GOLDENSOFTWARE

Surfer®

Powerful contouring, gridding & surface mapping system

Full User's Guide

Surfer® Registration Information

Your **Surfer** product key is located in the email download instructions 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.

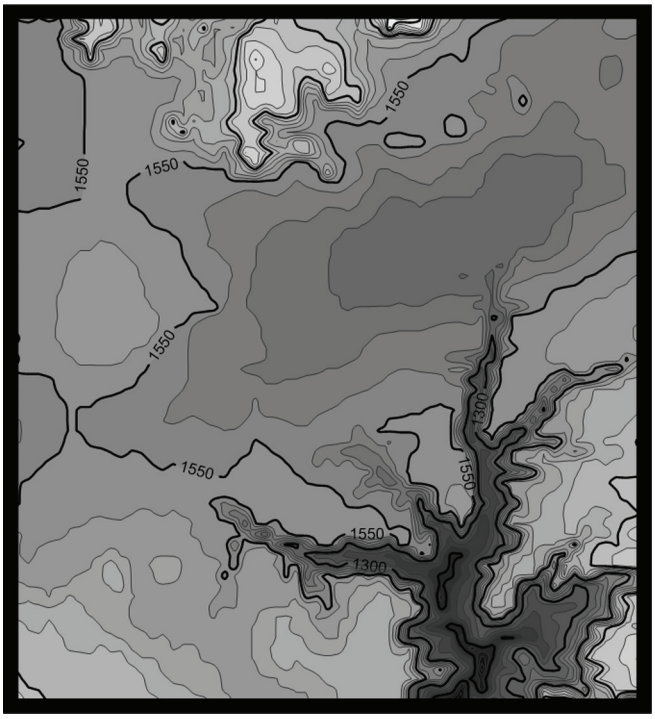
For future reference, write your product key on the line below.

_____

Surfer®

User's Guide

Contouring and 3D Surface Mapping
for Scientists and Engineers



Golden Software, LLC
809 14th Street, Golden, Colorado 80401-1866, U.S.A.
Phone: 303-279-1021 Fax: 303-279-0909
www.GoldenSoftware.com

COPYRIGHT NOTICE

Copyright Golden Software, LLC 2017

The **Surfer**® program is furnished under a license agreement. The **Surfer** software, quick start guide, and user's guide may be used or copied only in accordance with the terms of the agreement. It is against the law to copy the software, quick start guide, or user's guide on any medium except as specifically allowed in the license agreement. Contents are subject to change without notice.

Surfer is a registered trademark of Golden Software, LLC. All other trademarks are the property of their respective owners.

November 2017

Table of Contents

Chapter 1 - Introduction 1
- Scripter 1
- New Features 1
- Who Uses Surfer? 1
- System Requirements 2
- Installation Directions 2
- Updating Surfer 2
- Uninstalling Surfer 2
- Surfer Trial Functionality 3
- Three-Minute Tour 3
- Using Surfer 16
- Surfer Flow Chart 18
- Using Scripter 18
- Surfer User Interface 19
- Ribbon 21
- Quick Access Toolbar 23
- Tabbed Documents 24
- Changing the Layout 25
- Contents 27
- Properties 32
- Status Bar 34
- Menu and Tab Commands 35
- Worksheet Document 36
- Grid Editor 38
- File Types 42
- Gridding Overview 43
- Map Types 43
- Map Wizard 47
- Introduction to Map Layers 54
- Coordinate Systems 57
- Map Coordinate System Overview 58
- File Menu Commands 59
- Printing the Online Help 73
- Welcome to Surfer Help 74
- Welcome to Surfer Dialog 77
- Technical Support 79
- Register Product Key 80
- New Features 80

Chapter 2 - Tutorial 83
Tutorial Overview 83
Starting Surfer 84
Lesson 1 - Viewing and Creating Data 84
Lesson 2 - Using the Map Wizard 88
Lesson 3 - Changing Layer Properties 92
Lesson 4 - Modifying an Axis 102
Lesson 5 - Creating a Profile 103
Lesson 6 - Saving a Map 104
Lesson 7 - Creating a 3D Surface Map 105
Lesson 8 - Adding Transparency, Color Scales, and Titles 108
Lesson 9 - Creating Maps from Different Coordinate Systems 113
Optional Advanced Lessons 117
Chapter 3 - Data Files and the Worksheet 127
XYZ Data Files 127
Missing Entries 128
Multiple Columns of Information for Additional Maps 128
Additional Information in Data Files 129
Data as Numbers or Text 129
Data File Formats 130
Date/Time Formatting 130
Working with Date/Time Values 131
Date Time Formats 133
Opening a Worksheet Window 138
Worksheet Window 138
Working with Worksheet Data 142
Selecting Cells 145
Selecting a Column or Row Dividing Line 147
Hiding Columns or Rows 149
Displaying Hidden Columns or Rows 149
Worksheet Error Codes and Special Numeric Values 150
Worksheet Specifications 150
Worksheet Commands 151
Chapter 4 - Creating Grid Files 197
A Gridding Example 197
Grid Data 199
Introduction to Gridding Methods 237
General Gridding Recommendations 237
Choosing Methods Based on the Number of XYZ Data Points 238
Gridding Method Comparison 239

Exact and Smoothing Interpolators 242
Weighted Averaging 243
Inverse Distance to a Power 244
Kriging 246
Minimum Curvature 251
Modified Shepard's Method 255
Natural Neighbor 256
Nearest Neighbor 258
Polynomial Regression 259
Radial Basis Function 260
Triangulation with Linear Interpolation 263
Moving Average 264
Data Metrics 265
Local Polynomial 271
Grid from Server 274
Grid from Contours 275
Grid Function 279
Producing a Grid File from a Regular Array of XYZ Data 281
Chapter 5 - Introduction to Variograms 283
The Variogram 283
Kriging and Variograms 284
The Variogram Grid 285
Variogram Model 287
New Variogram 288
Data Page - New Variogram 289
General Page - New Variogram 291
Variogram Properties 292
Experimental Page 293
Smoothing a Variogram with Lag Width 297
Model Page 297
Statistics Page 308
Plot Page 310
Default Linear Variogram 311
Nugget Effect 312
Export Variogram 313
Using Variogram Results in Kriging 314
Suggested Reading - Variograms 314
Variogram Tutorial 315
Geostatistical References 332

Chapter 6 - Grid Editor 335
Grid Editor Window 336
Grid Editor Commands 337
Using the Grid Editor 337
Grid Editor - Node Labels 338
Grid Editor - Node Symbols 340
Grid Editor - Contours 341
Contour Levels - Grid Editor 344
Grid Editor - Color Fill 345
Select 347
Brush 347
Warp 351
Smooth 353
Push Down 356
Pull Up 359
Eraser 362
Eyedropper 365
Zoom In - Grid Editor 366
Zoom Out - Grid Editor 366
Grid Info 367
Update Layer 367
Open Grid 368
Save Grid As 371
Chapter 7 - Grid Operations 375
Assign NoData 375
Grid Filter 380
Grid Convert 390
Grid Spline Smooth 390
Assign Coordinate System - Grid 395
Grid Project 395
Grid Calculus 397
Grid Volume 419
Grid Math 431
Grid Transform 435
Grid Slice 438
Grid Residuals 443
Point Sample 445
Grid Mosaic 447
Grid Extract 451
Grid Info 453

Filtering Grid Files 453
Grid Operations References 454
Chapter 8 - Base Maps 455
Base (vector) Layers 455
Base (raster) Layers 456
Creating Base Maps 456
Adding Base Layers 457
Editing an Existing Base Map 457
Base Map from Data 458
View Data 460
Base Map from Server 461
Empty Base Map 461
Export Drawn Objects in Map Units 462
Placing Boundaries on Other Maps 463
Changing Properties in a Base Map 463
General Page - Base (vector) Layer 465
Symbology 467
General Page - Base (raster) Layer 484
Labels Page - Base Map 487
Chapter 9 - Contour Maps 491
Creating a Contour Map 491
Editing an Existing Contour Map 492
Contour Layer Properties 492
Map Properties 492
Drawing Contours 492
Contour Map - General Page 493
Contour Map - Levels Page 495
Advanced Levels 500
Contour Lines 507
Color Filled Contour Maps 518
Contour Labels 524
Hachures 528
Masking Portions of a Contour Map with a Base Map 531
Smoothing Contours 532
Chapter 10 - Post and Classed Post Maps 533
Post Map 533
Classed Post Map 541
Labels Page 553
Edit Post Labels 559
Data Files Used for Posting 560

Updating Post Map and Classed Post Map Data Files 561
Symbol Specifications in the Data File 561
Chapter 11 - 3D Surface Maps 565
Creating a 3D Surface 565
Editing an Existing 3D Surface 565
Adding a Map Layer 565
Overlaying Coincident 3D Surfaces 566
3D Surface Tips 566
3D Surface Properties 567
Map Properties 567
General Page - 3D Surface 567
Mesh Page - 3D Surface 570
Lighting Page 572
Overlays Page - 3D Surface 574
Chapter 12 - 3D Wireframe Maps 577
Creating a 3D Wireframe 577
Editing an Existing 3D Wireframe 577
Adding a Map Layer 577
Wireframe Properties 578
Map Properties 578
General Page - Wireframe 578
Z Levels 581
Color Zones 583
Line Spectrum Dialog 585
Level Files - Wireframe 586
Color Filled Wireframe 587
Specifying the Lines to Draw on a Wireframe 587
Adding Color Zones to a 3D Wireframe 588
Line Property Precedence 589
Wireframe Base 589
Smoothing a Wireframe 590
Wireframe NoData Regions 590
Chapter 13 - Color Reilef Maps 591
Creating a Color Relief Map 591
Editing an Existing Color Relief Map 592
Adding a Map Layer 592
Color Relief Layer and the 3D View 592
Color Relief Layer Properties 592
Map Properties 592
General Page - Color Relief 593

Chapter 14 - Shaded Relief Maps 597
Creating a Shaded Relief Map 598
Editing an Existing Shaded Relief Map 598
Adding a Map Layer 598
Shaded Relief Layer in the 3D View 598
Shaded Relief Layer Properties 599
Map Properties 599
General Page - Shaded Relief Map 599
Shading Methods 602
Chapter 15 - Grid Values Maps 603
Creating a Grid Values Map 603
Editing an Existing Grid Values Map 603
Grid Values Layer Properties 604
Map Properties 604
General Page - Grid Values Layer 604
Symbols Page - Grid Values Layer 607
Labels Page - Grid Values Layer 609
Chapter 16 - Watershed Maps 611
Creating a Watershed 611
Editing an Existing Watershed 612
Adding a Map Layer 612
Watershed Properties 612
Map Properties 613
Watershed References 613
General Page - Watershed 613
Chapter 17 - Vector Maps 619
1-Grid Vector Map 619
2-Grid Vector Map 622
Symbol Page - Vector Map 627
Scaling Page - Vector Map 629
Clipping Symbols on Vector Maps 631
Chapter 18 - Point Cloud Maps 633
Creating a Point Cloud Map 633
Editing an Existing Point Cloud Map 633
Adding a Map Layer 634
Saving a Point Cloud Map 634
Point Cloud Layer Properties 634
Map Properties 635
Import Points Dialog 635
General Page - Point Cloud Layer 641

Point Cloud Commands 642
Chapter 19 - Viewshed Layers 657
Add a Viewshed Layer 657
Viewsheds and 3D Surface Maps 657
Viewshed Properties 658
Map Properties 658
General Page - Viewshed 658
Chapter 20 - Downloading Layers from a Server 663
Grid from Server 663
Download Online Maps or Grids 663
Map Source Dialogs 672
Chapter 21 - Map Properties 677
View 677
Scale 677
Limits 677
Frame 677
Coordinate System 677
Info 677
Map Properties 678
View 678
Scale 682
Limits Page 688
Frame 692
Reload Map Data 693
Chapter 22 - Map Features 695
Map Layers 695
Map Limits 695
Layer Page 695
Scale Bar 696
Legends 699
Graticules 716
Profiles 721
Axes 725
Chapter 23 - 3D View 741
Contents 741
3D View Commands 751
Walk 751
Go to Home 752
Set Home 752
Create/Edit Fly-Through 753

