Horizontally** and **Distribute Vertically** commands. The **Align to Margins** command icon is highlighted when **Align to Margins** is enabled.

The default **Align to Margins** state is not checked. The **Align to Margins** option applies to all documents and windows open in **Surfer** in a single session. The setting is not persistent between **Surfer** sessions, i.e. if you close and reopen **Surfer**, **Align to Margins** will be turned off.

Align to Margins Example

When **Align to Margins** is enabled, the selected objects are individually aligned relative to the page when one of the **Align Objects** commands is clicked.

*The four rectangles were selected and **Layout | Arrange | Align | Top** was clicked while **Align to Margins** was checked. The four rectangles are aligned along the top margin.*

When **Align to Margins** is disabled, the selected objects are individually aligned relative to the bounding box when one of the **Align Objects** commands is clicked.

*The four rectangles were selected and **Layout |Arrange | Align | Top** was clicked while Align to **Margins** was not checked. The four rectangles are aligned along the top of the original bounding box.*

You can group the objects with the Group command to maintain the objects' positions relative to one another while aligning the group relative to the margins when **Align to Margins** is checked.

*The four rectangles were grouped with **Features | Group | Group**. Next the group was selected and **Layout | Arrange | Align | Top** was clicked while **Align to Margins** was checked. The top of the group is aligned along the top margin.*

Free Rotate

You can use the **Layout | Arrange | Free Rotate** command or the button to rotate an object with the mouse. Select the object then click the **Layout | Arrange | Free Rotate** command. The pointer will change to show next to it to indicate that the program is in rotate mode. To rotate an object, click on the screen, hold the left mouse button, and move the mouse. As the object is rotated the degrees of rotation are indicated in the status bar and the object's outline is displayed in the rotated view with a dashed line. To fix the position of the rotated object, release the mouse button.

Clear Rotation

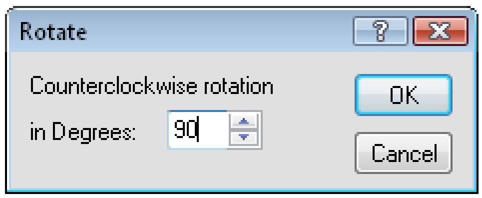
To clear rotation, use Home | Selection | Transform.

Exit Rotate Mode

Press the ESC on your keyboard or click the Select command to exit rotate mode.

Rotate

Use the **Layout | Arrange | Free Rotate | Rotate** command or the button to rotate an object by a specified number of degrees. After selecting the command, type the number of degrees to rotate the object into the **Rotate** dialog. Positive numbers rotate the object in a counterclockwise direction and negative numbers rotate the object in a clockwise direction.

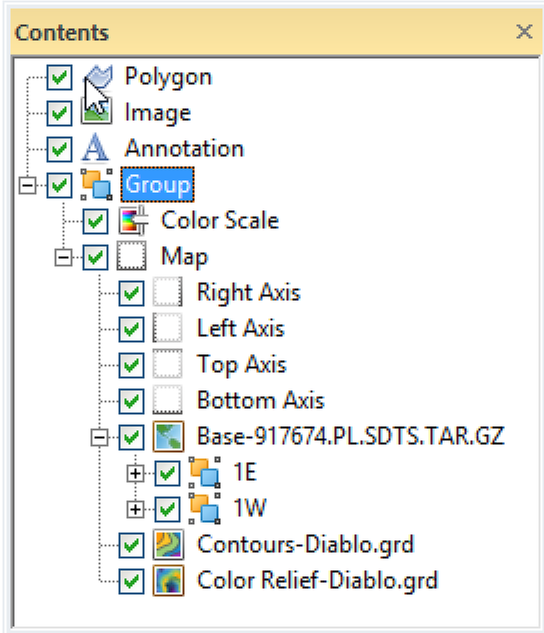


Specify the degree of Counterclockwise rotation in the ***Rotate*** *dialog.*

To clear rotation, use Home | Selection | Transform.

Group

The **Features | Group | Group** command or the button is used to group several independent objects into one *Group* object. Alternatively, select multiple objects and then right-click and select **Group**. Grouped objects can be a combination of several types of objects and they can be moved or resized as a single object.



Group objects to toggle visibility or arrange multiple objects quickly and easily.

Objects must be on the same level in the **Contents** window and, if applicable, under the same group or layer to be grouped. For example, top-level objects such as features, legends, and map frames can be grouped. Similarly, features within a single base layer can be grouped. If multiple objects can be selected, then the objects can be grouped. However, map layers cannot be grouped, and objects in different base layers or groups cannot be grouped.

Editing Grouped Objects

After objects are grouped together, the group is displayed in the Contents window as a *Group* object and sub-objects. Sub-objects can be edited, re-arranged, or deleted in the **Contents** window. Click on any of the sub-objects in the **Contents** window to display and change the properties for that object in the Properties window. A sub-object may also be deleted.

A group object can be transformed with the **Home | Selection | Transform** command.

Ungroup

The **Features | Group |Ungroup** command or the  button is used to separate objects that have been previously combined into a group object using the Features | Group | Group command.

A grouped object can also be ungrouped in the Contents window by right-clicking on the *Group* object and clicking **Ungroup**.

The **Features | Group | Ungroup** command can also be used on some imported files: Golden Software Boundary .GSB, Atlas Boundary .BNA, USGS .DLG, .LGO, .LGS, AutoCAD .DXF, Golden Software Blanking .BLN, Metafiles, and some pasted objects.

Chapter 31 - Changing the View

View Tab Commands

The **View** tab in the plot document has the following commands:

Fit to Window	Zooms to show all of the objects on screen
Page	Zooms to fit the full page on screen
Zoom	Zooms in, out, on selected objects, realtime, with rect-angles, actual size, to full screen, or previous view
Pan	Scrolls the window by "grabbing" the contents and dragging them
Hyperlink	View hyperlinks in the base layer attributes with the cursor
Redraw	Refreshes the image on screen
Auto Redraw	Turns the automatic display of redraw on or off
Rulers	Shows or hide the rulers
Grid	Shows or hide the drawing grid
Status Bar	Shows or hides the status bar
Contents	Shows or hides the **Contents** window
Properties	Shows or hides the **Properties** window
New Window	Opens a duplicate window of the active document
Cascade	Arranges the windows so they overlap
Arrange Icons	Arranges icons at the bottom of the application window
Tile Horizontal	Arranges windows horizontally so there are no over-lapping tiles
Tile Vertical	Arranges windows vertically so there are no over-lapping tiles
Reset Windows	Resets window display to the default layout

Fit to Window

Click the **View | Zoom | Fit to Window** command in the plot window or the  button, click the **Grid Editor | View | Fit to Window** command in the grid editor, click the **3D View | View | Fit to Window** command in the 3D view, or press CTRL+D on the keyboard to adjust the zoom so all objects fill the plot, grid editor, or 3D view window.

Page

Click the **View | Zoom | Page** command or the button or press CTRL + G to see extents of the page. The page outline is visible if the Show page rectangle option is checked in the **Options** dialog User Interface page.

Zoom In

The **View | Zoom | In** command increases the magnification of the image in the plot window. The command scrolls the window to keep the point of interest centered. The **3D View | View | Zoom In** command increases the magnification of the image in the 3D view. Click to zoom in toward the center of the 3D view window.

Zoom In - Plot

The **View | Zoom | In** command increases the magnification of the image in the plot window. The command scrolls the window to keep the point of interest centered.

Enter Zoom In Mode

To enter zoom in mode, click the **View | Zoom | In** command or the button in the plot window. The cursor will change to a to indicate zoom in mode. Left-click the area on which to center the magnified image.

Exit Zoom In Mode

Press the ESC key or click on another tool command, such as zoom, draw, or reshape, to end zoom in mode.

Zoom with A Wheel Mouse

You can use a wheel mouse to zoom realtime and pan in the plot window or grid node editor. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out. Hold down the wheel button straight down, and the cursor will turn to a closed hand. When the cursor is a , drag the mouse to pan the plot window. Be sure to click straight down with the scroll wheel. The zoom is changed so that the cursor location remains on the screen.

Zoom with the Keyboard

You can use keyboard commands to zoom in and out of the plot window. The default commands are CTRL + EQUAL SIGN (=) to zoom in, and CTRL + HYPHEN (-) to zoom out.

Zoom In - 3D View

Click the **3D View | View | Zoom In** command to change the cursor to zoom-in mode. Click the 3D view window to zoom in to the center of the 3D view window. Press ESC to return the cursor to Trackball mode, or click another **3D View | View** command. The 3D view can also be changed by adjusting the values in the *Environment* properties Camera page.

You can use a wheel mouse to zoom realtime in the 3D view window. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out.

Zoom Out

The **View | Zoom | Out** command decreases the magnification of the image in the plot window. The **3D View | View | Zoom Out** command decreases the magnification of the 3D view window.

Zoom Out - Plot

The **View | Zoom | Out** command decreases the magnification of the image in the plot window. The command scrolls the window to keep the point of interest centered.

Enter Zoom Out Mode

To enter zoom out mode, select the **View | Zoom | Out** command or the button in the plot window. The cursor will change to a to indicate zoom out mode. Left-click the area on which to center the magnified image.

Exit Zoom Out Mode

Press the ESC key or click on another tool command to end zoom out mode.

Zoom with A Wheel Mouse

You can use a wheel mouse to zoom realtime and pan in the plot window or grid node editor. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out. Hold down the wheel button straight down, and the cursor will turn to a closed hand. When the cursor is a , drag the mouse to pan the plot window. Be sure to click straight down with the scroll wheel. The zoom is changed so that the cursor location remains on the screen.

Zoom with the Keyboard

You can use keyboard commands to zoom in and out of the plot window. The default commands are CTRL+EQUAL SIGN(=) to zoom in and CTRL+HYPHEN (-) to zoom out.

Zoom Out - 3D View

Click the **3D View | View | Zoom Out** command to change the cursor to zoom-out mode. Click the 3D view window to zoom out from the center of the 3D view window. Press ESC to return the cursor to Trackball mode, or click another **3D View | View** command. The 3D view can also be changed by adjusting the values in the *Environment* properties Camera page.

You can use a wheel mouse to zoom realtime in the 3D view window. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out.

Zoom Selected

The **View | Zoom | Selected** command magnifies selected objects to the maximum size possible in the plot window.

Select an object in the plot window or **Contents** window and click the **View | Zoom | Selected** command or the  button to fit the view around the selected object.

Zoom with the Keyboard

You can use keyboard commands to zoom to the selected object in plot window. The default keyboard command for the **View | Zoom | Selected** command is CTRL+L.

Zoom Realtime

The **Realtime** command zooms in and out as the left mouse button is held down and the mouse is dragged up and down in the plot or 3D view window. The 3D view can also be changed by adjusting the values in the *Environment* properties Camera page.

Enter Zoom Realtime Mode

To enter zoom realtime mode, click the **View | Zoom | Realtime** command or the  button. The cursor will change to a  to indicate that you are in zoom realtime mode. Hold down the left mouse button and then drag up or down in the window to zoom in or out. As you drag the mouse up, the screen is zoomed in. As you drag the mouse down, the screen is zoomed out.

Exit Zoom Mode

Click another tool or press the ESC key to end zoom mode.

Zoom Realtime With A Wheel Mouse

You can use a wheel mouse to zoom realtime in the plot window, 3D view window, and grid editor. You can use a wheel mouse to pan in the plot window and grid node editor. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out. Hold down the wheel button straight down, and the cursor will turn to a closed hand. When the cursor is a , drag the mouse to pan the plot window. Be sure to click straight down with the scroll wheel. The zoom is changed so that the cursor location remains on the screen.

Zoom Rectangle

The **Rectangle** command allows magnification by drawing a rectangle around the area of interest in the plot window or grid editor.

Enter Zoom Rectangle Mode

Select the **View | Zoom | Rectangle** command or the button in the plot window or the **Grid Editor | View | Zoom Rectangle** command in the grid editor to enter zoom rectangle mode. The cursor will change to a to indicate that you are in zoom rectangle mode. Hold down the left mouse button and drag the dotted zoom rectangle around the area of interest to magnify it. When the left mouse button is released, the plot window will zoom to the selected area.

Exit Zoom Mode

Click another tool button or press the ESC key to end zoom mode.

Zoom with the Keyboard

You can use keyboard commands to zoom to a rectangle in plot window. The default keyboard command for the **Rectangle** command is CTRL+R.

Actual Size

The **View | Zoom | Actual Size** command or the button scales the drawing to the approximate size it will be when printed. The size is usually scaled up slightly on the display to allow an adequate size for displaying text.

Full Screen

Click the **View | Zoom | Full Screen** command or the button to scale the selected object to fit the monitor. The ribbon are not accessible when viewing at **Full Screen**.

Exit Full Screen Mode

Press the ESC key to return to the **Surfer** window. Alternatively, left-click anywhere on the screen.

Pan

The **Pan** command moves your location in the plot window, 3D view window, or grid node editor while retaining the current level of magnification. The 3D view can also be changed by adjusting the values in the *Environment* properties Camera page.

Enter Pan Mode

Click the **View | Pan | Pan** command or the button in the plot window, **3D View | View | Pan** command in the 3D view, or use the mouse wheel in the grid editor to pan the current view.

To pan the current view:

1. Click the **Pan** command.
2. Click on a portion of the current view.
3. Hold the left mouse button down while dragging the view to a new position.

Exit Pan Mode

Click on another toolbar button or press the ESC key to end pan mode.

Pan With A Wheel Mouse

You can use a wheel mouse to zoom realtime in the plot window, 3D view window, and grid editor. You can use a wheel mouse to pan in the plot window, grid node editor, and 3D view. Rotate the wheel forward to zoom in, or rotate the wheel backward to zoom out. Hold down the wheel button straight down, and the cursor will turn to a closed hand. When the cursor is a , drag the mouse to pan the plot window, grid node window, or 3D view. Be sure to click straight down with the scroll wheel. The zoom is changed so that the cursor location remains on the screen.

Redraw

Click the **View | Redraw | Redraw** command or the button or press the F5 key on the keyboard to redraw the window.

Auto Redraw

Auto Redraw is used to automatically redraw the image each time the window contents or view is changed. **Auto Redraw** is on by default, and this is indicated by a check mark beside the command. Click **View | Redraw | Auto Redraw** or the button to toggle the command on and off. If **Auto Redraw** is disabled use Redraw or press the F5 key on the keyboard to redraw the image.

Rulers

Click the **View | Show/Hide | Rulers** command or the button to toggle between showing and hiding the rulers on the top and left sides of the main plot window. When the **Rulers** command button is depressed, the rulers are shown. **Surfer** will remember your preference to have the ruler on or off when the program restarts.

You can also right-click on a ruler and click **Ruler and Grid Settings** to open the **Options** dialog Rulers and Grid page.

Drawing Grid

Click the **View | Show/Hide | Grid** command or the button or right-click on the plot window and click **Drawing Grid** to toggle between showing and hiding a grid which is superimposed over the plot window. The **Grid** command button is depressed to indicate that the grid is displayed. **Surfer** will remember your preference to have the drawing grid on or off when the program restarts.

The drawing grid is a series of evenly spaced dots, similar to graph paper, which is used to help align objects in the plot window.

You can control the number of grid divisions in a page unit by clicking **File | Options** and clicking on the Rulers and Grid page.

New Window

Click the **View | Windows | New Window** command or the button to create a duplicate window. You can display different views or different parts of the same document simultaneously by using **New Window**. Objects can be edited in either window and the changes appear in both windows.

Cascade

Click the **View | Windows | Cascade** command or the button to arrange multiple document windows in an overlapped fashion. Each window is offset a small amount from the previous window. Individual windows can be sized by dragging the window borders.

Arrange Icons

Click the **View | Windows | Arrange Icons** command or the button to arrange the icons for minimized windows at the bottom of the main window. If a maximized window exists, then some or all of the icons may be located underneath the window.

Tile Horizontal

Click the **View | Windows | Tile Horizontal** command or the button to arrange multiple document windows in a non-overlapped fashion such that the windows are oriented one above the other.

Tile Vertical

Click the **View | Windows | Tile Vertical** command or the button to arrange multiple windows in a non-overlapped fashion side by side.

Reset Windows

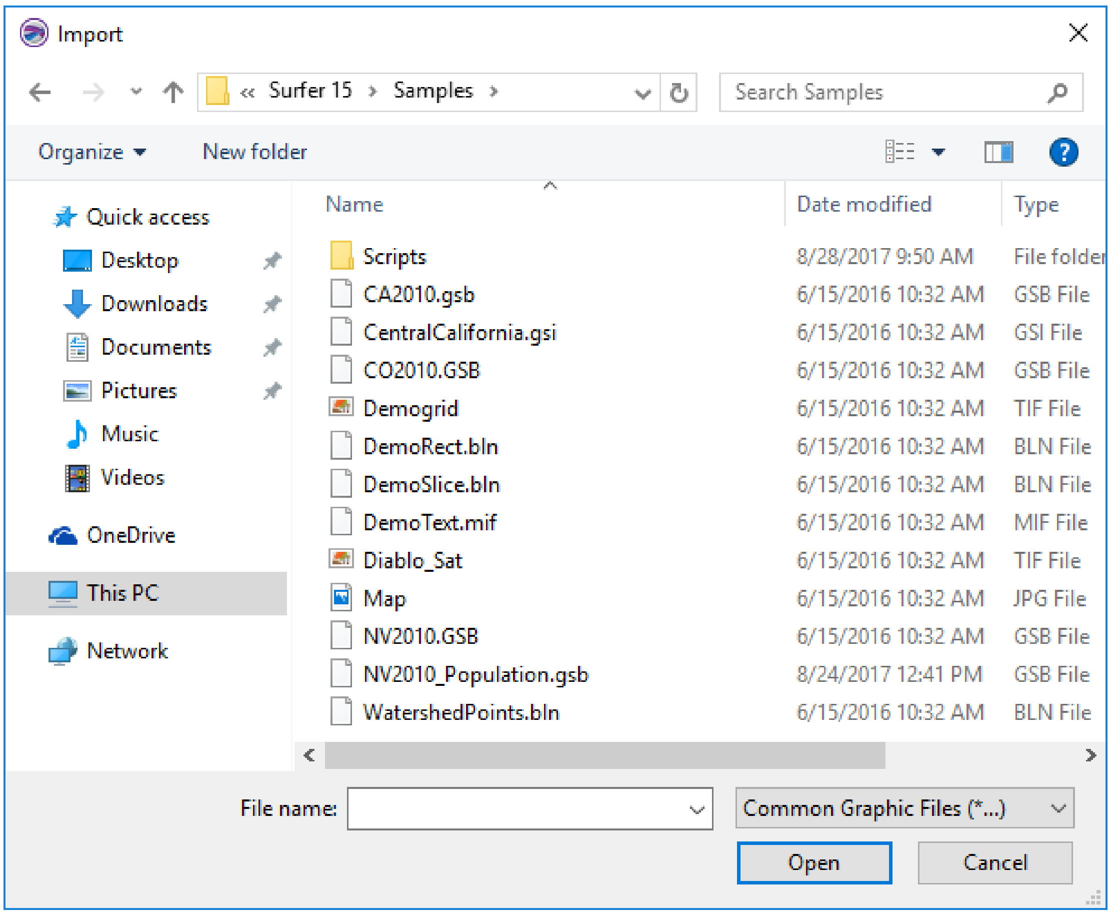
Click the **View | Windows | Reset Windows** command or the button to change the display of the program. This command resets the Contents and Properties windows back to the default size and position. It also resets all ribbon, quick access toolbar, and keyboard command customizations back to the defaults. This command is especially useful if your windows or managers become hidden by mistake.

You must restart **Surfer** in order for this command to take effect. Click *Yes* in the dialog, close the program, and reopen **Surfer**. The windows, ribbon, and quick access toolbar are now in the default conditions.

Chapter 32 - Importing, Exporting, and Printing

Import

The **Home | Insert | Graphic**, the **Home | New Map | Base**, and the **Home | Add to Map | Layer | Base** commands in the plot window open the **Import** dialog.



*Select files to import using the **Import** dialog.*

Look In

The *Look in* field shows the current directory. Click the down arrow to see the directory structure. Click on the folders to change directories.

Creating New Folders and Changing the View

The buttons to the right of the *Look in* field allow you to create new folders and change the view of the file list.

File List

The file list displays files in the current directory. The current directory is listed in the *Look in* field. The *Files of type* field controls the display of the file list. For example, if *Golden Software Boundary (*.GSB)* is listed in the *Files of type* field only .GSB files appear in the files list. To see all files in the directory, choose *All Files (*.*)* from the *Files of type* list.

Double-click on a file to open it or single-click the file and then click the *Open* button. To import more than one file, select the files you wish to import by holding CTRL while clicking, and then click *Open*. When adding images with **Home | Insert | Graphic**, each file is added as an image a group. When adding base maps with **Home | New Map | Base**, each file is added as a separate map.

File Name

The *File name* field shows the name of the selected file. Also, a path and file name can be typed into the box to open a file.

Open Files

Surfer tracks the files being used by maintaining a list of opened files for new maps and the files used in saved maps. The file(s) used in existing maps are shown in a compiled list in the *Open images* field of the **Import** dialog.

Files of Type

The *Files of type* field shows the file format to be opened. To change the file format, click the down arrow and select the file type from the list.

The *Common Graphic Files (*...)* format type is selected by default. This displays all the file formats that can be imported with **Home | Insert | Graphic** in the navigation pane. If a different format type is selected, **Surfer** will remember the setting until the end of the current session. When **Surfer** is restarted, the default format type will be used.

To see all files in the directory, choose *All Files (*.*)* from the *Files of type* list. Double-click on a file to open it or single-click the file and then click the *Open* button. The *All Files (*.*)* option shows all of the file formats in the current directory, even if the file type is not appropriate for the action chosen.

Import Format Types

The **Home | Insert | Graphic** command in the plot document opens the **Import** dialog. In the **Import** dialog, select one of the following formats to import objects into the plot.

- 000 IHO S-57 Navigation Chart (*.000)
- AN? ACR-NEMA Medical Image (*.an1, *.an2)
- BLN Golden Software Blanking (*.bln)
- BMP Windows Bitmap (*.bmp)
- BNA Atlas Boundary (*.bna)
- DICOm3 Medical Image (*.dic, *.dcm)
- DDF SDTS TVP (*.ddf, *.tar, *.tar.gz, *.zip, *.tgz)
- DGN MicroStation Design v7 (*.dgn)
- DLG USGS Digital Line Graph (*.dlg, *.lgo, *.lgs)
- DXF AutoCAD Drawing (*.dxf)
- E00 Esri ArcInfo Export Format (*.e00)
- ECW ERMapper (*.ecw)
- EMF Windows Enhanced Metafile (*.emf)
- GEOJSON JSON Data Interchange (*.geojson, *.json)
- GIF Image (*.gif)
- GML Geograpy Markup Language (*.gml)
- GPX GPS Exchange Format (*.gpx)
- GSB Golden Software Boundary (*.gsb)
- GSI Golden Software Interchange (*.gsi)
- JPG Compressed BItmap (*.jpg, *.jpeg)
- JPEG-2000 Bitmap (*.jp2, *.j2k, *.jpc, *.jpt, *.jpeg2000, *.j2000)
- KML Google Earth Keyhole Markup File Description (*.kml, *.kmz)
- MIF MapInfo Interchange Format (*.mif)
- PDF Adobe PDF (Vector or Raster) (*.pdf)
- PLT Golden Software PlotCall (*.plt)
- PLY Stanford PLY (*.ply)
- PNG Portable Network Graphics (*.png)
- PNM/PPM/PGM/PBM Image (*.pnm, *.ppm, *.pgn, *.pbm)
- RGB SGI-RGB Image (*.rgb, *.rgba, *.bw)
- RT Tiger/LINE (*.rt)
- SEG-P1 Exchange Format (*.sp1, *.seg)
- SHP Esri Shapefile (*.shp)
- SID LizardTech MrSID Image (*.sid)
- SUN Sun Raster Image (*.ras, *.sun)
- TAB MapInfo Table (Vector) (*.tab)
- TGA Targa (TrueVision) (*.tga)
- TIF Tagged Image (*.tif, *.tiff)
- VCT Idrisi Binary Vector (*.vct)
- VTK Visualization Toolkit (*.vtk)

- WMF Windows Metafile (*.wmf)
- X AVS X-Image (*.x, *.ximg)

Open Images

Surfer tracks the files being used by maintaining a list of opened images for new maps and the images used in saved maps. The images(s) used to build maps can be opened from the compiled list in the *Open Grids* field of the Open Grid dialog. The only exception is that vector data between base layers will not be shown.

Import Format Types

The **Home | Insert | Graphic** command in the plot document opens the **Import** dialog. In the **Import** dialog, select one of the following formats to import objects into the plot.

- 000 IHO S-57 Navigation Chart (*.000)
- AN? ACR-NEMA Medical Image (*.an1, *.an2)
- BLN Golden Software Blanking (*.bln)
- BMP Windows Bitmap (*.bmp)
- BNA Atlas Boundary (*.bna)
- DICOm3 Medical Image (*.dic, *.dcm)
- DDF SDTS TVP (*.ddf, *.tar, *.tar.gz, *.zip, *.tgz)
- DGN MicroStation Design v7 (*.dgn)
- DLG USGS Digital Line Graph (*.dlg, *.lgo, *.lgs)
- DXF AutoCAD Drawing (*.dxf)
- E00 Esri ArcInfo Export Format (*.e00)
- ECW ERMapper (*.ecw)
- EMF Windows Enhanced Metafile (*.emf)
- GEOJSON JSON Data Interchange (*.geojson, *.json)
- GIF Image (*.gif)
- GML Geograpy Markup Language (*.gml)
- GPX GPS Exchange Format (*.gpx)
- GSB Golden Software Boundary (*.gsb)
- GSI Golden Software Interchange (*.gsi)
- JPG Compressed BItmap (*.jpg, *.jpeg)
- JPEG-2000 Bitmap (*.jp2, *.j2k, *.jpc, *.jpt, *.jpeg2000, *.j2000)
- Images are typically imported into **Surfer** as a base map by clicking the Home | New Map | Base command.
- KML Google Earth Keyhole Markup File Description (*.kml, *.kmz)
- MIF MapInfo Interchange Format (*.mif)
- PDF Adobe PDF (Vector or Raster) (*.pdf)
- PLT Golden Software PlotCall (*.plt)
- PLY Stanford PLY (*.ply)
- PNG Portable Network Graphics (*.png)
- PNM/PPM/PGM/PBM Image (*.pnm, *.ppm, *.pgn, *.pbm)

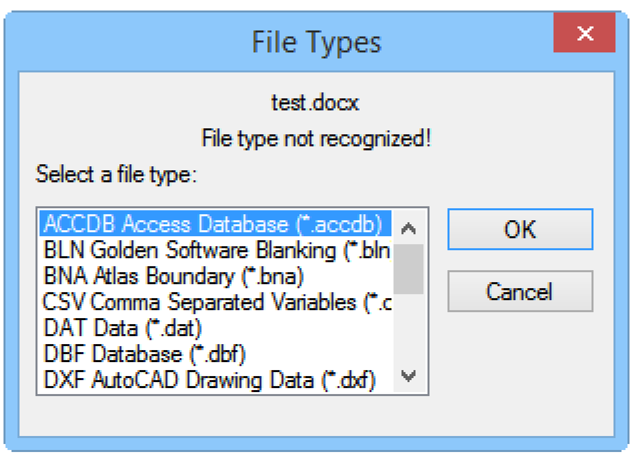
- RGB SGI-RGB Image (*.rgb, *.rgba, *.bw)
- RT Tiger/LINE (*.rt)
- SEG-P1 Exchange Format (*.sp1, *.seg)
- SHP Esri Shapefile (*.shp)
- SID LizardTech MrSID Image (*.sid)
- SUN Sun Raster Image (*.ras, *.sun)
- TAB MapInfo Table (Vector) (*.tab)
- TGA Targa (TrueVision) (*.tga)
- TIF Tagged Image (*.tif, *.tiff)
- VCT Idrisi Binary Vector (*.vct)
- VTK Visualization Toolkit (*.vtk)
- WMF Windows Metafile (*.wmf)
- X AVS X-Image (*.x, *.ximg)

Remarks

- To open Golden Software Blanking .BLN and Atlas Boundary .BNA files in the worksheet use File | Open in Worksheet rather than **Home | Insert | Graphic**.
- Images are typically imported into **Surfer** as a base map by clicking the Home | New Map | Base command.
- Where applicable, **Surfer** automatically imports all available attribute information.

Import File Types

The **File Types** dialog opens when a file is imported and the file type is not recognized.



*The **Select Format** dialog appears when an unsupported or unknown file type is selected.*

Select a File Type

If a file format is not recognized, select a supported file type from the *Select a file type* list in the **File Types** dialog. If the file type is not in the list, and if the file is not an ASCII text file, you will not be able to use the file directly. Save the file in another program as a different format.

Export

The **File | Export** command saves files as graphic files to use in other programs.

Click the **File | Export** command, or the button on the quick access toolbar, to open the **Export** dialog. The **File | Export** command is disabled if there are no objects in the document.

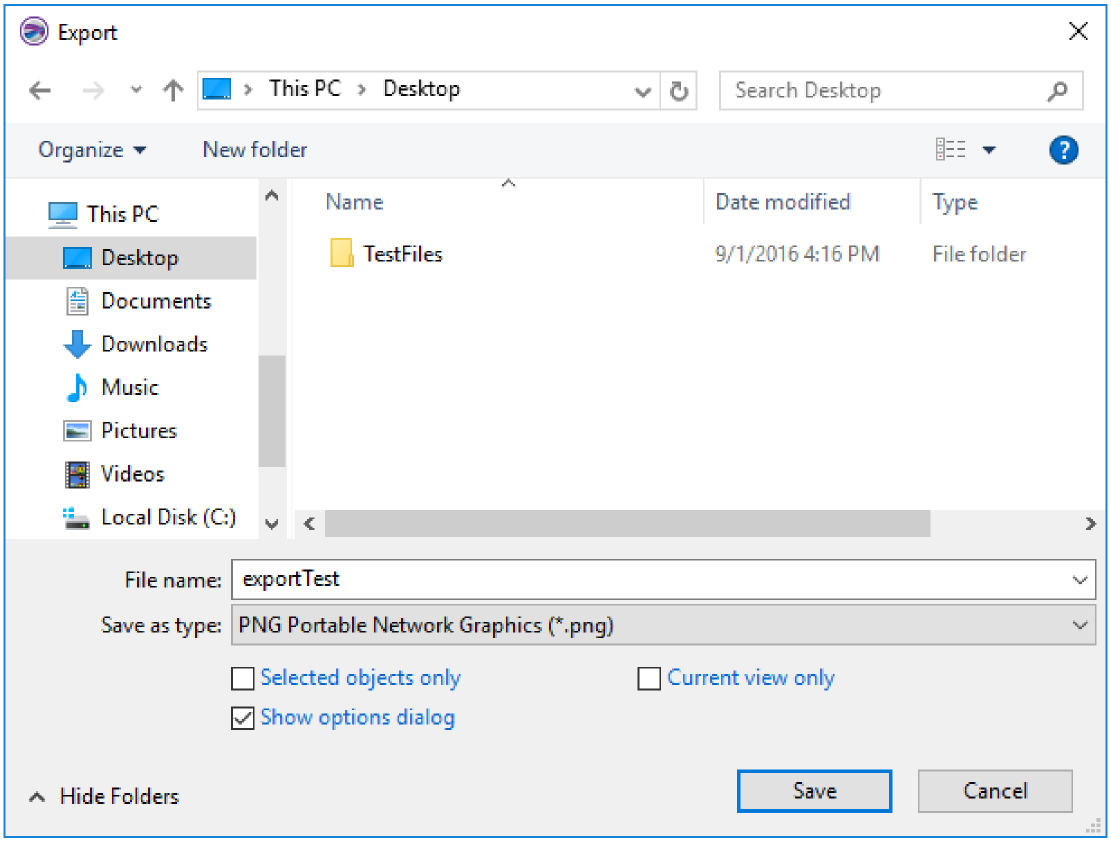
Ensure that edit mode is not enabled before using the **Export** command either by clearing the selection or selecting the object(s) you wish to export.

Attribute Information

Where applicable, the export filter exports attribute information for lines, polygons, and symbols. With contour maps, the **File | Export** command can be used to export Z information to an attribute field for BLN, BNA, CSV, DAT, GSB, GSI, KML, KMZ, MIF, and SHP files.

Export Dialog

Click the **File | Export** command to open the **Export** dialog.



*Specify the save location, file name, and file type in the **Export** dialog.*

Save In

The *Save in* field shows the current directory. Click the down arrow to see the directory structure and click on the folders to change directories. The buttons to the right of the *Save in* field allow you to create new folders and change the view of the file list.

File List

The file list displays the files using the extension specified in the *Save as type* box. A file can be overwritten by selecting it from the file list.

File Name

The *File name* box displays the name of the selected file, or type in the path and file name of the file to be exported.

Save As Type

The *Save as type* list box specifies the format of the file to be exported.

Selected Objects Only

Check the *Selected objects only* box to export selected objects rather than the entire plot.

Show Options Dialog

Check the *Show options dialog* option to display the Export Options dialog for the selected *Save as type*. If the *Show options dialog* option is selected when the *Save* button is clicked, the **Export Options** dialog appears. The Scaling page and Size and Color page of the **Export Options** dialog is uniform. Additional pages in the **Export Options** dialog may be available dependent on the export format type.

Current View Only

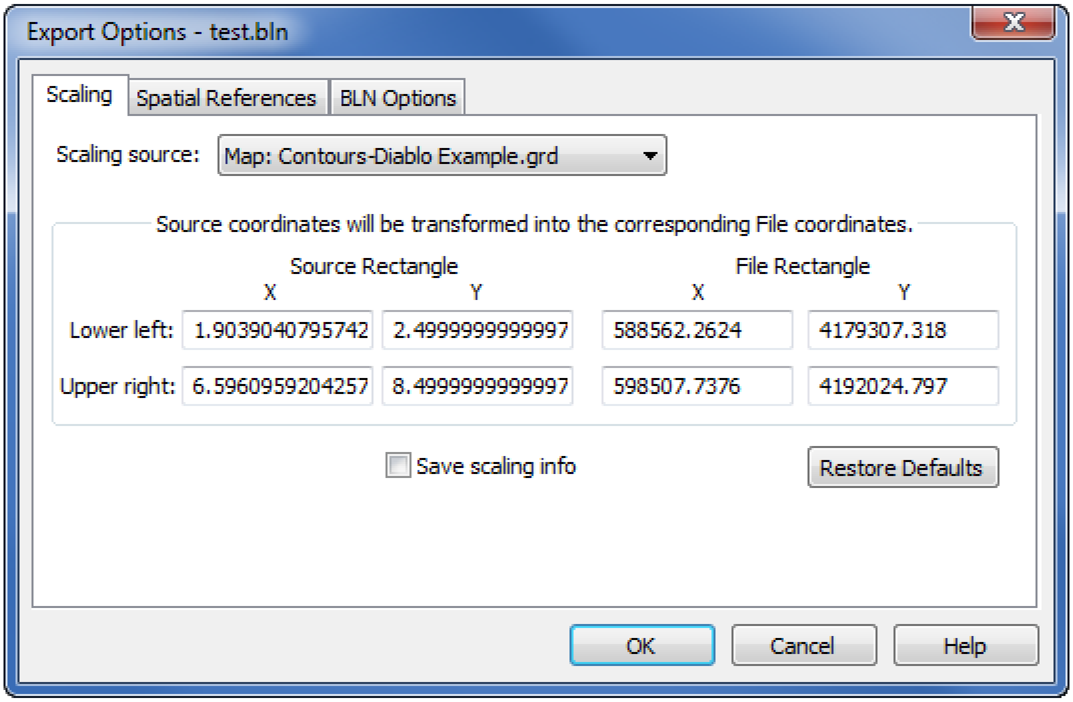
Check the *Current view only* check box to export only the portion of the plot document that is currently visible in the plot window. This option is useful if you wish to export only one portion of a plot, map, or surface. Zoom in to the region you wish to export before clicking the **Export** command. You can also resize the application window or plot window to fine tune the extent of the exported region. You can use the *Selected objects only* option to include only selected objects in the current view. Clear the *Current view only* check box to export the plot document's contents.

File Name

Export files typing a name into the *File name* box and then selecting the file type in the *Save as type* list. For example, typing MYPLOT in the *File name* box and choosing Tagged Image (TIFF) from the *Save as type* list results in MYPLOT.tif.

Export Options Dialog - Scaling Page

Many of the file formats have a **Scaling** page in the **Export Options** dialog.



Specify scaling options on the ***Scaling*** *page of the* ***Export Options*** *dialog.*

Scaling Source

Scaling information can be retrieved from multiple sources: *Paper space (page units)*, all map frame names in the **Surfer** file, any coordinate system for each image in the **Surfer** file, and *Saved settings*. To select the appropriate map, click on the existing map name and select the desired map from the list. All objects are exported with the coordinates from the *Scaling source*.

Surfer provides potentially useful scaling info whenever possible. If a single *Map* is selected in the plot window, the map coordinate system is used for the *Coordinate System*. If a single image is selected in the plot window, the coordinate system for the image is used for the *Coordinate System*. If more than one *Map* object is selected, the *Coordinate System* is set to *Paper space (page units)*. This can be changed to any of the *Map* coordinate systems in the list and all objects will be exported to this coordinate system.

If the application detects an unrotated 3-dimensional map object that is viewed from directly overhead (i.e., it is really a 2-D map object) and that map object is the only object being exported, it retrieves the (X,Y) data extents from the map and makes those the *Scaling source*. Otherwise, the application sets the *Scaling source* to *Paper space (page units)* so the coordinates will be the same as the document page units.

Paper Space

Paper space (page units) sets both the *Source Rectangle* and *File Rectangle* to page units.

Map Coordinates

Selecting any *map* or *image* from the *Scaling source* will load scaling info calculated by the application for the selected map or image.

Saved Settings

Saved settings will reload previously saved values.

Rectangle

Rectangle scaling is accomplished by specifying the corner points of a rectangle (in page units for the *Source Rectangle*) in the application document and the corner points of a rectangle (in *Scaling source* map units for *File Rectangle*). The document coordinates will be offset and/or scaled so the corner points of the document rectangle will have the desired coordinates.

The *Page Rectangle* lists two points on the page in the page coordinates. The *File Rectangle* lists the same two points on the page in the *Scaling source* coordinates.

Save Scaling Info

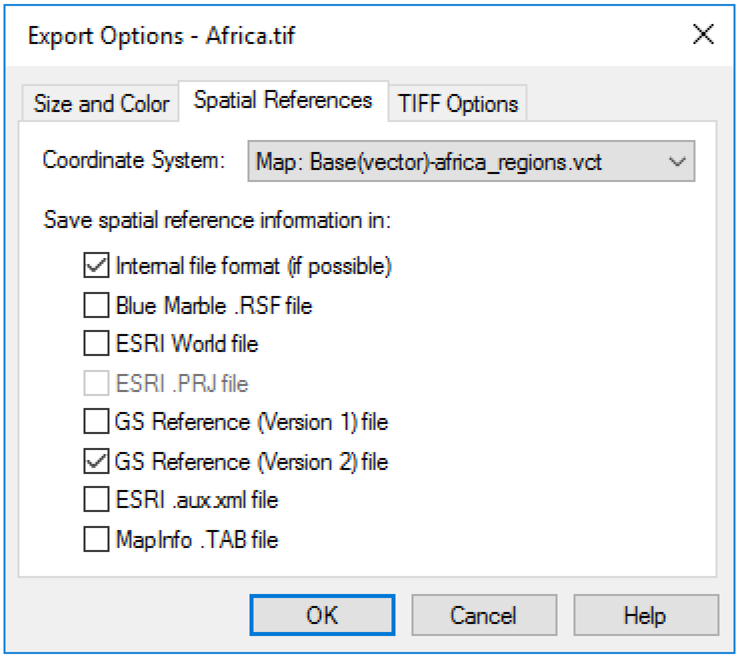
Check the box next to the *Save scaling info* option to save the scaling information to be stored for future use.

Defaults

Click the *Restore Defaults* button to set all options to default conditions. The scaling rectangles will, in turn, be reloaded with values from the default scaling source.

Export Options Dialog - Spatial References Page

Some applications associate spatial reference information (such as projection, datum, and georeference parameters) with files, to link the file to a specific region of the Earth’s surface. If the map that you are exporting has a coordinate system defined, the **Spatial References** tab will appear in the **Export Options** dialog.



*The **Spatial References** page allows you to choose how to save the projection, datum, and georeference information .*

Coordinate System

Next to *Coordinate System*, select the appropriate coordinate system to use when exporting the file. Available options are *Paper space (page units),* all map frame names in the **Surfer** file, and any coordinate system for each image in the **Surfer** file. To select the appropriate map, click on the existing map name and select the desired map from the list. All objects are exported in the selected *Coordinate System*.

If a single *Map* is selected in the plot window, the map coordinate system is used for the *Coordinate System*. If a single image is selected in the plot window, the coordinate system for the image is used for the Coordinate System. If more than one *Map* object is selected, the *Coordinate System* is set to *Paper space (page units)*. This can be changed to any of the *Map* coordinate systems in the list and all objects will be exported to this coordinate system.

To export only some of the objects to the desired coordinate system, select the objects first and check the *Export selected objects only* option on the **Export** dialog.

The *Coordinate System* option is not available for all file formats. With most vector formats (DXF, SHP, etc.), the *Coordinate System* option is on the Scaling tab.

Reference File Format

Most bitmap file storage formats don’t have a way to store the spatial reference information in the same file as the bitmap image. For these formats, the only way the spatial reference information can be saved is in a separate file. If the selected *Coordinate System* supports georeferencing information, the appropriate reference file options will be enabled. If the selected *Coordinate System* has a warp, only the reference file formats that support warp will be enabled.

Internal File Format (if Possible)

If the export format can internally store the georeference information, check this box. The spatial reference information will be stored (along with the image) in the selected format. One format that stores the coordinate reference is a TIF file, a vendor-independent format that can be imported into a variety of geographic software packages.

Blue Marble .RSF File

Blue Marble Geographics supports a text-based format for storing georeference information in a separate file. Various Blue Marble programs will import the information in this file when importing the bitmap file.

Esri World file

Environmental Sciences Research Institute (Esri) supports a text-based format for storing georeference information in a separate file. Various Esri programs will import the information in this file when importing the bitmap file. The first and third characters of the image file’s suffix, plus a final "w", are used for the world file suffix. Therefore, "mytown.tif" will have a world file called "mytown.tfw"; for "redlands.bmp", it will be "redlands.bpw".

Esri .PRJ File

Environmental Sciences Research Institute (Esri) supports a text-based format for storing georeference information in a separate file. Various Esri programs will import the information in this file when importing vector files, such as .SHP files.

GS Reference (Version 1) File

Golden Software supports a text-based format for storing spatial reference information in a separate file. Projection, datum and georeference information can be stored in this format. The file will be created having the same name as the exported bitmap, with the suffix ".GSR" appended. The .GSR file is an older format Golden Software reference file.

GS Reference (Version 2) File

Golden Software's .GSR2 file is a newer georeference file format that includes spatial reference information in a separate file. Projection, datum and georeference information can be stored in this format. The file will be created having the same name as the exported file, with the suffix ".GSR2" appended.

Esri .AUX.XML File

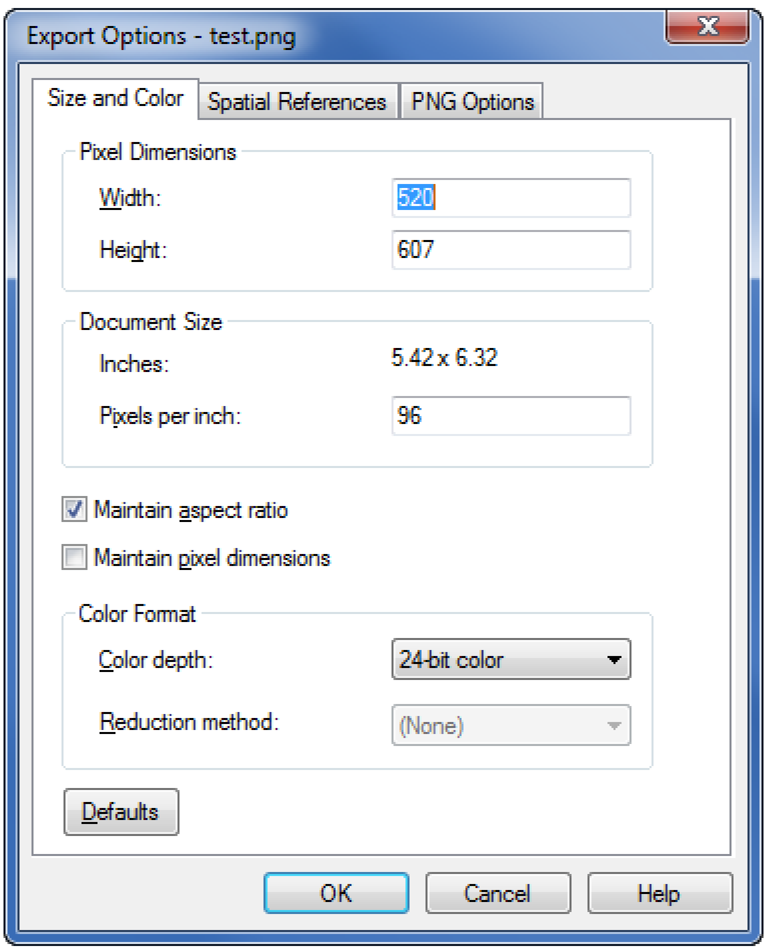
Environmental Sciences Research Institute (Esri) supports an xml-based format for storing georeference information in a separate file. Various Esri programs will import the information in this file when importing grid files, such as Surfer .GRD files.

MapInfo TAB File

MapInfo TAB files are text tables. A TAB file can include various types of information, including vector graphic information. A TAB file can be used as an external reference file for an image. Select the *MapInfo TAB file* option to include a MapInfo TAB reference file.

Export Options Dialog - Size and Color Page

All image export formats have a **Size and Color** page in the **Export Options** dialog.



*The **Size and Color** page of the **Export Options** dialog controls options for image export.*

Pixel Dimensions

Choose the *Width* and *Height* pixel settings for the image. The *Pixel Dimensions* indicate the number of pixels that are in the exported image. The larger the number of pixels, the larger the output image will be.

Document Size

The *Inches* displays the current selected image size in inches. The image size is updated when the *Pixel Dimensions* are adjusted. The size in inches is determined by dividing the number of *Pixel Dimensions* by the *Pixels per inch* and rounding to the hundredths.

When the *Pixels dimensions* have not been changed, the *Inches* is approximately equal to the size of the objects being exported. To determine this size, click once on the object to select it or select all objects that are being exported. The status bar will list the size of the selected objects.

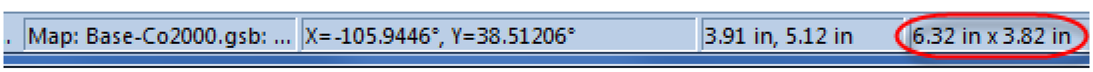
Pixels Per Inch

Choose the *Pixels per inch* to increase or decrease the resolution of the image being exported. If you choose to change the number of *Pixels per inch*, the *Width* and *Height* in the *Image Dimensions* changes accordingly. The *Pixels per inch* control how fine a resolution the output image will have. The larger the number of *Dots per inch*, the larger the *Pixel Dimensions* will be.

The *Pixels per inch* is set to 72 for all GIF images and cannot be changed. GIF images are always 72 DPI, by definition. For higher quality images, it is suggested that PNG, TIF, or BMP be used instead of GIF.

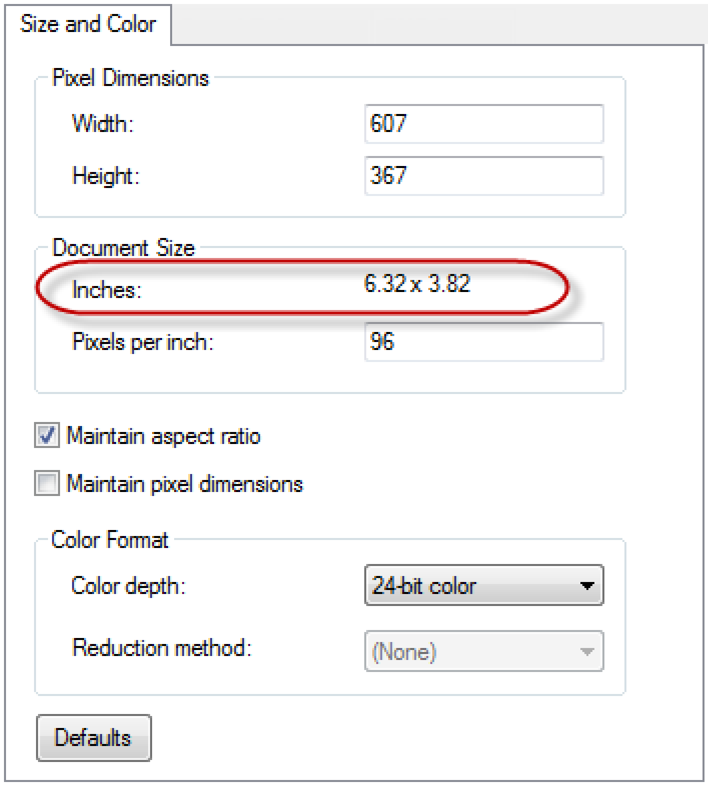
Example

For example, a base map is selected. The status bar reports that the size of the map is 6.32 in x 3.82 in.



The status bar shows the size of the objects being exported.

When this map is exported, the **Size and Color** tab shows the *Inches* as 6.32 by 3.82.



The size is approximately equal to the page size of the objects being exported.

Maintain Aspect Ratio

Check the *Maintain aspect ratio* box if you want the image to maintain an equal horizontal and vertical resolution. When unchecked, the output image may appear stretched in one of the directions.

Maintain Pixel Dimensions

Check the *Maintain pixel dimensions* box to export the image at the selected *Width* and *Height* , but with a different number of *Pixels per inch* . This results in the same number of pixels, but a different *Document Size* .

Color Format

The *Color Format* gives you the option to output your image with *Color depth.* The greater the color depth, the more faithfully the image will represent the colors assigned to objects in your document. Different output formats support different color depths. Some output formats support 256 colors only, while others also support True Color (16 million colors).

Select one of the options from the *Color Depth* list. The options are: *8-bit grayscale*, *16-bit grayscale*, *32-bit grayscale*, *1-bit color indexed*, *4-bit color indexed*, *8-bit color indexed*, *24-bit true color,* or *32-bit true color with alpha*.

For example, Windows .BMP format supports *monochrome*, *16 colors*, *256 colors* and *True Color*. Greater color depth will yield a better-looking image, but at the expense of requiring more memory and disk space to hold the image.

Reduction Method

If you select a color indexed *Color Depth*, you can choose a *Reduction method*. Select one of the options from the *Reduction method* list. The options are: *Ordered Dither*, *Diffuse Dither*, *Popularity*, *MedianCut555*, or *MedianCut888*.

If a *Color depth* of 256 or fewer colors is selected, you may specify the type of *Dithering* and the type of *Quantization* that is used to reduce the application's image to the selected number of colors.

Dithering determines how similar colors are distributed among clusters of pixels in the reduced image. Possible selections are *Diffuse* which uses a pseudo-random pattern, *Ordered* which uses a repeating pattern, or *None* which disables dithering.

Quantization determines how the colors for the exported image are selected from the palette of 16 million possible colors. Possible selections are *Popularity* which uses the most frequently occurring colors in the image, and *Median* which selects colors based on the 'median cut' method that tries to select the most even distribution of colors over the range of colors that appear in the image. The *Median* method can use either 5, 6, or 8 bits of sample data for each of the three color planes in the image, and the corresponding selections are *5:5:5 Median*, *6:6:6 Median*, and *8:8:8 Median*. Larger sample sizes require more memory to perform the conversion for export, so the smallest sample size that produces an acceptable image is recommended.

Defaults

Click the *Defaults* button to return the export options to the default selections.

Transparency

Checking the transparency options on the PNG Options tab, the GIF Options tab, or the TIF Options tab can result in an error message when exporting if the *Color depth* option or *Reduction method* does not allow transparency. Click *OK* on the error and change the *Color depth* to 4-bit, 8-bit, or 32-bit for PNG or TIF images. For either PNG or GIF images, change the *Reduction method* to MedianCut555, MedianCut888, or Popularity.

Transparent TIFF images in 4-bit and 8-bit modes use TIFF tag number 42113 (a.k.a. GDAL_NODATA) which is not yet widely supported by other applications. In applications that don't support this tag, the images appear with an opaque background (no transparency). If the application shows an opaque background where a transparent background should appear, re-export the file with 32-bit color depth. Most programs support transparent TIF files with 32-bit colors.

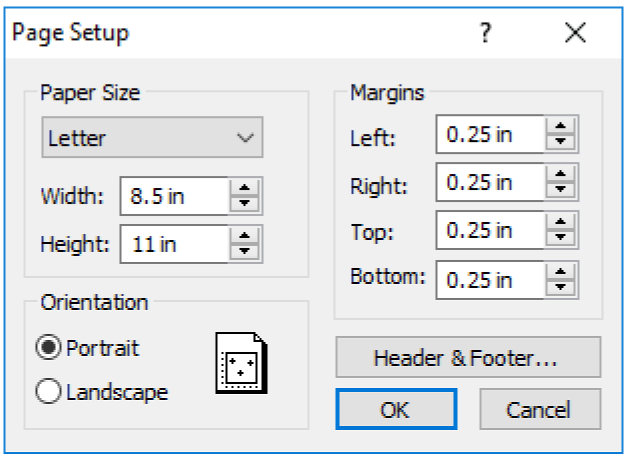
Page Setup

The **File | Page Setup** command in the plot window formats the document for printing. This includes paper size, orientation, margins, and a header and footer.

Page Setup Dialog

There are multiple ways to open the **Page Setup** dialog:

- Click the **File | Page Setup** command or the button
- Right-click on the plot window and click **Page Setup**
- Click the **Layout | Page Setup | Margins | Custom** command or the button.
- Click the **Layout | Page Setup | Size | More Sizes** command.



*Set page setup preferences in the **Page Setup** dialog.*

Paper Size

The *Paper Size* options control the size of the paper. Click the down arrow next to the *Paper Size* to select different paper dimensions. **Surfer** has several predefined page sizes, including *Letter, Tabloid/Ledger, Legal, Executive, A0, A1, A2, A3, A4, A5, B0, B1, B2, B3, B4,* and *B5*. For custom paper sizes, select *Custom* from the list and change the paper size in the *Width* and *Height* boxes.

Select the **Letter** (8.5"x11"), **Legal** (8.5"x14"), **Tabloid/Ledger** (17"x11"), or **A4** (210mm x 297mm) paper size by clicking the **Layout | Page Setup | Size** command.

Orientation

The *Orientation* group controls whether the page is set to *Portrait* or *Landscape* mode. Select *Portrait* to have a vertical page. Select *Landscape* to have a horizontal page.

Margins

Use the *Margins* group to set the page margins for all sides of the printed page. The default *Margins* are 0.25 inches (6.35 mm). Set the *Left*, *Right*, *Top*, and *Bottom* values in inches to any limits the printer will allow. Change the margins by entering new numbers into the *Left*, *Right*, *Top*, and *Bottom* boxes, or use the arrow buttons to scroll to new numbers.

Select **Normal**, **Narrow**, **Moderate**, or **Wide** margins by clicking the **Layout | Page Setup | Margins** command. **Narrow** is the default selection. **Normal** margins are 0.5 inches. **Moderate** margins are 0.5 inches at the *Top* and *Bottom* and 0.75 inches at the *Left* and *Right*. **Wide** margins are 0.5 inches at the *Top* and *Bottom* and 1.0 inches at the *Left* and *Right*.

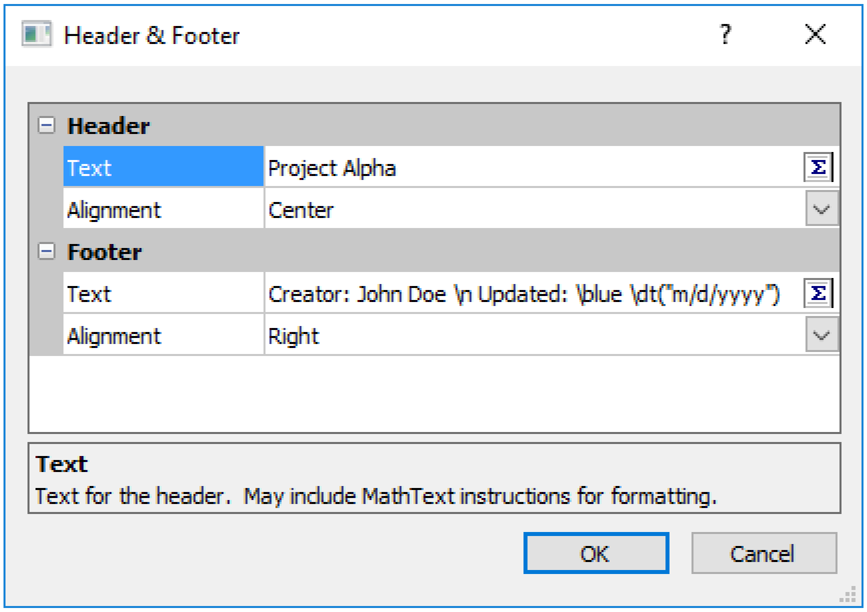
Setting the margins does not move an existing map on the page. Margins can be displayed on the page by checking the *Show margins* option on the User Interface page of the **Options** dialog.

Header & Footer

Click the *Header & Footer* button to set header and footer text and alignment in the Header & Footer dialog. The header and footer are displayed in the plot window. By default the header and footer are not included in printed or exported output. To include the header/footer in printed and exported output, check the *Include header/footer in export/print* check box in the **Options** dialog General page.

Header & Footer Dialog

Click the **Layout | Display | Header** or the button, or the **Layout | Display | Footer** command or the button to open the **Header & Footer** dialog. The dialog can also be opened by clicking the *Header & Footer* button in the Page Setup dialog. Set the header and footer text and alignment in the **Header & Footer** dialog. The header and footer are displayed in the plot window. By default the header and footer are not included in printed or exported output. To include the header/footer in printed and exported output, check the *Include header/footer in export/print* check box in the **Options** dialog General page.



*Set header and footer text and alignment in the **Header & Footer** dialog.*

Header and Footer Sections

The *Header* section and *Footer* section each contain fields for the *Text* and *Alignment*. Set the header properties in the *Header* section. Set the footer properties in the *Footer* section.

Text

Type the desired header or footer text in the *Text* field. Click the button to open the Text Editor to insert math text instructions. You can also type math text instructions directly into the *Text* field without opening the **Text Editor**.

Linked text can be added to headers and footers by typing the linked text variables in the *Text* field or in the **Text Editor**. Note that linked text is only supported in the headers and footers.

- Type <<*filename*>> to display the full name and path for the file, for example "C:\Users\My Name\Documents\My File.srf"
- Type <<*path*>> to display the file path, for example "C:\Users\My Name\Documents"
- Type <<*title*>> to display the file name and extension, for example "My File.srf"
- Type <<*date/time*>> to display the current date and/or time, where *date/-time* is any date/time format. The date and time update automatically each time the plot is changed or opened. For example, type <<ddd dd MMM YYYY HH:mm:ss>> to display the current date/time formatted as *Wed 16 Nov 2016 14:59:24*.

Alignment

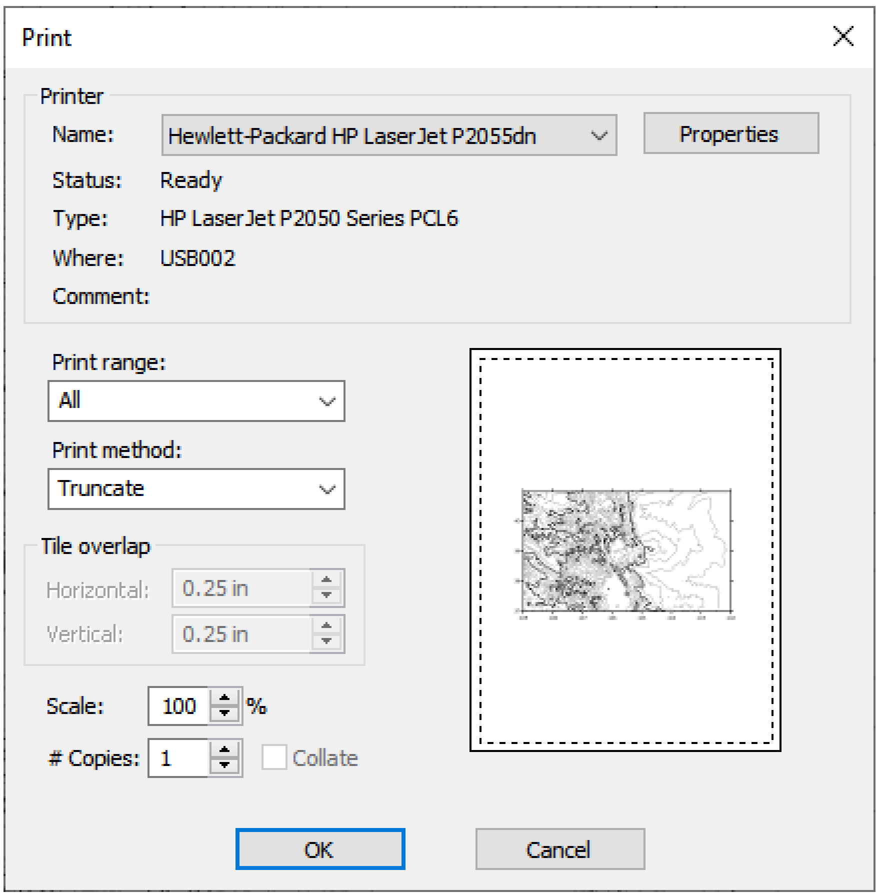
Set the header or footer text alignment relative to the page in the *Alignment* field. Click the current option and select *Left*, *Center*, or *Right* from the list. Use the Text Editor to set the text justification to the left, center, or right.

Print - Plot

Click the **File | Print** command or the button in the plot window to print the active document.

Print Dialog

The **File | Print** command in the plot window opens the **Print** dialog.



*Set your printer preferences in the **Print** dialog.*

Printer

The *Printer* section contains options to specify the printer. The printer *Status, Type, Where,* and *Comments* are listed below the printer *Name*.

Name

The default system printer is listed in the *Name* field. If more than one printer is installed on the computer, use the down arrow to the right of the name field to select a different printer.

Properties

Click the *Properties* button to specify a printer and the printer properties. For information on specific printer settings, see the owner's manual for the printer.

Print Method

The *Print Method* options control how the document is printed on the page.

- *Truncate* clips objects that extend past the margins.
- *Fit to Page* reduces the size of the plot so that it fits within the specified margins. Margins are set and displayed or hidden with the Layout tab commands.
- *Tile breaks the drawing into page size pieces and generates multiple pages of output.*
- *View prints the current view in the plot window.*

Print Range

The Print range options control how the worksheet pages are printed. *All* prints all the pages that contain data. *Selection* prints the selected worksheet cells.

Copies

Specify the number of copies to print in the *Number of copies* box. If two or more copies of multiple page documents are printed, check the *Collate* box to separate the copies into packets. Note that some printers do not allow multiple copies.

Collate

Check the *Collate* check box to collate the pages when printing multiple copies. Clear the *Collate* check box to repeat duplicates in the print order. For example assume two copies of a three-page document. When Collate is selected, the print order is p1, p2, p3, p1, p2, p3. When Collate is not selected, the print order is p1, p2, p2, p3, p3.

Overlap

Each page overlaps adjacent pages by the amount specified in the *Horizontal* and *Vertical* overlap fields. These options are only available when the *Print Method* is set to *Tile* . The *Overlap* is the amount of the map that is printed on both pages.

Scale

Scale is used when the *Print Method* is set to *Truncate* or *Tile* to reduce or increase the overall size of the drawing. A scale of 100% is actual size, 200% is twice as large, and 50% is half as large.

OK and Cancel

Click *OK* to start the print. Click *Cancel* to close the **Print** dialog without printing the plot document.

Page Setup - Worksheet

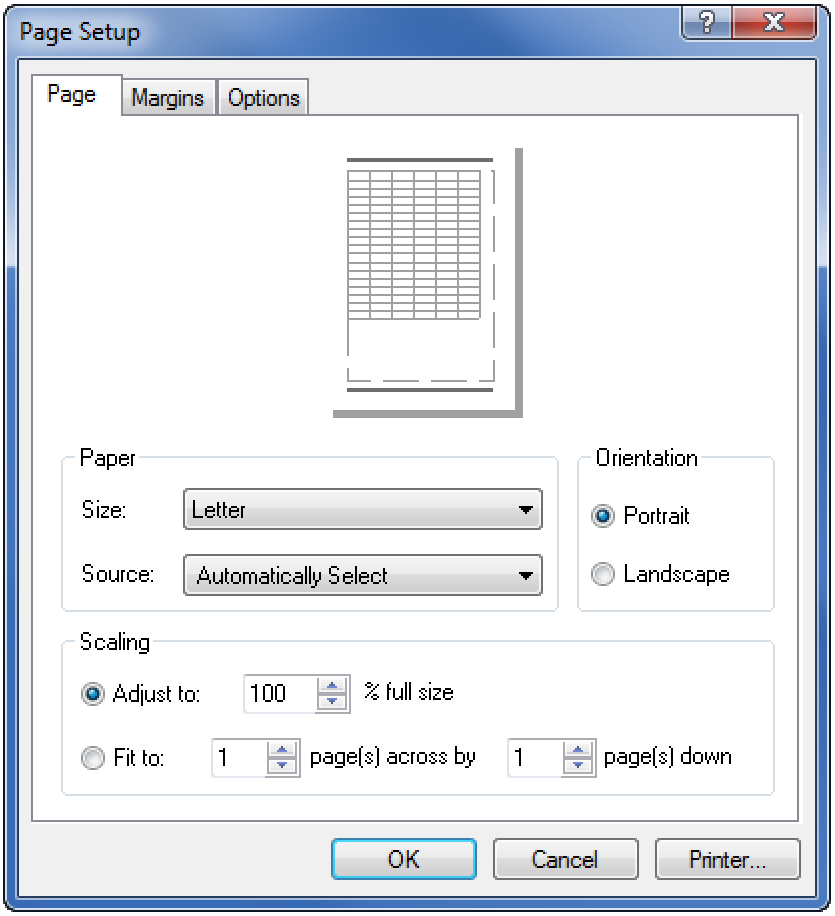
Before printing the worksheet, the page format of the worksheet can be set by clicking **File | Page Setup** or the button. The **Page Setup** dialog controls items such as printing headers and footers, centering the data on the page, showing grid lines, etc. There are three pages in the **Page Setup** dialog: **Page**, **Margins**, and **Options**. There is a *Printer* button at the bottom of the dialog that allows you to set the default printer regardless of which page is selected.

The worksheet **Page Setup** dialog has three pages:

Page	Set page size and scaling.
Margins	Set page margins, header and footer positions, and centering.
Options	Set grid lines, page order, and content of headers and footers.

Page Setup (Worksheet) - Page

Click on the **Page** tab in the **Page Setup** dialog to set page size and scaling. Click the File | Page Setup command in the worksheet to open the **Page Setup** dialog.



*Change page setup properties in the **Page Setup** dialog **Page** page.*

Paper

Use the *Paper* section to choose the paper *Size* and *Source* for the active printer.

Size

Click the down arrow next to the paper *Size* to change the size of the paper. The paper size options available for your printer are listed in the list.

Source

If your printer has multiple print trays, choose the paper *Source* by clicking the down arrow.

Orientation

The *Orientation* section controls whether the page is set to *Portrait* or *Landscape* mode. Select *Portrait* to have a vertical page. Select *Landscape* to have a horizontal page.

Scaling

The *Scaling* section controls the printed size of the worksheet. There are two options with *Scaling: Adjust to* and *Fit to* .

Adjust To

The *Adjust to ___ % full size* option sets the percent of full size that the worksheet will print. The arrow buttons are used to scroll up or down from 100% (full size), or values can be typed into the box. The *Adjust to ___ % full size* setting is independent of the *Fit to* option.

Fit To

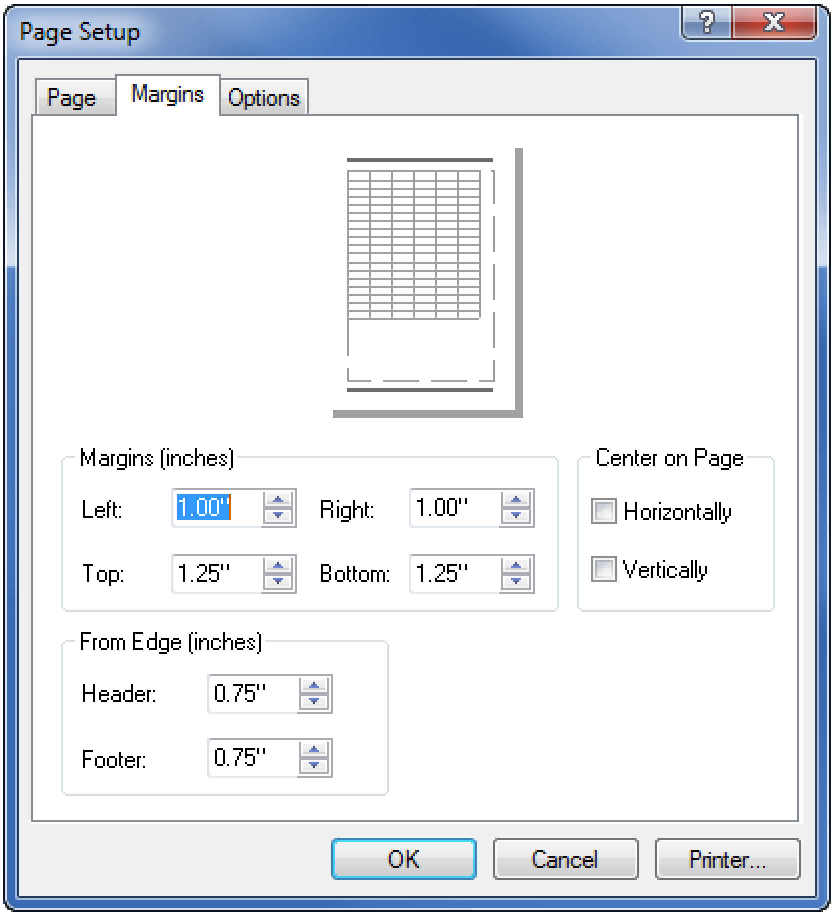
The amount of data in the worksheet determines how many pages are required to print the worksheet. The *Fit to __ page(s) across by __* option tells the program to print the worksheet at 100% scale or less. This option does not automatically scale the printed worksheet greater than 100%. This option is most useful when the worksheet is large and the number of printed pages needs to be limited.