Play Fly-Through 755
Record Fly-Through 756
Copy to Clipboard 756
Chapter 24 - Map Tools 757
Set Limits 757
Trackball 761
Measure 762
Overlay Maps 767
Combining Maps 768
Stack Maps 770
Creating Several Maps in the Same Plot Window 771
Aligning Several Maps on the Same Page 772
Masking with Background 772
Extract Grid or Data from Map 772
Chapter 25 - Layer Tools 775
Digitize 775
Creating a Blanking File with the Digitize Command 777
Break Apart Layer 777
Delete Map Layer (Break Apart Overlay) 778
Assigning Coordinates to a Base (raster) Layer 778
Georeference Image 779
Attribute Table 792
Query Objects 795
Track Cursor 798
Export Contours 800
Chapter 26 - Coordinate Systems 803
Map Coordinate System Overview 803
Source Coordinate System - Map Layer 804
Target Coordinate System - Map 805
Displaying Data with Different Coordinate Systems in a Single Map 807
Coordinate System Notes 807
Coordinate System Frequently Asked Questions 808
Assign Coordinate System 809
Introduction to Map Projections 818
Type of Predefined Coordinate Systems 825
Supported Projections 827
Golden Software Reference Files 860
Latitude and Longitude Coordinates 860
Latitude and Longitude in Decimal Degrees 861
How to Convert from NAD27 to NAD83 Using NTv2 862

Projection References 862
Chapter 27 - Creating and Editing Features 865
Text 866
Polyline 880
Polygon 881
Point 882
Spline Polyline 883
Range Ring 885
Rectangle 887
Rounded Rectangle 888
Ellipse 889
Create a North Arrow 890
Polyline to Polygon 892
Polygon to Polyline 893
Polyline to Points 893
Points to Polyline 894
Reshape 895
Thin 898
Smooth 900
Connect Polylines 902
Break Polyline 902
Break Polyline at Intersections 903
Union Polygons 903
Intersect Polygons 904
Difference of Polygons 905
Difference of Polygons, Union Polygons, and Intersect Polygons Dialogs 906
Buffer 906
Create Intersection Points 908
Triangulation 909
Thiessen Polygons 911
Split 912
Combine Islands/Lakes 913
Undo 914
Redo 915
Paste 915
Paste Special - Plot Document 916
Copy 917
Cut 918
Delete 918

Chapter 28 - Common Properties 919
Text and Font Properties 919
Fill Properties 923
Line Properties 930
Symbol Properties 939
Label Formats 942
Metafile Properties 946
General Page - Image Properties 947
Adding Color to Maps 949
Color Palette 959
Info Page 963
Chapter 29 - Selecting and Arranging Objects 971
Selecting a Map Layer and Changing the Object ID 971
Select Tool 973
Block Select 973
Select All 974
Deselect All 974
Invert Selection 974
Transform 974
Rename Object 975
Layout Tab Commands 976
Positioning and Sizing Objects 977
Resize Objects 978
Lock Position 978
Bring to Front 979
Bring Forward 980
Send to Back 980
Send Backward 981
Size Objects 981
Align Objects 982
Distribute Horizontally 983
Distribute Vertically 984
Align to Margins 985
Free Rotate 987
Rotate 987
Group 988
Ungroup 989
Editing Groups 989
Chapter 30 - Changing the View 991
Fit to Window 991

Page 991
Zoom In 991
Zoom Out 992
Zoom Selected 993
Zoom Realtime 993
Zoom Rectangle 994
Actual Size 994
Full Screen 995
Pan 995
Redraw 995
Auto Redraw 995
Rulers 996
Drawing Grid 996
New Window 996
Cascade 996
Arrange Icons 996
Tile Horizontal 997
Tile Vertical 997
Reset Windows 997
Chapter 31 - Importing, Exporting, and Printing 999
Import 999
Export 1002
Page Setup - Plot 1012
Header & Footer Dialog 1013
Print - Plot 1015
Page Setup - Worksheet 1016
Print - Worksheet 1022
Chapter 32 - Options, Defaults, and Customizations 1025
Options - General 1025
Options - Updates 1028
Options - User Interface 1030
Options - Selection 1033
Options - Rendering 1034
Options - Printing 1035
Options - Rulers and Grid 1036
Options - Default Properties 1038
Default Settings 1039
Customize 1047
Chapter 33 - Automating Surfer 1056
Start the Scripter Program 1056

Scripter Windows 1057
Working with Scripts 1059
Scripter BASIC Language 1059
Visual BASIC Compatibility 1059
Using Scripter 1060
Using Scripter Help 1061
Suggested Reading - Scripter 1062
Writing Scripts 1062
Running Scripts 1063
Debugging Scripts 1064
Program Statements 1067
Line Continuation 1067
Comments 1068
Double Quotes and Text 1068
Operators 1068
Flow Control 1068
Optional Arguments and Named Arguments 1069
Named and Positional Arguments 1070
Subroutines and Functions 1070
Specifying Cell Coordinates 1072
Code, Class, and Object Modules 1075
Type Library References 1078
The Object Browser 1078
Variables 1080
Object Variables 1081
Array Variables 1081
User-Defined Types 1082
Global Variables 1082
Coordinate Arrays 1083
Getting User Input 1083
Creating Dialogs 1084
UserDialog Example 1086
Surfer Object Model 1087
Overview of Surfer Objects 1089
A Brief Introduction to the Major Surfer Objects 1089
Derived Objects 1094
Using Collection Objects 1095
Parent and Application Properties 1096
PlotWindow, WksWindow, and GridWindow Objects 1096
Using Surfer Objects 1097

Object List 1098
Object Hierarchy 1100
Improve Automation Performance 1102
Automation Examples 1102
Appendix A - Mathematical Functions 1109
Data Types 1109
Variable Names 1109
Precedence 1109
Trigonometric Functions 1110
Bessel Functions 1110
Exponential Functions 1110
Miscellaneous Functions 1111
Statistical Functions of Intervals 1111
String Comparison 1112
Boolean Expressions 1114
Examples 1115
Appendix B - Math Text Instructions 1117
Encapsulate Math Text Instruction 1117
Percentage Instructions 1117
Instructions that Change Typefaces, Sizes, and Styles 1118
Instructions that Change Text Color 1118
Instructions that Change Text Position 1119
Instructions Used to Insert Special Characters or Date and Time 1119
Examples of Math Text Instructions 1120
Appendix C - File Formats 1123
File Format Chart 1123
File Type Not Recognized 1130
ASCII .DAT, .TXT, .CSV Data Files 1131
Image (Bitmap) File Descriptions 1141
?10G Globe DEM data File Description 1148
000 IHO S-57 Navigation Chart File Description 1149
ACCDB Microsoft Access File Description 1150
ADF Esri ArcInfo Binary Grid File Description 1152
AM Amira Mesh [.AM, .COL] Files 1153
ASC Arc/Info ASCII Grid 1156
ASI Amira Stacked Slices [.ASI] File Description 1157
BIL Band Interleaved (BIL, BIP, BSQ) File Description 1159
BLN Golden Software Blanking File Description 1160
BNA Atlas Boundary File Description 1167
CLR Color Spectrum File Format 1174

CLS Classed Post File Format 1175
CPS CPS-3 Grid File Format 1177
DBF ASCII Database [.DBF] File Description 1177
DDF SDTS TVP Topological Vector Profile and Raster Profile File Description 1178
DEM SDTS DEM File Description 1182
DEM USGS Digital Elevation Model File Description 1183
DGN Microstation Design v7 File Description 1188
DIC DICOm3 Medical Image File Description 1189
DLG USGS Digital Line Graph File Description 1191
DOS USGS ETOPO5 File Description 1199
DTED Digital Terrain Elevation Data [.DTED] File Description 1200
DXF AutoCAD DXF File Description 1201
E00 Esri ArcInfo Export Format (E00) File Description 1213
E00 DEM Grid File Description 1216
ECW ER Mapper [.ECW] File Description 1216
EMF Windows Enhanced Metafile File Description 1219
EPS Encapsulated PostScript File Description 1223
ERS ER Mapper Grid [.ERS] File Description 1227
FLD AVS Field [.FLD] File Descriptions 1228
FLT, HDR Esri Binary Float Grid [.FLT, .HDR] File Description 1231
GEOJSON GeoJSON Data Interchange Format File Description 1231
GEOREF Golden Software Georeference File Description 1232
GIF Graphics Interchange Format 1233
GML Geography Markup Language File Description 1235
GPX GPS Exchange Format File Description 1235
GRD Geosoft Grid File Description 1236
GRD Golden Software Grid file format 1237
GRIB Global Weather Data Grid File Description 1244
GSB Golden Software Boundary File Description 1246
GSI Golden Software Interchange File Description 1249
GXF Grid Exchange [.GXF] File Description 1253
HDF Hierarchical Data Format [.HDF] File Description 1256
HDR GTopo30 [.HDR, .STX, .DEM] File Description 1259
HGT NASA SRTM Data File Description 1259
HTM HTML Image Map File Description 1261
IMG Analyze 7.5 Medical Image Files 1264
IMG ERDAS Imagine Image File Description 1266
INFO Leica Confocal Scanning File Description 1267
JPG File Interchange Format File Description 1267
JP2 JPEG2000 Interchange Format File Description 1269

KML KMZ Google Earth Keyhole Markup File Description 1271
LAS LiDAR Binary File Description 1279
LAT Iris Explorer [.LAT] File Description 1285
LVL Level File Format 1287
MIF MapInfo Interchange Format File Description 1290
NC NetCDF File Description 1294
PDF Adobe Acrobat Portable Document Format [.PDF] File Description 1296
PLT Golden Software PlotCall File Description 1305
PLY Stanford Polygon File Description 1307
PNG Portable Network Graphic File Description 1309
PNM Portable Any Map File Description 1310
RAS, SUN Sun Raster Image (SUN, RAS) File Description 1311
RAW Binary Lattice File Description 1311
RGB Silicon Graphics (RGB) Image File Description 1319
RST Idrisi Raster Image [.RST, .IMG] File Description 1319
RT* TIGER/Line File Description 1320
SEG SEG-SP1 Data File Format 1320
SHP Esri Shapefile File Description 1322
SID LizardTech MrSID File Description 1332
STK Metamorph File Description 1335
STL STereoLithography 3D Mesh File Description 1336
SVG Scalable Vector Graphics File Description 1339
SYLK Spreadsheet [.SLK] File Description 1340
TAB MapInfo Table File Description 1341
TGA Truevision Targa [.TGA] File Description 1342
TIF Tagged Image File Description 1343
TXT Formatted Text Grid [.FTG, .CSV, .DAT, .TXT] File Description 1346
VCT Idrisi Binary Vector File Description 1350
VTK Visual Tool Kit File Description 1351
WMF Windows Metafile Description 1353
X AVS X-Image [.XIMG] File Description 1357
XLSX XLSM XLS Excel Workbook File 1357
XYZ Grid [.DAT] File Description 1363
XYZ Points [.CSV, .DAT, .TXT, .XYZ] File Description 1363
ZMAP Zmap Grid File Format 1365
Index 1369

Chapter 1 - Introduction

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 | Math** 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 that automate **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
- In the web help at http://surferhelp.goldensoftware.com/#t=topics%2Fnew_features.htm
- In the program, click the help button , and click on the *New Features* page in the *Introduction* book

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, 8 (excluding RT), 10 or higher
- 512MB RAM minimum for simple data sets, 1GB RAM recommended
- At least 500MB free hard disk space
- 1024x768 or higher monitor resolution with a minimum 16-bit color depth

Installation Directions

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 **Surfer 15** over any previous versions of **Surfer**. **Surfer 15** can coexist with older versions (e.g. **Surfer 14**) as long as both versions are installed in different directories. By default the program installation directories are different. For detailed installation directions see the *Readme.rtf* file.

To install Surfer from a download:

1. Log into Windows under the account for the individual who will be licensed to use **Surfer**.
2. Download **Surfer** according to the emailed directions you received.
3. Double-click on the downloaded file to begin the installation process.
4. Once the installation is complete, run **Surfer**.
5. 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** (e.g. **Surfer 15.0** to **Surfer 15.1**), you will be prompted to download 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 15** from the list of installed applications. Click the *Uninstall* button to uninstall **Surfer**.

Windows 8

From the *Start* screen, right-click the **Surfer 15** 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 15** tile and click *Uninstall* at the bottom of the screen.