Printer

The active printer can be changed by clicking the *Printer* button at the bottom of the **Page Setup** dialog.

Page Setup (Worksheet) - Margins

Use the **Margins** page in the **Page Setup** dialog to set page margins, header and footer positions, and centering. Click the File | Page Setup command in the worksheet to open the **Page Setup** dialog.



*Change page margin properties on the **Page Setup** dialog **Margins** page.*

Margins

Use the *Margins (inches)* group to set the page margins for all sides of the printed page. Set the *Left*, *Right*, *Top*, and *Bottom* values in inches to any limits the printer will allow. The margins are for the worksheet printout and are independent of the settings used for *Headers* or *Footers*.

Center on Page

The *Center on Page* group options automatically center the printout *Horizontally*, *Vertically*, or both. If neither option is selected, the worksheet prints in the upper left corner of the page.

From Edge (inches)

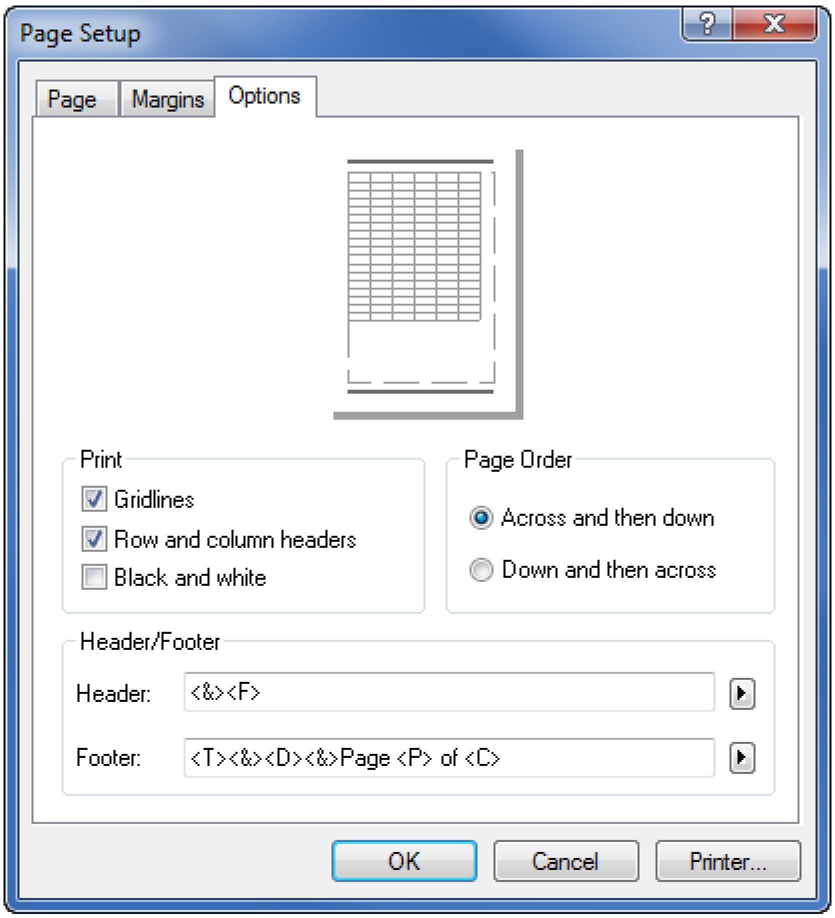
Headers and *Footers* are the only items that print outside of the margins. The *From Edge (inches)* group options controls how far the *Header* or *Footer* is printed from the edge of the page. If these values are greater than the *Top* or *Bottom* margins, it is possible that the worksheet data can print over the header or footer. The text that is printed for the header and footer is controlled on the Options page.

Printer

The active printer can be changed by clicking the *Printer* button at the bottom of the *Page Setup* dialog.

Page Setup (Worksheet) - Options

Use the **Options** page in the worksheet **Page Setup** dialog to set grid lines, page order, and content of the header and footer. Click the File | Page Setup command in the worksheet to open the **Page Setup** dialog. Click on the **Options** tab at the top of the dialog.



*Change page option properties in the **Page Setup** dialog **Options** page.*

Print

The *Print* section controls how the worksheet information is printed.

Gridlines

Check the *Gridlines* option to draw grid lines separating each column and row.

Row and Column Headers

Check the *Row and column headers* option to print the column letters and row numbers of the worksheet.

Black and White

If cells contain color backgrounds (set from the Data | Format | Format Cells command), check the *Black and white* option to print the worksheet in only black and white.

Page Order

The *Page Order* section controls the order in which multiple pages are printed. The *Across and then down* option prints from left to right first, and then moves down and prints left to right again. The *Down and then across* option prints the worksheet from top to bottom first, and then moves to the right and prints top to bottom again.

Header/Footer

The *Header/Footer* group controls the type of information included in the worksheet data print out. The plot window does not have header/footer options. The *Header* appears at the top of the page, and the *Footer* appears at the bottom of the page. The header and footer are spaced from the edge of the page based on the *From Edge* option o, the Margins page. Descriptive text can be typed in the *Header* and *Footer* boxes, or click the arrows to the right of the boxes and click the items in the list.

Automatic header/footer codes:

- *File Name* (<F) prints the name of the active file. The drive and path are not included.
- *Page Number* (<P>) prints the page number for each page. When several pages are printed, the order of printing is controlled from the *Page Order* section.
- *Total Page Count* (<C>) prints the total number of pages that are required to print out the worksheet with the specified scaling parameters.
- *Current Date* (<D>) prints the current date.
- *Current Time* (<T) prints the current time.
- *Left/Center/Right Separator* (<&>) separates the header and footer text so it is spread out across the page. Too many separators can actually push text off the page. If this happens, remove the <&> separator, and use spaces instead.

Printer

The active printer can be changed by clicking the *Printer* button at the bottom of the worksheet **Page Setup** dialog.

Examples

For a six page document, <&><&>Page <P> of <C> would print (on the right side of the first page):

```
                              Page 1 of 6
```

Enter Joe Smith<&><FD to print out a name, file name, and date:

```
 Joe Smith COLORADO.DAT 01/05/10
```

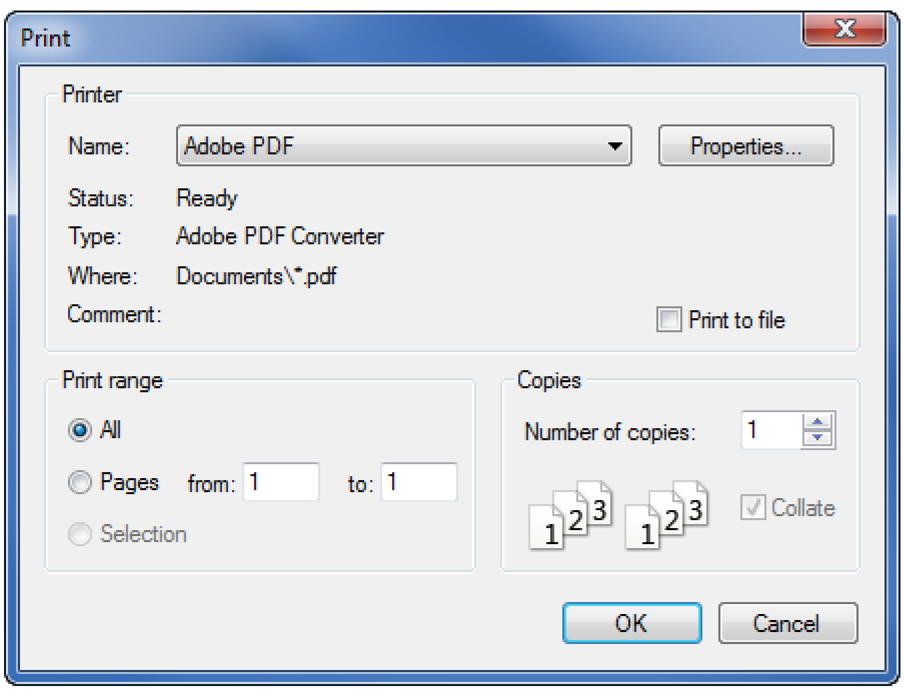
To print centered text use the "&" operator one time, such as <&><F.

```
            FILENAME.DAT
```

Print - Worksheet

Click the **File | Print** command or the button to print the contents of the worksheet to the active printer or to a .PRN file. To control the display of data on the printed page, refer to the [File | Page Setup](#) command. While the worksheet is spooling, a dialog indicates that printing is progressing.

The **File | Print** command in the worksheet opens the **Print** dialog.



*Use the **Print** dialog to specify the printing options.*

Printer

The *Printer* section contains options to specify the printer.

- The default system printer is listed in the *Name* field. If more than one printer is installed on the computer, use the down arrow to the right of the printer name to select a different printer.

- The printer *Status, Type, Where,* and *Comment* are listed below the printer *Name*.

- Click the *Properties* button to set printer specific properties. For information on specific printer settings, see the owner's manual for the printer.
- The *Print to file* check box allows you to print the data to a .PRN file. .PRN files are ASCII text files. When this option is checked, click *OK* in the **Print** dialog after setting other printing options. The **Print to File** dialog will open. Enter a path and file name in the **Print to File** dialog, and click *Save* .

Print Range

The *Print range* options control how the worksheet pages are printed.

- *All* prints all the pages that contain data.
- *Pages* prints the pages specified. Enter the starting page in the *from* box. Enter the ending page in the *to* box.
- *Selection* prints the selected worksheet cells.

Number of Copies

Specify the number of copies to print in the *Number of copies* box. If two or more copies of multiple page documents are printed, check the *Collate* box to separate the copies into packets. Note that some printers do not allow multiple copies.

Collate

Check the *Collate* box to collate the pages when multiple page documents are printed two or more times.

OK or Cancel

Click *OK* to start the print. Click *Cancel* to abort the print and return to the worksheet window.

Chapter 33 - Options, Defaults, and Customizations

Options

The default settings of **Surfer** and advanced options are customized by clicking the **File | Options** command or the button. By adjusting the settings in the **Options** dialog, you can customize **Surfer** to suit your individual preferences and work habits. The settings are automatically saved and restored whenever **Surfer** is restarted. Changes made in the **Options** dialog affect all subsequent documents.

The **Options** dialog contains the following pages:

General	Set the basic window features such as what type of map will be opened with grid and data files, file open/save paths, significant digits, and undo levels.
Updates	Allow **Surfer** to automatically check for updates.
User Interface	Set the interface and tab style, help in the **Properties** window, and the startup splash screen.
Selection	Set the selection handle size and tolerance size for selecting objects.
Rendering	Set antialiasing properties.
Printing	Controls printer page size settings.
Rulers and Grid	Control the display of the rulers and grid.
3D View	Set the time period for the timing of updates in the 3D View and for the maximum number of points in point cloud layers.
Default Properties	Set line, fill, symbol, font, and label format default attributes.

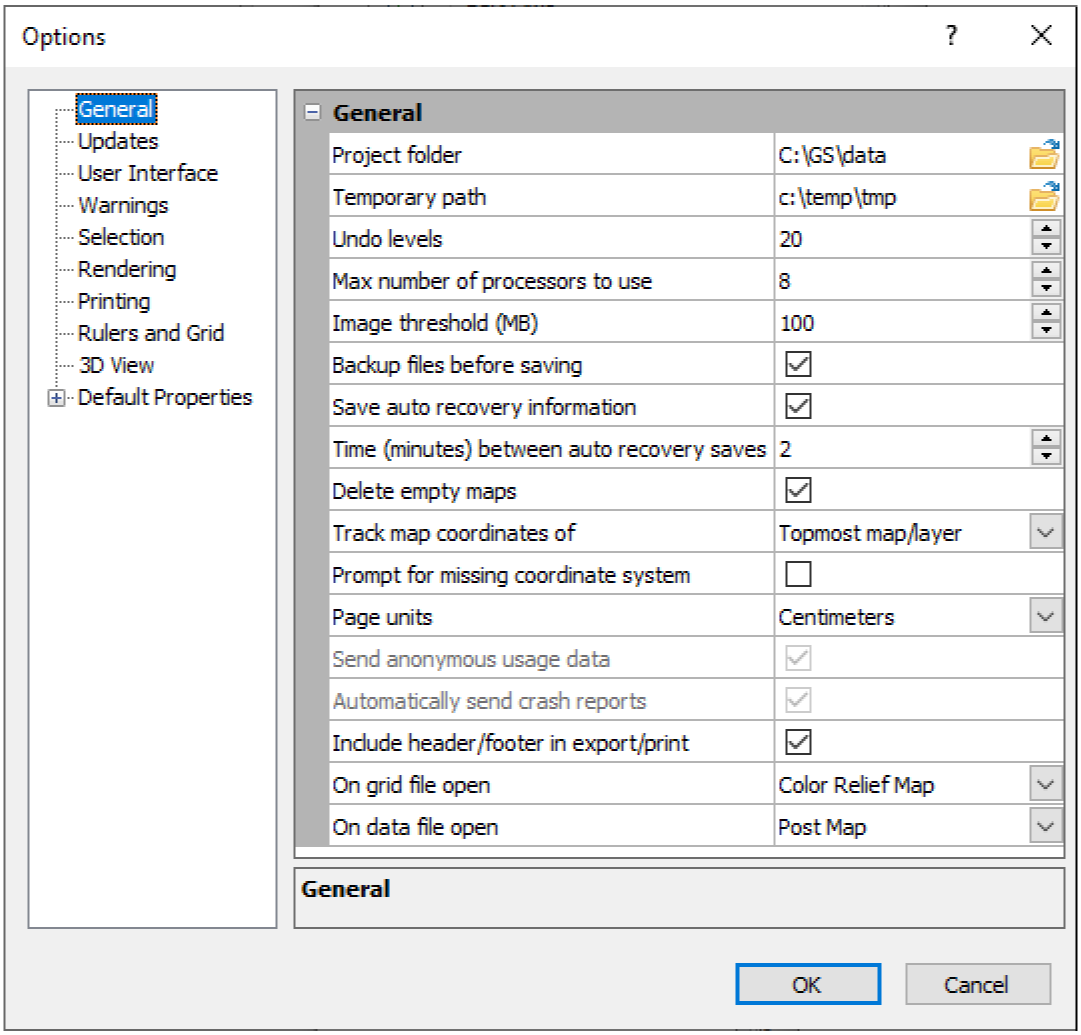
Advanced default options are available for most menu commands by clicking the File | Defaults command.

Options - General

Set defaults (e.g. open/save paths and undo levels) on the **General** page in the Options dialog.

Options Dialog

Click the **File | Options** command or the button to open the **Options** dialog. Click on the *General* option on the left side of the dialog to open the **General** page.



Customize General *options in the* ***Options*** *dialog.*

Project Folder

Enter a path into the *Project folder* box to set the default path for opening and saving files. Alternatively, click the button to browse for a path. This option sets the initial directory displayed in all the open, import, save as, and export dialogs. **Surfer** must be closed and reopened for the *Project folder* change to take effect. This option can also be set with the *Set Project Folder* button in the Welcome to Surfer dialog. If the *Project folder* is set in the **Welcome to Surfer** dialog, it is not necessary to restart **Surfer**.

To set the *Project folder* to the Windows default path, delete the current path and leave the option blank. Click *OK* , and the default Windows path will be used for the *Project folder*.

Temporary Path

Enter a path into the *Temporary path* box to set the default path for temporary files. Alternatively, click the button to browse for a path. To set this to the default path, delete the current path and leave the option blank. After *OK* is clicked, the default Windows path will be written to the *Temporary path*.

Undo Levels

Set the number of commands to undo in the *Undo levels* box. The maximum number of Undo levels is 100. Once the maximum number of actions has been performed, the oldest action is dropped off the list as new actions are added. Undo can consume significant amounts of memory, so this option should probably be left between 3 and 10 if memory is at a premium. Set the undo levels to 0 to disable **Undo** completely. Use the to increase or decrease the number of undo levels or highlight the existing value and type a new value between zero and 100.

Maximum Number of Processors to Use

The *Max number of processors to use* option sets the maximum number of processors **Surfer** can concurrently use during multi-threaded operations. The minimum value is 1 and the maximum value is the number of physical processors on the PC. The *Max number of processors to use* is set to the maximum value by default. Reduce the *Max number of processors to use* value if you wish to reserve some of your PC's processing power for other applications during processor-intensive processes, such as gridding.

Image Threshold (MB)

The *Image threshold (MB)* option controls the number of megabytes that can be used before switching images to disk-based images. This option can be set to a value between 0 and 16384. If an imported image is larger than the image threshold, it will be stored in a tiled bitmap format which uses minimal internal memory but can result in some performance degradation. Increase this value if your computer has a lot of internal RAM. Lower this value if you are experiencing very sluggish performance, "Out of memory" errors, or crashes when using large images. Any images already imported must be re-imported to change their internal storage format. Use the to increase or decrease the number of megabytes used or highlight the existing value and type a new value between zero and 16384.

Backup Files Before Saving

If *Backup files before saving* is checked, a backup copy of an existing file is created before saving the document. The backup copy is saved with a .BAK extension. If an identically named backup file already exists, it will be overwritten.

Save Auto Recovery Information

Select the *Save auto recovery information* option to periodically save changes to a temporary file that is automatically restored if **Surfer** closes unexpectedly. Large files may take a moment to save and can result in periodic slowdowns while using **Surfer** with this option enabled.

Auto Recover Save Interval

Specify the number of minutes between auto-recovery saves in the *Auto recover save interval* option. The interval can be a number between 1 and 30 minutes.

Delete Empty Maps

Check the *Delete empty maps* option to delete empty map frames after combining the map layers with other maps. When this option is not checked, the empty map object and axes are still drawn after the map layers are moved. By default, **Surfer** checks this box so that empty maps are automatically deleted.

Track Map Coordinates

The *Track map coordinates of* option controls which map is used to display the XYZ coordinates and units in the status bar and which map is used when the Track Cursor option is enabled. The options are *Topmost map/layer* or *Selected map/layer*. When the *Topmost map/layer* is chosen, the coordinates and units being shown in the status bar and by the **Map Tools | Layer Tools | Track Cursor** command or the button are those for the top layer of the map at the top of the **Contents** window. When *Selected map/layer* is chosen, the coordinates and units being shown are those for the selected map or layer.

Prompt For Missing Coordinate System

The *Prompt for missing coordinate system* option controls whether **Surfer** prompts for a coordinate system if an unreferenced data file, grid, or base map is loaded. Check this option to enable the prompt for a coordinate system. Uncheck this option to disable the dialog prompt when loading data, grids, and base maps that are unreferenced.

Regardless of whether the *Prompt for missing coordinate system* is checked or not, loading a file with the **File | Reload Map Data** command or by clicking the button in the **Properties** window for the map layer will not prompt for the coordinate system. The coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. In the **Properties** window, click on the **Coordinate System** tab and set the coordinate system to the appropriate new system.

Page Units

Page Units are the units used to measure distances on the printed page. Set *Page Units* to *Inches* or *Centimeters*. The location of the cursor on the page is listed in page units in the status bar. The page units are displayed on the Rulers.

Track Usage

Check the box next to *Track usage* to allow **Surfer** to send anonymous usage data to Golden Software. Sending usage data will help improve **Surfer** to meet our users' needs. This property does not have a default setting; its initial setting is determined based on your selection to opt-in or opt-out of the Customer Experience Improvement Program during the **Surfer** install process. **Surfer** must be restarted for this setting to be applied. A message is displayed reminding you to restart **Surfer** after changing this setting.

Include Header/Footer in Export and Print

Check the *Include header/footer in export/print* check box to include the header/footer when printing or exporting the plot window. By default the header and footer are not included in printed and exported output. Clear the *Include header/footer in export print* check box to only display the header/footer in the plot window.

On Grid File Open

The *On Grid File Open* option controls the map type that is opened in the plot window when a grid file is selected by clicking the **File | Open** command. The default is set to the Color Relief Map. Click the ▾ to see the list of seven different map types and click on the map type to change the default and then click *OK*.

On Data File Open

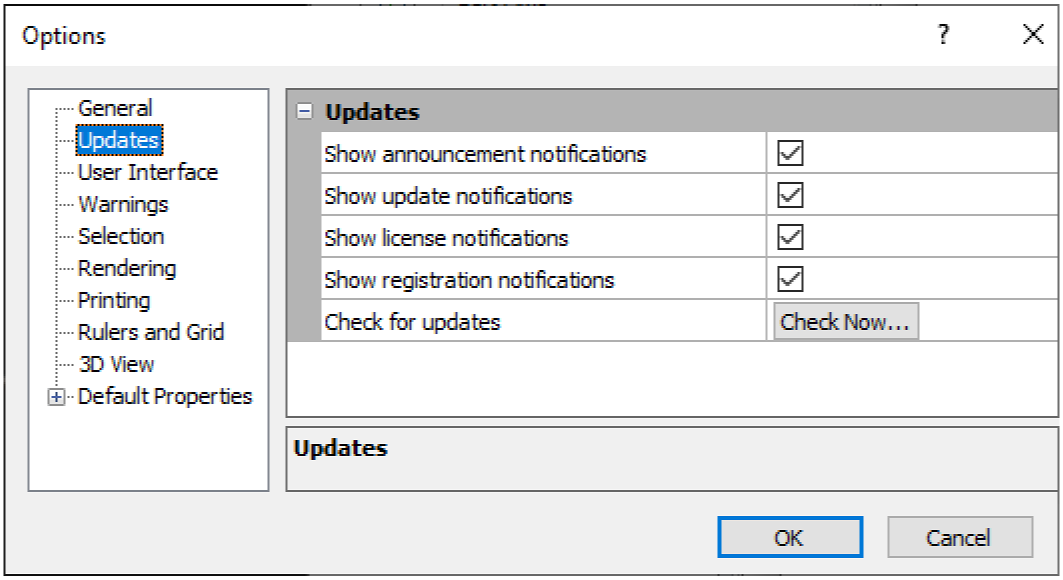
The *On Data File Open* option controls what file type opens when a data file is selected by clicking the **File | Open** command. Either one of three map types opens in a plot window or a worksheet editor is opened. The default is set to the Post Map and can be changed by clicking the drop down arrow ▾. Click on a file type and then click *OK*.

Options - Updates

Set automatic update preferences on the **Updates** page in the Options dialog.

Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on *Updates* on the left side of the dialog to open the **Updates** page.



Customize the Updates *options in the* ***Options*** *dialog.*

Show Announcement Notifications

Check the *Show announcement notifications* to allow Golden Software to automatically check for program and company announcements. This could include information about a new product release, tips for use of Golden Software programs, special offers, or an update to the program.

Show Update Notifications

Check *Show update notifications* to allow Golden Software to automatically check for program updates. The program will link to the Golden Software server to see if any program updates are available. Program updates include fixes to errors or problems that are found in the program. It is recommended that you keep this box checked so that your version of **Surfer** is always up to date.

Show License Notifications

Check *Show license notifications* to display messages regarding the status of your software licenses. For example, you might see a message notifying you that your software maintenance is about to expire or if a trial version you are using is about to expire.

Show Registration Notifications

Check *Show registration notifications* to display messages about registering your software product key. You may want to register the software under your name so you can manage the product license and receive important information and updates. See Register Product Key.

Check For Updates

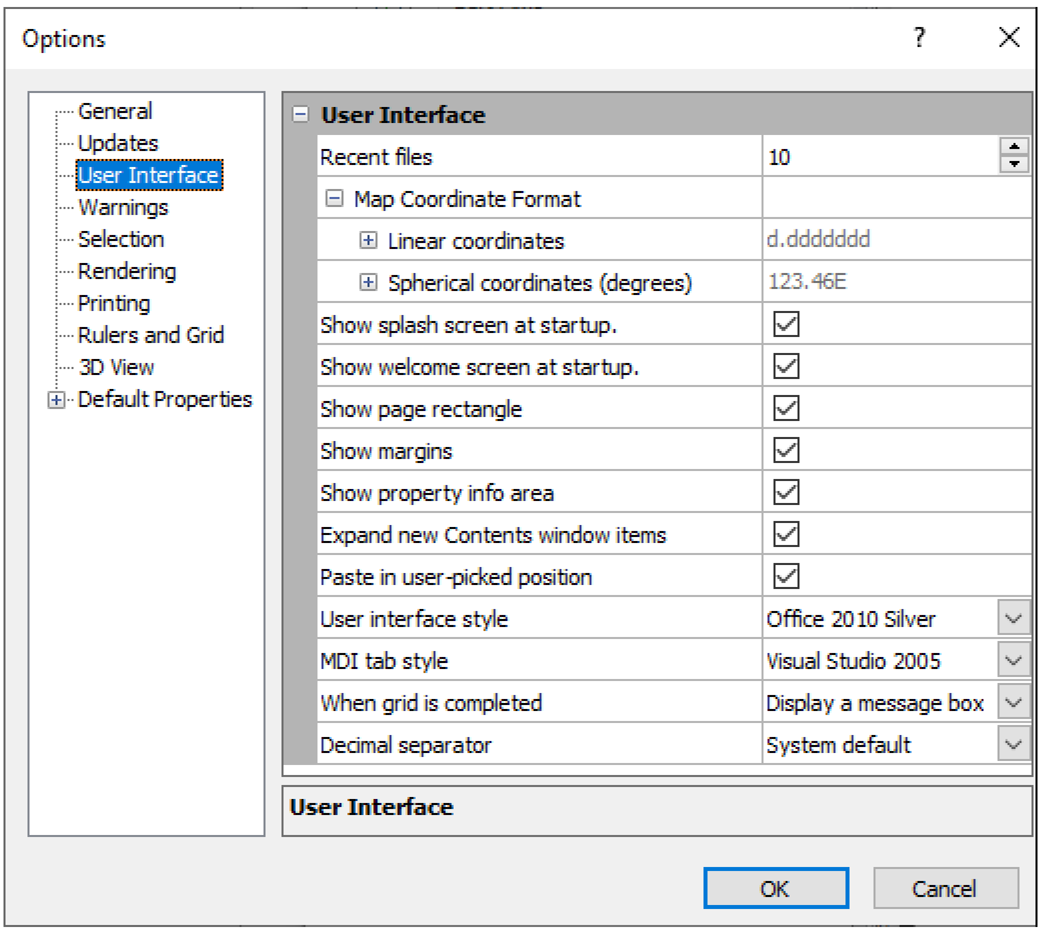
Click the Check Now... next to the *Check for updates* command to check for program updates. Before using this command, make sure your computer is connected to the Internet. Follow the directions in the **Internet Update** dialog to complete the update if an update is available.

Options - User Interface

Set the user interface environment on the **User Interface** page in the Options dialog.

The Options Dialog

Click the **File | Options** command or the button to open the **Options** dialog. Click on the *User Interface* option on the left side of the dialog to open the **User Interface** page.



Customize the User Interface *options in the* ***Options*** *dialog.*

Recent Files

The *Recent files* option controls the number of recently used files listed near the bottom of the **File** menu. The default value is 10. This option can be set to a value between 0 and 16.

Map Coordinate Format

Set the format for the map coordinates section of the status bar in the *Map coordinate format* section. The status bar displays the cursor location coordinates of the tracked map or layer. Select which map or layer is tracked in the **Options** dialog General page. Click the to the left of *Map coordinate format* to expand the format options. The *Linear coordinates* section controls the coordinate format when displaying linear units in the status bar, i.e. when tracking the cursor location on a projected or unreferenced map/layer. The *Spherical coordinates (degrees)* section controls the coordinate format when displaying spherical units in the status bar, i.e. when tracking the cursor location on an unprojected latitude/longitude map or layer. The format options for both the *Linear coordinates* and *Spherical coordinates (degrees)* are the standard label format properties. Using *DMS (Lat/long)* is NOT recommended for the *Linear coordinates* selection.

Show Splash Screen at Startup

Check the box next to *Show splash screen at startup* to allow **Surfer** to display the **Surfer** splash screen and copyright notice while the program is loading. Uncheck this box to disable the splash screen display.

Show Welcome Screen at Startup

Check the box next to *Show welcome screen at startup* to show the Welcome to Surfer dialog when **Surfer** starts. Uncheck this box to disable the **Welcome to Surfer** dialog.

Show Page Rectangle

Check the box next to the *Show page rectangle* option to turn on the display of the paper representation in the plot window. Uncheck this box to turn off the display of the paper representation. The page size is set by clicking the File | Page Setup command.

Show Margins

Check the box next to the *Show margins* option to turn on the display of the page margin in the plot window. Uncheck this box to turn off the display of the page margin. Margins are set by clicking the File | Page Setup command.

Show Property Information Area

Check the box next to the *Show property info area* option to display a short help statement for each selected command in the Properties window. Uncheck this box to turn off the help area in the **Properties** window.

Always Show Drop Arrows in Properties Window

Check the box next to *Always show drop arrows in Properties window* to display drop arrows for all combo-box properties in the **Properties** window. When this option is active, fields containing lists can be easily differentiated from fields containing text or value input boxes. When the *Always show drop arrows in Properties window* option is unchecked the drop arrow is only visible when the property is selected. Turn this option off when the **Properties** window is narrow, and more of the properties' text will be shown.

Expand New Contents Window Items

Check the box next to the *Expand new Contents window items* option to force the Contents window to expand all new object hierarchies in the tree control. If this option is turned off, the objects contain a ⊞ button. Click the ⊞ to manually expand additional levels.

Paste in User-picked Position

Check the box next to *Paste in user-picked position* to manually position objects with the cursor when using the Paste command. When the *Paste in user-picked position* check box is checked, a shaded bounding box is displayed around the cursor after **Paste** is used. Click in the desired location to paste the object or objects.

Clear the *Paste in user-picked position* check box to automatically paste objects in the center of the plot window. This is the default behavior.

User Interface Style

Specify the interface style in the *User interface style* list. Available options are *Windows XP, Office XP, Office 2000, Office 2003, Visual Studio 2005, Visual Studio 2008, Visual Studio 2010, Office 2010 Black, Office 2010 Blue* , and *Office 2010 Silver*. The worksheet window colors are dark when *Office 2010 Black* is selected, blue when *Office 2010 Blue* is selected, and silver when any other *User interface style* is selected. The default style is *Visual Studio 2008*.

MDI Tab Style

Specify the tabbed document view style in the *MDI tab style* list. The options are *None, 3D, One Note,* and *Visual Studio 2005.* The plot window, worksheet window, and grid node editor window have a tab appearance when multiple windows are open when the *MDI tab style* is set to any option other than *None*. The default style is *Visual Studio 2005*.

Notifications for Creating a Grid File

There are three notification options you can choose from in the *When grid is completed* list: *Do Nothing*, *Play a Sound*, or *Display a Message Box*. This action is performed after most gridding operations to inform you that the operation has completed.

Decimal Separator

The *Decimal separator* controls which character is used to separate the whole number portion from the decimal portion in a number. To change the *Decimal separator*, click on the existing option and select the desired option from the list. Available options are: *System default, Period,* and *Comma*. The default is to use the *System default*. *System default* defers treatment of decimal separators to Windows. The *Period* option displays a period (.) to separate the numbers before and after the decimal. The *Comma* option display a decimal comma (,) to separate the numbers before and after the decimal. When this option is changed, all maps that are opened will display the selected character for the decimal separator.

When using *System default*, the setting is controlled by Windows. To set the Windows local, in the Windows Control Panel, under *Region and Language*, set the *Format*. In locale's where the period is the separator, it will be used in **Surfer**. In locale's where the comma is the separator, it will be used in **Surfer**. All .SRF files will use the format specified by Windows. This means that older .SRF files may appear differently depending on the format. Some changes may be in axis labels.

The *Decimal separator* setting is mainly used to import data files from different locales correctly in your instance of **Surfer**. For example, a CSV file uses the period (.) for the decimal separator and a comma (,) for the data delimiter in the English locale. However a CSV file uses the comma (,) for the decimal separator and a semicolon (;) for the data delimiter in the German locale. You can correctly import data files from other locales by updating your *Decimal separator* setting. This means it is not necessary to update your Windows locale to import and open data files from other locales.

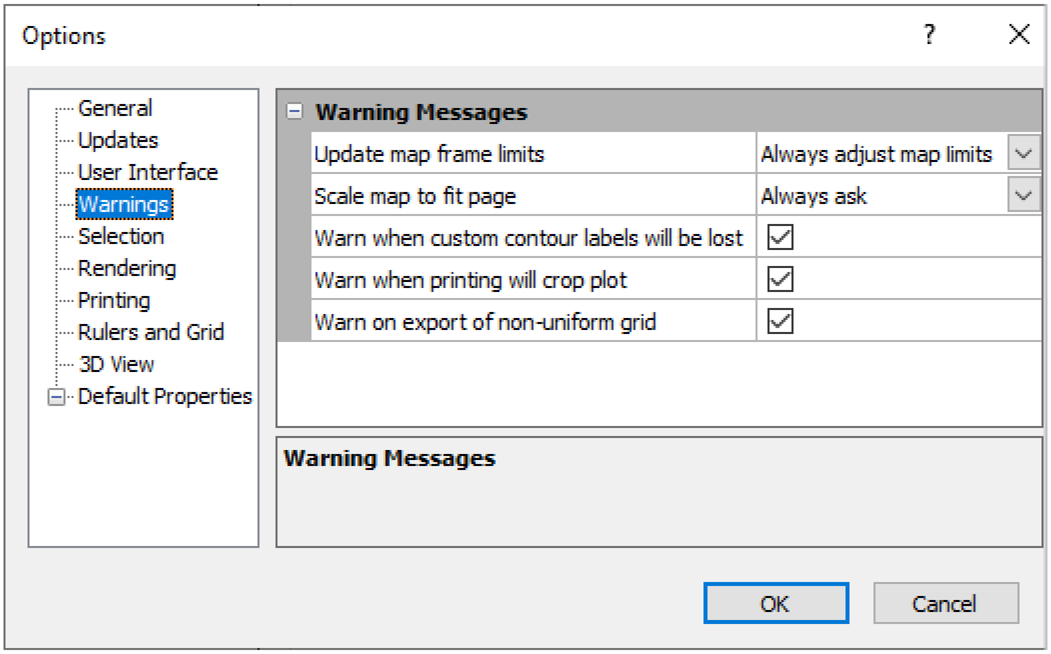
Options - Warnings

Set the options when specific warnings are displayed or suppressed on the **Warnings** page in the Options dialog. In addition to displaying or suppressing a warning, default actions can be defined for two of the warnings.

The Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *Warnings* option on the left side of the dialog to open the **Warning Messages** page.

Note: The warnings that have the option to be disabled also have a check box at the bottom of the warning dialog to no longer show the warning.