Windows 10

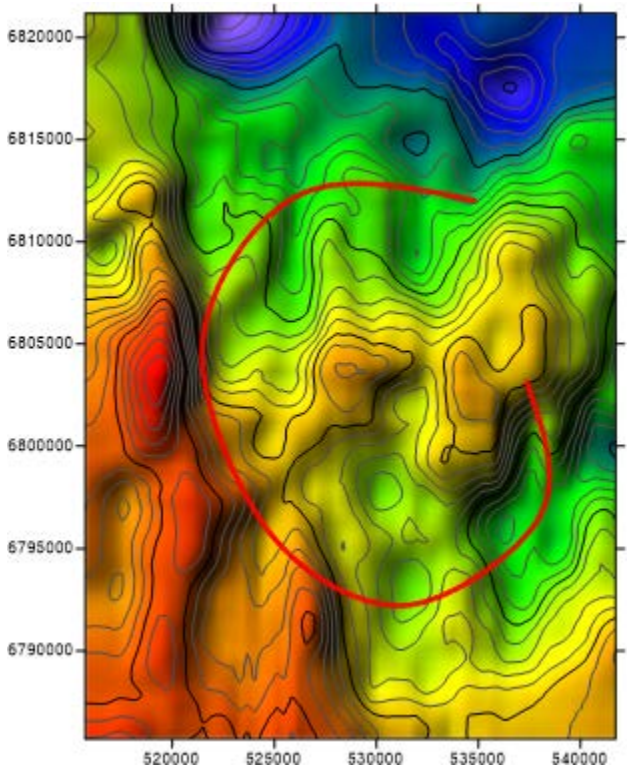
Select *Settings* in the **Start** menu. In *Settings*, select **System | Apps & features**. Select **Surfer 15** and then click *Uninstall*. To uninstall **Surfer** from the Windows Control Panel, click **Programs | Programs and Features**. Select **Surfer 15** and click *Uninstall*.

Surfer Trial Functionality

The **Surfer** trial is a fully functioning time-limited trial. This means that commands work exactly as the command works 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 some 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 file.



To see the example files:

1. Open **Surfer**.
2. Click the File | Open command.
3. In the **Open** dialog, navigate to the **Surfer** Samples folder. The **Surfer** Samples folder is located in C:\Program Files\Golden Software\Surfer 15\ 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.

Overview of Sample Surfer .SRF Files

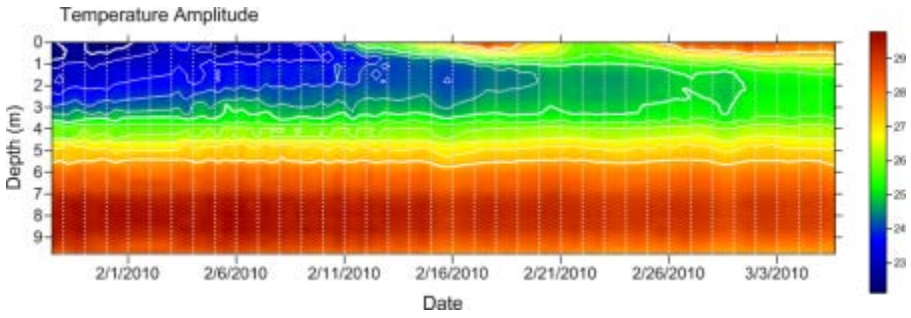
Click on the links below to see an image of the sample file and a brief explanation of what the sample file contains.

3DView.SRF

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. Select the map and click **Map Tools | View | 3D View** to open a 3D view. Click **3D View | Fly-Through | Play** to view the example fly-through.

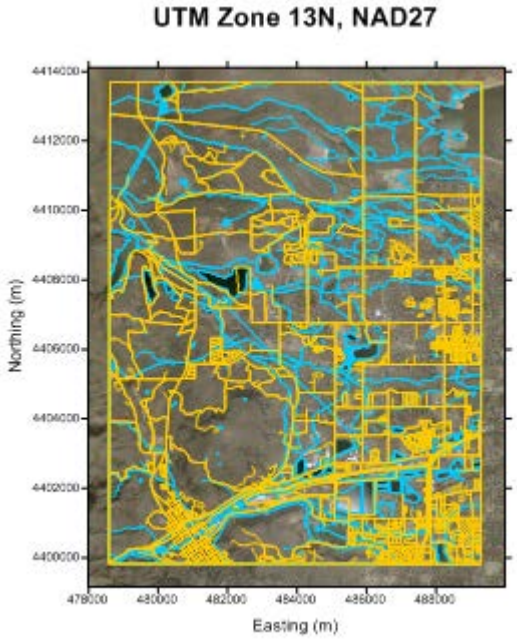
Axes.SRF

The Axes.SRF file contains a contour map layer and color relief map layer overlaid. The grid file used for the two map layers is the same and includes dates as the X values. The X Axis is displayed using date formatting.



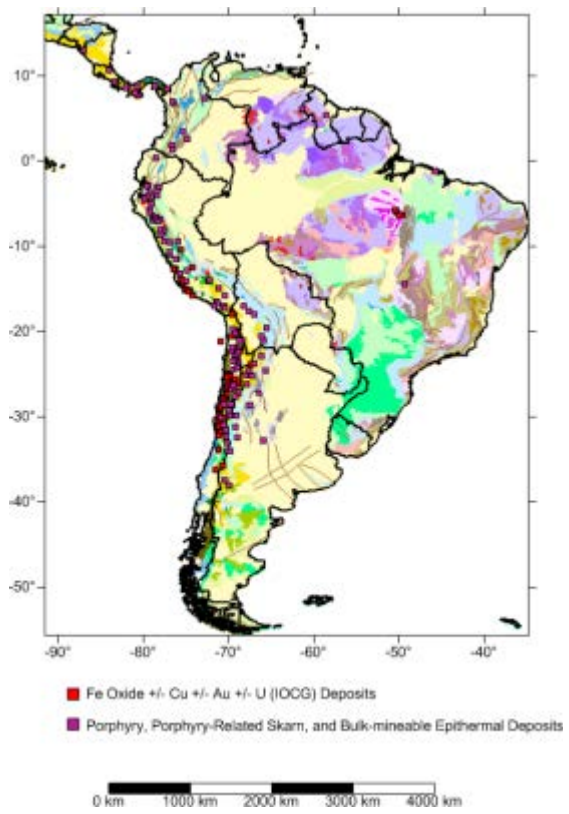
Base.SRF

The Base.SRF sample file displays three base map layers showing road transportation, stream hydrology, and a USGS urban area satellite image for Golden, Colorado, USA. The individual polygons and polylines that make up the base maps can be edited or deleted by expanding the base map layer in the **Contents** window.



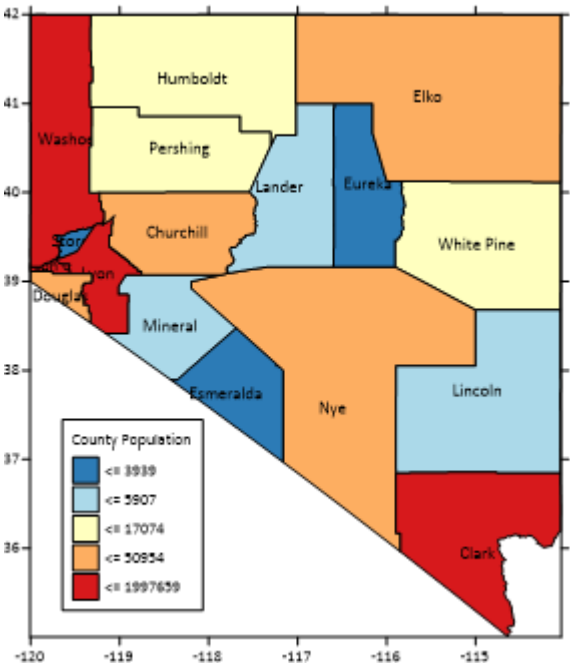
BaseMapFromServer.SRF

The BaseMapFromServer.SRF file contains five base maps of South America, showing Distribution of various minerals, national boundaries, and generalized geology. All base maps were created by downloading images from online servers.



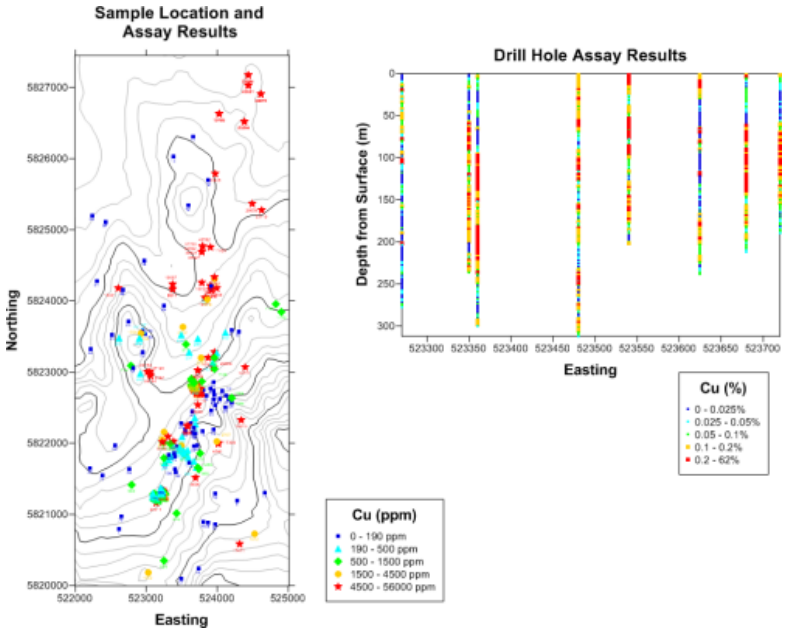
BaseSymbology.SRF

The *BaseSymbology.SRF* sample file includes a base (vector) layer with classed colors symbology applied to a map of Nevada. Counties are classified and colored by population. A legend is included to indicate the upper class values for each of the five classes.



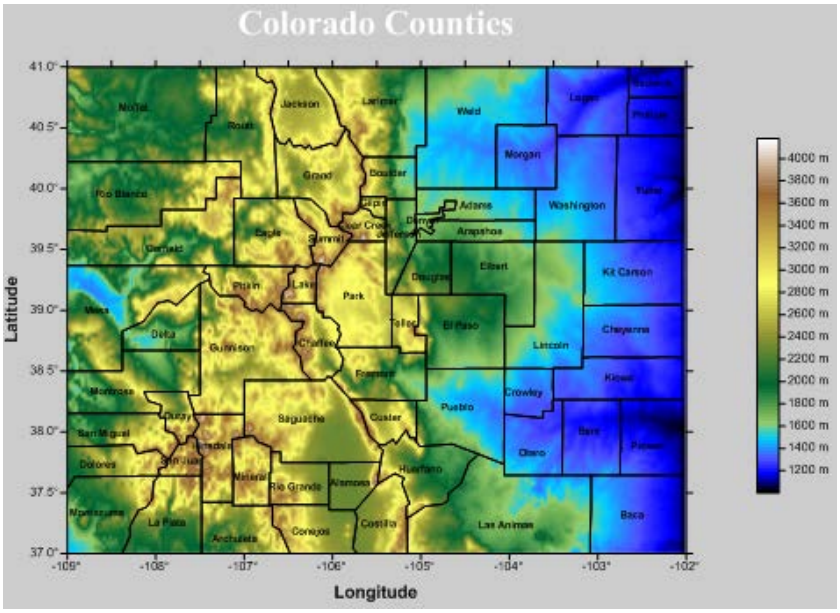
Classed Post.SRF

The Classed Post.SRF sample file displays two maps. The left map is a contour map with a classed post map layer displaying the sample location 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.



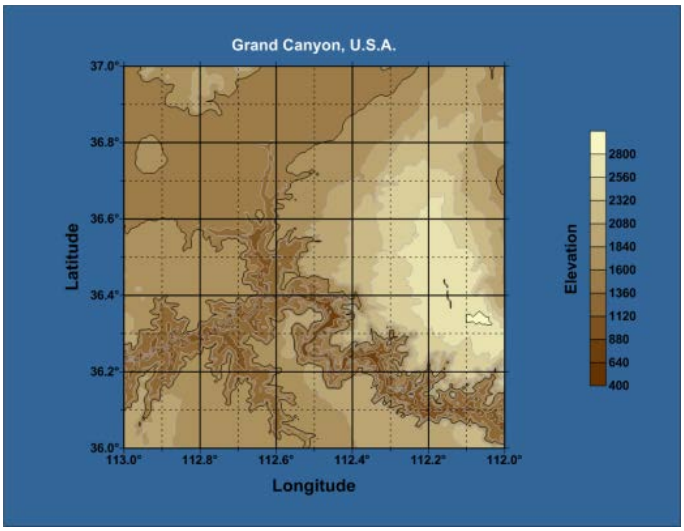
ColorRelief.SRF

The ColorRelief.SRF sample file displays a color relief map with a base map layer and a post map layer. The color relief map displays the elevation changes across the state of Colorado. A color scale has been added to show the elevation values. The base map layer displays the county boundaries. The post map layer displays the county names.