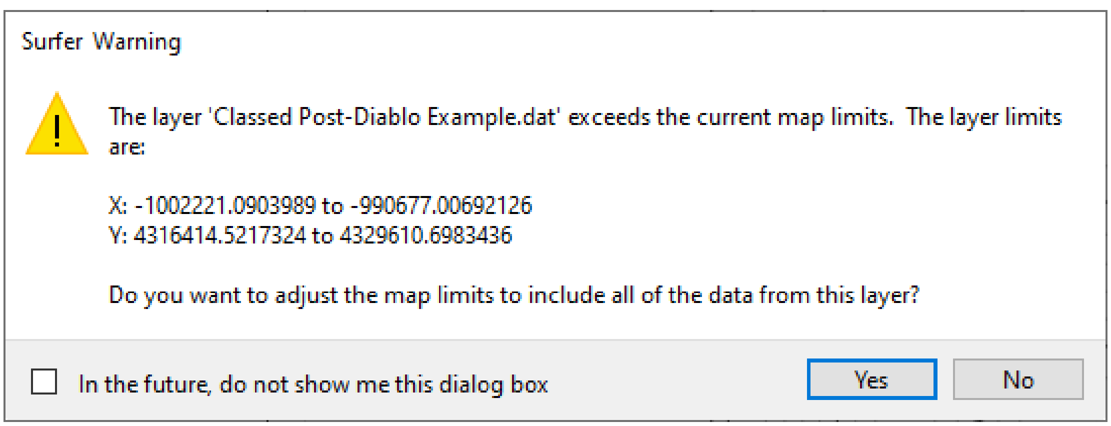


Customize the Warnings *options in the* ***Options*** *dialog.*

Update Map Frame Limits

The *Update map frame limits* option controls the warning that occurs when an action would increase the limits of the map, such as adding a layer with larger limits.

The following image is an example of this type of warning.



Update map frame limits example warning

Select one of the following options from the *Update map frame limits* list to control when this type of warning is displayed and to define the default action.

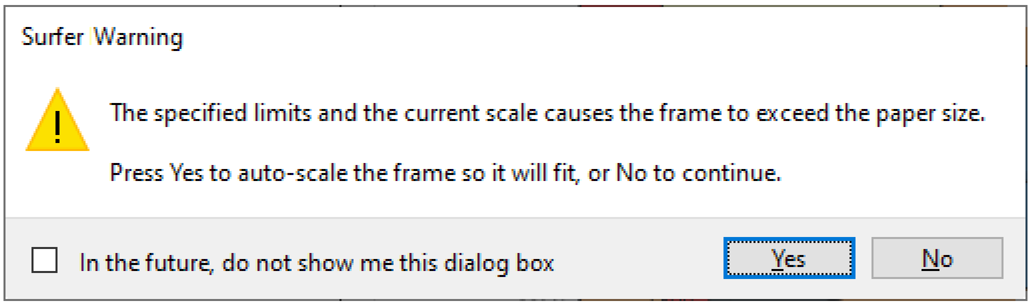
- Select *Always ask* to display the warning whenever an action increases the current map's limits.

- Select *Always adjust map limits* to suppress the warning and allow the change to the map limits.
- Select *Keep existing limits* to suppress the warning but not allow the change to the map limits.

Scale Map to Fit Page

The *Scale map to fit page* option controls the warning that occurs when a change to the limits of a map would make the map larger than the selected page paper size at the current scale.

The following image is an example of this type of warning.



Scale map to fit page example warning

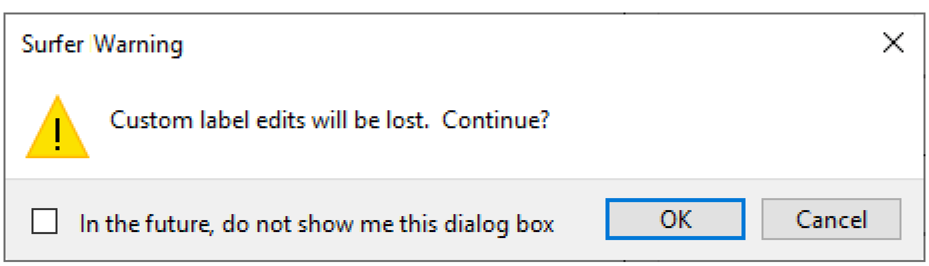
Select one of the following options from the *Scale map to fit page* list to control when this type of warning is displayed and to define the default action.

- Select *Always ask* to display the warning whenever an action increases the map size beyond the page size.
- Select *Always adjust map scale* to suppress the warning and change the map scale.
- Select *Keep map scale* to suppress the warning and not change the map scale.

Warn When Custom Contour Labels Will Be Lost

Check the *Warn when custom contour labels will be lost* check box to always display this warning or uncheck the check box to always suppress the warning. This warning occurs when a change to a map would eliminate custom labels, such as filling contours or creating a color scale bar.

The following image is an example of this type of warning.

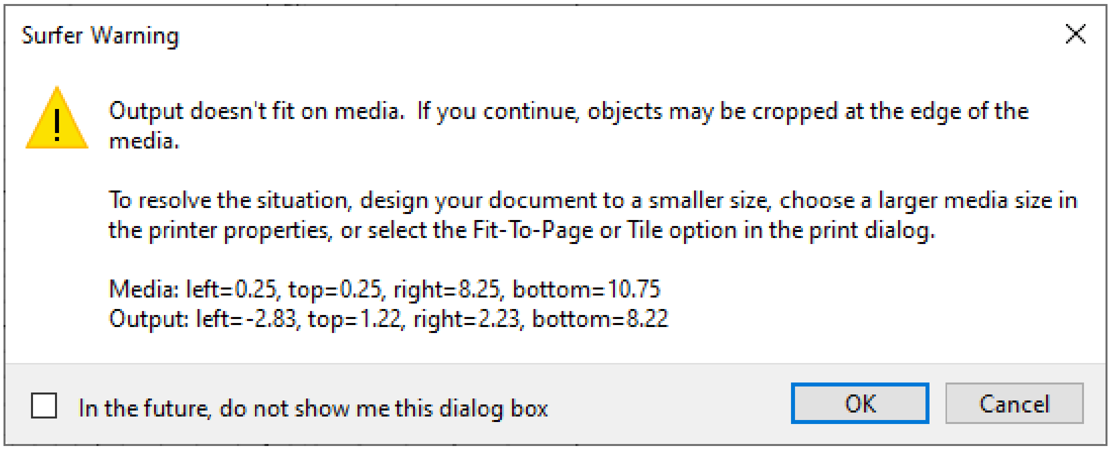


Warn when custom contour labels will be lost example warning

Warn When Printing Will Crop Map

Check the *Warn when printing will crop map* check box to always display this warning or uncheck the check box to always suppress the warning. This warning occurs when printing a map and part of the map will not be on the printed page.

The following image is an example of this type of warning.

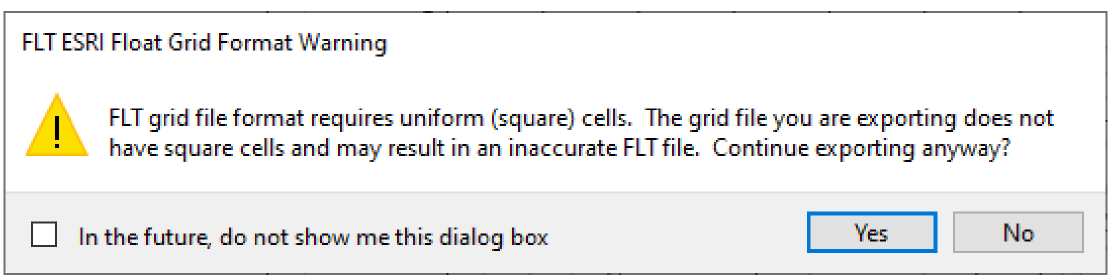


Warn when printing will crop map example warning

Warn on Export of Non-Uniform Grid

Check the *Warn on export of non-uniform grid* check box to always display this warning or uncheck the check box to always suppress the warning. This warning occurs when saving a grid with non-uniform grid cells to a format that requires uniform grid cells (e.g. Arc/Info ASCII Grid *.ASC or ESRI Float Grid *.FLT).

The following image is an example of this type of warning.



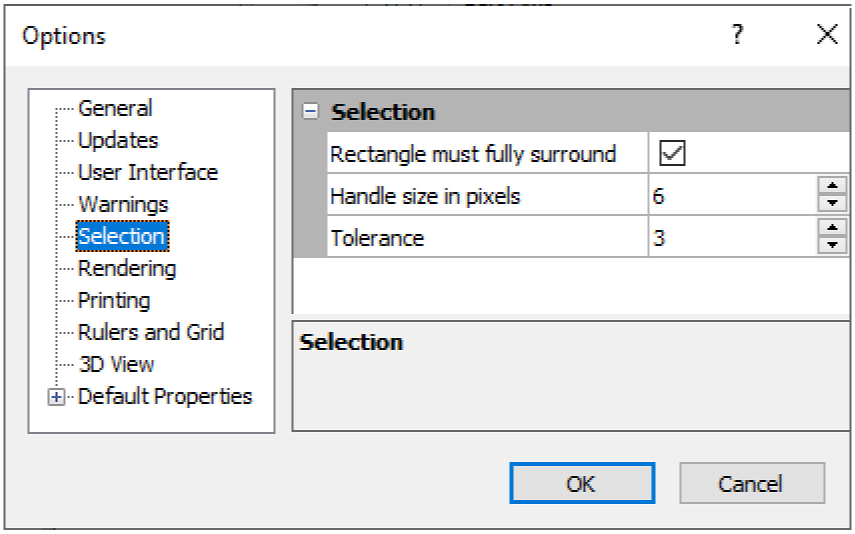
Warn on export of non-uniform grid example warning

Options - Selection

Set the options for how objects are selected on the **Selection** page in the Options dialog.

The Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *Selection* option on the left side of the dialog to open the **Selection** page.



Customize the Selection *options in the **Options** dialog.*

Rectangle Must Fully Surround

The *Rectangle must fully surround* option controls how the block select feature functions. If the box is checked, the block select rectangle must be drawn completely around the object to select it. If any portion of the object extends beyond the block select rectangle, the object is not selected. If this option is unchecked, the block select rectangle only needs to partially intersect an object to select it.

Handle Size

The *Handle size in pixels* option controls the width and height in pixels of the selection handles that appear around selected objects. The handle size setting range is from zero to 25 pixels. A setting of 25 will create large size selection handles. A setting of zero will create no selection handles. The default setting is a handle size of six pixels. The handle size scales with zoom. Click the to increase or decrease the value or highlight the existing value and type a new number to change the size.

Tolerance

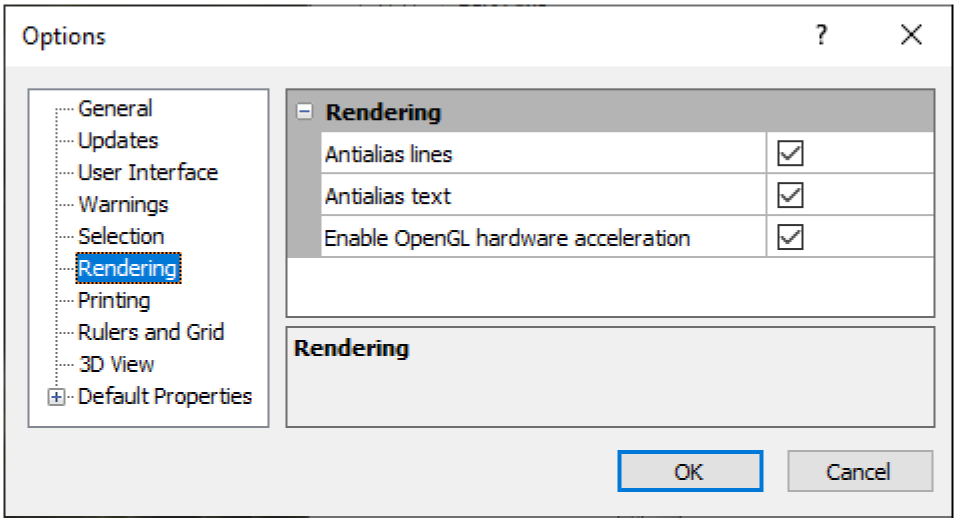
The *Tolerance* option controls how far away the cursor can be from an object when clicking to select the object. The tolerance setting range is from zero to 25 pixels. When the tolerance size is zero, the cursor must be directly on the object to select it. The tolerance size does not affect the size of the bounding box, it only controls the distance the cursor can be from the object to select it. The default tolerance setting is three pixels. The tolerance scales with zoom. Click the to increase or decrease the value or highlight the existing value and type a new number to change the tolerance.

Options - Rendering

Set the antialias options on the **Rendering** page in the Options dialog.

The Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *Rendering* option on the left side of the dialog to open the **Rendering** page.



Set the Rendering *options in the* ***Options*** *dialog.*

Antialiased Lines

Check the box next to *Antialias lines* to allow redraw to diminish jagged edges and create a smooth appearance for all lines in the plot window. Uncheck this box to disable the smoothing and make lines crisper.

Antialiased Text

Check the box next to *Antialias text* to allow redraw to diminish jagged edges and create a smooth appearance for all text and symbols in the plot window. Uncheck this box to disable the smoothing and make text and symbols crisper.

Enable OpenGL Hardware Acceleration

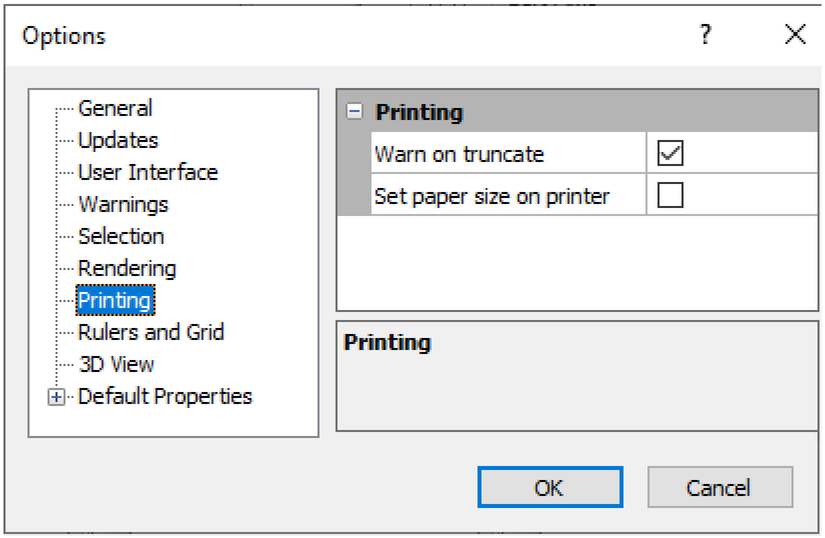
Check the box next to *Enable OpenGL hardware acceleration* to use the OpenGL driver included with your video card in the 2D plot view. If you experience any difficulties when creating 3D surface maps or see errors or crashes when displaying the 3D surface map, uncheck the box. Surfer must be restarted after making this change. This option does not affect the display or rendering in the 3D view.

Options - Printing

Set printing page size defaults (i.e. warning of invalid page sizes and controlling the page size based on the selected printer) on the **Printing** page in the Options dialog.

The Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *Printing* option on the left side of the dialog to open the **Printing** page.



Customize Printing *options in the* ***Options*** *dialog.*

Warn on Truncate

The *Warn on truncate* option controls if a warning message appears when printing if objects are outside the page margin. By default this option is checked. When the box is checked, if the map or any drawn objects are outside the page margins, a warning message appears. Click *OK* on the warning and reset the page margins or page size using the File | Page Setup command. When the *Warn on truncate* option is unchecked, if the map or any drawn objects are located outside the page margins, no warning message appears. The print is truncated to the page margins.

Set Paper Size on Printer

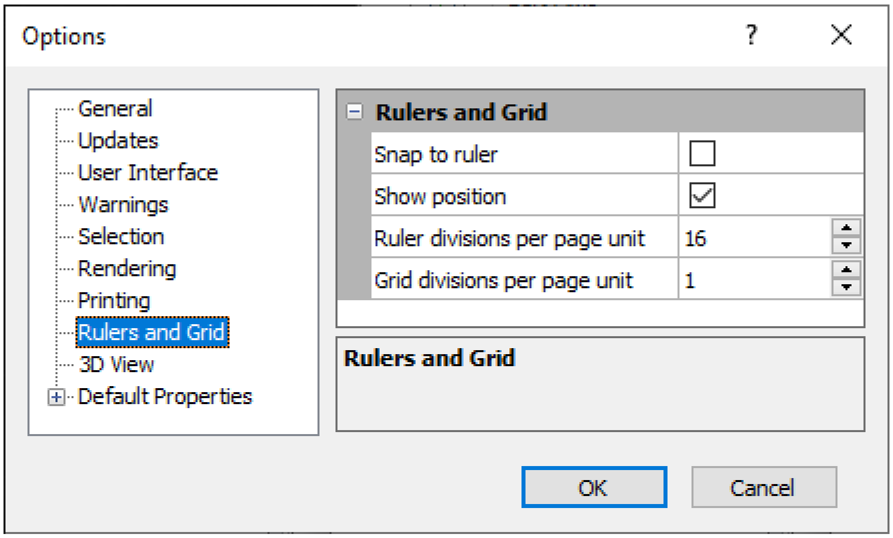
The *Set paper size on printer* option controls the page sizes available for printing. By default, this option is unchecked. When unchecked, the paper size on the **Page Setup** dialog is not automatically sent to the printer. The printer paper size will need to be set separately in the File | Print dialog. When the *Set paper size on printer* option is checked, **Surfer** automatically uses the paper size set in the **Page Setup** dialog for the printer paper size. If the **Page Setup** dialog specifies a page size that is not supported by the printer, no page size is transferred to the printer and the page size must still be specified in the **Print** dialog.

Options - Rulers and Grid

Set ruler and grid properties on the **Rulers and Grid** page in the Options dialog. Rulers appear along the top and left edge of the plot window, and are used to position and align objects. Rulers use the current page units. To change the *Page units*, click on the **General** page.

The Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *Rulers and Grid* option on the left side of the dialog to open the **Rulers and Grid** page.



Specify Rulers and Grid *options in the* ***Options*** *dialog.*

Snap to Ruler

Snap to ruler causes the cursor to snap to the ruler divisions as objects are drawn or moved. This allows objects to be easily aligned with the ruler division marks. Check this box to snap the cursor to the ruler divisions. Uncheck this box to disable the automatic snapping.

Show Position

When the *Show position* option is checked, the current cursor position is displayed on the rulers. As the cursor is moved, the position indicator moves within the rulers to show the exact page position of the cursor. Uncheck this box to disable the cursor position in the rulers.

Ruler Divisions Per Page Unit

The *Ruler divisions per page unit* option controls the number of divisions per page unit on the ruler. Select a value between 1and 99. To change the value, highlight the existing value and type a new number or click the to increase or decrease the number of divisions. The number of divisions are in page units. Set the page units on the General page of the **Options** dialog.

Grid Divisions Per Page Unit

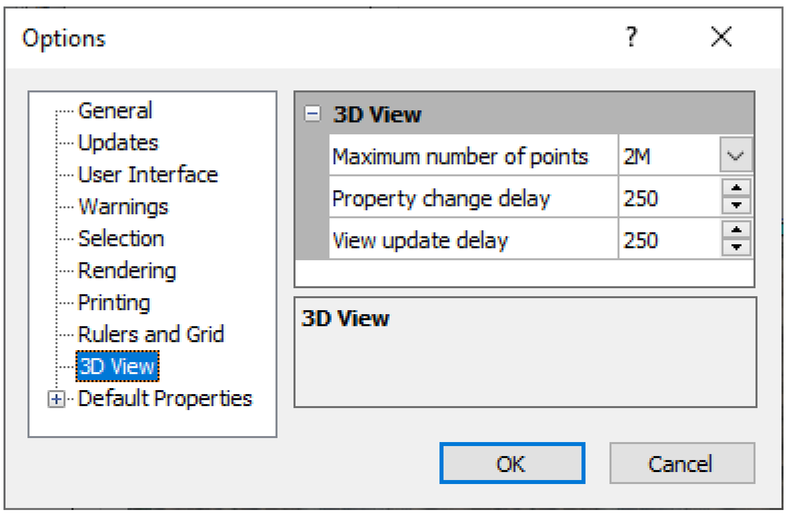
The *Grid divisions per page unit* option controls the number of divisions per page unit on the grid. Select a value between 1 and 99. To change the value, highlight the existing value and type a new number or click the to increase or decrease the number of divisions. The number of divisions are in page units. Set the page units on the General page of the **Options** dialog.

Options - 3D View

Set the options for the timing of updates in the 3D View and for the maximum number of points in point cloud layers.

The 3D View Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *3D View* option on the left side of the dialog to open the **3D View** page.



Customize the 3D View *options in the* ***Options*** *dialog.*

Maximum Number of Points

Large point cloud layers can use a lot of resources to render in the 3D view. The number of points displayed in the 3D view is limited by the *Maximum number of points* option. If **Surfer** is not responsive or slow to respond when viewing a point cloud in the 3D view, reduce the *Maximum number of points* option. The default is 2 million (2M) points. Select 1M, 2M, 4M, 8M, 12M, or 16M from the list. Any open 3D view windows must be closed and reopened after changing the *Maximum number of points* value in the **Options** dialog to apply the change.

Property Change Delay

The *Property change delay* option sets the length of time in milliseconds of no user input before applying all property changes in the 3D view. You may want to increase the value of this option if you are changing multiple property values and are experiencing lags in responsiveness.

View Update Delay

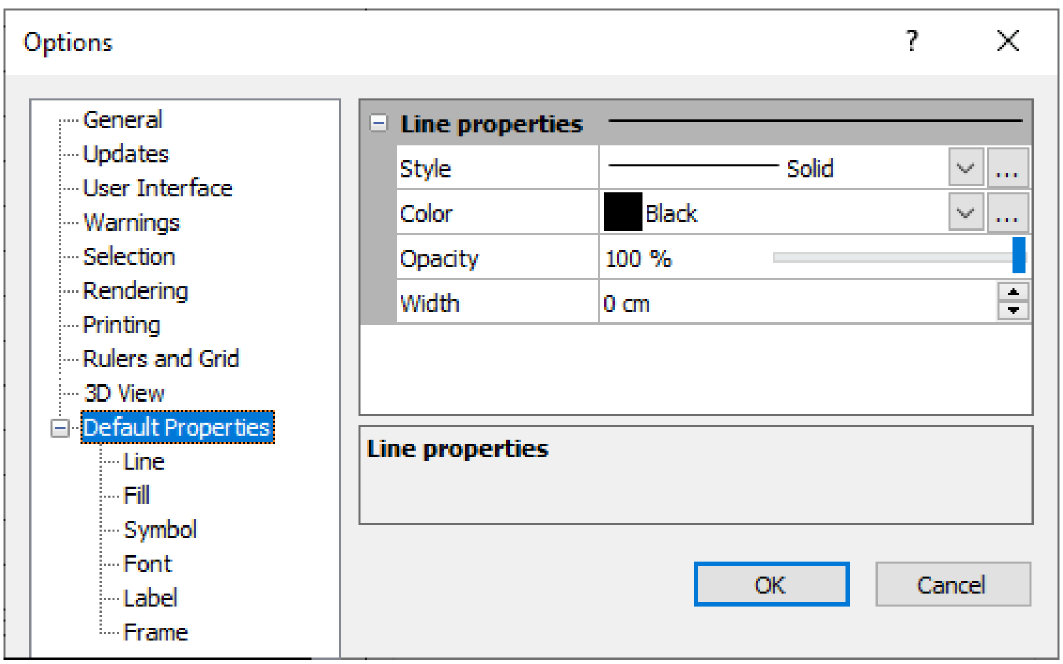
The *View update delay* option sets the length of time in milliseconds of no user input before re-rendering the 3D View. You may want to increase the value of this option if you are making frequent changes and are experiencing lags in responsiveness.

Options - Default Properties

Use the **Default Properties** pages in the Options dialog to specify the default line, fill, symbol, font, label, and frame properties used when creating new objects.

The Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *Line, Fill, Symbol, Font, Label* or *Frame* link under the *Default Properties* option on the left side of the dialog to open the appropriate default setting page.



Customize the Default Properties *in the* ***Options*** *dialog.*

Line

Line properties are used to specify the default line style, color, opacity, and width. Some examples of where this is used include axes, stand-alone lines, and the outer border of rectangles, ellipses, and polygons. Click the *Line* option on the left side of the dialog to open the **Line Properties** page. Make any changes and click *OK* to make the default line changes.

Fill

Fill properties specify the fill pattern, foreground and background colors, foreground and background opacity, and allow custom images to be imported in the file. The fill properties are used to color the interior of polygons, rectangles, and ellipses. Click the *Fill* option on the left side of the dialog to open the **Fill Properties** page. Make any changes and click *OK* to make the default fill changes.

Symbol

Symbol properties include the symbol, symbol set, fill color, fill opacity, line color, line opacity, and size. These properties are used for stand-alone symbols, post maps, and imported graphics that do not use a specific symbol. Click the *Symbol* option on the left side of the dialog to open the **Symbol Properties** page. Make any changes and click *OK* to make the default symbol changes.

Font

Font properties specify the default face, style, size, color, opacity, and text alignment. Some examples of where this is used include stand-alone text, axis labels, post map labels, and contour labels. Click the *Font* option on the left side of the dialog to open the **Font Properties** page. Make any changes and click *OK* to make the default font changes.

Label Format

Label format properties specify numeric display for labels. The label type, prefix, suffix, number of significant digits, thousands symbol, and absolute value may be set. Click the *Label Format* option on the left side of the dialog to open the **Label Format** page. Make any changes and click *OK* to make the default label changes.

Frame

Frame properties specify the type and format of border that can be applied to several different objects, such as map scale bars, color scale bars, and legends. Click the *Frame* option on the left side of the dialog to open the **Frame** page. Make any changes and click *OK* to make the default frame changes.

Default Settings

The **File | Defaults** command or the  button opens the **Defaults** dialog. The **Defaults** dialog controls most of the default settings used in commands for **Surfer**. These settings are saved in a .SET setting file, and are reloaded whenever **Surfer** is started. Using the hierarchical list in the **Defaults** dialog, you can customize each default setting independently. This is an advanced feature, and the settings should not be changed indiscriminately. Minimal error checking is performed, and it is possible to specify values completely inappropriate for some data sets. Some inappropriate values may cause maps to appear incorrectly.

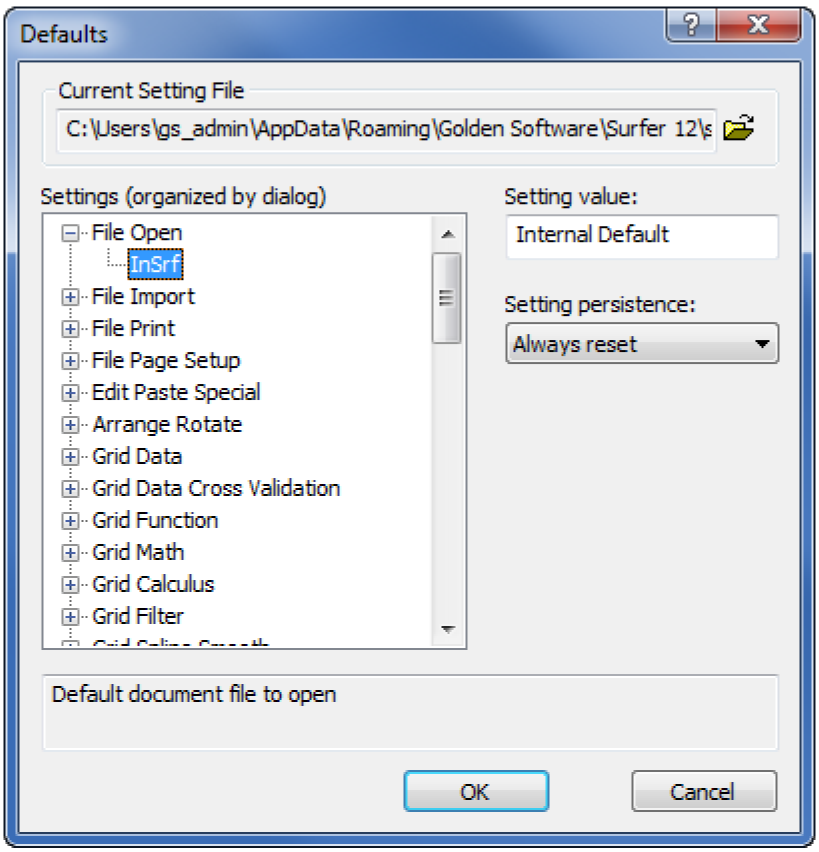
Changing Defaults in the Dialog

An easy way to make changes to default settings is to set the *Setting value* to *Internal Default* and set the *Setting Persistence* to *All sessions*. Click OK in the **Defaults** dialog. Then, open the dialog or command that should be changed. Make any changes in the dialog or command and click *OK.* You can confirm the change by clicking the **File | Defaults** command again and selecting the desired setting. The *Setting value* should now show the options from the dialog.

You must click *OK* in the dialog that uses the setting for the setting to be updated. If you *Cancel* the dialog, the changes are not recorded in the setting file.

Defaults Dialog

Click the **File | Defaults** command to open the **Defaults** dialog.



Customize the Advanced options in the ***Options*** *dialog.*

Current Setting File

The *Current Setting File* is the file **Surfer** is currently using to extract the defaults. Any changes made to the defaults in the various dialogs are saved in the current setting file when **Surfer** is shut down. By default, **Surfer** uses a file named SURFER.SET. This file is located by default in the C:\Users\<user name>\AppData\Roaming\Golden Software\Surfer\ directory. It is also possible to specify a different setting file by clicking the button.

Settings (organized by dialog)

The *Settings (organized by dialog)* list displays the default settings organized hierarchically by dialog. To change a setting, click the ⊞ button next to the dialog you wish to modify. Individual settings can be highlighted by clicking on them or by using the ARROW keys on the keyboard. As each individual setting is highlighted, a brief description appears in the text box at the bottom of the dialog.

Setting Value

Enter the new value of the default setting in the *Setting value* control. This control changes depending on the type of setting highlighted in the *Settings (organized by dialog)* list. Numeric values are entered into a box. Settings with a discrete number of choices are displayed in a list. The values entered in this control are not checked for errors, so be sure to specify an appropriate value for the highlighted setting.

All settings have a value called *Internal Default*. Changing the *Setting value* to *Internal Default* allows **Surfer** to determine the best value for the current setting. If the *Setting value* control is an edit box, you can specify the *Internal Default* value by deleting the contents of the box so it is empty.

Several default settings require the value to be specified in a particular format. The type of format required is determined from the comments at the bottom of the **Defaults** dialog. See the Formats for Attribute Values topic for a list of these special values.

Setting Persistence

The *Setting persistence* option specifies how the setting is updated when the user changes the setting during normal operation. The options are:

Always reset	Do not update the default setting when it is changed in a dialog. Every time the dialog is invoked, the setting is reset to the value in the setting file.
Current session only	This option saves changes made to the setting within the dialog during the current session only. The settings are not written to the setting file and are not used the next time **Surfer** is started.
All sessions	This option saves the changes made to the setting within the dialog during the current session, and writes the changes to the setting file to be used the next time **Surfer** is started.

Using Custom Setting Files

SET files allow custom settings to be saved and reused in **Surfer** each time the program is opened.

Creating a Custom .SET File

To create a custom setting file:

1. Make a backup of the default SURFER.SET file. The SURFER.SET file is located in the User's AppData directory. This is C:\Users\<User Name>\AppData\Roaming\Golden Software\Surfer by default.
2. Rename the new copy, keeping the .SET extension.
3. Click the File | Defaults command.
4. Click the  button next to the *Current Setting File.*
5. Select the new .SET file and click *Open*.
6. Modify the setting you wish to change.
7. Save the changes to the file by clicking *OK* in the **Options** dialog.

Using the Custom .SET File

After the preferences are set, you can rename the set file, keeping the .SET extension. Multiple .SET files can exist on the same computer. Each setting file can use a different set of defaults. You can manually change the setting file **Surfer** uses by clicking **File | Defaults**. Click the  button next to the *Current Setting File* field to select a different .SET file.

Alternatively, to use a different setting file with **Surfer**, start **Surfer** with the **/SET** switch on the command line. To do this, click the Windows Start button and select **Run**. Then, enter the following information:

"Path to Surfer.exe" /SET "Path to new .SET file"

An example may look like:

"c:\Program Files\Golden Software\Surfer\Surfer.exe" /SET "C:\Documents and Settings\<User Name>\Application Data\Golden Software\Surfer\custom.set"

To make this entire process easier, you can create a new shortcut on the desktop and specify the `/SET` switch in the *Target:* field of the shortcut properties dialog. See your Windows documentation for details on how to create shortcuts. To create a new shortcut:

1. Right-click on the Windows Desktop and choose **New | Shortcut**.
2. Type in the "Path to Surfer.exe" /SET "Path to new .SET file" in the box.
3. Click *Next*.

4. Type a name for the shortcut, such as Surfer.exe SET name.
5. Click *Finish* and the icon is created.

Using Old .SET Files

You can also use old **Surfer** setting files. When using older **Surfer** setting files, the hierarchy displayed in the **File | Defaults** dialog is only a single level due to lack of nesting information in the old format.

Be sure you copy the [.SET] file in the correct place for it to be used, C:\Users\(username)\AppData\Roaming\Golden Software\Surfer.

Formats for Attribute Values

Several settings use specially formatted values to specify all the required information when using Default Settings. When specifying the values for these settings, you must include all the parameters as discussed below.

Minimal error checking is performed, and it is possible to specify values completely inappropriate for some data sets. Some inappropriate *Setting values* may cause maps to appear incorrectly.

Colors

Colors are specified by name as they appear in the dialog color palettes, by Rxxx Gyyy Bzzz values, or by Rxxx Gyyy Bzzz Aaaa values. For detailed syntax information and an example, see color property syntax.

Line Properties

Lines are specified by the line color, name, and width. For detailed syntax information and an example, see line property syntax.

Fill Properties

Fills are specified by color, pattern, and additional options. For detailed syntax information and an example, see fill property syntax.

Font Properties

Fonts are specified by font name, size, color, and style. For detailed syntax information and an example, see font property syntax.

Symbol Properties

Symbols are specified by symbol number, symbol set, color, size, and opacity. For detailed syntax information and an example, see symbol property syntax.

Numeric Label Properties

The numeric label is specified by prefix, suffix, font properties, and numeric properties. For detailed syntax information and an example, see numeric label property syntax.

Color Syntax

Colors are specified by name as they appear in the color palettes. The names must be enclosed in double quotes. For example, "Ocean Green."

Alternatively, a color is specified using the special syntax:

"Rxxx Gyyy Bzzz"

or

"Rxxx Gyyy Bzzz Aaaa"

where xxx, yyy, and zzz specify a red, green, and blue color component and aaa specifies the opacity value, respectively. Each component can range from 0 to 255.

Example

`"R0 G0 B0"`	All components are 0, resulting in black
`"R0 G255 B0"`	Pure green
`"R255 G255 B255"`	All components are at full intensity, resulting in bright white.
`"R255 G255 B255 A45"`	All components are at full intensity, resulting in bright white. Opacity is set to 18%.

Line Property Syntax

In Default Settings, line properties are specified using the syntax:

```
"Color" "Style" Width
```

Parameter	**Description**
Color	The line color name. See Color List for details.
Style	The style of the line as it appears in the dialog line style palette. The name must be enclosed in double quotes.
Width	The width of the line in thousandths of an inch (mils). This should not be enclosed in quotes.