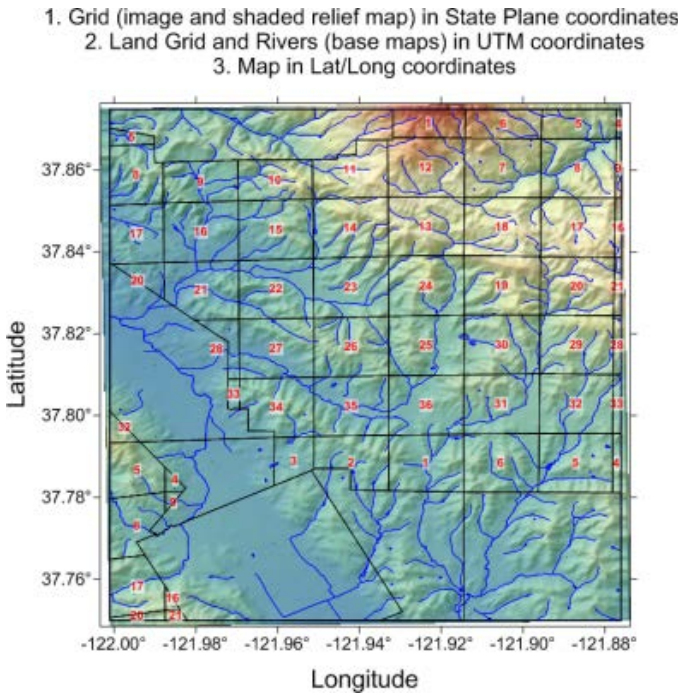
Contours.SRF

The Contours.SRF sample file displays a contour map of the Grand Canyon, USA. The left axis and bottom axis have the major and minor grid lines shown.



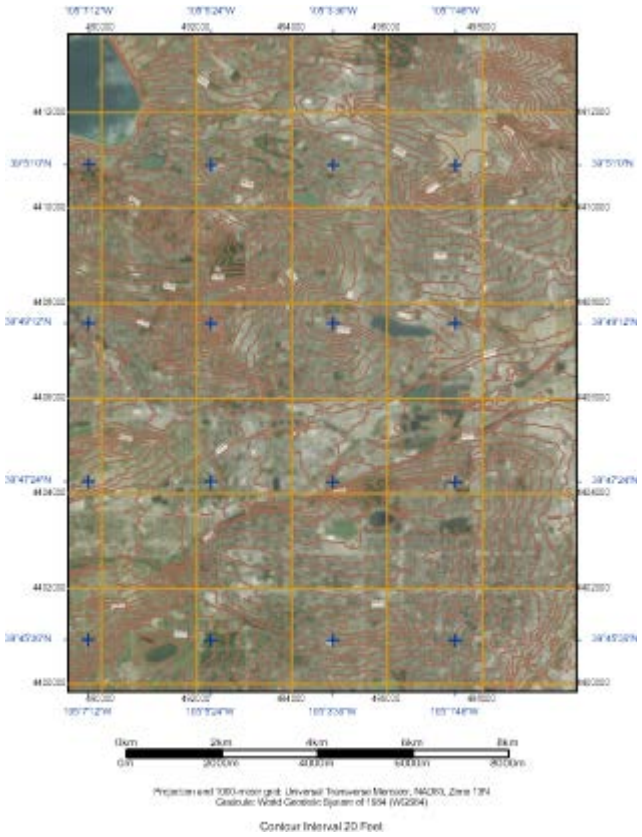
CoordinateSystems.SRF

The CoordinateSystem.SRF sample file displays two base map layers showing stream hydrology, and a land grid. An image map showing topography for the Mt. Diablo region is overlaid. The base map layers use a UTM coordinate system. The image map layer uses a state plane coordinate system. The final map is displayed in latitude and longitude coordinates.



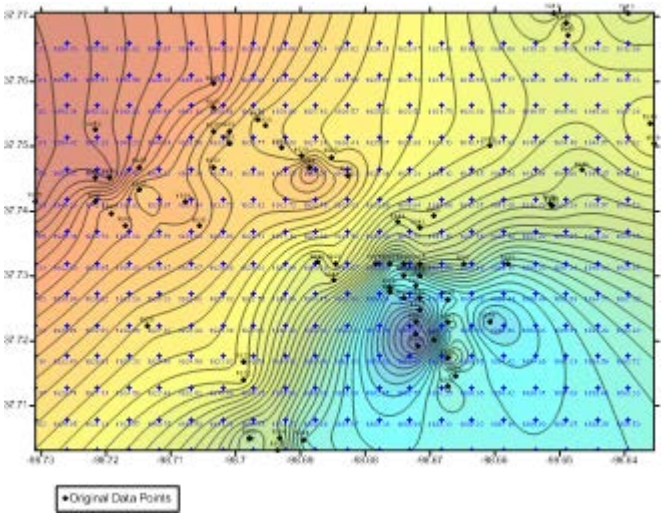
Graticule.SRF

The Graticule.SRF file shows a contour layer overlaid on a downloaded base layer. The map also includes a graticule, scale bars, and grid lines.



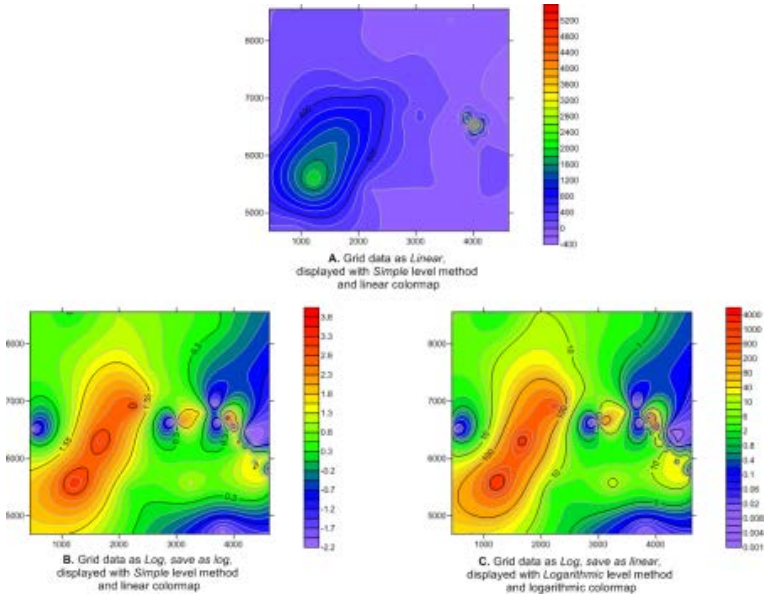
GridValues.SRF

The *GridValues.srf* sample file displays a grid values, post, and filled contour layer overlaid in a single map. The grid values layer is used to visualize the grid geometry or post grid values on the map.



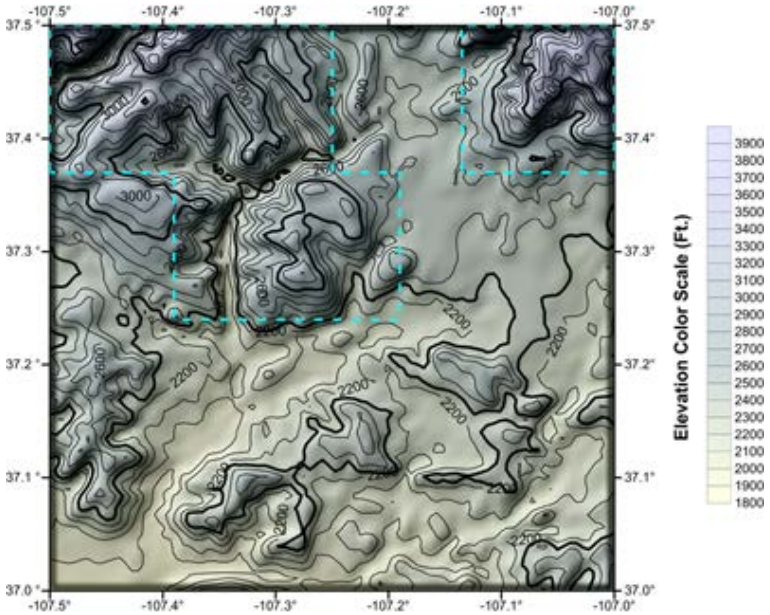
Logarithmic.SRF

The Logarithmic.SRF file contains the same data gridded in three different ways. The top map shows the data gridded in the normal *Linear Z* method. The bottom two maps show the data gridded in the *Log Z* method. The map on the left has the Z data saved in log form. The map on the right has the Z data saved in linear form, but the contour level method is set to logarithmic.



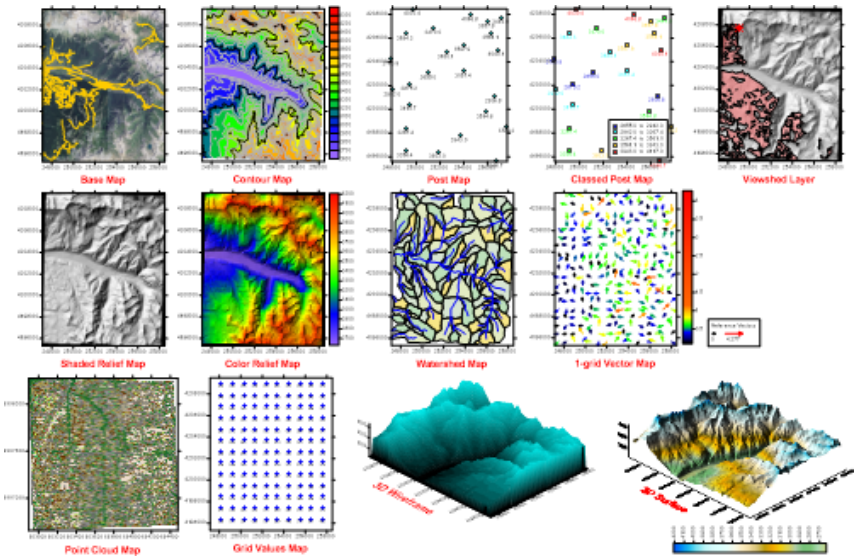
Map Layers.SRF

The MapLayers.SRF sample file displays a map with a contour layer, a shaded relief layer, and a base layer defining a national forest boundary. The contour map is semi-transparent, allowing the ability to see through the filled contours to the shaded relief map layer. The base map boundary was created from a blanking file .BLN. Blanking files can be created from known points, or by digitizing points. The color scale displays the elevation values for the contour map.



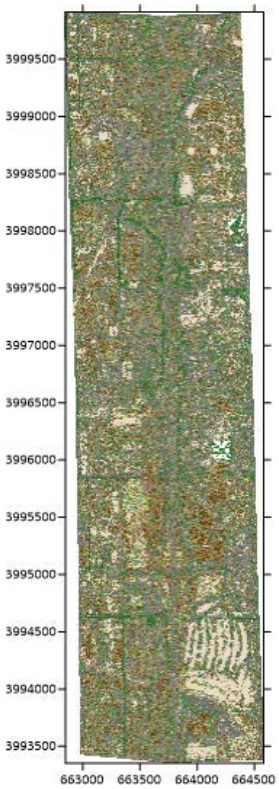
MapTypes.SRF

The MapTypes.SRF file contains a sample map for each of the map types. All grid based map types are created from the same Telluride.grd file. The classed post and post map are created from the same data file, which contains elevation of various locations in the same Telluride area.



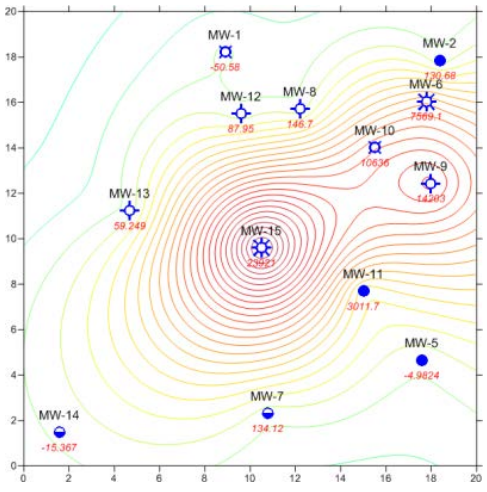
PointCloud.SRF

The PointCloud.SRF sample file includes a point cloud map with four source LAZ files. The map is a section of Las Vegas, NV. The point cloud is colored by intensity with a custom colormap.



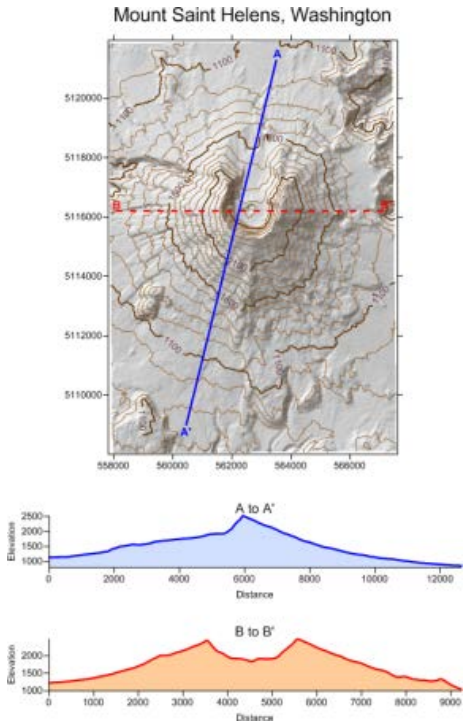
Post.SRF

The Post.SRF sample file displays a post map with a contour map layer. The post map is using a symbol column from the worksheet and contains multiple labels. The contour map is using a gradational line color.



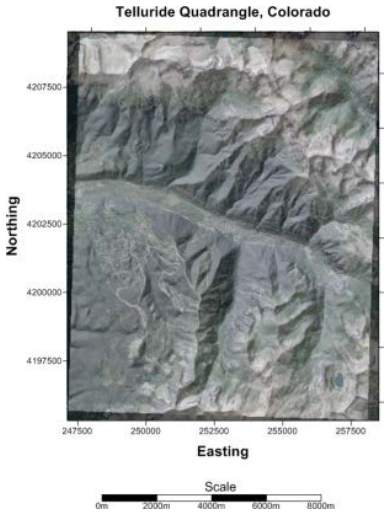
Profile.SRF

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 two profiles lines are displayed, showing the elevation across the profile.



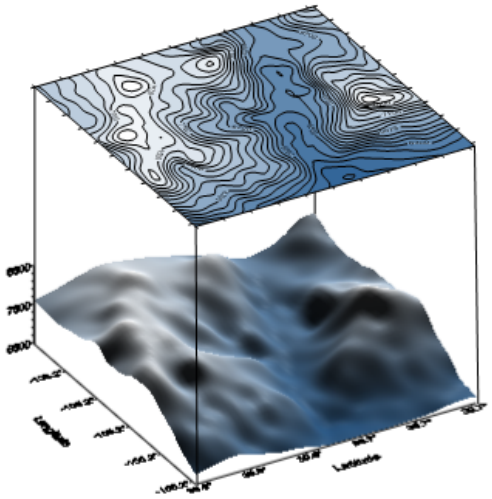
Shaded Relief.SRF

The Shaded Relief.SRF sample file displays a shaded relief map of the Telluride Quadrangle, Colorado on an aerial image of the same area. A map scale has been added.



Stacked Maps.SRF

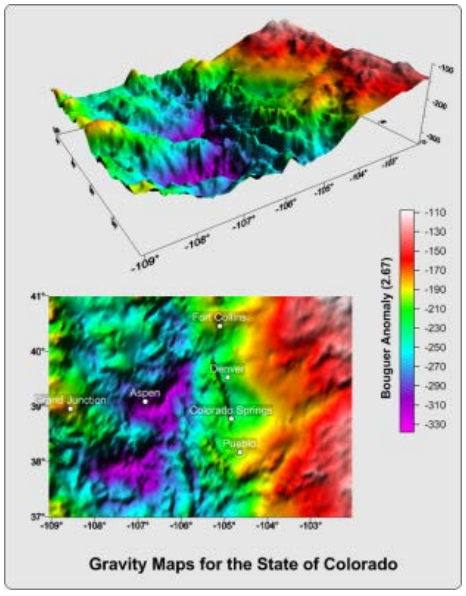
The Stacked Maps.SRF sample file displays two maps. The top map is a contour map, and the bottom map is a 3D surface of the same grid file. The two maps were aligned with the **Map Tools | Map Tools | Stack Maps** command and polylines were added. The top contour map has an index contour, where every fifth line is bold.



Southwest Corner of the
Morrison Quadrangle, Colorado

Surface.SRF

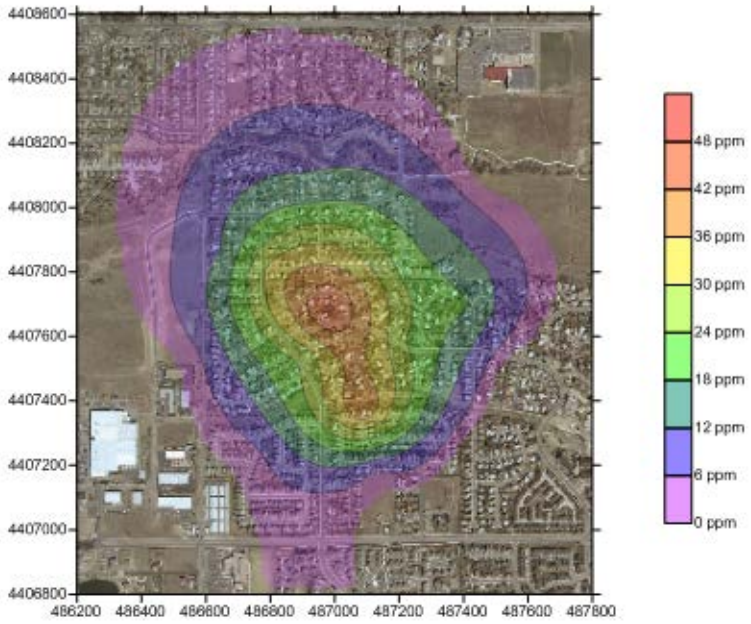
The Surface.SRF sample file displays three maps. The top map is a 3D surface gravity map for the state of Colorado. The bottom maps display the same 3D surface map from a different view, and a post map of some Colorado cities for reference. A color scale has been added to display the values for the Bouguer anomaly.



Gravity Maps for the State of Colorado

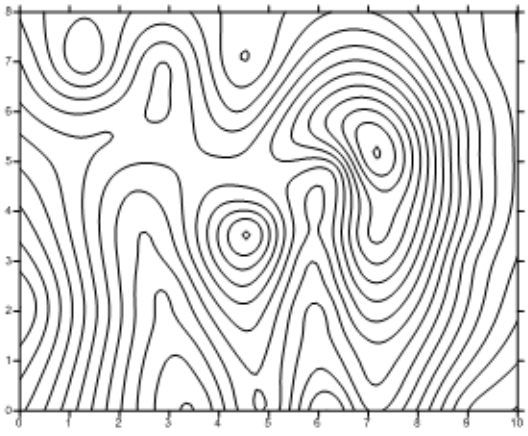
Transparent.SRF

The Transparent.SRF file displays a contour map overlaid on a base map created from an aerial photograph. The contour map is filled with a partially transparent pattern.



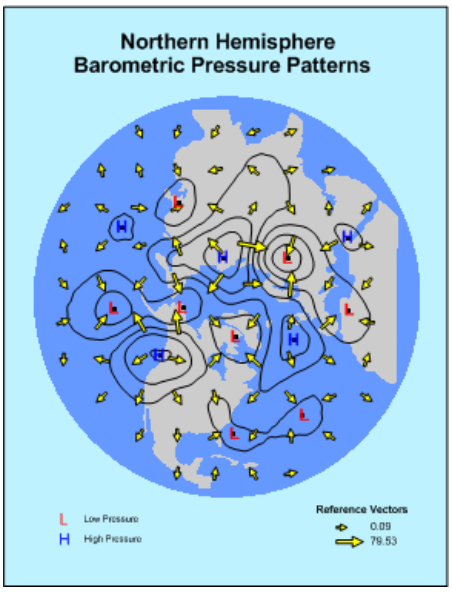
Tutorial.SRF

The Tutorial.SRF file displays a sample contour map.



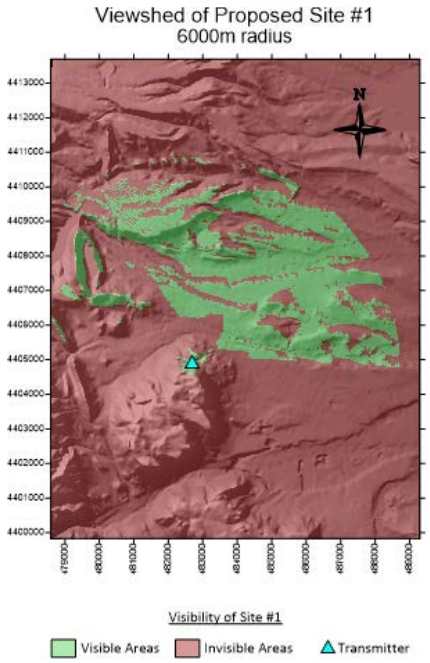
Vectors.SRF

The Vectors.SRF sample file displays a vector map with contour and post layers. The vectors show barometric pressure patterns in the Northern Hemisphere. A legend has been added to show the reference vector sizes.



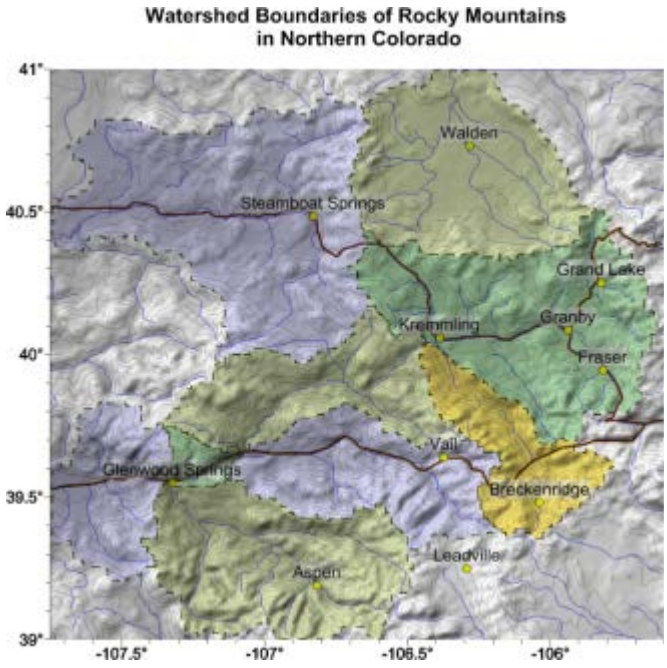
Viewshed.SRF

The Viewshed.SRF sample file shows a shaded relief layer with viewshed layers added. The transmitter location is represented by the blue triangle symbol. The visible regions are highlighted in green, and the invisible regions are highlighted in red. The viewshed radius is limited to 6000km.



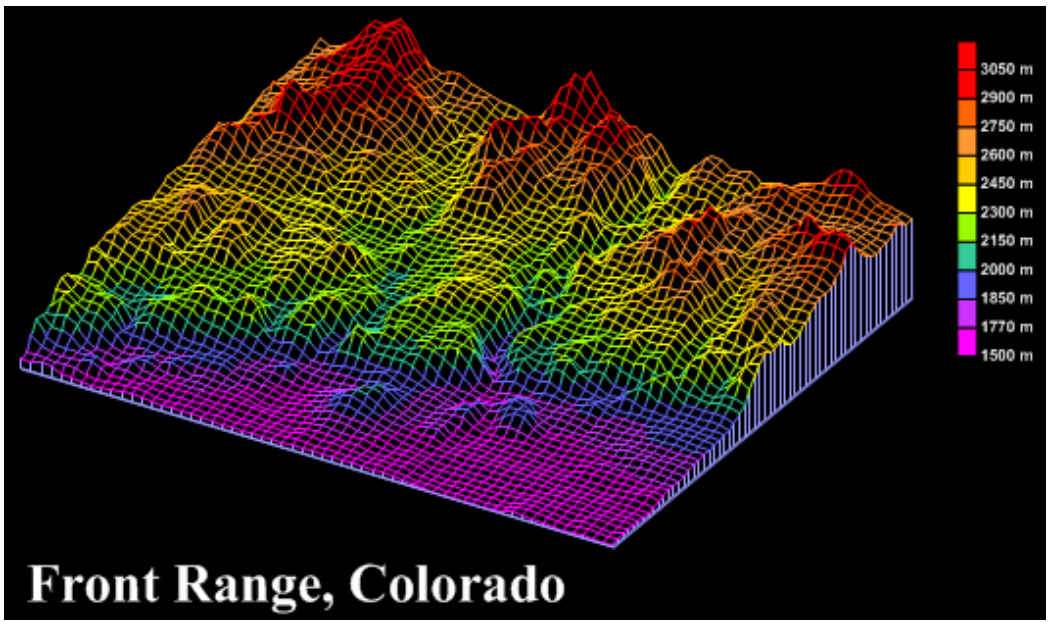
Watershed.SRF

The *Watershed.srf* sample file displays a post map, two base maps, a contour map, a watershed map, and a surface map layer. The map shows roads, rivers, and the watershed delineation of a section of the Rocky Mountains in Colorado.