Example

```
"R0 G255 B50" ".1 in. Dash" 10
```

Fill Property Syntax

In Default Settings, fill properties are specified using the syntax:

```
"version "fgColor" "bgColor" "Pattern" offset scale angle coverage"
```

Parameter	Description
version	Defines the version of the fill property. Must be 1.
fgColor	The RGBA color value for the foreground color. See Colors for details.
bgColor	The background color name. See Colors for details.
Pattern	The fill pattern name as it appears in the dialog fill pattern palette. The name must be enclosed in double quotes.
offset	The offset value has an X and Y component. The offset controls the location of the image pattern within the geometry.
scale	The scale value has an X and Y component. The scale controls the density of the image pattern.
angle	The angle value determines the rotation of the pattern. This should always be 0.
coverage	The coverage value determines whether an image is stretched or tiled. 0 = tiled, 1 = stretched.

Note that all items are enclosed in quote marks.

Example

This example shows a fill property with the following options:

version = 1

fgColor = "R0 G0 B255 A99" indicating a slightly transparent blue

bgColor = "R0 G0 B255 A255" indicating a slightly transparent red

Pattern = "Gneiss 1", an image of a cut gneiss

Offset = 1 and 10, indicating 1 in the X direction and 10 in the Y direction

Scale = 0.9 and 0.8, indicating 0.9 in the X direction and 0.8 in the Y direction

Angle = 0

Coverage = 1, indicating that the pattern is to be stretched.

```
"1 "R0 G0 B255 A99" "R255 G0 B0 A38" "Gneiss 1" 1 10 0.9 0.8 0 1"
```

Font Property Syntax

In Default Settings, font properties are specified using the syntax:

```
"FaceName" "Color" Style Size
```

Parameter	Description
`FaceName`	The font face name as it appears in the Font Properties section in the **Properties** window. The name must be enclosed in double quotes.
`Color`	The font color name. See Color List for details.
`Style`	Add the following values to get the style you want: None = 0 Bold = 16 Italics = 32 Strikethrough = 64 Underline = 128 The style is entered as a number, and is not enclosed in double quotes.
`Size`	The size of the text in points. This is entered as a number and is not enclosed in double quotes.

Example

```
"Calibri" "Black" 0 6
```

Symbol Property Syntax

In Default Settings, symbol properties are specified using the syntax:

```
""SymSet" "FillColor" "LineColor" BitFlags Index Size"
```

Parameter	Description
SymSet	SymSet is the symbol set or face name as it appears in the Symbol Properties. The name must be enclosed in double quotes.
Fill Color	Fill color is the RGBA color for the interior of the symbol. See Color List for details.
Line Color	Line color is the RGBA color for the line around the symbol. See Color List for details.
Bit Flags	Bit flags is no longer used and is only included for backward compatibility. In older versions of Surfer, the Bit Flags was used to indicate whether the symbol Line Color was used. In the current version the value should be set to 0.
Index	Index is the 0-based index of the symbol to use as it appears in the Symbol Properties dialog. This is entered as a number, and does not use double quotes.
Size	Size is the size of the symbol page units (inches or centimeters). This is entered as a number.

Note that all items are enclosed in quote marks.

Example

This example shows a symbol with the following properties:

SymSet = "GSI Default Symbols"

Fill Color = "R255 G0 B0 A50" indicating a slightly transparent red

Line Color = "R0 G0 B255 A255" indicating a fully opaque blue

Bit Flags = 0

Index = 10

Size = 0.5 inches (or centimeters)

```
""GSI Default Symbols" "R255 G0 B0 A50" "R0 G0 B255 A255" 0 10 0.5"
```

Numeric Label Property Syntax

In Default Settings, numeric label properties are specified using the syntax:

```
Version Type Digits Style DateFormat TimeFormat "Prefix" "Suffix"
```

Parameter	**Description**
Version	Version is the version of the label format. This should be set to 5.
Type	Type is the numeric format. This is entered as a number, and is not enclosed in double quotes. 0 = Fixed 1 = Exponential 2 = Compact 4 = Date/Time
Digits	Digits refers to the number of decimal digits after the decimal place in the label. This value can range from 0 to 15. This is entered as a number, and is not enclosed in double quotes.

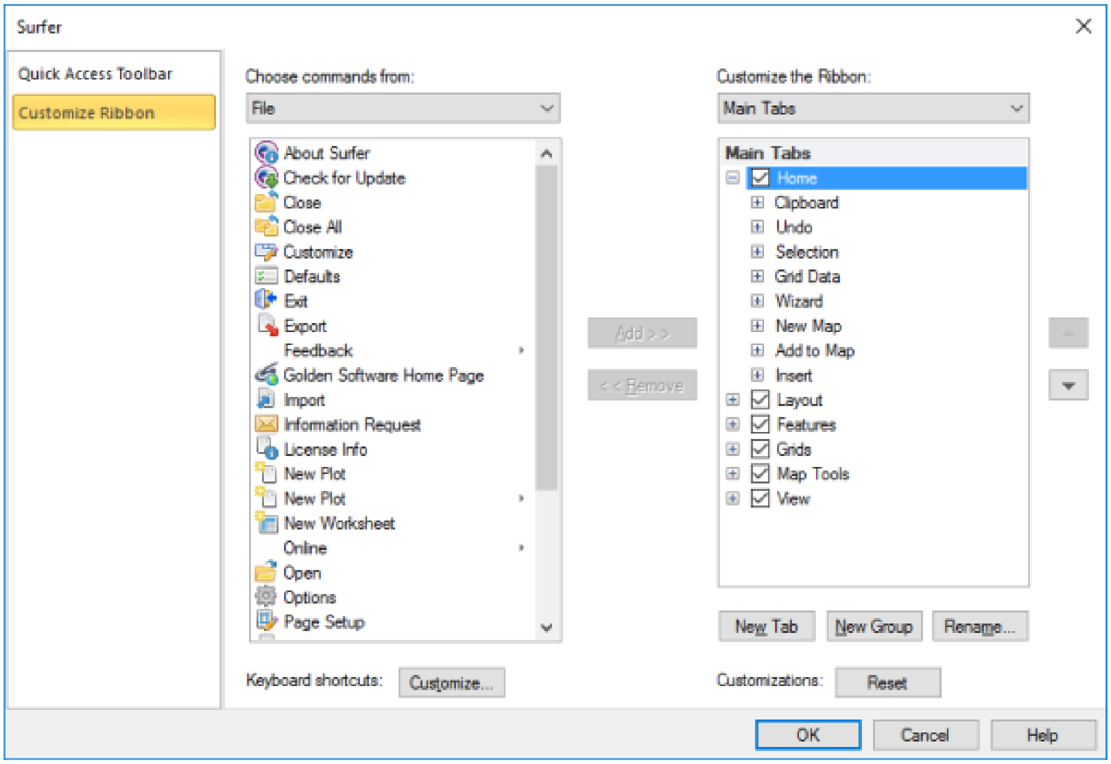
Style	Label style is entered as a number, and is not enclosed in double quotes. 0 = None (no styles) 1 = Thousands (separate thousands with a comma) 2 = Absolute Value (show numbers as absolute values) 3 = Thousands and Absolute Value (use thousands and absolute values)
DateFormat	The date format is entered as a number and is not enclosed in double quotes. The number values are listed on the wksDateFmtType Values page. Select the desired format and use the *Value* from this page in the label property syntax. For example, a value of 2 would set the format to DATEFMT_M_D_YY, which would display the format as 9/7/98.
TimeFormat	The time format is entered as a number and is not enclosed in double quotes. The number values are listed on the wksTimeFmtType Values page. Select the desired format and use the *Value* from this page in the label property syntax. For example, a value of 2 would set the format to TIMEFMT_H_MM_AMPM, which would display the format as 1:03 PM.
Prefix	The Prefix text is text that appears at the beginning of every label. This must be enclosed in double quotes.
Suffix	The Suffix text is text that appears at the end of every label. This must be enclosed in double quotes.

Example

```
0 8 0 "Pre" "Post"
```

Customize

Customize the **Surfer** ribbon, Quick Access Toolbar, and keyboard shortcuts by clicking **File | Customize Ribbon** or the button or by right-clicking on the ribbon and selecting **Customize the Ribbon**. Customize the ribbon by clicking *Customize Ribbon* on the left side of the dialog. Customize the Quick Access Toolbar by clicking *Quick Access Toolbar* on the left side of the dialog. Customize the keyboard commands by clicking *Customize* at the bottom of the dialog next to *Keyboard shortcuts*.



*The **Customize** dialog lets you create keyboard shortcuts for common actions and change the appearance of the ribbon and Quick Access Toolbar.*

Customizing the Ribbon

The ribbon is customizable in **Surfer**. To customize the commands in the ribbon, right-click on the ribbon and select **Customize the Ribbon**.

In the **Customize** dialog, you can add new tabs, add groups, hide existing tabs or custom groups, and add commands to any custom group. You can also rearrange the tabs into an order that fits your needs better.

To customize the commands in the **Customize** dialog, right-click on the ribbon and select **Customize the Ribbon**. In the **Customize Ribbon** dialog, use the following options.

Tab options:

- To add a custom tab, set the *Customize the Ribbon* section to *All Tabs*. Click in the list on the right side of the dialog where the custom tab should be located and click the *New Tab* button.
- To delete custom tab, right-click on the tab name in the list on the right side of the dialog and select **Delete**.

- To rename a default or custom tab, click on the tab name in the list on the right side of the dialog. Click the *Rename* button. Type the new name and press OK to make the change.
- To hide a default or custom tab, uncheck the box next to the tab name on the right side of the dialog. Only checked tabs will be displayed.
- To change the order of default or custom tabs, click on the tab name that should be moved in the list on the right side of the dialog. Click the up and down arrow buttons on the far right side of the dialog to move the selected tab up or down. Default tabs must remain in their major group.

Group options:

- To add a custom group to a default or custom tab, click on the ⊞ next to the tab name. Click in the list of group names where the new group should be located and click the *New Group* button.
- To delete a default or custom group on any tab, right-click on the group name in the list on the right side of the dialog and select **Delete**.
- To rename a default or custom group on any tab, click on the group name in the list on the right side of the dialog. Click the *Rename* button. Type the new name and click OK to make the change.
- To change the order of default or custom groups on any tab, click on the group name that should be moved in the list on the right side of the dialog. Click the up and down arrow buttons on the far right side of the dialog to move the selected group up or down in the list.
- To replace a default group with a custom group, right-click on the default group name and select **Delete**. Click the *New Group* button. Add the desired commands to the new group that you want displayed. Rename the new group, if desired.

Command options:

Commands can only be added to or deleted from custom groups. Commands can only be rearranged or renamed in custom groups. If you wish to edit commands in default groups, the default group should be hidden and a new custom group should be created with the same commands.

- To add a command to a custom group, set the *Choose commands from list:* to *All Tabs* so that all commands are listed on the left side of the dialog. Select the desired command that should be added. On the right side of the dialog, click the ⊞ next to the custom group name. Click on the desired position in the list of commands. If no commands exist in the group yet, click on the group name. Click the *Add>>* button and the command is added to the custom group.

- To delete a command from a custom group, right-click on the command name in the list on the right side of the dialog and select **Delete**. Only commands from custom groups can be deleted.
- To rename a command in a custom group, click on the command name in the list on the right side of the dialog. Click the *Rename* button. Type the new name and click OK to make the change. Only commands in custom groups can be renamed.
- To change the order of commands in a custom group, click on the command name that should be moved in the list on the right side of the dialog. Click the up and down arrow buttons on the far right side of the dialog to move the selected command up or down in the list.

Reset the Ribbon

To reset all customizations on the ribbon, click the *Reset* button at the bottom of the **Customize Ribbon** dialog.

Customizing the Quick Access Toolbar

The Quick Access Toolbar is a customizable toolbar. One method that can be used to add commands to the Quick Access Toolbar is to right-click on the command in the ribbon and choose **Add to Quick Access Toolbar**. The command is automatically added to the end of the toolbar.

To customize the commands on the Quick Access Toolbar, right-click on the Quick Access Toolbar or ribbon and select **Customize Quick Access Toolbar**.

In the **Customize** dialog,

1. To add a command, select the command from the list on the left that you want to add. Click the *Add>>* button and the command is added to the list on the right.
2. To add a separator between commands, select *<Separator>* and click *Add>>*. Move the separator to the desired position.
3. To delete a command, select the command from the list on the right. Click the *<<Remove* button and the command is removed from the list on the right.
4. To rearrange commands or move separators, click on the command or separator name from the list on the right that you want to move. Click the up and down arrow buttons on the far right to move the command up or down the list. Commands are shown in the exact order that they are displayed in

the Quick Access Toolbar.

5. To reset the Quick Access Toolbar to the default display, click the *Reset* button below the list on the right side of the dialog.

6. Click OK and all changes are made.

Keyboard Shortcuts

Click the *Customize* button next to *Keyboard shortcuts* to add, remove, or change the keyboard shortcuts in **Surfer**. The Customize Keyboard dialog is displayed.

Keyboard Commands

Keyboard commands can be used to increase efficiency and precision in the **Surfer** environment.

Plot Window

You can use the keyboard to move the pointer within the plot window, to select objects, and to move objects.

- The ARROW keys move the pointer within the plot window.
- Pressing the SPACEBAR is equivalent to clicking the left mouse button.
- Press CTRL+TAB to change the active page in a dialog.

Menu Commands

The keyboard can be used to access the menu commands.

- Hold down the ALT key and pressing an underlined letter in the menu bar.
- When the menu is displayed you can access a command by pressing the underlined letter in the command.

Dialogs

The keyboard can be used to move around in a dialog.

- The TAB key moves between the options in the dialog. As you use the TAB key to move through the dialog, the options are highlighted as they become active.
- The SPACEBAR is used to simulate mouse clicks, allowing you to toggle check boxes or press buttons that provide you with access to other dialogs or close the current dialog.

- Use the underlined hot keys by holding down the ALT key and typing the letter. This moves you immediately to the desired option.

Customize Keyboard Commands

To customize keyboard commands use the **File | Customize** command.

Default Keyboard Commands

Use these keyboard commands.

General

CTRL+F4	Close the plot window
ALT+SPACE	Display the application control menu
ALT+HYPHEN	Display the plot window control menu
CTRL+F6	Next document window
CTRL+SHIFT+F6	Previous document window
CTRL+TAB	Switch between **Surfer** windows
F10 or ALT	Activate the menu bar
CTRL+ESC	Display the Windows start menu
ALT+TAB	Switch to the last active application
ALT+ENTER	Activate the **Properties** window
ALT+F11	Activate the **Contents** window

Help

F1	Open help for the currently selected command or dialog
SHIFT+F1	Click on a command or dialog to open the detailed help topic
SHIFT+F10	Opens the context-menu for the selected object(s)

File

CTRL+N	Open a New plot window
CTRL + W	Open a New worksheet
CTRL+O	Open a file
CTRL+S	Save a **Surfer** .SRF file
CTRL+I	Import a file into the current **Surfer** window
CTRL+E	Export the current **Surfer** window
CTRL+P	Print the drawing in the current plot window
ALT+F4	Close **Surfer**

Edit

CTRL+A	Select all objects in the current plot window
CTRL+X or	Cut the selected objects to the clipboard
CTRL+C or	Copy the selected objects to the clipboard
CTRL+V or	Paste the clipboard contents into the plot window
CTRL+Y	Redo the previous Undo command

CTRL+Z or	Undo the last command
DELETE	Delete the selected objects
CTRL+SHIFT+A	Deselect All objects in the current plot window
F2	Rename the selected object

Plot Window

Use these plot window commands.

Order Objects

SHIFT+PAGE UP	Move selected to front
SHIFT+PAGE DOWN	Move selected object to back
CTRL+PAGE UP	Move selected object forward
CTRL+PAGE DOWN	Move selected object backward

View

F5	Redraw the screen
CTRL + D	Fit to window
CTRL + G	Page
CTRL + R	Zoom to Rectangle
CTRL + L	Zoom to Selected
CTRL + +	Zoom In
CTRL + -	Zoom Out
F11	Zoom Full Screen

Digitize Coordinates Dialog

CTRL+O	Open a file
CTRL+S	Save as a .BLN or .DAT file
CTRL+A	Save the file as a new .BLN or .DAT file
CTRL+Z or	Undo the last command
CTRL+Y	Redo the last undo
CTRL+X	Cut the selected text to the clipboard
CTRL+C or	Copy the selected text to the clipboard
CTRL+V or	Paste the clipboard contents into the digitizer window
DELETE	Delete the selected text
CTRL+F	Find
CTRL+H	Replace

Grid Editor Window

CTRL+N	Open a new plot window
CTRL+W	Opens a new worksheet window
CTRL+O	Open a grid file
CTRL+S	Save the edited grid file
F5	Redraw the screen

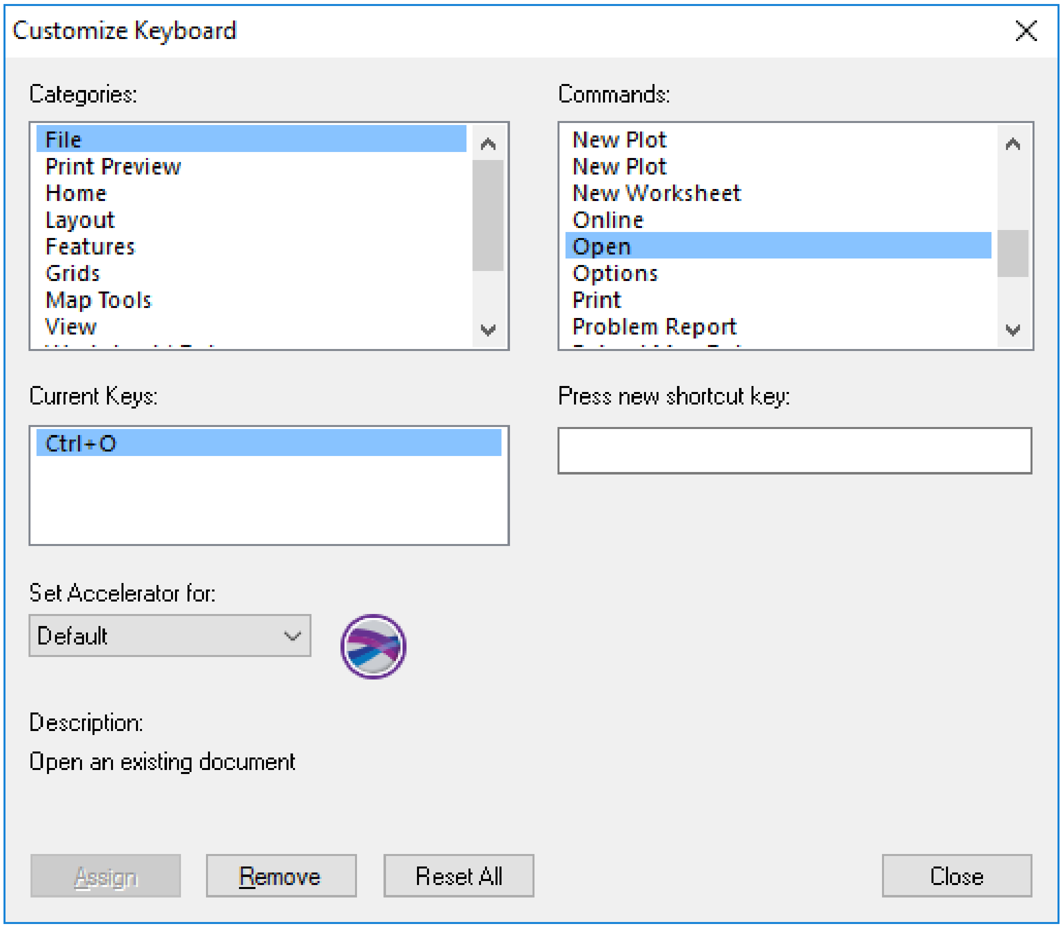
CTRL+B	Assign NoData to the selected node
ARROW	Move to the next node in the direction of the arrow

Worksheet Window

ARROW KEYS	Move to adjacent cell
F2	Edit active cell
DELETE	Clears contents of marked cells
END	When data are in the column: Go to the last row containing data. When data are not in the column: Go to the end of the document.
ENTER	Preserve the typed contents in the cell
HOME	When data are in the column: Go to the first row with data. When data are not in the column: Go to the beginning of the document.
PAGE UP	Scroll the worksheet up one screen
PAGE DOWN	Scroll the worksheet down one screen
CTRL + HOME	Go to cell A1
CTRL + Z or ALT+BACKSPACE	Undo the last command
CTRL + Y	Redo the previous Undo command
CTRL + X or SHIFT+DEL	Cut the selected objects to the clipboard
CTRL + C or CTRL + INSERT	Copy the selected objects to the clipboard
CTRL+V or	Paste the clipboard contents into the plot window
DELETE	Clears contents of the currently selected cell
CTRL + R	Opens the Insert dialog
CTRL + D	Opens the Delete dialog
CTRL + F	Opens the **Find and Replace** dialog, Find page
F3	Find next command
CTRL + H	Find and Replace dialog, Replace page

Customize Keyboard

Use the **Customize Keyboard** dialog to assign keyboard shortcuts to commands. To open the **Customize Keyboard** dialog click the *Customize* button next to *Keyboard shortcuts* in the Customize dialog.



*Use the **Customize Keyboard** dialog to customize keyboard shortcuts.*

Category, Commands, and Description

Menu and Tab titles (*File*, *Home*, etc.) are listed in the *Category* field. When a category is selected, the *Commands* list displays the current commands within the category. Also, when a command is selected, an explanation of the command appears in the Description section at the bottom of the dialog.

You can also list all commands by selecting *All Commands* at the bottom of the *Categories* list.

Accelerator Menu Application

You can choose to assign the accelerator (keyboard shortcut) to the *Surfer*, *Worksheet*, or *Default* menus in the *Set Accelerator* for list.

Current Shortcut Keys

When a command is selected in the *Commands* list, the accelerator keys are listed in the *CurrentKeys* list. The *Current Keys* list only displays accelerators for the current group of menus selected in the *Set Accelerator* for list.

Press New Shortcut Key

To create a shortcut, select a command, place the cursor in the *Press new shortcut key* field, type the shortcut on your keyboard, and then click **Assign**. If a shortcut is already assigned to a command a message appears below the *Press new shortcut key* field and the **Assign** button is disabled.

Removing Shortcuts

To remove a shortcut, select the shortcut in the *Current Keys* list and click **Remove**.

Resetting Shortcuts

To reset all shortcuts to the defaults, click **Reset All**.

Chapter 34 - Automating Surfer

Introduction to Scripter

Golden Software ScripterTMis a program for developing and running scripts.Surferoperations can be controlled automatically by scripts. A script is a text file containing a series of instructions carried out by a script interpreter program when the script is run. Instructions are written in a Visual BASIC-like programming language.

You can do practically everything with a script that you can do manually with the mouse or from your keyboard. Scripts are useful for automating repetitive tasks, consolidating a complicated sequence of steps, or acting as a "front end" to help novice users access Surfer's capabilities without having to become familiar with Surfer. In addition, scripts can integrate the features of several programs. For example, you could, with a single script, open a data file in Microsoft Excel, transfer the data to **Surfer**, create a map, and copy the map to Microsoft Word.

Scripter offers many features to help you write, edit, and debug scripts. Its features include language syntax coloring, a list of the procedures defined in the script, an object browser for examining procedures available in external objects, a visual dialog editor, break points, single-step execution (including options to step over and to step out of procedures), a watch window for displaying the values of script variables, and others. The Surfer Automation object model is displayed with the Surfer Object Hierarchy tree.

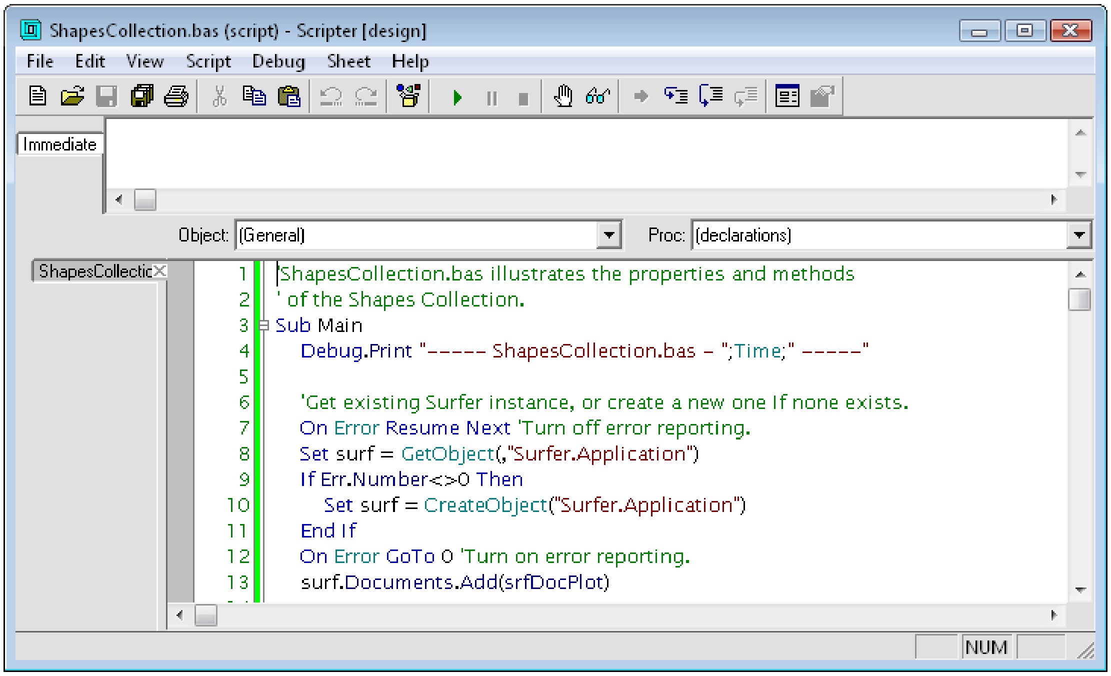
As an example, suppose you are monitoring groundwater levels in an area. Each week you collect XYZ data, create a contour map from this data, print the map, and save it in a **Surfer** file [.SRF]. The example scripts provide examples that automate these tasks. The sample script asks for the XYZ data file, creates a grid file, produces a contour map, prints the map, and saves it in a file. A person using the "front-end" script would not have to know anything about **Surfer** in order to create a contour map from the raw water level data.

Golden Software recommends that Windows 7 users run the program as administrator for best results.

Start the Scripter Program

To start the **Scripter** program, select it from the Windows Start menu. **Scripter** is installed in the same program group as **Surfer**. If **Scripter** is not present in the Windows Start menu, installation of **Scripter** may have been skipped when **Surfer** was installed. You can install **Scripter** from the **Surfer** download instructions. If you need new download instructions, contact technical support.

Scripter will open whichever version of **Surfer** was opened last.



*This is a script in the main window of the Golden Software **Scripter** program.*

Scripter Windows

When **Scripter** is first started, you are presented with a text editor window containing the lines `Sub Main`, followed by a blank line, and then `End Sub`. This is the code editor window where you type script instructions and where the contents of script files are displayed.

The code window acts as a text editor, similar to the Windows Notepad program, with a few enhancements to facilitate script writing:

- After you press the ENTER key, tabs or spaces are automatically inserted at the beginning of the next line to maintain the same indentation level as the previous line.
- Key words and symbols of the BASIC language are displayed in different colors. You can use the **View | Colors** command to modify the colors used to display the various elements of the programming language.
- A light horizontal divider line is automatically drawn between sections of your script. The divider lines help you to locate the start of subroutine and function definitions.

Above the code editor window is a bar containing the *Object* and *Proc* (procedure) lists. Selecting items from these lists moves the various sections of your script file into view. The object and procedure lists are useful when your script file becomes large.

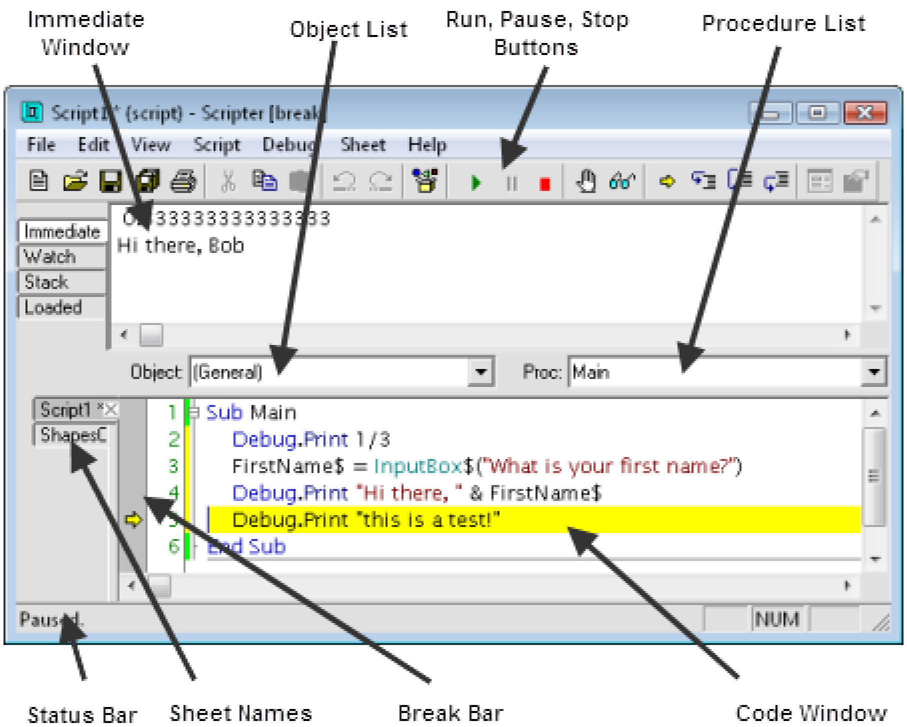
Above the object and procedure lists, you may see a blank window area with a tab on top that reads **Immediate**. If this window is not visible, select the **View | Always Split** command to make it appear. The immediate window is used to execute one-line instructions immediately. When you type an instruction into this window and press the ENTER key, **Scripter** carries out the instruction.

In addition to being a scratch area for evaluating language statements, the immediate window shows debugging information. The output from the `Debug.Print` statement and the value of variables selected with the **Debug | Quick Watch** command are printed in the immediate window. While a script program is running, **Watch**, **Stack**, and **Loaded** tabs are added at the top of the immediate window area. Click these tabs for information that may be useful for debugging. See Debugging Scripts for more information on the immediate, watch, stack, and loaded windows.

Along the left edge of the code window are code sheet tabs. When you select either the **File | New** command or the **File | Open** command, **Scripter** creates a new code sheet and inserts a new sheet tab. Each tab corresponds to one of the code sheets. Clicking once on a tab makes that sheet the current sheet. Double-clicking a tab closes the sheet.

Between the sheet tabs and the code window is an area called the "break bar." When a script is paused, a yellow arrow in the break bar shows which line is next to execute. The break bar also shows which lines have break points. Set a break point by clicking on the break bar. A red dot appears in the break bar, and the adjacent line in the code window is highlighted. When a line marked as a break point is about to be executed, **Scripter** pauses program execution. To clear a break point, click on the red dot in the break bar. See Debugging Scripts for more information on break points.

A status bar along the bottom of the **Scripter** window shows information about the current state of the program. The **View | Show/Hide | Status Bar** command hides or reveals the status bar. Before running a script, make sure that the status bar is visible because messages about typographical and syntax errors are displayed in the status bar.



*This **Scripter** application window is shown while execution of a script is paused.*

Working with Scripts

To create a new script, select the **File | New** command. A blank script sheet is created. You can start typing script instructions into this sheet. If you edit more than one sheet at a time, click the sheet tabs to switch between them or select the **Sheet | 1**, **Sheet | 2**, etc. menu commands. You can edit up to nine code sheets at the same time.

New Modules

To create a custom ActiveX object, select **File | New Module**, and choose either **Object Module** or **Class Module** (choosing **Code Module** is the same as the **File | New** command).

Existing Scripts and Modules

To open an existing script, select the **File | Open** command. To open a script you opened recently, click its name at the bottom of the **File** menu. To open other modules used by your script, select the **Sheet | Open Uses** command. Golden Software provides examples to help get you started.

Saving Scripts

Once a script is complete, save the script by using the **File | Save** or **File | Save As** commands. If a script has not been changed since the last save, the **Save** command is grayed out.

Closing Scripts

To close the active script, use **File | Close**, use **Sheet | Close**, or double-click the sheet tab of the sheet. Close all open scripts with **Sheet | Close All**.

Scripter BASIC Language

The online help describes the major elements of the **Scripter** BASIC programming language, but it does not explain the concepts of writing computer programs. Many good books on the subject of programming with BASIC (**B**eginner's **A**ll-purpose **S**ymbolic **I**nstruction **C**ode) have been written. If you are not moderately familiar with writing computer programs, we suggest that you refer to one of the books listed in the Suggested Reading topic.

Scripts are text files that contain a sequence of instructions to be carried out by the **Scripter** program. Each instruction specifies a task such as defining a variable or displaying a message on the screen. When the **Scripter** program processes the script, the instructions are carried out one at a time, from top to bottom.

Execution of a script begins with the first statement of the subroutine called `Main`. All scripts must therefore include the `Sub Main` and `End Sub` statements. Execution proceeds line-by-line until the end of the `Main` procedure, until an `End` statement is executed, or until an error occurs.

Visual BASIC Compatibility

The **Scripter** BASIC programming language is compatible with the Visual BASIC for Applications language (VBA). Scripts that run in **Scripter** will work in a VBA environment with few or no modifications. **Scripter** programs will also work under Microsoft Visual BASIC. Unlike most Visual BASIC programs, however, **Scripter** programs are not event-driven. **Scripter** programs are procedural. They start with the first statement of the Main procedure, and end when the Main procedure ends.

Some statements available in VBA are not supported in **Scripter** BASIC:

- The VBA *Collection* object
- The VBA *Clipboard* object
- GoSub
- On...GoSub

- On...Goto
- GoSub…Return
- Multiple statements on one line (separated by ":")
- All Financial functions
- Resume at current line
- Erl
- Option Compare
- Conditional compilation
- With Events
- LinkExecute
- LinkPoke
- LinkRequest
- LinkSend
- Line numbers
- LoadPicture

Conversely, some features of the **Scripter** BASIC language are not supported by VBA. Do not use the following features if you want to transfer your scripts from **Scripter** into VBA:

- Clipboard function
- CallersLine
- User dialogs
- PortInt
- MacroRunThis
- MacroDir
- Wait instruction
- MacroRun
- DDEExecute
- DDEPoke
- DDERequest
- DDEInitiate
- DDETerminateAll
- DDETerminate

Using Scripter For Automation

Tasks can be automated in **Surfer** using Golden Software's **Scripter** program or any ActiveX Automation-compatible client, such as Visual BASIC. A script is a text file containing a series of instructions for execution when the script is run. **Scripter** can be used to perform almost any task in Surfer. Scripts are useful for automating repetitive tasks and consolidating a sequence of steps. **Scripter** is installed in the same location as Surfer. Refer to the *Surfer Automation* topic in the help for more information about **Scripter**. We have included several example scripts so that you can quickly see some of **Scripter's** capabilities.

To run a sample script file:

1. Open **Scripter** by navigating to the installation folder, C:\Program Files\Golden Software\Surfer\Scripter. If you are running a 32-bit version of **Surfer** on a 64-bit version of Windows, navigate to C:\Program Files (x86)\Golden Software\Surfer\Scripter. Right-click on the *Scripter.exe* application file and select **Run as administrator**.

2. Choose the **File | Open** command.

3. Select a sample script .BAS file. These are located in the C:\Program Files\Golden Software\Surfer\Samples\Scripts folder or, if you are running a 32-bit version of Surfer on a 64-bit version of Windows, the C:\Program Files (x86)\Golden Software\Surfer\Samples\Scripts folder.