Wireframe.SRF

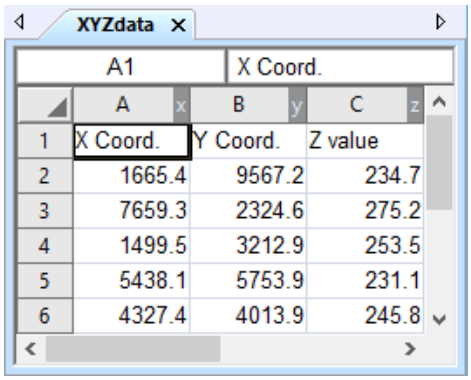
The Wireframe.SRF sample file displays a 3D wireframe map of the Colorado Front Range. A color scale has been added to show the elevation values.



Using Surfer

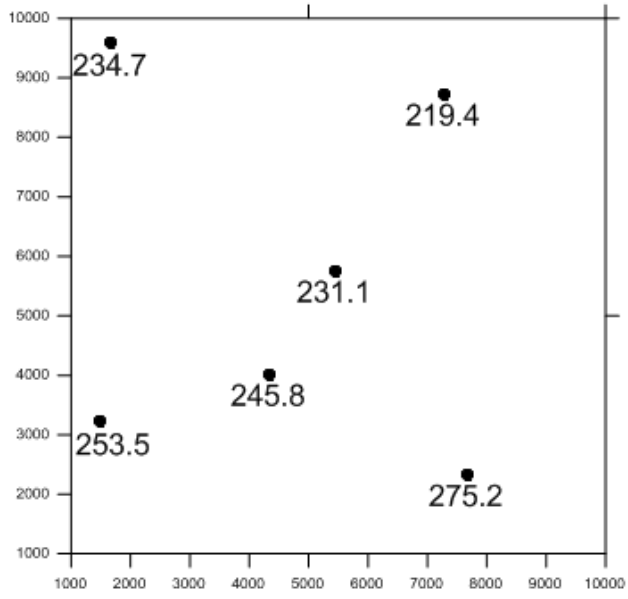
The most common application of **Surfer** is to create a grid-based map from an XYZ data file. The Grid Data command uses an XYZ data file to produce a grid file. The grid file is then used by most of the **Home | New Map** commands to produce maps. Post maps and base maps do not use grid files. The 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).



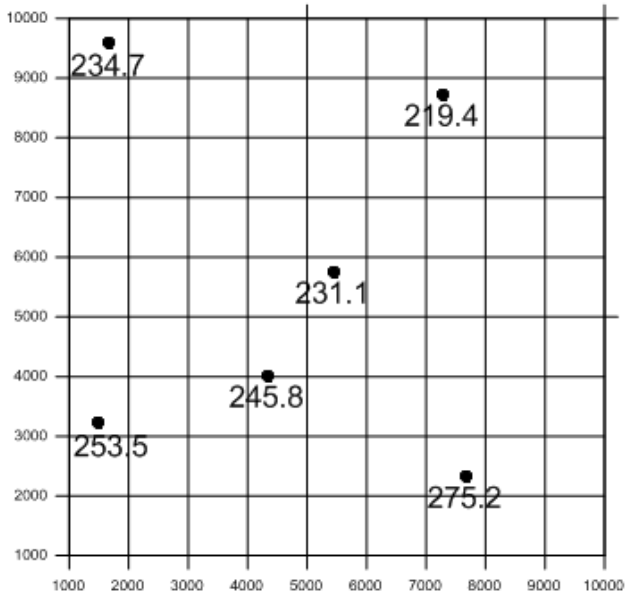
Start with irregular XYZ data in three columns.

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



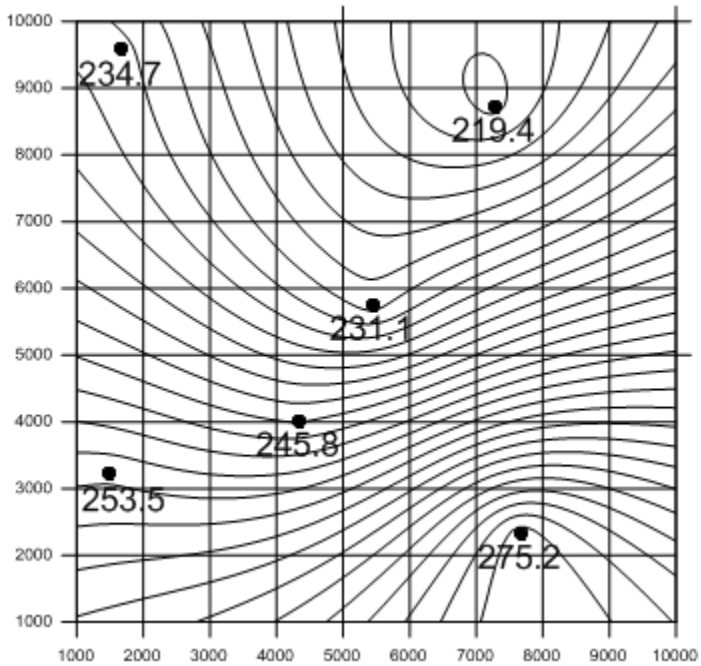
A post map displays the original XYZ data locations.

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



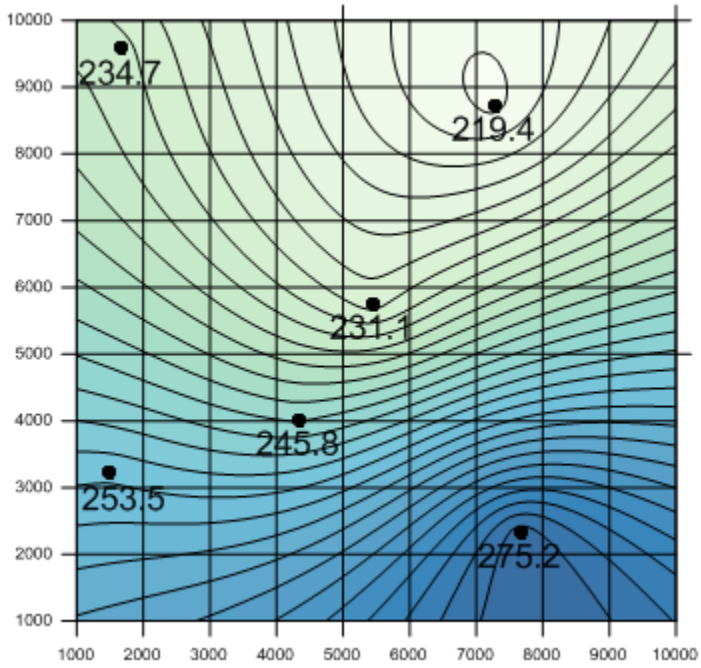
Gridding interpolates a Z value at the intersection of each row and column in the grid file. This fills the holes in the data. Here the rows and columns are represented by grid lines.

4. To create a map, select the map type from the **Home | New Map** commands. Select the grid file from step two. Grid-based maps include contour, 3D surface, 3D wireframe, color relief, shaded relief, vector, watershed, viewshed, 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.

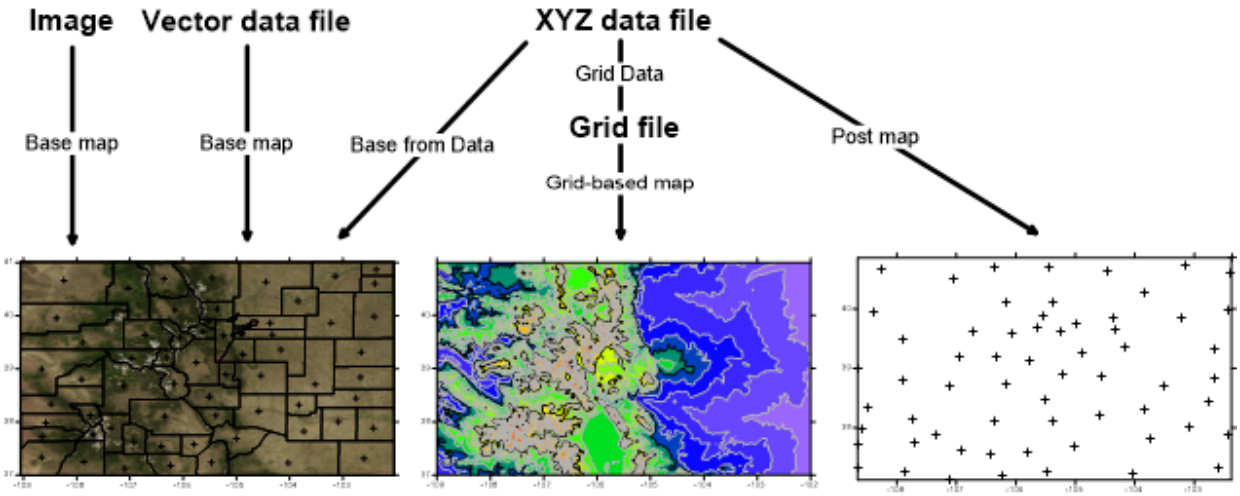


The contour map layer is filled with a gradational color fill.

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

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.

Using Scripter

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

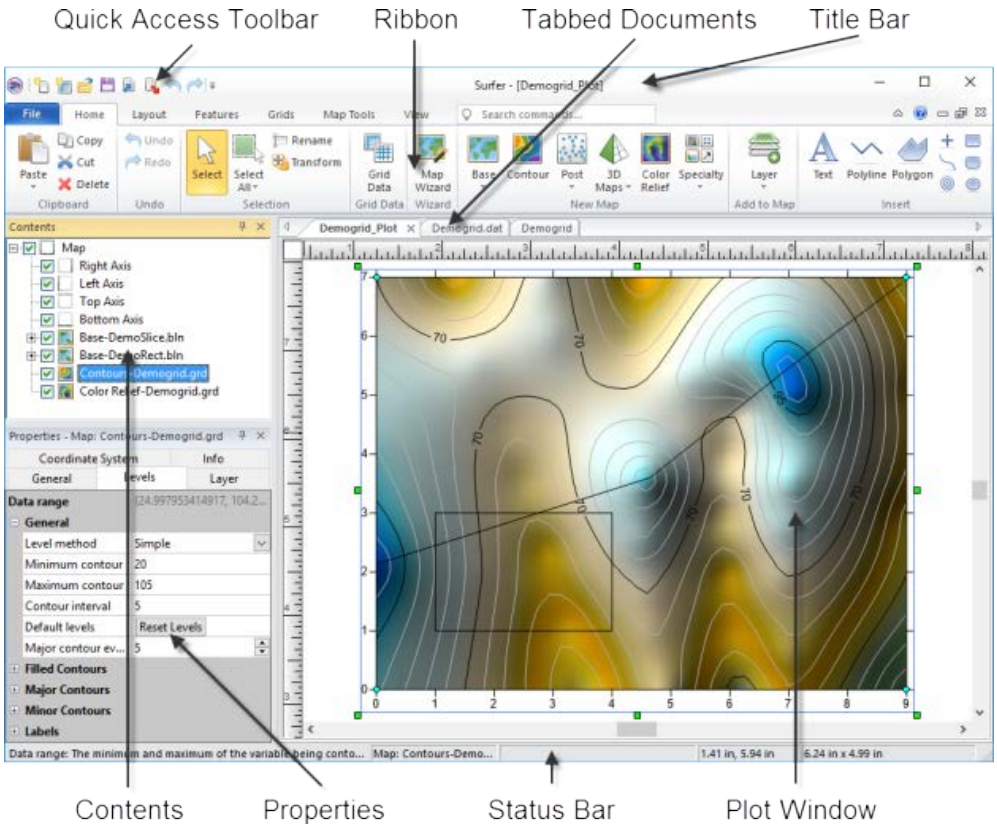
instructions for execution when the script is run. **Scripter** can be used to perform almost any task in **Surfer**. Scripts are useful for automating repetitive tasks and consolidating a sequence of steps. **Scripter** is installed in the same location as **Surfer**. Refer to the *Surfer Automation* help book 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 15\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 15\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 15\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 15\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.
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.