4. Click the **Script | Run** command and the script is executed. Most sample scripts open **Surfer** and display a map in the plot window.

Using Scripter Help

For more information on **Scripter** program menu commands, select the **Help | Contents** command in **Scripter**. Press the F1 key for more information about the **Scripter** windows or the active dialog. The **Help | Surfer Automation Help** command shows all **Surfer**-specific methods and properties.

The online help, shown when you select the **Help | BASIC Language Help** command (or press SHIFT + F1), explains all of the BASIC language statements and functions. Each help topic describes the purpose of a statement, and shows the syntax (the order of keywords and other symbols) to use when writing an instruction. The syntax examples in the online help use a shorthand method to explain the possible variations in usage:

`Sub, End, True`	Words with the initial letter capitalized indicate language-specific keywords.
`name`	Lower-case words with a dotted underline are placeholders for information you supply.
`[param]`	Items inside square brackets are optional. These may be omitted from the statement.
`{Until | While}`	Items between curly braces, and separated by a vertical bar are lists of options. You must use one of the items in the list.
`[Private | Public]`	Items between square braces, and separated by a vertical bar are lists of optional choices. You may use one of the items in the list, or none of them.
`...`	An ellipsis mark indicates that the preceding item in the syntax example is repeated. Repeated items are usually optional and separated by commas.
`; , . ( )`	Other symbols must be typed as shown in the syntax example, with the exception of the underscore "_" character.

_	The underscore is used to show that a sample line has been split.

Suggested Reading - Scripter

For additional help in learning how to program or for more information about the Visual BASIC for Applications (VBA) language (which is nearly identical to theScripterBASIC language) we recommend the following books:

Harris, Matthew (1997), *Teach Yourself Visual BASIC 5 for Applications in 21 Days*, Third Edition, SAMS Publishing, Indianapolis, IN, 1248 pp.

Lomax, Paul (1998),,

Wang, Wallace (1998), *Visual BASIC 6.0 for Windows for Dummies*, IDG Books Worldwide, Foster City, CA, 477 pp.

Writing Scripts

To create a script, you must type the script text into the **Scripter** code window, or edit an existing script. When you want to create a new script that is specific for your circumstances, you will most likely start with an empty **Scripter** window and type the entire script. If you want to perform a routine task such as creating a grid file or a contour map, you can probably open an existing script file and edit the file to meet you specific needs. **Surfer** comes with several sample scripts that you can modify as desired.

Consider a script that creates a grid file from an XYZ data file, and then creates a contour map from the grid file:

```
Sub Main
     ' Create a programmable object to represent the Surfer program
     Set SurferApp = CreateObject("Surfer.Application")
     ' Make the Surfer window visible
     SurferApp.Visible = True
     ' Add a new plot document
     Set plot = SurferApp.Documents.Add
     ' Prompt for name of the data file
     DataFile$ = GetFilePath$()
     ' Invoke the Surfer Grid Data command.
     SurferApp.GridData DataFile := DataFile$, OutGrid := "out.grd",
     ShowReport := False
     ' Invoke the AddContour map method.
     ' The AddContourMap method is a member of the Shapes collection
     object.
     ' The Shapes collection object, in turn, is one of the
     ' properties of the plot document object.
     plot.Shapes.AddContourMap "out.grd"
```

```
End Sub
```

When you execute the script, **Surfer** is automatically started and a plot window is displayed. The grid file is created (the progress of the operation is indicated on the **Surfer** status bar), and the contour map is drawn in the plot window. When the script execution is complete, you are returned to the **Scripter** window. The **Surfer** window remains open, although the **Scripter** window is the active window.

Writing comments in your scripts to explain how they work can save you time and frustration when you later need to modify the script. The apostrophe character (') signals the start of a comment.

Running Scripts

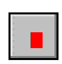
Scripts are nothing but plain text files. You could create script files with any text-editing program, such as the Windows Notepad. The usefulness of scripts is not realized until the script instructions are carried out. The **Scripter** program is designed to interpret script instructions and to carry them out. The process of carrying out the instructions in a script is called running or executing the script.

Running Scripts in Scripter

Scripts are placed in the code window either by typing a new script from scratch or by loading the script with the **File | Open** command.

To run the script in the **Scripter** code window, select the **Script | Run** command, press the F5 key, or click the  button. **Scripter** examines the script instructions, and, if all the instructions are recognized, it begins to perform each instruction in turn.

More often than not, however, a newly typed script will not work correctly the first time it is run. Even simple typographical errors will cause the script to fail. For information on finding and fixing errors in scripts, see the Debugging Scripts.

Select the **Script | End** command or click the  button to stop executing a script. This may be necessary when you want to edit a script after a run-time error occurs, or when you accidentally start a script and you want to cancel the execution.

Running Scripts from the Command Line

You can run scripts from a command prompt without having to manually load and execute the script in **Scripter**. The same commands that you would type at a command prompt may also be entered as the "target" for a shortcut in order to link a shortcut button to a script. Enter the following to run a script from the command line or to link a shortcut to particular script file:

```
<Scripter path> -x filename.bas
```

where `<Scripter path>` represents the path to the **Scripter** program file. (for example, "C:\Program Files\Golden Software\Surfer\Scripter.exe"), and `filename.bas` represents the name of the script to run. The space between the `-x` and the file name is required. This command opens the **Scripter** window, loads the specified script file, and runs the specified script. When the script terminates - either successfully or unsuccessfully - the **Scripter** window closes.

To load a script file but not execute it, the following command can be used:

```
<Scripter path> filename.bas
```

This opens the **Scripter** window and automatically loads the specified script file. The **Scripter** window remains open.

Passing a Command Line Argument to the Script

A single command line argument can be passed to a script. The command line argument can be any text that is used in the script. For example, a file name can be passed and used to create a map. Enter the following to run a script with a command line argument:

```
<Scripter path> -x filename.bas command
```

where `command` represents the commands that should be passed to the script. Access the command line argument from the script using the `Command$` option. For example, if the script name is variable.bas and it can be run with any grid file, you could use:

```
"c:\program files\golden software\Surfer\scripter.exe" -x "c:\tem-
p\variable.bas" "c:\temp\sample.grd"
```

Do not quote the arguments in the command. To use the data in the script, use:

```
'Return the argument to the Immediate window
Debug.Print Command$

'Create a contour map from the grid
Plot1.Shapes.AddContourMap(GridFileName:=Command$)
```

Debugging Scripts

Bugs are errors in a script that keep it from performing as intended. Debugging is the process of locating and fixing these errors. The most common bugs are typographical errors in scripts or malformed instructions. **Scripter** detects these types of errors immediately when you try to run a script. The program beeps, highlights the line containing the error in red (or whatever color has been set with **View | Colors** command), and displays an error message on the status bar.

Viewing Errors

Before running a script, verify that the **View | Show/Hide | Status Bar** command is enabled, otherwise you will not see the error message. To resolve the errors that **Scripter** immediately detects, you usually must interpret the error message and correct the indicated problem. Typical errors are typing mistakes, unbalanced parentheses, misuse of a BASIC language instruction, or failure to declare variables in a DIM statement (if you use the OPTION EXPLICIT statement). If you do not see an obvious problem, refer to the online BASIC language help to make sure you are using the right syntax.

Run-Time Errors

Scripts which encounter errors midway through script execution may be fixed much the same way as syntax errors. The error message should guide your actions. Some run-time errors cannot be detected until they are executed, such as when you try to open a file that does not exist. In these cases, you need to check for error conditions in your scripts. For example, use the DIR function to make sure a file exists before trying to open it. Alternatively, you can use the ON ERROR GOTO statement to specify an error handling section to execute when a procedure encounters a run-time error:

```
Sub OpenFile(srf As Object, filename As String)
     On Error Goto ErrLabel
     srf.Documents.Open filename
     Exit Sub

     ErrLabel:
     MsgBox "Unable to open file " & filename
     Exit ' Must use RESUME or EXIT at end of error handling code
End Sub
```

Script Runs Incorrectly

Most difficult to correct are scripts which run, but do not work as expected. Fixing these scripts is hard because you do not know which line or statement is causing the problem. **Scripter** provides a number of debugging features to help you locate the source of problems.

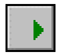
Debug.Print

Probably the simplest debugging technique is to insert instructions into your script to monitor the progress of the script and display the values of variables at various points throughout the script. Use the `Debug.Print` statement to display information in the **Scripter** immediate window:

```
Debug.Print "The value of variable X is "; X
```

To clear the contents of the immediate window, select the text in the window and press either DEL or BACKSPACE.

Stop or Pause

Insert the STOP instruction to pause script execution where you think there might be a problem. While the script is paused, you can examine and change the values of program variables. If a running script appears unresponsive, it may be stuck in an infinite loop. Select the **Script | Pause** command or click the  button to pause the script. To resume executing a paused script, select the **Script | Run** command or click the  button.

Viewing Variable Values

While a script is paused, there are several ways to view the value of a variable:

- In the immediate window, type a question mark, followed by the variable name and press ENTER. The current value of the variable is displayed.
- In the code window, place the cursor on the variable name you want to examine (that is, click on the variable name in the code window). Press SHIFT+F9, select the **Debug | Quick Watch** command, or click the  button on the toolbar. The current value of the variable is displayed in the immediate window.
- To continuously monitor a variable's value, click on the variable name in the code window, and press CTRL+F9 or select the **Debug | Add Watch** command. Alternatively, type the variable name in the watch window and press ENTER. The variable name and its value are displayed in the watch window. Every time script execution pauses, the variable value is automatically updated. To clear a variable from the watch window, highlight the line showing the variable value and press the DEL or BACKSPACE key.

Changing Variable Values

To change the value of a variable, type an assignment expression in the immediate window and press ENTER. For example, type "A=5" (without quotes) and press ENTER to assign a new value to the variable named "A."

Step

A powerful debugging technique is to watch **Scripter** execute your script one line at a time. This lets you check the effect of each instruction and verify that the script is doing what you expect. While stepping through a script, you can examine and change the values of script variables. Select the **Script | Run** command or click the  button to resume script execution at full speed after stepping through script instructions.

- To execute your script one line at a time press the F8 key, or select the **Debug | Step Into** command. The first line of the script is executed (or, if the script was paused, the next highlighted line is executed). The next line is highlighted and a yellow arrow appears to the left of the next line. To execute the highlighted instruction, press F8 again.

- If a statement calls a subroutine or function that is defined within your script, the highlight will move into the called procedure. To keep from tracing execution into a called procedure, press SHIFT+F8 or select the **Debug | Step Over** command. This executes the whole subroutine or function in a single step.

- If you accidentally step into a procedure, press CTRL+F8 or select the **Debug | Step Out** command. This executes all remaining instructions in a procedure, and returns the highlight to the instruction that called the procedure.

- If you do not see the next highlighted instruction, select the **Debug | Show Next Statement** command to scroll the highlighted line into view.

- Sometimes you may want to skip the execution of some instructions or you may want to execute the same instructions several times without restarting the script. To change the next instruction line, click on the line you want to execute next and select the **Debug | Set Next Statement** command.

Breakpoint

Watching **Scripter** execute every line of the script may be too time consuming. In this case, a breakpoint pauses the script where you think there might be a problem. A breakpoint is a line of code that you mark. When **Scripter** encounters a line marked as a breakpoint, it pauses the script just as if it had executed a STOP instruction. Breakpoints are more convenient than STOP instructions because they may be set and cleared while a script is paused, whereas STOP instructions may be changed only after a script has ended.

- To set a breakpoint, click in the break bar area next to the line you want to mark. The break bar is the area to the left of the code window, between the sheet tabs and the code window. Alternatively, click on the line you want to mark, and press F9 or select the **Debug | Toggle Break** command. The line becomes highlighted in red, and a round marker appears in the break bar area.

- To clear a breakpoint, click on the round marker, or move the cursor to the marked line and press F9 or select the **Debug | Toggle Break** command again. You can clear all breakpoints by pressing SHIFT+CTRL+F9 or selecting the **Debug | Clear All Breaks** command.

A quick alternative to setting a breakpoint is the **Debug | Step To Cursor** command. This command has the same effect as setting a breakpoint on the current line, running the script, and then clearing the breakpoint after script execution has paused on the current line.

Trace

To check flow of execution through your script without having to watch each line of the script being executed, try using the TRACE function. To activate the trace function type "Trace" (without the quotes) in the immediate window and press ENTER. Trace On is displayed in the immediate window. As the script is run, the location of every instruction being executed is printed in the immediate window. After the script finishes, the trace function is automatically disabled.

Stack

If you nest procedure calls (that is, one procedure calls another procedure, which calls yet another procedure, and so forth), the stack window may be useful. When a script is paused, the stack window lists the procedures that have been called, and the order in which they were called. For instance, if the `Main` procedure calls procedure "A" which in turn calls procedure "B," the stack window displays three lines, one for each of the called procedures. Clicking on a line in the stack window moves the corresponding procedure into view in the code window.

Module Files

Click the loaded window tab in the immediate window area to see which module files are currently being interpreted by **Scripter**. The loaded files include the current script file and any modules it includes with the '#Uses statement.

Program Statements

Statements are individual instructions to **Scripter** that carry out a specific operation. Statements are case insensitive and are typically written one to a line. To enter two or more statements on the same line, separate the statements with colons. For example:

```
a = 5 : b = 5 * a
```

Scripter BASIC requires flow control statements (IF, WHILE, DO, etc.) and declaration statements (DIM, PUBLIC, TYPE, SUB, etc) to be placed on a line by themselves.

Line Continuation

To break a line into two lines in **Scripter**, use a space followed by an underscore " _". You must include the space for the continuation to work properly in the script. A backslash continuation " \ " from earlier versions of **Scripter** is not supported. Comments are not allowed after the continuation character.

Example

```
SurferApp.GridData("C:\Program Files\Golden
ExclusionFilter:=x4, xMin:=4, Algorithm:=srfKriging,
```

Comments

Writing comments in your scripts to explain how they work can save you time and frustration when you later need to modify the script. The apostrophe character (') signals the start of a comment. **Scripter** ignores all text following the apostrophe up to the end of the line. Comments can be placed on their own line, or they may be placed at the end of a line. For example:

```
SurferApp.GridData "demogrid.dat" ' This creates a grid file from
demogrid.dat
```

In addition, you can use the REM statement to add a remark in the script. However, REM statements can only be used at the beginning of a line.

Double Quotes and Text

In **Scripter**, text strings must be enclosed in double quotes. File names, for example, must be surrounded by double quotes. If quotes are missing, the text may be mistaken for a variable name.

```
Debug.Print "This text string is printed in Scripter's immediate win-
dow"
```

Operators

Operators are symbols that direct a script to perform basic calculations, such as addition, exponentiation, string concatenation, number comparison, and others. The language supports several arithmetic, comparison, and logical operators. In **Scripter**, select **Help | BASIC Language Help** command and search for "Operators" to see a complete list.

Flow Control

When you run a script, execution starts with the Sub Main statement and continues line-by-line until the End Sub statement at the end of the main procedure or until an End statement is encountered. SEveral flow control statements allow you to change this line-by-line progression according to conditions encountered by your script. The **Scripter** BASIC language includes a variety of looping and branching statements that is typical for modern programming languages. The flow control statements include the following (see the online BASIC language help for details on the syntax of these statements):

IF...END IF

`IF...END IF` executes a statement only if a condition is true.

IF...ELSE...END IF

The alternate of `IF...END IF` form executes one statement if a condition is true and a different statement if the condition is false.

SELECT CASE...END SELECT

`SELECT CASE...END SELECT` branches to one of several statements. This compares the value of an expression to several test values and executes the statements associated with the test value that matches the expression value.

DO...LOOP

`DO...LOOP` is the basic looping statement. This statement loops either while a condition is true or until a condition becomes true and tests the condition either at the top of the loop or at the bottom of the loop.

This and all other loop structures may be stopped before the test condition has been met by placing an `Exit` statement in the body of the loop.

WHILE...WEND

`WHILE...WEND` loops while a condition is true and tests the condition at the top of the loop.

FOR...NEXT

`FOR...NEXT` loops a number of times and increments (or decrements) an index variable each time through the loop.

FOR EACH...NEXT

`FOR EACH...NEXT` iterates through all the elements in a collection object. Several **Surfer** automation objects are collection objects. The `For...Each` statement is a convenient way to process each element of a collection.

For example, the following code fragment closes all the documents in the **Surfer** *Documents* collection object:

```
'Assume that several documents are already open and that "SurferApp"
' is the name of a variable which refers to the Surfer Application
object
```

```
Dim documents, doc As Object
Set documents = SurferApp.Documents
For Each doc In documents
     doc.Close
Next
```

Optional Arguments and Named Arguments

Many procedures, especially the methods provided by Surfer Automation objects, accept a large number of arguments. Some of the arguments are required. Every required argument must be supplied or the script will fail to run. Some arguments are optional. Optional arguments may be omitted and the procedure will assume a default value for the missing arguments.

For example, the PlotDocument object’s "PrintOut" method accepts up to seven arguments, all of which are optional:

```
plot.PrintOut(Method, SelectionOnly, NumCopies, Collate, _
xOverlap, yOverlap, Scale )
```

Since the arguments are optional, you can skip all or some of them when calling the procedure. To print three copies at fifty-percent scale, for example, you would supply just the NumCopies, and Scale argument values. These arguments must be listed in the correct position, separated by commas, as shown below:

```
Set SurferApp = CreateObject("Surfer.Application")
Set plot = SurferApp.Documents.Add
plot.PrintOut , , 3, , , , 50
```

Although only two of the seven argument values are supplied in this example, the appropriate number of commas must be used to mark the positions of the missing arguments. Since inserting the right number of commas can be troublesome, you can supply the arguments by name rather than by position. Named arguments are specified by the argument name followed by a colon and an equal sign (":="), followed by the argument value:

```
Set SurferApp = CreateObject("Surfer.Application")
Set plot = SurferApp.Documents.Add
plot.PrintOut Scale := 50, NumCopies := 3
```

Named arguments may be listed in any order without regard to the order they appear in the procedure's definition.

Named and Positional Arguments

In **Scripter**, named arguments must have a colon plus equal sign (:=), not just an equal sign as in earlier versions of **Scripter**. The arguments can be named or you can use positional statements separated by commas. The following two examples show the same line written with the two different methods.

Named arguments:

```
SurferApp.GridData(SurferApp.Path+"\Samples\Demogrid.dat", _
ExclusionFilter:="x<4", xMin:=4, Algorithm:=srfKriging, DupMeth-
od:=srfDupFirst)
```

Positional arguments:

```
SurferApp.GridData(SurferApp.Path+"\Samples\Demogrid.dat", _
,,,"x<4",srfDupFirst,,,,,4,,,,srfKriging)
```

Subroutines and Functions

Writing a long or complicated script may be easier to manage if you divide the script into smaller pieces called procedures. A procedure is a separate sequence of instructions that you can call from multiple places within your script. The BASIC language provides many predefined procedures for performing frequently needed tasks, and, in fact, the methods provided by the **Surfer** automation objects are themselves procedures.

When you call a procedure, the instructions in the procedure are executed. When the procedure finishes its task, it returns control to the instruction that called the procedure. The **Scripter** BASIC language distinguishes two types of procedures: functions and subroutines. Functions return a value, whereas subroutines do not.

Subroutines and functions may accept one or more values, called arguments. Arguments are passed to a procedure by listing them after the procedure name. If there is more than one argument, the arguments must be separated by commas. For example:

```
x = Cos (0) 'Returns the cosine of the argument (returns 1)
a$ = Left("AgAuPb",2) ' Returns the left-most characters (returns
"Ag")
Wait 5 ' Waits for 5 seconds
```

`Cos`, `Left`, and `Wait` are procedures built-in to the BASIC language. `Cos` and `Left` are functions which return values to the caller. `Wait` is a subroutine, and, by definition, it does not return a value. The `Wait` subroutine waits for the number of seconds specified by its argument (5 seconds in this example) before returning control to the calling location.

The arguments passed to a function must be enclosed in parentheses if the function's return value is used. If the function's return value is not used, the arguments may be listed without enclosing them in parentheses. Arguments passed to a subroutine are never enclosed in parentheses.

Writing Subroutines

To define subroutines within a script, use the `Sub` statement. Subroutine and function definitions cannot be nested within other procedures. That is, the `Sub` statement must appear after the `End Sub` statement of any preceding subroutine definitions. The syntax for a subroutine definition is:

```
Sub name ( arguments )
     statements
End Sub
```

Where `name` represents the name you want to give the subroutine, `arguments` represents a list of arguments names and types, and `statements` represents the instructions that comprise the body of the subroutine. There is no limit to the number of instructions you can include between the `Sub` and the `End Sub` lines. Consider the definition of a Main procedure and another subroutine:

```
Sub Main
     MultipleBeep 25 ' call the MultipleBeep subroutine
End Sub
Sub MultipleBeep (count As Integer)
     For i = 1 To count
          Beep
          Wait 0.5 ' Wait one-half second between beeps
     Next
End Sub
```

Each time the `MultipleBeep` procedure is called, the instructions between its `Sub` and `End Sub` statements are executed.

If the subroutine accepts arguments, the arguments are defined following the subroutine name using a format similar to the `Dim` statement. The argument definition list must be enclosed in parentheses, and argument definitions are separated by commas if there is more than one. When a subroutine is called, the variables listed in the argument list are automatically assigned the values passed in from the calling procedure.

Writing Functions

Functions are defined using the Function statement much the same as subroutines are defined with the Sub statement. Like subroutines, function definitions cannot be nested within other procedures. Unlike subrouties, functions can return a value to the calling procedure. The syntax of a function definition is:

```
Function name ( arguments ) As type
     statments
```

```
End Function
```

Where `name` is the function name you want to use, `arguments` is a list of arguments names and types, `type` is the type of the value returned by the function, and `statements` represents the instructions in the body of the function. To return a value from a function, assign a value to a variable with the same name as the function itself. For example:

```
Function hypotenuse (a As Double, b As Double) As Double
     c = a * a + b * b ' The built-in Sqr function computes the square
     root
     hypotenuse = Sqr (c) ' Set the function's return value
End Function
```

You define the list of arguments accepted by a function the same way as you define the arguments accepted by subroutines.

Built-in Functions and Procedures

Numerous useful functions and subroutines are built into the **Scripter** BASIC language. These routines can help you perform some of the most commonly required programming tasks. Functions for processing strings, performing mathematical calculations, error handling, working with disk files, and many others are available.

If you are not already familiar with the Visual BASIC for Applications programming language, it will be worth your time to review the list of available subroutines. This list is found by selecting **Help | BASIC Language Help** in **Scripter**.

Next: Using Surfer Objects

Specifying Cell Coordinates

Cell ranges can be specified in various ways for the **WksDocument** Cells and Range methods:

[1] Cells("A1") is a one argument single cell

[2] Cells("A1:C5") is a one argument range of cells

[3] Cells("A:E) is a one argument range of whole-columns

[4] Cells("1:5") is a one argument range of whole-rows

[5] Cells(Range object) is a one argument range of cells

[6] Cells(1,"A") -or- Cells(1,1) is a two argument single cell

[7] Cells("A1","C5") is a two argument range of cells

[8] Cells(1,"A",5,"C") -or- Cells(1,1,5,3) is a four argument range of cells

Column ranges can be specified in various ways for the **WksDocument** Columns method:

[1] Columns(1,5) or ("A","E) is a two argument range of columns

[2] Columns(A:

[3] Columns("A1:E1") is a one argument range of columns [the row coordinates are ignored]

[4] Columns(Range object) is a one argument range of columns [the row coordinates are ignored]

[5] Columns("A5") is a one argument single column [the row coordinate is ignored]

[6] Columns(1) is a one argument single column

Row ranges can be specified in various ways for the **WksDocument** Rows method:

[1] Rows(1,5) is a two argument range of rows

[2] Rows("1:5") is a one argument range of rows

[3] Rows("A5:A10") is a one argument range of rows [the column coordinates are ignored]

[4] Rows(Range object) is a one argument range of rows [the column coordinates are ignored]

[5] Rows("A5") is a one argument single row [the column coordinate is ignored]

[6] Rows(1) is a one argument single row

Also, the Cells, Columns, and Rows methods work slightly differently when invoked on a **WksRange** object than when invoked on a **WksDocument** object. When invoked on a **WksRange** object, the coordinates are relative to the upper-left corner of the range. For example, Range.Cells(A1) refers to whatever the upper-left corner of the Range happens to be, like so:

```
Set Wks = Surfer.Documents.Add(srfDocWks)
Set RangeObject1 = Wks.Cells(C5:E10)
' RangeObject2 now contains the cell "C5"
Set RangeObject2 = RangeObject1.Cells("A1")

' RangeObject3 now contains the cell "C5"
Set RangeObject3 = RangeObject1.Cells(1,1)
```

```
' RangeObject4 now contains the cell "D6"
Set RangeObject4 = RangeObject1.Cells(2,2)
```

In addition, you can use a single numeric argument in the Range.Cells() method to sequentially access each cell in the range, like so:

```
' Note: RangeObject1 equals C5:E10
Set RangeObject5 = RangeObject1.Cells(1) ; cell "C5"
Set RangeObject6 = RangeObject1.Cells(2) ; cell "D5"
Set RangeObject7 = RangeObject1.Cells(3) ; cell "E5"
' There are three cells in the first row of RangeObject1.
' Cell #4 is in the second row…
Set RangeObject8 = RangeObject1.Cells(4) ; cell "C6"
Set RangeObject9 = RangeObject1.Cells(5) ; cell "D6"
Set RangeObject10 = RangeObject1.Cells(6) ; cell "E6"
' Cell #7 is in the third row…
Set RangeObject11 = RangeObject1.Cells(7) ; cell "C7"
```

There are some special cases when the **WksRange** objects' Cells, Columns, and Rows methods are called. The behavior for these special cases is explained in these notes:

WksRange.Cells() Method:

[1] Coordinates are relative to the top, left of the current (base) range

[2] The returned range can extend beyond the original range

[3] Rows are limited to the original range if a whole-column sub-range is specified

[4] Columns are limited to the original range if a whole-row sub-range is specified

[5] Cells are indexed across and then down

Examples:

Item	Base Range	Specified Sub-Range	Range Returned
[1]	Wks.Range (B10:C20).	Cells("A1")	"B10"
[2]	Wks.Range (B10:C20).	Cells("A1:C30")	"B10:D39"
[3]	Wks.Range (B10:C20).	Cells("A:C")	"B10:D20"
[4]	Wks.Range ("B10:C20").	Cells("1:5")	"B10:C14"
[5]	Wks.Range ("B10:C20").	Cells(n)	n=1 "B10", n=2 "C10", n=3 "B11", etc.

WksRange.Rows Method

Columns are limited to the original range (the same as if a whole-row sub-range were supplied to the Range.Cells method).

Example:

Base Range	**Specified Sub-Range**	**Range returned**
Wks.Range ("B10:C20").	Rows("1:5")	"B10:C14"

WksRange.Columns Method

Rows are limited to the original range (the same as if a whole-column sub-range were supplied to the Range.Cells method)

Example:

Base Range	**Specified Sub-Range**	**Range returned**
Wks.Range ("B10:C20").	Columns("A:C")	"B10:D20"

Code, Class, and Object Modules

If you create very large scripts, or frequently reuse the same subroutines in several different scripts, you may want to split your script code into more than one file. Each script file is called a module.

A script an call subroutines and functions defined in other modules. In addition to procedures, global variables, type definitions, and enumeration definitions may be shared among modules. Just as procedures make long scripts easier to manage and debug, modules make large script projects easier to manage.

Module Types

The **File | New Module** command in **Scripter** adds new code sheets to the workspace. Each sheet is stored in a separate file. When routines in one code sheet are used by other sheets, the code sheets are called modules. **Scripter** supports three types of modules:

- Code modules are used for stand-alone scripts and for storing libraries of useful procedures that can be called from other modules. The scripts described in this chapter are code modules, which contain a Main subroutine. Code modules without a Main subroutine cannot be run, but the routines contained in them can be shared by other scripts. Code modules are stored in files with a [.BAS] extension.

- Class modules are used to define objects that you can use in other modules. A class module defines the properties and methods that the object supports. Other modules access the object's properties and methods using the same syntax that is used to access **Surfer** automation objects. Unlike **Surfer** objects, new instances of the object defined in a class module are created using the NEW keyword. Class modules are stored in files with a [.CLS] extension.
- Object modules are identical to class modules, except that when a script uses the object defined in an object module, one instance of the object is automatically created. Additional instances of an object defined in an object module can be created with the NEW keyword. Object modules are stored in files with an [.OBM] extension.

The '#Uses Line

Before using the procedures and objects defined in another module, a script must indicate the name of the file containing the procedure or object definitions. You must place a '#`Uses` statement at the beginning of a script, before any procedure definitions, to instruct **Scripter** to load all modules used by the script. For example:

```
'#Uses "c:\utils.bas"
'#Uses "test.cls"
Sub Main
' use the procedures and object defined in UTILS.BAS and TEST.CLS
End Sub
```

Scripter does not permit cyclic `'#Uses` statements. That is, if module A uses module B, module B cannot use procedures from module A.

Private and Public Definitions

By default, all subroutines, functions, and user-defined types (including enumeration definitions) may be accessed from other modules. To prevent other modules from accessing procedures or user-defined types precede the definition with the `Private` keyword:

```
Private Sub MyBeep
Beep : Beep
End Sub
```

In contrast to procedures, the global variables defined at the top of one module are not available to other modules unless they are declared using a `Public` statement. When used for variable declarations, the `Public` statement has the same syntax as the `Dim` statement:

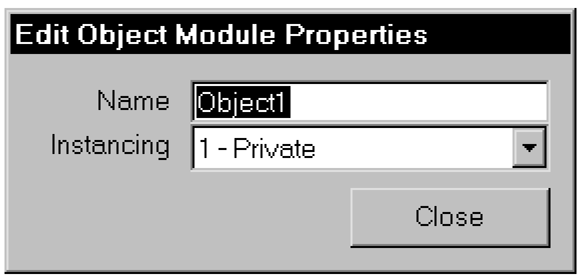
```
Public universal_data As String
```

The names of all definitions, even private ones, are visible in other modules. To avoid errors due to name conflicts you must avoid using the same procedure, type, and variable names in more than one module. A common technique for avoiding name conflicts is to append a prefix to the names of global variables and procedures. For example, if you write a module of text-processing functions, you might prefix each function name with `txt` (e.g., `txtFunction1`).

Module Properties

To set the name by which other modules refer to an object defined in a class or object module, select the **Edit | Properties** command. The **Edit Class Module Properties** dialog appears. Type the name that you want other scripts to use when referring to the object defined in the module. The instancing options control how other applications access the object defined in the module, but these options are not relevant to scripts executed within **Scripter**. Code modules do not have module properties.

When an object module is used in a script, one instance of the object defined in the module is automatically created. The name of the object that is automatically created is the name specified in the **Edit Object Module Properties** dialog.



*When working with an object module or class module, open the **Edit Object Module Properties** or **Edit Class Module Properties** dialog with Edit | Properties .*

Defining Object Properties and Methods

Class and object modules define the properties and methods of objects. To define the methods for an object, simply define public subroutines and functions. All the public procedures in a class or object module are methods that users of the object can call.

The properties of an object typically correspond to private global variables defined in the module. To allow users of the object to access the variable values, you provide "property get" and "property set" procedures. Use the `Property Get` statement to define a function that returns the value of a property. Use the `Property Let` statement (or the `Property Set` statement if the property is an object reference) to define a subroutine that changes the value of a property.

Two special subroutines are called when an object is first created and just before it is finally destroyed. In a class module, these subroutines are called "Class_Initialize" and "Class_Terminate." In an object module, these subroutines are called "Object_Initialize" and "Object_Terminate." These subroutines do not take any arguments.

Example: Defining an Object in a Class Module

The following class module demonstrates how to define an object. The sample &<<![endif]-->defines a property named Radius and a method named Draw.

```
' Declare a private global variable for storing the property called
"Radius"
Dim cirRadius As Double

' Define the initialization subroutine
Private Sub Class_Initialize
cirRadius = 99
End Sub

' Define the termination subroutine
Private Sub Class_Terminate
End Sub

' Define the "property get" function to retrieve the Radius property
Property Get Radius() As Double
Radius = cirRadius
End Property

' Define the "property let" procedure to change the Radius value
Property Let Radius(val As Double)
cirRadius = val
End Property

Sub Draw
' Method performs some action here
End Sub
```

Scripts that use the class module would access the object as follows (assuming the module is stored in the file CIRCLE.CLS and that the object name entered in the **Edit Object Module Properties** dialog is `MyCircleObject`)

```
'#Uses "circle.cls"
Sub Main
Dim x As New MyCircleObject
x.Radius = 7 ' sets the value of the Radius property
x.Draw ' calls the Draw method
Debug.Print x.Radius ' prints 7 in the immediate window
End Sub
```

Type Library References

Many application programs expose their services with objects. These objects may be used in your scripts, just as the **Surfer** automation objects are used. Before you can use another application's objects, you must add a reference the application's type library. A type library is a file that describes the objects, properties, and methods provided by an application. The **Surfer** type library is automatically referenced.

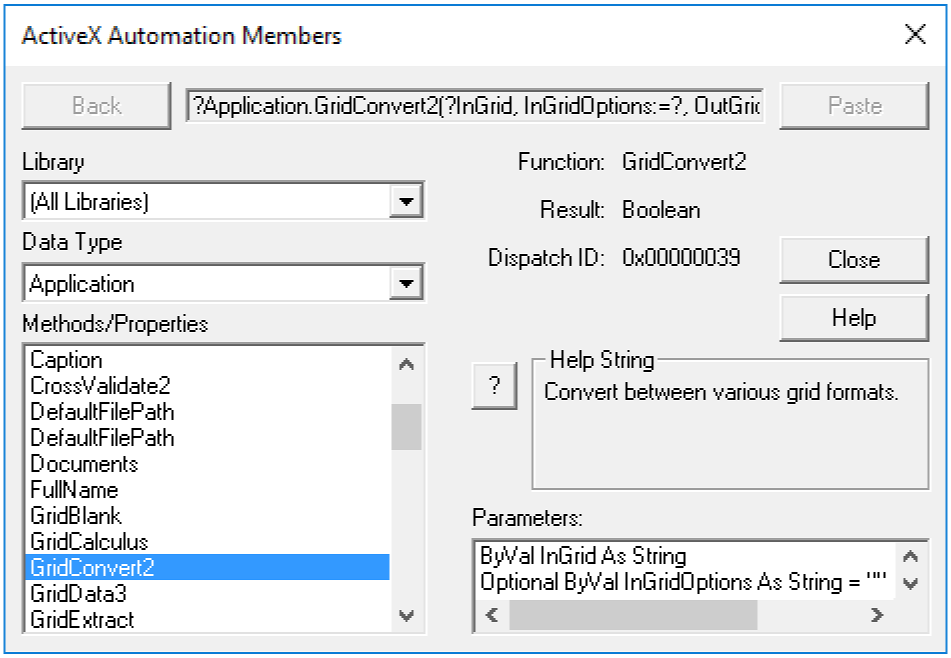
To add a type library reference to the current script module:

1. Select the **Edit | References** command to open the **References** dialog.
2. Click on the check box next to the type library whose objects you want to use in your script.
3. Click on the up and down arrows to adjust the relative priority of the checked references. If two libraries describe items with the same name, the item described in the higher-listed library is used by your script.
4. Click the OK button when finished adding type library references to your script.