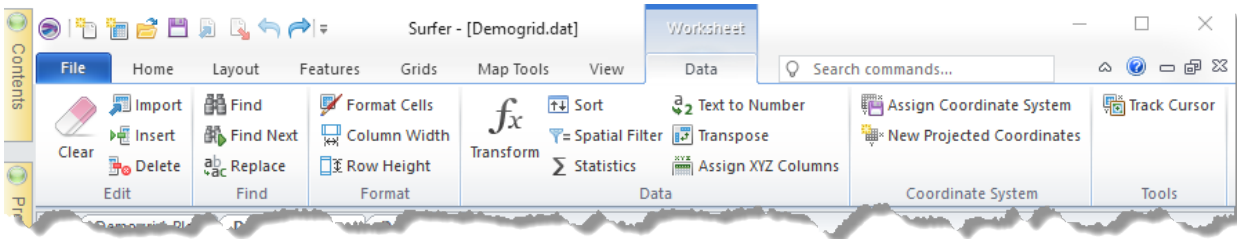
Opening Windows

Selecting the File | Open command opens any of the three window types, depending on the type of file selected. 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.

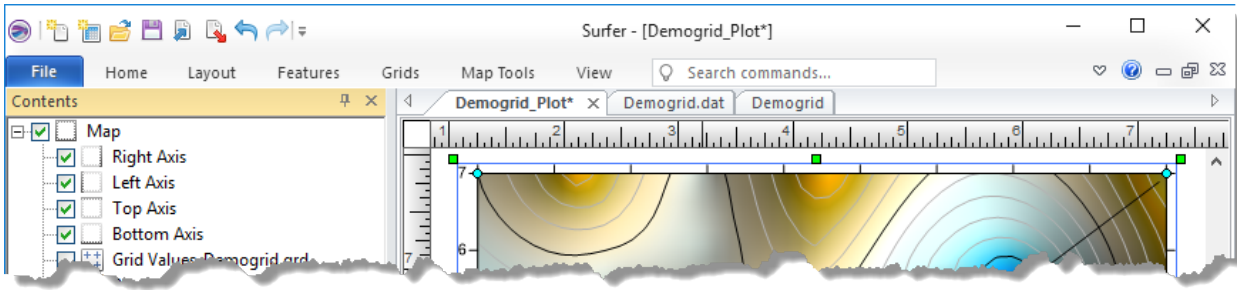
Above the Ribbon are a number of tabs, such as **Home**, **Features**, and **Map Tools**. Clicking or scrolling to a tab displays the commands located in this section of the ribbon. The tabs have commands that are organized into a group. For instance, all the commands for adding drawn objects are on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

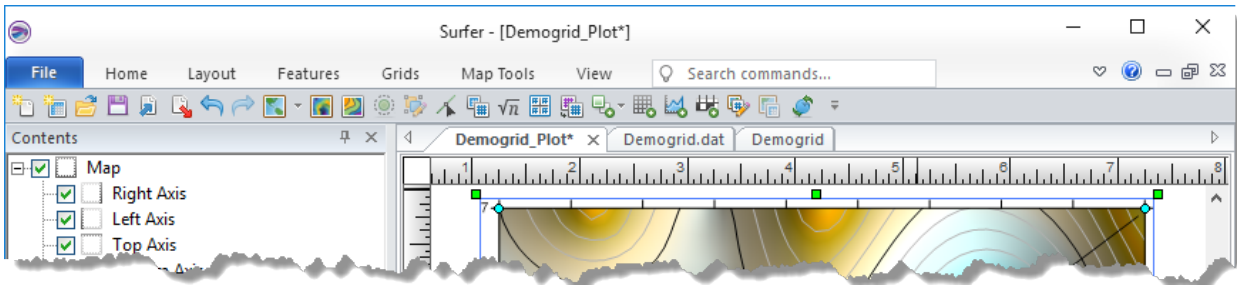
1. To add a command, select the command from the list on the left that you want to add. Click the *Add>>* button and the command is added to the list on the right.
2. To add a separator between commands, set the *Choose commands from* to Main on the left side of the dialog. Select *<Separator>* and click *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

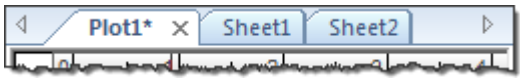
When viewing in tabbed document mode, the tabs may be dragged to reorder them. Left-click on a tab, hold the left mouse button, drag to a new location, and release the mouse button to move the tab to a new location.

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, or 3D view window, **Properties** window, **Contents** window, and ribbon display in a docked view by default. However, they can also be displayed as floating windows. The visibility, size, and position of each item may also be changed.

Visibility

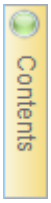
Use the **View | Show / Hide** commands to toggle the display of the rulers, drawing grid, status bar, Contents window, and Properties window. Alternatively, click the or buttons in the **Contents** and **Properties** windows to auto-hide or close the windows.

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.

Position the mouse pointer over the tab to view the window. Move your mouse away from the window and the window "hides" again. You can also 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.



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

Size

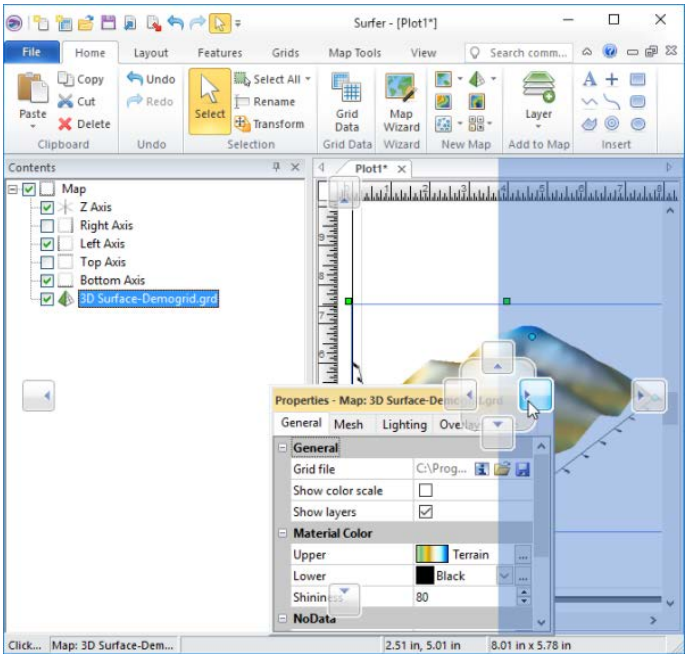
You can drag the sides of the application window, **Contents** window, **Properties** window, or document window to change its size. If a window is docked, its left and right bounds are indicated by a cursor, and its upper and lower bounds are indicated by a cursor. Click and drag the cursor to change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



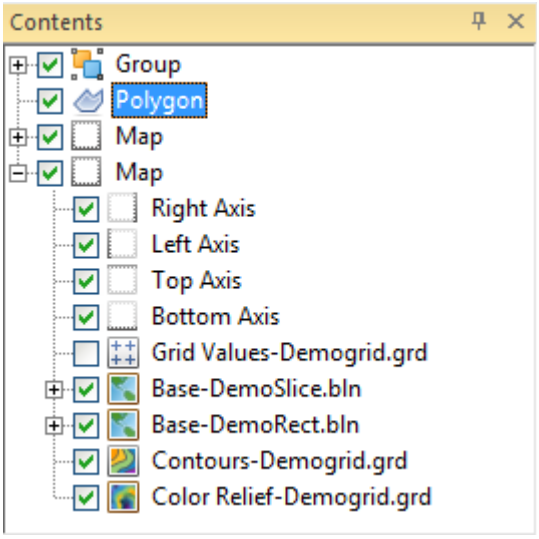
*This image displays the **Properties** window being docked to the right side of the **Surfer** plot window.*

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

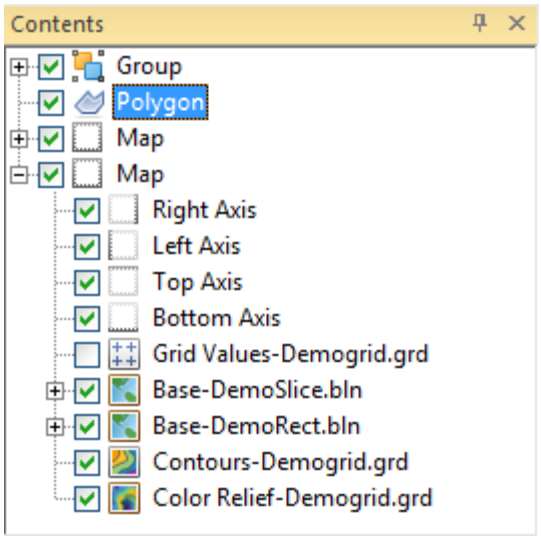
The **Contents** window can be docked on the edge of the **Surfer** window or floated as a dialog. The **Contents** window is displayed in a docked view by default. The manager can also be detached to display as a floating window. Double-click on the **Contents** window title bar to toggle between floating 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.



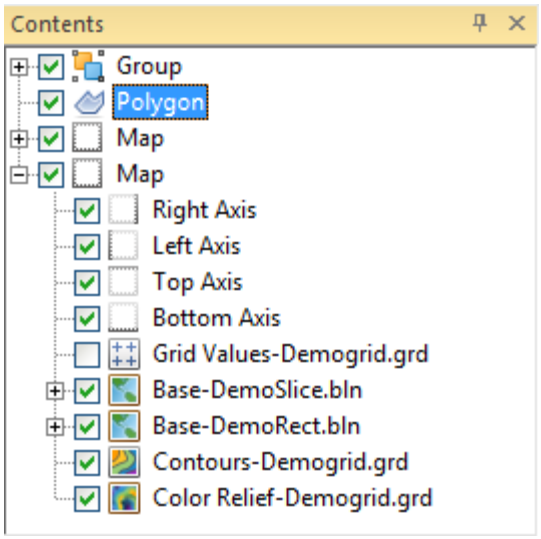
The + sign to left of the Group indicates it is collapsed. The - sign to the left of the Map indicates it is expanded.

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.

Object Visibility

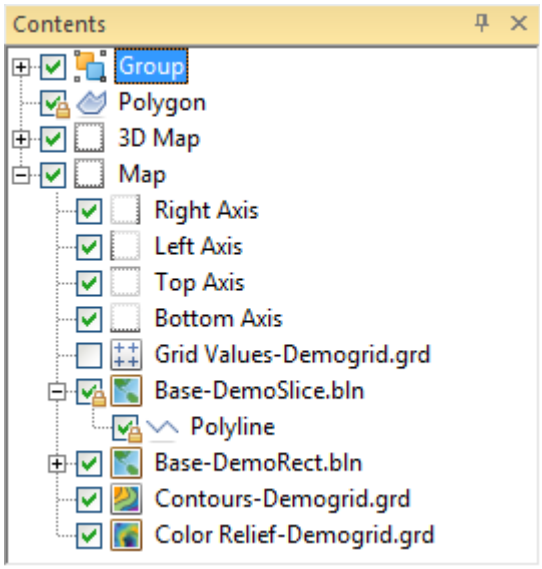
Each object in the **Contents** window includes an icon indicating the type of object and a text label for the object. All objects also have a check box that indicates if the object is visible. A ☑ indicates the object is visible. A ☐ indicates the object is not visible. Click on the check box to change 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 the axes and layers within the *Map* will also be hidden. Note that if a surface is made invisible, any overlays also become invisible.



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

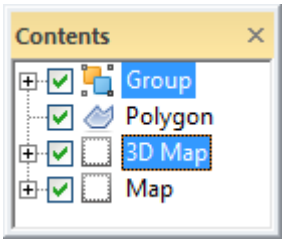
To display the properties for an object, click once on the object in the **Contents** window or in the plot window. The properties are displayed in the Properties window. To display a context menu of available actions for an object, right-click on that object. When the **Properties** window is hidden 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.