The Object Browser

The **ActiveX Automation Members** dialog shows the methods and properties of the objects available to your script. This dialog provides a quick way to check the names of objects or to determine the argument list for a method.

Choose the **Debug | Browse** command to open the **ActiveX Automation Members** dialog.



*Use the **Debug | Browse** command in **Scripter** to open the **ActiveX Automation Members** dialog.*

Library

The *Library* list shows object libraries available for use in the current script. These are the same libraries checked in the **References** dialog. See the *Type Library References* for information about adding type library references in your script.

Data Type

The *Data Type* list shows objects available in the type library selected from the *Library* list.

Methods/Properties

The *Methods/Properties* list shows methods and properties available from the object selected in the *Data Type* list.

Browser Path Edit Box

The text box along the top of the dialog demonstrates the usage of the selected method or property. Click the *Paste* button to insert this sample into the code window.

Browser Info Area

Information about the item selected in the *Methods/Properties* list is shown along the right half of the dialog. The top line shows the selected item's type (function, property, or property get) and its name. The second line, labeled *Result*, shows the type of value returned by the method or property. The third line, labeled *Dispatch ID*, shows a value that uniquely identifies a property of method (this value is not used in scripts).

Follow Result

If a method or property returns a reference to another object in the type library, the *Result* label is replaced by a button labeled *Follow Value*. Click the *Follow Value* button to see the definition of the object returned by the method or property. After clicking the *Follow Value* button, the *Back* button is enabled. Click the *Back* button to return to the definition you were previously viewing.

Help String

The *Help String* group shows a short description of the item selected in the *Methods/Properties* list. Click the *?* button to view the help file associated with the type library. Not all type libraries have help files available.

Browser Parameter List

If the item selected in the *Methods/Properties* list is a method that accepts parameters, a *Parameters* list shows the names and types of all parameters used by the method.

Close

Click the *Close* button to dismiss the object browser dialog.

Variables

In **Scripter**, a variable is a symbolic name for a value. A variable name starts with a letter and may contain digits. Variable names cannot be the same as a reserved word. Because the **Scripter** code window displays variable names in black and reserved words in color, you can see when you have selected a variable name that conflicts with a reserved word.

Variables may be one of several types. The type of a variable determines what kind of data it may contain. See the following table for the possible variable types. In addition to the built-in data types, the **Scripter** language supports user-defined compound data types, user-defined enumeration types, and user-defined objects (defined in object modules and class modules).

The type of a variable is declared in a `DIM` statement. The syntax of a `DIM` statement is:

```
Dim varname As type
```

where `varname` is the name of the variable being declared and `type` is the variable's data type. Variables not declared in a DIM statement are a variant type, unless the variable name ends with one of the type-definition characters. If a variable name ends with one of the special type-definition characters, listed below, its type is recognized based on this character.

Type	Type-Definition Character	Description of Type
Integer	%	A 16-bit integer value
PortInt	?	(Portable Integer) A 16- or 32-bit integer value
Long	&	A 32-bit integer value
Single	!	A 32-bit floating-point value
Double	#	A 64-bit floating-point value
Currency	@	A 64-bit fixed-point value
String	$	A text string of any length
Byte	(none)	An 8-bit unsigned integer value
Boolean	(none)	A true or false value
Date	(none)	A 64-bit floating-point value
Object	(none)	A reference to an ActiveX object
Variant	(none)	Capable of holding any type of value

Using the `DIM` statement to declare the variable type is optional. Variables can be used without first being declared in a `DIM` statement, but this practice is not recommended for any script longer than a few dozen lines. To enforce this policy, an `OPTION EXPLICIT` statement should be placed at the top of long scripts. The `OPTION EXPLICIT` statement makes it an error to use any variable without first declaring it. Using this option lets you find typographical errors in variable names before a script is run. Without this option, typographical errors in variable names are usually detected only when the script fails to produce the expected results.

Next: Object Variables

Object Variables

In **Scripter**, object variables contain references to ActiveX objects. Creating the program *Application* object is an example of declaring an object variable:

```
Dim SurferApp As Object
Set SurferApp = CreateObject("Surfer.Application")
```

In this example, a `DIM` statement declares that the variable named `SurferApp` holds a reference to an object. The built-in `CreateObject` function returns a reference to a **Surfer** Application object, and the `SET` statement assigns this object reference to the `SurferApp` variable. Unlike variables of other types, which can be assigned new values simply with an equal sign (=), object variables must be assigned values with a `SET` statement.

Next: Array Variables

Array Variables

Array variables store a list or table of values. A single variable name refers to the entire collection, and individual values are distinguished by their numeric indices (their "subscripts"). The maximum number of values that can be stored in an array must be defined by a `DIM` statement. The elements of an array are accessed by using the variable name followed by a left parenthesis, the index of an array element, and a right parenthesis.

```
Dim month (11) As String
month (0) = "January"
month (1) = "February"
...
month (11) = "December"
```

Array subscripts begin with `zero`, unless an `Option Base` statement is used at the start of a script. Notice that in the previous example an array whose maximum subscript value is 11 actually has room for twelve elements because the subscripts start with zero.

The `Dim` statement can reserve only a constant number of elements for an array. If the maximum number of elements cannot be known in advance, a *dynamic array* may be used. A dynamic array is an array whose number of elements can be changed while a script is running. The `Redim` statement changes the maximum number of values that can be stored in a dynamic array. Refer to **Help | BASIC Language Help** in **Scripter** for mor information on `Dim` and `Redim`.

Next: User-Defined Types

User-Defined Types

A collection of related variables can be grouped together under one name. The TYPE statement defines the elements of a user-defined type.

```
Type measurement
     julianday As Integer
     level As Double
End Type
```

The TYPE definitions must appear at the top of a script file, before any subroutines. The TYPE...END TYPE statement defines a new type; it does not create a variable of that type. Variables of the user-defined type must be declared in a DIM statement. The elements of a user-defined type variable are accessed by using the variable name followed by a period and the element name:

```
Dim m As measurement
m.julianday = 192
m.level = 12.3
Debug.Print m.julianday ' prints 192 in the Immediate window
Debug.Print m.level ' prints 12.3 in the Immediate window
```

Next: Global Variables

Global Variables

In **Scripter**, variables declared in the body of a subroutine or function are available only within that procedure. If you want to share the same variable throughout a script file, then you can define it at the top of the file, before any subroutine definitions. Variables declared at the top of the file are available to all subroutines in the file; hence, they are called "global" variables.

The `PUBLIC` keyword may be substituted for the `DIM` keyword to allow a global variable to be used in other modules. For more information on modules, see **Code**, **Class**, and **Object Modules**.

Coordinate Arrays

Coordinates are passed to and from **Surfer** as arrays of doubles with alternating X and Y coordinates.

For example, the triangle specified by the coordinates (x1,y1), (x2,y2), (x3,y3) would be passed in an array with the elements arranged like so: x1,y1,x2,y2,x3,y3. **Surfer** is flexible about the dimension of the array so long as the X and Y coordinates are contiguous (no empty elements) and alternating. To use arrays in **Surfer** from within VB:

1. Singly dimensioned array:

```
Dim coordinates(1 to 6) As Double
coordinates(1)=x1
coordinates(2)=y1
coordinates(3)=x2
coordinates(4)=y2
coordinates(5)=x3
coordinates(6)=y3
```

2. Doubly dimensioned array:

```
Dim Points(1 To 2,1 To NumPoints) As Double
```

```
Points(1,1) = x1: Points(2,1) = y1
Points(1,2) = x2: Points(2,2) = y2
Points(1,3) = x3: Points(2,3) = y3
```

The first dimension is used for the X and Y coordinate, the second dimension refers to the vertex index.

3. You can use the **Array** statement for initialization, and then copy the results to a double array:

```
coordinates = Array(x1,y1,x2,y2,x3,y3)
dim Points(1 to 6) As Double
For i=1 to 6
Points(i) =
Next
```

Getting User Input

The **Scripter** language provides several predefined functions for prompting the script user for information. The `GetFilePath` function displays a standard Windows file open dialog. This allows the script user to select a file. The name of the selected file is returned by the function. The `InputBox` function allows the user to enter an arbitrary line of text. This function returns the line of text entered.

```
UserText$ = InputBox(Enter something here:) 'show prompt
Debug.Print UserText$ 'show line in Immediate window
```

This is a simple example of the InputBox function. The text entered in the box is returned to the script.

In addition to these simple input routines, **Scripter** supports user-defined dialogs. You can design your own dialogs and process their input in any manner you choose.

Creating Dialogs

Scripter contains a dialog editor that you can use to design customized dialogs. Select **Edit | UserDialog Editor** to visually design a dialog. You can control the size and placement of the components of the dialog, as well as customize the text included in the dialog.

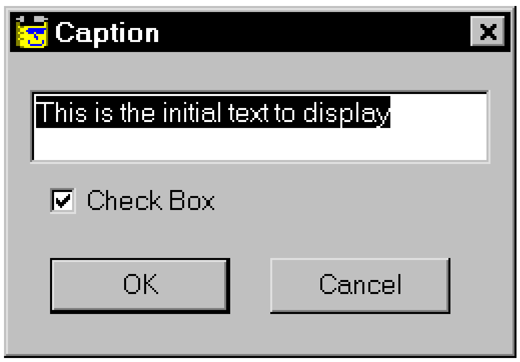
- **Dialog Function:** The dialog function is the name of a special function that is called when various events happen in a dialog. Define a dialog function to control the behavior of a dialog and to retrieve its input.
- **Field Name:** The field name is the name used to refer to a component.
- **Caption:**The caption is the text displayed within a component.
- **Quoted:** When not quoted, the caption property gives the name of variable that contains the text to display for the caption. When quoted, the caption property is the literal text to display.
- **Array of Items:** The array of items is the name of a string array variable that contains the strings to display in a list. The array variable must be initialized before the dialog is invoked.
- **Type:** The type is the behavior of some components varies depending on which option is selected. Refer to the online help for descriptions of the available component types.
- **Option Group:**For option buttons, the field name is used to refer to a group of option buttons. Only one option button within a group may be checked.
- **Comment:** A comment is the text to insert in the dialog definition block.

UserDialog Example

The following function demonstrates how to define, display, and extract the values entered in a user dialog.

```
Function MyInputBox As String
     ' Define the dialog template. This definition
     ' is inserted by the UserDialog editor.
     Begin Dialog UserDialog 250,112,"Caption"
          TextBox 10,14,230,28,.Text1
          CheckBox 20,49,160,14,Check Box,.Check1
          OKButton 20,77,90,21
          CancelButton 130,77,90,21
     End Dialog

     ' Declare a dialog variable
```

```
        Dim dlgvar As UserDialog

        ' Initialize the dialog controls
        dlgvar.Text1 = "This is the initial text to display"
        dlgvar.Check1 = True ' start with check box checked

        ' Display the dialog and wait for the OK or Cancel button to be
        pressed
        result = Dialog(dlgvar)

        ' Extract the information entered into the dialog
        If result = -1 Then ' check to see if OK button was pressed
             MyInputBox = dlgvar.Text1
             If dlgvar.Check1 Then Debug.Print "The Check Box was
             Checked!!"
        End If
End Function
```



This script creates the dialog displayed above.

To perform processing while a user dialog is active, define a special "dialog function." The dialog function will be called when various dialog events occur. To define a dialog function:

1. While designing the dialog, double-click in a blank portion of the dialog design area to activate the **Edit UserDialog Properties** dialog.

2. Enter a name for the *Dialog Function* property of the dialog. This property gives the name of a function that will be called when dialog events occur.

3. When you save the dialog, **Scripter** asks you if it should create a skeleton dialog function. Click the *Yes* button, and **Scripter** inserts the basic instructions for a dialog function into your script.

Refer to the DialogFunc help topic in theHelp | BASIC Language Helpfor more information about how to process dialog events in a dialog function.

Surfer Object Model

Surfer provides ActiveX Automation objects which allow scripts to control practically every feature of Surfer. These objects can be accessed from **Scripter** or from any Automation-enabled environment, such as Visual BASIC, Windows Scripting Host, or Excel.

Accessing Surfer

The means of accessing Surfer Automation objects varies depending on the scripting tool and language being used. With the Golden Software **Scripter** program and other applications compatible with Visual Basic, the **CreateObject** function creates a Surfer Application object:

```
Set x = CreateObject("Surfer.Application")
```

In this sample, a variable named "x" is assigned the value returned by the VBCreateObject function. The CreateObject function finds the name Surfer-.Application in the system registry, automatically activates Surfer, and returns a reference to the SurferApplication object. For an introduction to the Scripter programming language, see the Scripter BASIC Language help.

After creating an *Application* object, you can access other Surfer objects through the properties and methods of the *Application* object. Surfer supplies nearly 60 different kinds of ActiveX Automation objects. These objects are organized in a hierarchy, with the *Application* object at the root. This object hierarchy is known as an object model.

Methods and Properties

Objects have attributes associated with them. The Grid object, for example, has an attribute called "xMin" which represents the minimum X coordinate of the grid. An object's properties, also known as its **properties**, may be examined and changed by scripts. Some properties may be examined, but may not be changed. These are called read-only properties.

In addition to having properties, many Automation objects perform services. A *Grid* object, for example, will perform the "SaveFile" service when a script instructs it to do so. The services that an object performs are called its **methods** . Methods often require parameters to control how their services are performed. The *Grid* object's "SaveFile" method, for example, accepts two parameters: a file name and a file format code.

The way you access Surfer Automation objects' properties and methods varies depending on the programming language you use. In the Golden Software **Scripter** language, properties and methods are accessed by typing the name of an object variable, followed by a period, and the name of the property or method:

```
Set x = CreateObject("Surfer.Application")
```

```
x.Visible = True
```

This sample creates an *Application* object and sets the *Application* object's "Visible" property to the special "True" value predefined by the **Scripter** language.

You can find all the properties and methods which are available for each of the Automation objects, as well as a description of the objects themselves, in the online help. To view the online help, select either the **Help | Automation Help** command from the **Surfer** menu, or the **Help | Surfer Automation Help** command in **Scripter**. The **ActiveX Automation Members** dialog also shows you the available objects, properties, and methods. To view this dialog, select the **Debug | Browse** command in **Scripter**.

Collections

Several non-visible objects, called **collection objects**, are used to organize the object hierarchy. Collection objects are containers for groups of other, related, objects. For example, the *Overlays* collection contains the objects in a composite map. An *Overlays* collection object could contain a *BaseLayer* object, *Wireframe* object, and a *PostLayer* object if these three objects were overlaid together.

All collection objects, regardless of what the collection contains, can be processed using similar techniques. Thus, you retrieve an object from a collection of maps (an *Overlays* collection) the same way you retrieve an object from a collection of axes (an *Axes* collection). Collection objects also provide methods to create new objects.

Object Model Chart

The object model chart shows you which objects provide access to other objects in the hierarchy. Although the *Application* object is at the top of the hierarchy, not all objects are directly accessible from the *Application* object. To access many objects you must traverse from the *Application* object through one or more layers of sub-objects. People often refer to "drilling" or "boring" through the object hierarchy to describe this traversal through several objects to obtain an object you want to use.

To "drill through" the object hierarchy you must know which properties and methods of an object provide access to the next level of objects. Overview of Surfer Objects, discusses the most commonly used objects and the properties and methods which provide access to other objects in the hierarchy.

Every object represents a specific part of Surfer. For example, the Wireframe object represents a wireframe, and the Axis object represents a map axis. Some objects do not represent a visible entity, but organize other objects into groups. The Shapes object, for example, provides the means to create new maps and new drawing primitives (rectangles, symbols, polygons, etc.), but the *Shapes* object itself is never visible.

Overview of Surfer Objects

Learning to use the Surfer Automation objects in a script may appear daunting at first. Most tasks, however, can be accomplished using just a few Surfer objects. Once you become familiar with these primary objects and learn how to "drill through" the object hierarchy, you will be able to access most of Surfer**'s** features from your scripts.

The online help is the complete reference for all of the Surfer Automation objects, their properties, and their methods. The object model chart should serve as your guide for navigating through the object hierarchy. Use the **Help | Automation Help** command in Surfer, or the **Help | Surfer Automation Help** in **Scripter** to see the online help reference.

This section shows how to access the Surfer Automation objects using the Scripter BASIC language. If you are not familiar with computer programming, you may benefit from a programming tutorial. See the Suggested Reading topic for recommendations. Refer to the online help *Writing Scripts* topic for more information about **Scripter** BASIC.

A Brief Introduction to the Major Surfer Objects

The following will introduce some of the most important Surfer automation objects. This is by no means comprehensive. Please see Objects, Properties, and Methods for a complete list.

Application Object

The **Application** object represents the Surfer program. All navigation through the Surfer object model begins with the **Application** object. The **Application** object is a single instance of Surfer and it is the root of all objects in Surfer. External programs will typically create an instance of the **Application** object during initialization. In VB this is done using the function as in:

```
Set SurferApp = CreateObject("Surfer.Application")
```

The CreateObject function activates a new instance of Surfer, and returns a reference to the **Application** object to your script. If Surfer is already running, and you do not want to start a new instance of Surfer, use the VB GetObject function instead of CreateObject:

```
Dim SurferApp As Object
Set SurferApp = GetObject(,"Surfer.Application")
```

The GetObject function obtains the **Application** object from the currently running instance of Surfer. If Surfer is not already running, the function will fail. Call the **Application** object’s Quit method to close Surfer from a script.

When Surfer is started by a script, its main window is initially hidden. To make the Surfer window visible, you must set the **Application** object's Visible property to True:

```
Set SurferApp = GetObject(,"Surfer.Application")
SurferApp.Visible = True
```

Most of the gridding-related operations are methods of the Application object. Use the GridData, GridFunction, GridMath, GridFilter, GridSplineSmooth, GridAssignNoData, GridConvert, GridVolume, GridSlice, GridTransform, GridExtract, and GridCalculus methods to create new grids and perform computations with grids.

Other methods and properties of the **Application** object let you move and resize the Surfer window, adjust the main window state (maximized, minimized, hidden), change the window caption, and modify preference settings.

The **Application** object provides two important collections that allow access to the next level of objects in the hierarchy. Use theDocumentsproperty to obtain a reference to the **Documents** collection object, and use the Windows property to obtain a reference to the **Windows** collection object. The **Documents** and **Windows** collection objects provide access to the other objects in the object hierarchy. You can access the currently active document and window objects with the **Application** object's ActiveDocument and ActiveWindow properties.

Set the ScreenUpdating property to False to disable screen updating and make your scripts execute more quickly. Turning off screen updating can greatly increase the performance of Automation when you want to perform a number of actions that cause screen redraws.

See Application Object for additional information.

Documents Collection

The **Documents** collection contains all open documents. The **Documents** collection provides the means to access plot and worksheet documents. Use the Add method to create new plot or worksheet documents. Surfer provides the pre-defined values srfDocPlot and srfDocWks for specifying the type of document to create:

```
Set srf = CreateObject("Surfer.Application")
srf.Documents.Add srfDocPlot ' Create a new, blank plot document
srf.Documents.Add srfDocWks ' Create a new, blank worksheet document
```

The Add method returns a reference to the **Document** object that was created. New grid documents are not created with the *Documents* collection Add method. You can create grid files by calling the Application object's grid-related objects.

Use the Open method to open existing plot, worksheet, and grid files. The Open method returns a reference to the opened document.

Use the Item method to retrieve a reference to an open plot or worksheet document object. When iterating through a collection of documents, the document Type and Index properties can be used to determine the type and position of the document within the collection.

See Documents Collection for additional information.

Windows Collection

The **Windows** collection provides access to all windows within Surfer. The **Windows** collection returned by the Application object contains all the windows in the application, including the plot window, the worksheet window, and the grid editor views. To create new **Window** objects, call a Document object's NewWindow method.

See Windows Collection for additional information.

PlotDocument Object

A **PlotDocument** object corresponds to an SRF file. **PlotDocument** objects are created by the Documents.Add or Documents.Open methods of the Application object. All drawing objects within the document are stored in the Shapes collection. All selected objects are part of the Selection collection.

The **PlotDocument** provides access to all the drawing and map objects. Use methods and properties in the **PlotDocument** object to save Surfer files [.SRF], to import and export files, to print, and to change the default line, fill, font, and symbol for a plot.

Use the Shapes, Selection, and Window properties to access the collection objects having the same names. These collection objects provide access to the next layer of objects in the object hierarchy. Use the Shapes collection object to create new drawing and map objects.

See PlotDocument Object for additional information.

WksDocument Object

The **WksDocument** object represents a worksheet document. WksDocument objects are created by the Documents.Add or Documents.Open methods. Cells within a worksheet are accessed through the Cells, Rows, and Columns methods. The active cell and current selection are accessed through a WksWindow object. You can perform a mathematical transformation on worksheet cells and merge files together with the WksDocument object. The WksRange object can be accessed using the Cells, Rows, Columns, and UsedRange properties.

See WksDocument Object for additional information.

PlotWindow, WksWindow, and GridWindow Objects

It is important to distinguish between windows and documents. Document objects represent a file on disk containing the actual data to be represented. Windows are used to view that data. There can be multiple windows viewing the same document at once.

The Window object encapsulates all the methods and properties associated with a Surfer window. Methods exist to open, close, move and size the window.

When a document is closed, all windows viewing that document are closed as well. The reverse relationship exists as well. When the last window viewing a document is closed the document is closed.

See PlotWindow, WksWindow, and GridWindow Objects for additional information.

Shapes Collection

Within Automation, graphic drawing and map objects are called **Shapes**. The **Shapes** collection contains all of the drawing and map objects in a document. This object is also used to create new shapes and add them to the collection. Methods exist to enumerate all the shapes in the collection, add objects to the Selection collection, and create new drawing and map shapes.

One of the most common uses of the **Shapes** collection is to create new shapes. There is a unique method to create each type of shape. For example, to create a contour map use the AddContourMap method, as in:

```
Sub Main

Set SurferApp = CreateObject("Surfer.Application")
SurferApp.Visible = True

'Create a new, empty plot document
Set Doc = SurferApp.Documents.Add srfDocPlot

'Get the Shapes collection
Set Shapes = Doc.Shapes

'Create the contour map
Set MapFrame = Shapes.AddContourMap(Demogrid.grd)

End Sub
```

See Shapes Collection for additional information.

Selection Collection

The **Selection** collection contains all the properties and methods associated with the collection of selected shapes. The **Selection** collection defines methods that operate on one or more selected shapes. Most commands within Surfer that require selected objects are part of the **Selection** collection.

To select a particular shape, set the shape's Selected property to True. When a **Shape** object is selected, it is added to the **Selection** collection. When it is deselected, it is removed from the **Selection** collection. Note that the shape is neither created nor destroyed by selection operations. It is merely added or removed from the **Selection** collection.

The **Selection** collection allows all selected shapes to be enumerated, and contains most methods that operate on multiple objects. This includes moving and sizing one or more shapes; cut, copy, and paste; and overlaying and stacking multiple maps.

See Selection Collection for additional information.

MapFrame Object

You create new maps using the Add<map type>Map methods of the Shapes collection (AddVectorBaseMap, AddReliefMap, AddContourMap, and so forth). These methods return a reference to a MapFrame object - not, as you might expect, a reference to the object of the specific map type that was created. The **MapFrame** object contains the coordinate system, axes, and all properties common to the maps contained within it. The **MapFrame** defines the coordinate system and axes for one or more overlays.

You add layers to an existing **MapFrame** object with the Add<map type>Layer methods (such as AddContourLayer). These methods return a reference to the specific map layer that has been created.

When a new map object is created, usually the **MapFrame** object is returned rather than the contained overlay. For example, Shapes.AddContourMap returns the containing **MapFrame** object rather than the contour map overlay. To access the overlay within the **MapFrame**, use the Overlays collection as in:

```
Set MapFrame = Shapes.AddContourMap(demogrid.grd)
Set ContourMap = MapFrame.Overlays(1)
```

Use the Overlays property of the **MapFrame** to obtain the **Overlays** collection object. The **Overlays** collection contains all the individual map objects associated with a map frame. When a new map frame is created, the **Overlays** collection contains just one map item. To obtain a reference to a newly created map, access the first item of the collection:

```
Sub Main

Set SurferApp = CreateObject("Surfer.Application")
```

```
SurferApp.Visible = True

'Create a blank plot
Set plot = SurferApp.Documents.Add srfDocPlot

'Add a new contour map. This will return a MapFrame object
Set mapframe = plot.Shapes.AddContourMap demogrid.grd

'Get a ContourMap object from the MapFrame
Set contourmap = mapframe.Overlays(1)

End Sub
```

Use the **MapFrame** object to set map limits, adjust the three-dimensional view, adjust the scale, and change the size of the map. Access the Axes collection with the **MapFrame** object's **Axes** property. Use the **Axes** collection object to obtain the individual **Axis** objects associated with a map. Access the **ScaleBars** collection through the **MapFrame** object's ScaleBars property. Use the **ScaleBars** collection to add scale bars to a map and to access the **ScaleBar** objects associated with a map.

See MapFrame Object for additional information.

Derived Objects

Several objects shown in the object list share common features. For example, the PlotDocument object and WksDocument object each provide SaveAs, Activate, and Close methods. These common features are inherited from a predecessor Document object.

Derived objects inherit all the properties and methods of the predecessor object. In the online reference, the help topic for derived objects shows just the properties and methods unique to the object. The predecessor object is accessed through the *Derived from* link. Remember that all the properties and methods of a predecessor object are available as well. Derived objects and their predecessor objects include:

Document Object

The Document object is a predecessor of the PlotDocument and WksDocument objects.

Window Object

The Window object is a predecessor of the PlotWindow, WksWindow, and GridWindow objects.

Shape Object

The Shape object is a predecessor of all objects which can be moved and resized. These include the MapFrame, Variogram, Composite, all of the basic drawing objects (Rectangle, Ellipse, Symbol, Text, Polyline, Polygon, and RangeRing), various map component objects (Axis, ScaleBar, Profile, VectorLegend and PostLegend), and the predecessor objects listed below (Layer, Legend, and ColorScale objects).

Layer Object

The Layer object is a predecessor of all the map layer objects (ClassedPostLayer, ContourLayer, ColorReliefLayer, PostLayer, RasterBaseLayer, ReliefLayer, VectorBaseLayer, VectorLayer, ViewshedLayer, and WatershedLayer) and the Graticule object.

Legend Object

The Legend object is a predecessor of the VectorLegend and PostLegend objects.

ColorScale Object

The ColorScale object is a predecessor of the ContinuousColorScale and DiscreteColorScale objects.

Using Collection Objects

Surfer groups most objects in collections. Although these collections contain different types of data, they can be processed using similar techniques.

Count Property

All collection objects have a read-only property named Count which gives the number of objects in the collection.

Item Method

Every collection also has a method called Item which retrieves one of the objects contained in the collection. The Item method accepts a single argument specifying either an index number or (for most collections) the name of the object to retrieve. An index number is a value between 1 and the value returned from the collection’s "Count" property. The name used to identify items in a collection varies:

- In a Documents collection, the individual Document objects are identified by the file name (or the caption, if the document has not been saved in a file yet).

- In a Windows collection, individual Window objects are identified by the window caption.
- In collections containing objects derived from the Shape object (Shapes, Selection, Axes, ScaleBars, and Overlays collection objects), the individual *Shape* objects are identified by the object ID assigned to the object.
- In the Levels collection, individual Level objects may be retrieved by their index number only. The *Level* objects cannot be retrieved by name.

A shorthand syntax for retrieving items from a collection is to enclose the index or name of the desired object in parentheses immediately following the name of a collection object (similar to accessing an element in an array). Thus, the following two instructions have the same effect:

```
' Assume "docs" is a variable holding a reference to the Documents collection
docs.Item(1) ' retrieves the first object in the collection
docs(1) ' shorthand, retrieves the first object in the collection
```

Objects contained in collections are typically derived from a base object. When you use the Item method to retrieve an object from a collection, the method returns a reference to the base object. To determine the actual type of the returned object, most base objects contain a Type method. For example, the *Windows* collection may contain *PlotWindow*, *WksWindow*, or *GridWindow* objects. When you retrieve a window from the *Windows* collection, a reference to a generic *Window* object is returned. Use the Window.Typeproperty to determine if the returned object is a PlotWindow, WksWindow, or GridWindowobject.

Add Method

Many collections have one or more Add methods for creating new items and adding them to the collection. For example, to add a rectangle to the *Shapes* collection, you would use the AddRectangle method:

```
Shapes.AddRectangle 2, 2, 4, 4
```

Close and Delete Methods

The objects contained by collections are automatically removed when the contained object is deleted or closed. For example, call a *Document* object’s Close method to close a document or call the *Shape* object’s Delete method to remove any *Shape*-derived object.

Parent and Application Properties

Every Automation object provides a Parent and an Application property. The Application property returns a reference to the top-level Application object. This is a convenient way to access the *Application* object, particularly when passing Automation objects as arguments to subroutine and function calls.

The Parent property returns a reference to the collection object that an object is contained in, or the controlling object. If an object is not contained by a collection object, the Parent property typically returns a reference to the *Application* object.

PlotWindow, WksWindow, and GridWindow Objects

It is important to distinguish between windows and documents. Document objects represent a file on disk containing the actual data to be represented. Windows are used to view that data. There can be multiple windows viewing the same document at once.

The Window object encapsulates all the methods and properties associated with a Surfer window. Methods exist to open, close, move and size the window.

When a document is closed, all windows viewing that document are closed as well. The reverse relationship exists as well. When the last window viewing a document is closed the document is closed.

Using Surfer Objects

To access Surfer commands from your script you must create a Surfer *Application* object. To create an *Application* object, call the CreateObject function with `Surfer.Application` as the argument. The *Application* object is the root of an object hierarchy. Access all other Surfer Automation objects though the properties and methods of the *Application* object.

Every object has properties and methods associated with it. **Properties** are values describing the state of an object. **Methods** are actions an object can perform. Access properties and methods by typing the name of an object variable, followed by a period, followed by the property or method name.

You can use object properties as you would use variables: assign values to properties, branch based on the value of a property, or use the value of a property in calculations. You call an object's methods as you would call subroutines and functions. Use the return values from methods the same as you would use return values from functions.

When you "drill through" the object hierarchy, you can store references to intermediate objects in variables, or you can string together long sequences of object references. For example, you can set the default font for a plot document in a single line:

```
' Assume "SurferApp" is a variable holding a reference to the Applic-
ation object
SurferApp.Documents.Item(1).DefaultFont.Bold = True
```

Alternatively, you can store each intermediate object in variables as you traverse the object hierarchy:

```
' Assume "SurferApp" is a variable holding a reference to the Applic-
ation object
Set docs = SurferApp.Documents
Set plot = docs.Item(1)
Set font = plot.DefaultFont
font.Bold = True
```

The second form - storing intermediate objects - is more efficient if you are performing several actions with the same object. A third alternative is to use the `WITH…END WITH` statement:

```
' Assume "SurferApp" is a variable holding a reference to the Applic-
ation object
With SurferApp.Documents.Item(1).DefaultFont
.Bold = True
.Size = 12
.Color = SurferAppColorBananaYellow
End With
```

Object List

The following table provides a list of all the **Surfer** automation objects. The table also provides the interfaces that are dispatched when writing a late-binding script for **Surfer**.

The interfaces from which the current interfaces have been extended are included in the table, if applicable. If you are writing an early binding script and would like to use the properties and methods available in the latest version, be sure to dispatch the current interface.

The current interfaces include all of the properties and methods available in **Surfer**. The properties and methods specific to the former interfaces can be found in the older application help files.

Object	Interface (current)	Interface (former)
Application	IApplication5	IApplication4
Axes	IAxes	
Axis	IAxis2	IAxis

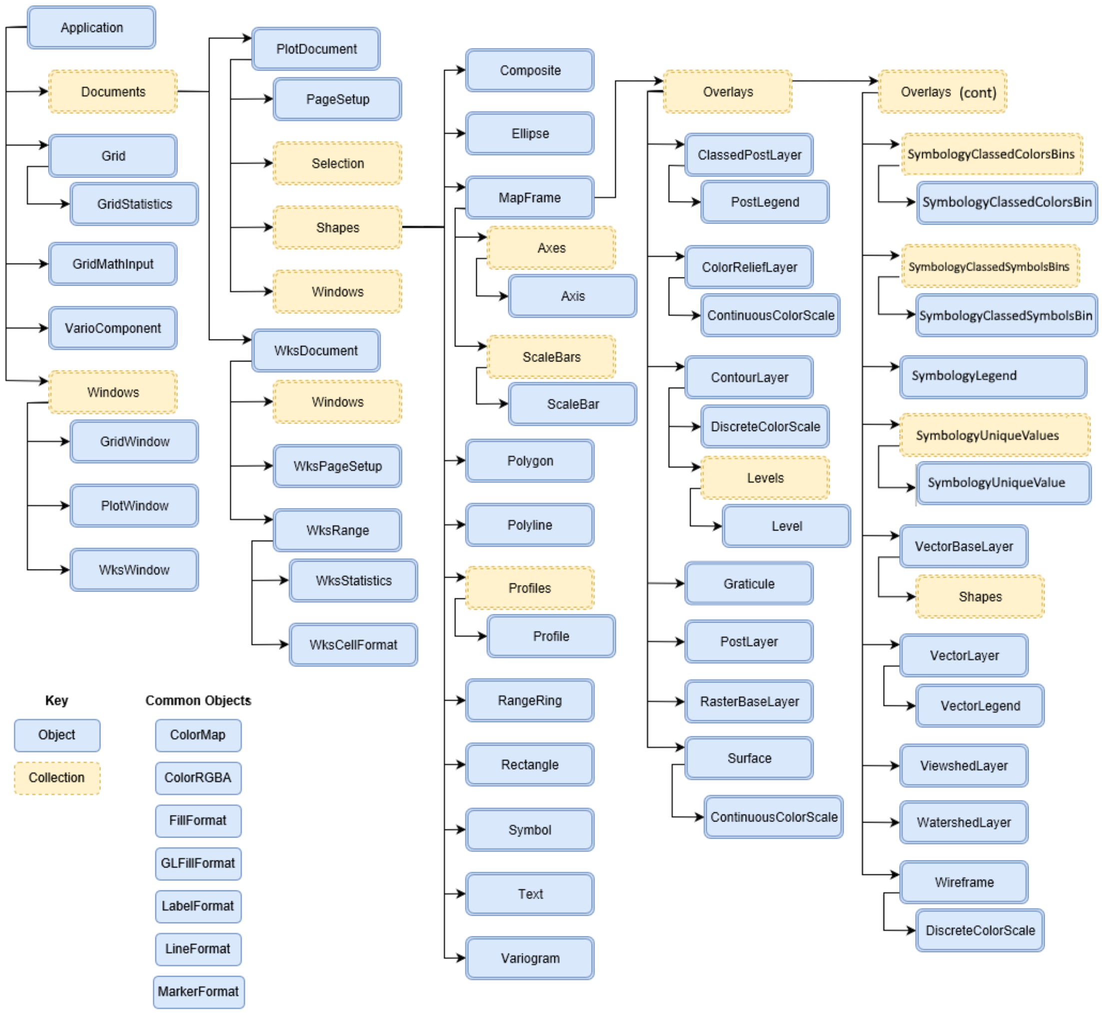
ClassedPostLayer	IClassedPostlayer2	IClassedPostLayer, IClassedPostMap
ColorMap	IColorMap2	IColorMap
ColorNode	IColorNode	
ColorNodes	IColorNodes	
ColorReliefLayer	IColorReliefLayer2	IColorReliefLayer, IImageLayer, IImageMap
ColorRGBA	IColorRGBA	
ColorScale	IColorScale	
Composite	IComposite2	IComposite
ContinuousColorScale	IContinuousColorScale	
ContourLayer	IContourLayer	IContourMap
DiscreteColorScale	IDiscreteColorScale	
Document	IDocument	
Documents	IDocuments	
Ellipse	IEllipse	
FillFormat	IFillFormat	
FontFormat	IFontFormat	
GLFillFormat	IGLFillFormat	
Graticule	IGraticule	
Grid	IGrid3	IGrid2
GridMathInput	IGridMathInput	
GridStatistics	IGridStatistics	
GridWindow	IGridWindow	
ImageLayer	IImageLayer2	
LabelFormat	ILabelFormat3	ILabelFormat2
Layer	ILayer	
Legend	ILegend	
LegendLayers	ILegendLayers	
Level	ILevel	
Levels	ILevels	
LineFormat	ILineFormat	
MapFrame	IMapFrame3	IMapFrame2
MarkerFormat	IMarkerFormat	
Overlays	IOverlays	
PageSetup	IPageSetup	
PlotDocument	IPlotDocument3	IPlotDocument2
PlotWindow	IPlotWindow	
Polygon	IPolygon	

Polygon3D	IPolygon3D	
Polyline	IPolyline2	IPolyline
Polyline3D	IPolyline3D	
PostLayer	IPostLayer2	IPostLayer, IPostMap
PostLegend	IPostLegend2	IPostLegend
Profile	IProfile	
Profiles	IProfiles	
RangeRing	IRangeRing	
RasterBaseLayer	IRasterBaseLayer2	IRasterBaseLayer
Rectangle	IRectangle	
ReliefLayer	IReliefLayer	
Ruler	IRuler	
ScaleBar	IScaleBar3	IScaleBar2, IScaleBar
ScaleBars	IScaleBars	
Selection	ISelection3	ISelection2, ISelection
Shape	IShape	
Shapes	IShapes7	IShapes6, IShapes5, IShapes4, IShapes3, IShapes2, IShapes
Surface	ISurface	
Symbol	ISymbol	
Sym-bologyClassedColorsBin	ISym-bologyClassedColorsBin	
Sym-bologyClassedColorsBins	ISym-bologyClassedColorsBins	
Sym-bologyClassedSymbolsBin	ISym-bologyClassedSymbolsBin	
Sym-bologyClassedSymbolsBins	ISym-bologyClassedSymbolsBin	
Text	IText	
TopLegend	ITopLegend	
VarioComponent	IVarioComponent	
Variogram	IVariogram	
VectorBaseLayer	IVectorBaseLayer2	IVectorBaseLayer
VectorLayer	IVectorLayer	IVectorMap
VectorLegend	IVectorLegend2	IVectorLegend
ViewshedLayer	IViewshedLayer	
WatershedLayer	IWatershedLayer2	IWatershedLayer

Window	IWindow	
Windows	IWindows	
Wireframe	IWireframe	
WksCellFormat	IWksCellFormat	
WksDocument	IWksDocument	
WksPageSetup	IWksPageSetup	
WksRange	IWksRange	
WksStatistics	IWksStatistics	
WksWindow	IWksWindow	
WksWindows	IWindows	
Wireframe	IWireframe	
WksCellFormat	IWksCellFormat	
WksDocument	IWksDocument	
WksPageSetup	IWksPageSetup	
WksRange	IWksRange	
WksStatistics	IWksStatistics	
WksWindow	IWksWindow	

Object Hierarchy

The following image shows the **Surfer** automation object model. To access a particular object you must traverse the object hierarchy using the methods and properties provided by the objects. This chart shows objects that provide access to other objects. Collection objects are containers for groups of related objects. Non-container objects represent a specific part of **Surfer**.



Improve Automation Performance

Turning off screen updating can greatly increase the performance of Automation when you want to perform a number of actions that cause screen redraws. You can achieve this efficiency by setting ScreenUpdating to false, performing the repetitive operations, and then turning *ScreenUpdating* back on. When *ScreenUpdating* is turned back on, all windows will be redrawn in the new state.

Automation Examples

In addition to the examples found in the Table of Contents *Examples* book within the *Surfer Automation* book, there are many more examples for your use.

- The SCRIPTS folder in the **Surfer** directory contains sample [.BAS] files to use in **Scripter**. By default, the SCRIPTS folder is located at: C:\Program Files\Golden Software\Surfer\Samples\Scripts.
- In addition, each object, method, and property contains an example in the Automation help. The methods and properties contain only a few lines for the example.
- Click on the *Used By* link for an example on how to tunnel through the hierarchy to the method or property.
- Some methods and properties contain a link showing a full example of the object.
- Sample scripts are available for download at the Where Can I Find Surfer Script Examples article in the **Surfer** support Knowledge Base.

Creating and Printing a Contour Map

This example automates the process of creating a contour map to visualize changes in groundwater levels. The script prompts the user for a data file, creates a grid from the file, creates a contour map, prints the map, and saves the map in a Surfer file [.SRF].

```
Sub Main
'Declare object and string variables used in the script
Dim SurferApp, Plot, ContourMapFrame,ContourLayerAs Object
Dim InFile, GridFile,

'Create the Surfer Application object and assign it to the "SurferApp"
variable
Set SurferApp = CreateObject("Surfer.Application")
SurferApp.Visible = True 'Make Surfer visible

'Prompt the user for the name of the data file to process.
InFile = GetFilePath("","DAT;TXT;CSV;XLS",CurDir(), Select data file,
0)
If InFile = "" Then End 'Can't continue: no file was selected

'Get file name without the extension (for example, "week5.txt" becomes
"week5")
BaseName = InFile
ExtStart = InStrRev(InFile,.)
If ExtStart 1 Then

'Create a grid from the specified data file using the Kriging
algorithm
GridFile = BaseName + .
```

```
SurferApp.GridData DataFile:=InFile, Algorithm:=srfKriging, _
DupMethod:=srfDupNone, ShowReport:=False, OutGrid:=GridFile

'Create a plot document in Surfer and assign it to the variable named
"Plot"
Set Plot = SurferApp.Documents.Add(srfDocPlot)

'Create a contour map. Assign the map frame to the "ContourMapFrame
variable
Set ContourMapFrame =

'Assign the contour map properties to the variable named "Con-
tourLayer"
Set ContourLayer = ContourMapFrame.Overlays(1)

ContourLayer.FillContours = True 'Fill the contour map levels
Plot.PrintOut 'Print the page on the default printer
Plot.SaveAs(BaseName + ".srf") 'Save the map

End Sub
```

Opening, Saving, and Closing Documents

The Documents collection provides access to the **Surfer** file commands. Use the *Document* object’s "Open" method to open an existing plot, worksheet, or grid file. Use the Add method to create a blank plot or worksheet. The SaveAll method saves all open documents, and the CloseAll method closes all open documents. To save or close an individual document, use the *Document* object’s Save, SaveAs, and Close methods.

```
Sub Main

     Set SurferApp = CreateObject("Surfer.Application")
     SurferApp.Visible = True

     ' Create a blank plot
     SurferApp.Documents.Add srfDocPlot

     ' Create a blank worksheet
     SurferApp.Documents.Add SurferAppDocWks

     ' Open an existing plot
     filename$ = GetFilePath(,SRF)
     if filename$ Then
     SurferApp.Documents.Open filename$
     End If

     ' Open the sheet named "Sheet1" from an Excel file
     filename$ = GetFilePath(,XLS)
```

```
        If filename$ Then
        SurferApp.Documents.Open filename$, Sheet=Sheet1
        End If

        ' Close the active document
        SurferApp.ActiveDocument.Close

        ' Save the active document using its current name
        If Not SurferApp.ActiveDocument.Saved Then
        SurferApp.ActiveDocument.Save
        End If

        ' Save the document whose window caption is "Plot1"
        SurferApp.Documents("Plot1").SaveAs "MyDocument

        ' Close all documents
        SurferApp.Documents.CloseAll

End Sub
```

Creating a Variogram with Scripter

The Variogram object represents a variogram. A *Variogram* object is a type of Shape object. Create a new *Variogram* using the Shapes collection's AddVariogram method. The AddVariogram method returns a reference to a *Variogram* object. Use the properties and methods of the *Variogram* object to access information about the computed variogram and to modify the options.

A VarioComponent object represent one component of a variogram model. When you use the Kriging algorithm to create a grid file, you can specify the variogram model to use by passing an array of *VarioComponent* objects to the Application object's GridData method. Create a *VarioComponent* by calling the *Application* object's NewVarioComponent method. Use the *Variogram* object's Model property to obtain the array of *VarioComponent* objects used by a variogram.

The following subroutine creates a *Variogram* object, and then creates a grid file using the computed variogram components.

```
Sub GridWithComputedVariogram(plot As Object,
' Create a variogram.
Set v = plot.Shapes.AddVariogram (

' Create a grid using the computed variogram
plot.Application.GridData DataFile:=datafile, Algorithm := Surfer-
AppKriging, _
KrigVariogram := v.Model, ShowReport := False, OutGrid := grid

' Delete the variogram
v.Delete
```

```
End Sub
```

The next subroutine creates two *VarioComponent* objects and creates a grid based on these variogram components.

```
Sub GridWithStaticVariogram(plot As Object, datafile As String, grid
As String)
' Get the Application object
Set SurferApp = plot.Application

 'Create two VarioComponent objects and store them in an array
Dim components(1 To 2) As Object
Set components(1) = SurferApp.NewVarioComponent(Surfer-
AppVarNugget,1.0,2.0,3.0)
Set components(2) = SurferApp.NewVarioComponent(Surfer-
AppVarLinear,1.0,2.7,1.5)

'Create a grid using the two variogram components
SurferApp.GridData DataFile:=datafile, Algorithm := SurferAppKriging,
_
KrigVariogram := components, ShowReport := False, OutGrid := grid
End Sub
```

Finally, the following Main procedure demonstrates how you might use the preceding subroutines:

```
Sub Main
' Start Surfer, set the default directory, and create a plot document
Set SurferApp = CreateObject("Surfer.Application")
SurferApp.Visible = True
SurferApp.DefaultFilePath = SurferApp.Path + "\samples"
Set plot = SurferApp.Documents.Add(srfDocPlot)

' Create a grid using a pre-determined variogram model
GridWithStaticVariogram plot, "demogrid.dat, VarioDemo1

' Create a grid using a computed variogram model
GridWithComputedVariogram plot, "demogrid.dat, VarioDemo2

' Create contour maps to compare the two grids
plot.Shapes.AddContourMap "VarioDemo1"
plot.Shapes.AddContourMap "VarioDemo2"

End Sub
```

Overlaying Maps with Automation

The overlay map script is compatible with **Surfer 8** and below. In **Surfer 9** use add map layers. In **Surfer 10** and above, use one of the add map layers methods (such as AddVectorBaseLayer) or use the OverlayMaps method.

To overlay maps in a script, you must first select the select the map or MapFrame objects to be overlain, and then call the Selection collection's OverlayMaps method. The following subroutine demonstrates how to overlay maps:

```
Sub OverlayMapsExample(mapframe1 As Object, mapframe2 As Object)
' Retrieve the parent PlotDocument object
Set plot = mapframe1.Parent

' Clear all selections and then select the two MapFrame objects
plot.Selection.DeselectAll
mapframe1.Selected = True
mapframe2.Selected = True

' Overlay the selected maps
plot.Selection.OverlayMaps
End Sub
```

The subroutine would be used like this:

```
Sub Main
' Start Surfer and create a blank plot document
Set SurferApp = CreateObject("Surfer.Application")
SurferApp.Visible = True
Set plot = SurferApp.Documents.Add(srfDocPlot)

' Create a contour map and a vector map
gridfile$ = SurferApp.Path + \samples\demogrid.grd
Set map1 = plot.Shapes.AddContourMap(gridfile$)
Set map2 = Plot.Shapes.AddVectorMap(gridfile$)

' Call the example subroutine to overlay the maps
OverlayMapsExample map1, map2

End Sub
```

Stacking maps is similar to overlaying them. Call the **Selection** collection's StackMaps method rather than the OverlayMaps method.

Modifying Axes

Axis settings may be changed using the Axis object. You access *Axis* objects through the Axes collection object, which in turn is accessed through a MapFrame object. The following example does not use subroutines, but you could write the script with subroutines instead.

```
Sub Main
' Start Surfer and create a blank plot document
Set SurferApp = CreateObject("Surfer.Application")
SurferApp.Visible = True
Set plot = SurferApp.Documents.Add(srfDocPlot)

' Create a contour map
gridfile$ = SurferApp.Path + \samples\demogrid.grd
Set mapframe = plot.Shapes.AddContourMap(gridfile$)

' Turn off screen redrawing to speed up this procedure
old_update_setting =
mapframe.Application.ScreenUpdating = False

' Adjust the left and bottom axes
For Each axis In mapframe.Axes
If axis.AxisType = SurferAppATLeft Or
With axis
.MajorTickType = SurferAppTickCross
.MinorTickType = SurferAppTickIn
.MinorTickLength = 0.1
.MinorTicksPerMajor = 9
.ShowMajorGridLines = True
.AxisLine.Width= 0.03
End With
End If
Next

' Restore the previous screen setting
mapframe.Application.ScreenUpdating =
End Sub
```

Appendix A - Mathematical Functions

Mathematical Functions

Mathematical Functions are used to modify data with the grid Function, Math, Grid Data, and Variogram commands in the plot window, and with the Transform command in the worksheet window.

Data Types

The expression evaluator supports 32-bit signed integer numbers, double-precision floating-point numbers, a boolean value, a text string of 0 to 256 characters, and time stamp values.

Variable Names

Variable names must begin with a column letter (i.e. A), row number (i.e. _1), or cell location (i.e. A2), which may be followed by other letters, numbers, or underscores (_), up to a maximum of 256 characters per variable name.

The variable names are not case sensitive. For example, **sum(a..z)**, **sum(A..z)**, and **sum(A..Z)** all refer to the same variable.

Precedence

The mathematical expression can consist of constants, variables (such as column letters), or functions (outlined below). The formulas follow standard precedence rules. Spaces are used in the equation for clarity.

Formulas are specified using standard precedence rules. Operators, in order of decreasing precedence, are:

( )	parentheses
-	minus (or negative sign)
* /	multiplication and division
+ -	addition and subtraction

The expression evaluator treats operators with the following precedence:

1. !, NOT, ~
2. ^
3. *, /, %
4. +, -
5. <<, >>
6. <, >, <=, >=

7. ==,!=,<>
8. &
9. XOR
10. |
11. &&, AND
12. ||, OR
13. ?:
14. IF

Operators of equal precedence are evaluated from left to right within the equation. Parentheses are used to override precedence, and expressions with the parentheses are performed first.

The following built-in functions are supported.

Trigonometric Functions

All trigonometric functions are carried out in radians. If the data are in degrees, use the d2r(x) conversion function (in the *Miscellaneous Functions* section below) to convert degree data to radians and then use the trigonometric functions.

sin(x)	sine of angle x
cos(x)	cosine of angle x
tan(x)	tangent of angle x, the value of x must not be an odd multiple of $\pi/2$
asin(x)	Arcsine in the range $-\pi/2$ to $\pi/2$, x must be between -1 and 1
acos(x)	Arccosine in the range 0 to π, x must be between -1 and 1
atan(x)	Arctangent in the range $-\pi/2$ to $\pi/2$
atan2(y,x)	Arctangent of y/x in the range $-\pi$ to π

Bessel Functions

j0(x) j1(x) jn(n,x)	Bessel functions of the first kind at x of orders 0, 1, and n, respectively
y0(x) y1(x) yn(n,x)	Return the Bessel functions of the second kind at x, of orders 0, 1, and n, respectively. For y0, y1, and yn, the value of x must not be negative.

Exponential Functions

exp(x)	exponential function of x (e to the x)
sinh(x)	hyperbolic sine of angle x
cosh(x)	hyperbolic cosine of angle x

tanh(x)	hyperbolic tangent of angle x
ln(x)	natural logarithm (base *e*) of x, x must be positive
log10(x)	base 10 logarithm of x, x must be positive
pow(x,y)	x raised to the y^{th} power Error conditions result if: x is zero and y is negative or zero, x is negative and y is not an integer, an overflow results.
x^y	x raised to the y^{th} power, see above

Miscellaneous Functions

min(x,y)	smaller of x and y
max(x,y)	larger of x and y
randn(x,y)	an approximately normally (Gaussian) distributed real random number with mean x and standard deviation y
randu(x)	a uniformly distributed real random number with mean of x from the interval [0,x]
row()	returns the row number
ceil(x)	next whole number greater than or equal to x
floor(x)	next whole number less than or equal to x
pi()	returns the value of Pi. To limit to a specific number of digits, use Round(Pi(),y) where Y is the number of digits after the decimal point
round(x, y)	X rounded to a the nearest number with Y digits after the decimal point
sqrt(x)	square root of x, x must not be negative
fabs(x)	absolute value of x
fmod(x,y)	floating point remainder of x/y, if y is zero, fmod returns zero
d2r(x)	convert angle x from degrees to radians. Example: sin(d2r (30)) computes the sine of 30 degrees. Sin(30) computes the sine of 30 radians
r2d(x)	convert angle x from radians to degrees
sign(x)	evaluate the sign of a number, returns -1 for x < 0, 0 for x = 0, and 1 for x>0

Statistical Functions of Intervals

sum(a..z)	calculates the sum of a range of columns in a row
sum(_1.._5)	calculates the sum of a range of rows in a column
avg(a..z)	calculates the average of a range of columns in a row
avg(_1.._5)	calculates the average of a range of rows in a column

std(a..z)	calculates the (population) standard deviation of a range of columns in a row
std(_1.._5)	calculates the population standard deviation of a range of rows in a column
rowmin (a..z)	finds the minimum value of a range of columns in a row
rowmin(_ 1.._5)	finds the minimum value of a range of rows in a column
rowmax (a..z)	finds the maximum value of a range of columns in a row
rowmax(_ 1.._5)	finds the maximum value of a range of rows in a column

- The *Statistical Functions of Intervals* functions operate row-wise on an interval of columns or column-wise on an interval of rows.
- The *Statistical Functions of Intervals* do not operate on cells. The equation B1 = avg(A1..A6) will result in only the average of row 1, not the average of the specified cells.
- For example, SUM(A..Z) computes the sum of the twenty-six columns A, B, C, ..., Z. It does this for each row separately.
- Replace 'a..z' by any valid interval of columns or rows, such as W..AC or _4.._612. There must be exactly two periods between the column or row labels. The labels may be given in reverse order, such as SUM(Z..A).
- The *Statistical Functions of Intervals* functions can only be used in the worksheet. These functions cannot be used in **Grid Math**.

String Comparison

atof(x)	convert a string x to floating-point value
atoi(x)	convert a string x to an integer value
ftoa(x,y)	convert a floating-point number x to a string with y digits after the decimal point
strlen(x)	length of string x in characters
strcmp (x,y)	compare string x with y and return 1 if x>y, -1 if x<y, or 0 if x=y
stricmp (x,y)	compare string x with y without regard to the case of any letters in the strings
strncmp (x,y,z)	compare the first z characters of string x with y
strnicmp (x,y,z)	compare the first z characters of string x with y without regard to the case of any letters in the strings

String comparison functions work with strings, not numbers. Any rows or columns containing numbers result in blanks. In each of the string comparison functions, 1 is returned if string x is greater than string y, -1 is returned if string x is less than string y, and 0 if string x = string y. In the three-parameter comparison functions, the third parameter, z, specifies the number of characters to compare. For example, a z value of 3 compares the x and y strings' first three characters and ignores any characters after the third.

The comparisons are based on the standard ASCII table:

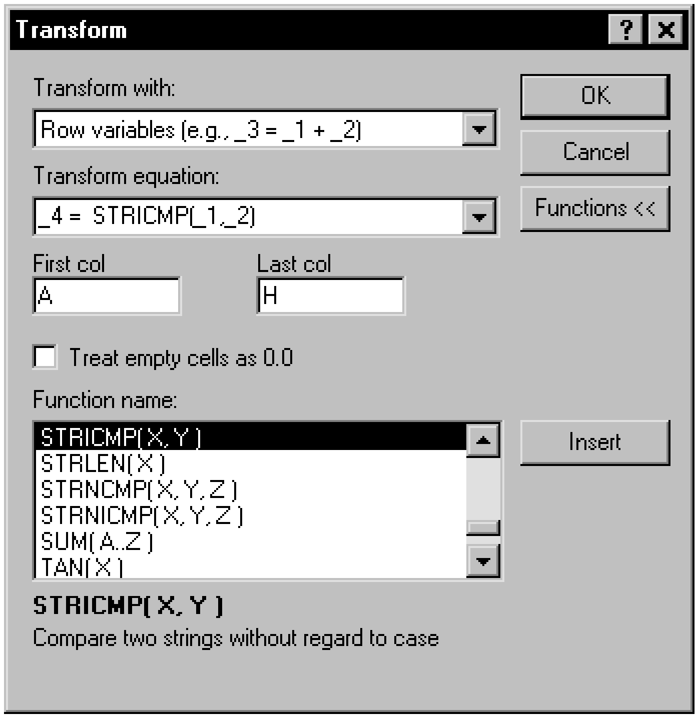
1. numeric values (disregarded in string comparisons as mentioned above)
2. cells starting with a space character
3. common punctuation
4. numeric text (numbers entered as text)
5. less common punctuation
6. uppercase letters
7. even less common punctuation
8. lower case letters
9. uncommon punctuation
10. blank cells (disregarded in string comparisons)

0	1	2	3	4	5	6	7	8	9
space	!	"	#	$	%	&	'	(	)
*	+	,	-	.	/	"0"	"1"	"2"	"3"
"4"	"5"	"6"	"7"	"8"	"9"	:	;	<	=
>	?	@	A	B	C	D	E	F	G
H	I	J	K	L	M	N	O	P	Q
R	S	T	U	V	W	X	Y	Z	[
\	]	^	_	`	a	b	c	d	e
f	g	h	i	j	k	l	m	n	o
p	q	r	s	t	u	v	w	x	y
z	{	|	}	~	blank				

This is the ASCII table order. The table is read left to right, top to bottom. Items appearing toward the upper left corner are less than the items appearing toward the lower left corner.

Example of a String Comparison

The following example shows a comparison of two strings in rows 1 and 2 with the results displayed in row 4. STRCMP returns a 1 if row 1 is greater than row 2; -1 if row 1 is less than row 2, and 0 if row 1 is equal to row 2.



This equation compares the strings in rows 1 and 2 and places the results in row 4.

	A	B	C	D	E	F	G
1	1	abc	abc	*abc	ABC	abc	abc
2	A	zyx	zyx	abc	abc		abc
3							
4		-1	1	-1	-1		0

This table includes results of the string comparisons based on the ASCII table. Cells containing numbers and blanks are ignored.

Boolean Expressions

Boolean expressions include:

- **Logical Operators** (and, or, xor, not),
- **Comparison Operators** (=, <>, <, >, <=, >=),
- the **IF Function -** condition, true_value, false_value)

The words AND, OR, XOR, NOT, and IF are reserved keywords and may not be used as variable names.

Logical Operators (and, or, xor, not)

SYMBOL	NAME	DESCRIPTION
AND	AND	The result is true if both operands are true.
&&	AND	The result is true if both operands are true.
!	Logical NOT	Inverts the boolean value. True becomes false, false becomes true.

<table>
<tr><td>NOT</td><td>Logical NOT</td><td>Inverts the boolean value. True becomes false, false becomes true.</td></tr>
<tr><td>&</td><td>AND</td><td>The result is true if both operands are true.</td></tr>
<tr><td>|</td><td>OR</td><td>The result is true if either of the two operands is true</td></tr>
<tr><td>XOR</td><td>Exclusive-OR (XOR)</td><td>The result is true if both operands differ.</td></tr>
<tr><td>||</td><td>OR</td><td>The result is true if either of the two operands is true.</td></tr>
<tr><td>OR</td><td>OR</td><td>The result is true if either of the two operands is true.</td></tr>
</table>

Comparison Operators (=, <>, <, >, <=, >=)

SYMBOL	NAME	DESCRIPTION
~	Bitwise NOT	Inverts the bits in an integer
*	Multiply	Multiplies the two operands
/	Divide	Divides the first operand by the second
%	Remainder	Integer remainder of the first operand divided by the second
+	Add	Adds the two operands
-	Subtract	Subtracts the second operand from the first
<<	Shift Left	Shifts the operand to the left
>>	Shift Right	Shifts the operand to the right
<	Less Than	In the example, A1 < B1, the result is true if A1 is less than B1
<=	Less Than or Equal To	Result is true if the ordinal value of p1 is less than or equal to p2
>	Greater Than	Result is true if the ordinal value of p1 is greater than p2
>=	Greater Than or Equal To	Result is true if the ordinal value of p1 is greater than or equal to p2
==	Equal To	Result is true if the operands have identical values
!=	Not Equal To	Result is true if the operands do not have identical values
<>	Not Equal To	Result is true if the operands do not have identical values

IF Function - IF(condition,

SYMBOL	NAME	EXAMPLE	DESCRIPTION

IF	Conditional Evaluation	IF (p1,p2,p3)	IF(condition,true_value,false_value) i.e. If p1 is true, the result will be p2. If p1 is false, the result will be p3.
IF	Conditional Evaluation	p1?p2:p3	condition?true_value:false_value i.e. If p1 is true, the result will be p2. If p1 is false, the result will be p3.

Examples

The following are examples of mathematical function syntax used in **Surfer**. With **Grids | New Grid | Function** and **Grids | Calculate | Math** use X,Y, and Z for the variables. If you use **Data | Data | Transform** in the worksheet, replace X,Y, and Z with column letters (A), row numbers (remember the underscore, _1 is row 1), or cell locations (A1).

Equation	Mathematical Function Syntax
x^2	x^2 or pow(x,2)
$\ln(x)$	ln(x)
$\log_{10} x$	log10(x)
$1 - e^{-x}$	(1-exp(-X))
$1 - e^{-x^2}$	(1-exp(-X*X))
$1 - \frac{\sin(x)}{x}$	1-(sin(x)/x)
$\frac{x^2}{1 + x^2}$	x^2/(1+x^2)
$2x - x^2$	(2 * X) - X^2
$(x^2 + y^2)(\sin(8 \times \tan^{-1} xy))$	(x^2+ y^2)*(sin(8*atan(x*y)))

Appendix B - Math Text Instructions

Math Text Instruction Syntax

The math text instruction set offers advanced formatting of text in the **Surfer** plot window. Math text instructions can be used to change the typeface, size, color, weight, and style of text on a character-by-character basis. Greek letters and mathematical symbols can be written in the plot window using math text instructions. The math text instructions also allow for the detailed placement of characters and symbols; thus, superscripts, subscripts, and the superposition of characters are possible. The worksheet cells used as post labels in a post plot can contain math text instructions. In general, the clipboard can be used to cut and paste math text instructions.

All text within the plot window can be controlled by math text instructions. For example, an axis label, title, or text block can include math text instructions in the text string in the Properties window. In the **Properties** window, add the math text instructions below directly in the *Title text, Prefix, Suffix,* or *Text* option. Do not open the **Text Editor** as text in the Text Editor cannot use math text instructions. To use different text properties in the **Text Editor** , highlight the text to change and click the appropriate button or command in the dialog.

Unless otherwise indicated, all math text instructions begin with a backslash ("\"), and end with a single space. For example, the instruction "\up50" shifts the baseline of the text up 50 percent of the current text height. All characters from the beginning backslash through the ending single space are interpreted as instructions by the math text interpreter, and are not included in the resulting label.

Each line in a text block starts with the default text properties such as typeface, size, color, and style. (Note that some typefaces, such as Symbol, do not support bold or italicized text.) A line of text within a text block uses the current properties until a math text instruction is encountered. All text following an instruction is modified according to the instruction. For example, if the typeface is changed in the middle of a text string, the text following the instruction will use the new typeface until the end of the line of text is reached, or until another instruction affecting the typeface is encountered.

Encapsulate Math Text Instruction

Math text instructions can also be encapsulated so they are not carried out over an entire line. A left curly brace ("{") instructs the math text system to remember all of the text properties in effect at that point. A right curly brace ("}") restores the properties to what they were at the matching left curly brace. This allows the insertion of special text in the middle of an otherwise uniform text block. The only instructions this does not apply to are text baseline instructions

\dnX and \upX), and the position instructions (\rpX and \spX). Curly braces can be nested.

To incorporate a backslash, right curly brace, or left curly brace as a text character in a text block, precede them with a backslash when entering the text string. For example, "\\" produces "\", and "\{" produces "{".

Percentage Instructions

Instructions based on a percentage, such as font size, are cumulative. This means that a second percentage change within a text block is interpreted as a percentage of the first percentage change. For example, if the font is scaled by 50%, and later in the same text block the font is scaled by 50% again, the font size after the second percentage would be 25% of the original font size.

Note: Instruction names are case insensitive (\fs50, \FS50, \Fs50 or \fS50 are all valid).

Note: The \sp and \rp instructions only refer to positions on the same line.

Instructions that Change Typefaces, Sizes, and Styles

Math Text Instruction	**Result**
`\b`	All text after the \b command is bold.
`\f"X"`	Change to the typeface named X. These are the names listed in the Face list box in the **Text** dialog in the plot window. Typeface names are case sensitive. Enclose the face name in double quotes. If the typeface is not found, a generic stick typeface is used in place of the unfound typeface.
`\fsX`	Change font size to X% of current font size. For example, a value of 200 for X increases the font size by two, and a value of 50 for X decreases the font size by one-half.
`\i`	All text after the \i instruction is italicized.
`\plain`	This sets the text to "plain" text with normal weight, no italics, no underlining, and no strikethrough.
`\strike`	Strikethrough the text.
`\ul`	All text after the \ul instruction is underlined.

Instructions that Change Text Color

The following instructions are provided to make it easy to set basic text colors:

Math Text Instruction	**Result**
`\black`	Sets the text color to black.
`\blue`	Sets the text color to blue.
`\green`	Sets the text color to green.
`\cyan`	Sets the text color to cyan.

\red	Sets the text color to red.
\magenta	Sets the text color to magenta.
\yellow	Sets the text color to yellow.
\white	Sets the text color to white.
\gray	Sets the text color to gray.

The following instructions allow the text color to be set to an arbitrary RGBA (Red,Green,Blue, Alpha) value:

Math Text Instruction	Result
\rgbrX	Sets the amount of red in an RGBA text color (X=0 to 255).
\rgbgX	Sets the amount of green in an RGBA text color (X=0 to 255).
\rgbbX	Sets the amount of blue in an RGBA text color (X=0 to 255).
\rgbaX	Sets the alpha value in the RGBA text color (X=0 to 255).
\color (r,g,b,a)	Sets the amount or red, green, blue, and alpha value in the RGBA text color. r, g, b, and a are all a value between 0 and 255.

Instructions that Change Text Position

Math Text Instruction	Result
\dnX	Moves text baseline down X% of current font size (subscript). This instruction produces subscripts or returns the baseline to the original position following a \upX instruction. If a font size (\fsX) instruction follows the \dnX instruction, any subsequent \dnX or \upX instructions are relative to the changed font size.
\n	Creates a new line in the text block. This works similar to a carriage return - line feed combination. This should be used instead of the \dnX to create a new line of text.
\rpX	Restores the current position to position #X (X = 1 to 20). This instruction is used in conjunction with the \spX instruction. Any text following this instruction begins at the position defined with the \spX instruction. If the \rpX instruction is used without first setting a position with the \spX instruction, the position for the text is returned to the stating position for the text block.

\spX	Saves the current position as position #X (X = 1 to 20). The position is the up, down, left, and right areas within the text block. When the \spX instruction is used, the current location within the text block is assigned a position number. Return to this position using the \rpX instruction. Specify the position number assigned with the \spX instruction when using the \rpX instruction. These instructions are most useful when placing both superscript and subscripts after the same character.
\upX	Moves the text baseline up X% of current font size (superscript). This instruction is used to produce superscripts or to return the baseline to the original position following a \dnX instruction. If a font size (\fsX) instruction follows the \spX instruction, any subsequent \upX or \dnX instructions are relative to the changed font size.

Instructions Used to Insert Special Characters or Date and Time

Math Text Instruction	**Result**
\aX	Insert an ANSI character with code given by X (0 to 255). Generally, this instruction is needed for characters with ANSI numbers beyond the normal limits of the keyboard. For example, use the ANSI character code to include an integral sign in a text string. By determining the character set, followed by the \aX command to specify the correct character number. For example, an integral sign is located at ANSI position 242 in a symbol set called Symbol. Type \f"Symbol" \a242 for the integral to appear in the text block. The Character Map program is an accessory program in the Windows installations. Use the Character Map to display each of the available character sets with the ANSI code. The ANSI code is displayed in the right half of the Character Map status bar when a character is selected.
\date	Inserts the current date. Be sure to follow this instruction with a space, even if no other text follows the date. The date is updated every time the text is redrawn.
\time	Inserts the current time. Be sure to follow this instruction with a space, even if no other text follows the time. The time is updated every time the text is redrawn.
\dt ("format")	Inserts the current date and/or time as indicated by a custom format string. Be sure to follow this instruction with a space, even if no other text follows the closing parenthesis. The date and or time is updated every time the text is redrawn.

Examples of Math Text Instructions

Due to page size limitations, some of these examples contain multiple lines of math text instructions. These examples must be entered on one line for the text to be displayed correctly.

Math Text Instruction	**Result**
`CO\dn50 2`	CO_2
`x\up50 2`	x_2

`\sp1 {\fs200 \f"Symbol" \a229 \sp2 }{\rp1`
`\dn90 \fs75 i=1\rp1 \up220 n}\rp2 \up25`
`X\dn50 {\fs75 i}\up50 Y\dn50 {\fs75 i}\up50`
`= S{\dn50 {\fs75 XY}\up50 }`
	$\sum_{i=1}^{n} X_i Y_i = S_{XY}$

`{\i Avogadro} Constant = 6.022 x 10{\up50` `{\fs75 23}\dn50 }`	*Avogadro* Constant = 6.022 x 10^{23}

`{\f"Symbol" \a209 }{\up50 {\fs50 2}\dn50 }`
`{\f"Symbol" \a102 } = {\f"Symbol" \a182 }`
`{\up50 {\fs50 2}\dn50 }{\f"Symbol" \a102 }/`
`{\f"Symbol" \a182 }x{\up50 {\fs50 2}\dn50 }`
`+ {\f"Symbol" \a182 }{\up50 {\fs50 2}\dn50`
`}{\f"Symbol" \a102 }/{\f"Symbol" \a182 }y`
`{\up50 {\fs50 2}\dn50 }`
	$\nabla^2\phi = \partial^2\phi/\partial x^2 + \partial^2\phi/\partial y^2$

`{\fs200 N}\sp1 \up100 \fs75 5 \rp1 \dn50 1`	N_1^5
`sin{\up50 {\fs50 2}\dn50 }(X) + cos{\up50` `{\fs50 2}\dn50 }(X) = 1`	$\sin^2(X) + \cos^2(X) = 1$
`104\f"Symbol" \a176 \f"Arial" 37'`	104°37'
`a\dn25 {\fs75 c} \n ___ \n b\dn25 {\fs75 c}`	$\frac{a_c}{b_c}$
`{\f"Symbol" d}\up50 {\fs75 234}\dn50 U`	$\delta^{234}U$

Index

B

D

F

G

H

J

K

L

M

Q

W

Z