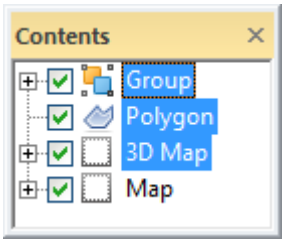
Selecting Objects

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. Only one nested object can be selected at a time. For example, it is not possible to select two axes at once.

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.



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



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

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. 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. Alternatively, you can 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 Map Tools | Layer Tools | Break Apart. Alternatively, right-click on the map layer and select **Break Apart Layer**.

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* help topic or post map *General* help topic respectively.

Opening and Closing the Properties Window

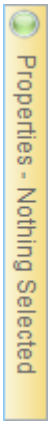
The **Properties** window is opened and closed with the **View | Show / Hide | Properties** command. Clear the **Properties** check box to close the **Properties** window. Check the **Properties** check box to open the **Contents** window. Alternatively, you can click on the button in the title bar of 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.

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

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.



*The **Properties** tab view.*

Changing the Properties Window Location - Floating vs. Docking

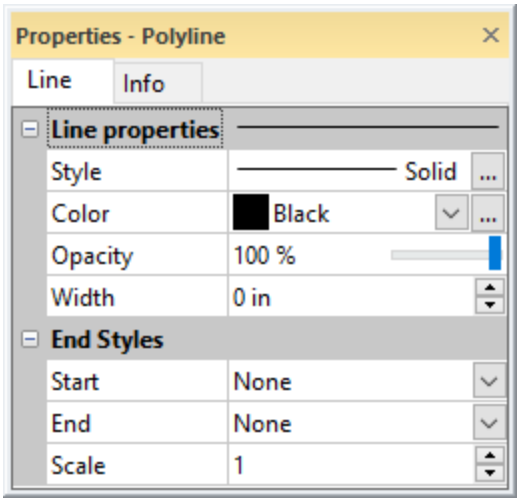
The **Properties** window can be docked on the edge of the **Surfer** window or floated as a dialog. The **Properties** window is displayed in a docked view by default. The window can also be detached to display as a floating window. Double-click on the **Properties** window title bar to toggle between 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.

Changing Properties

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.



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

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

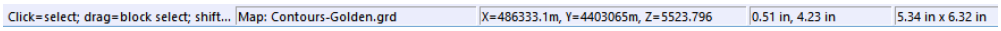
If the *Show property info area* is checked on the **Options** dialog User Interface page, a short help statement for each selected command in the **Properties** window.

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

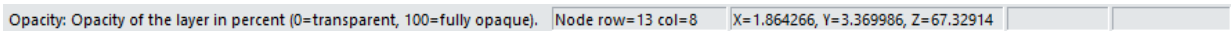
The status bar is divided into five sections. The left section displays information about the selected command or item in the **Properties** window. The second section shows the selected object name or the number of objects/points in the selection. The middle section shows the cursor coordinates in map 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

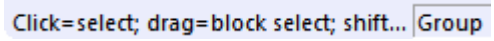
When viewing a grid in the grid editor, the first three sections of the status bar display a description for the selected property in the Properties window, the active grid node grid coordinates, and the map coordinates of the cursor location.



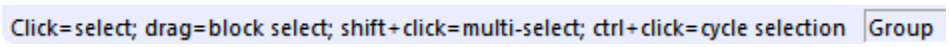
The status bar displays different information when viewing a grid in the grid editor.

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.



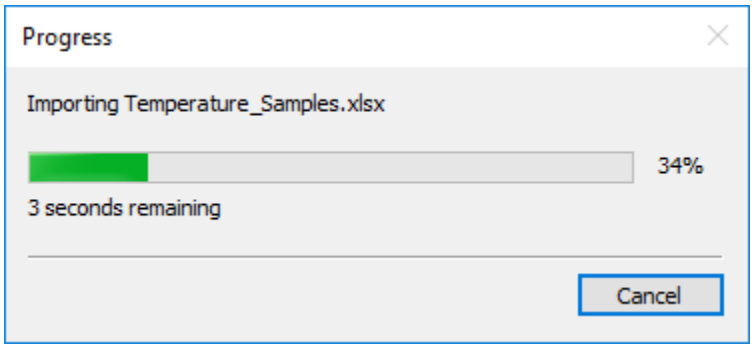
A portion of the status bar. The "..." in the left section indicates there is additional text.



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

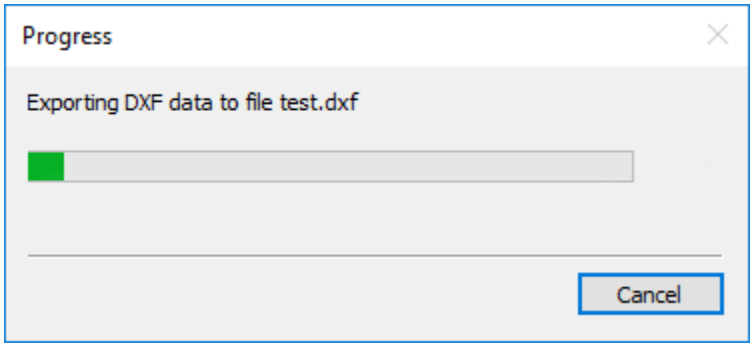
Progress

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 in the **Point Cloud** tab. 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 in the **3D View** tab. 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** tab. 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, reproject 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.

Worksheet Document

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.

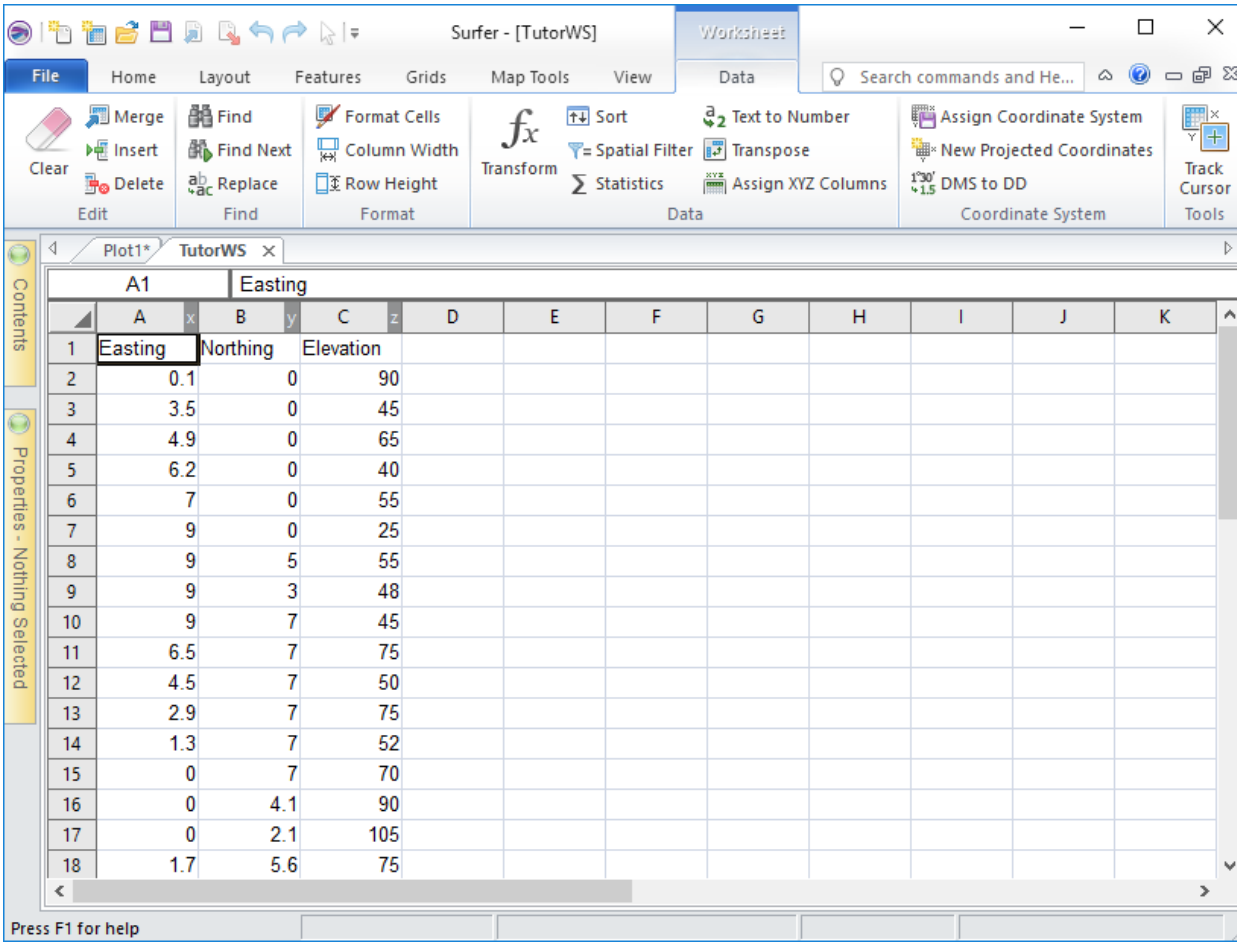
The Application/Document Control menu commands control the size and position of the application window or the document window.

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.



	A	B	C
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
11	6.5	7	75
12	4.5	7	50
13	2.9	7	75
14	1.3	7	52
15	0	7	70
16	0	4.1	90
17	0	2.1	105
18	1.7	5.6	75

*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**, and **Map Tools | Edit Layer | Grid** commands open the grid editor as a new document.

- The **File | Open** command opens the grid editor when a grid or image file is selected in the Open dialog.
- The **Grids | Editor | Grid Editor** command opens 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.

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.

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 normalized red, green, blue, and alpha.

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:

$$I = .30*1.0 + .59*0.0 + .11*0.0 = 0.3$$

Note the normalization process converted the color's (0 to 255) range to (0.0 to 1.0).

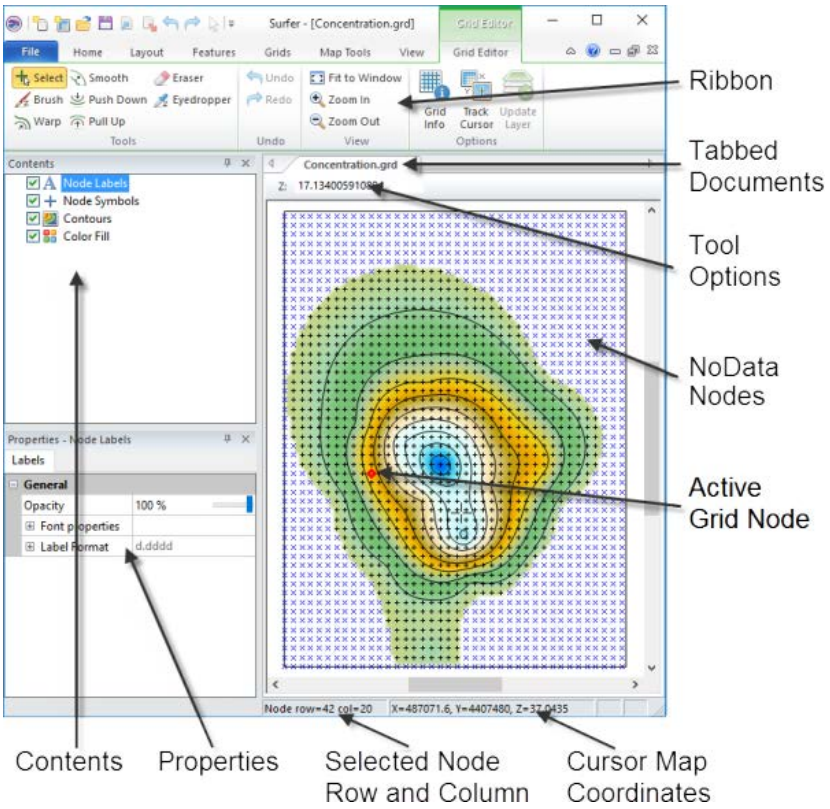
Grayscale Image

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, and Color Fill with the **Contents** window.
Properties	Edit Node Labels, Node Symbols, Contours, and Color Fill display 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
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 you wish to edit in the plot document **Contents** window. Only the grid for this map layer will be edited, even when multiple layers 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. Alternatively, click the **File | Open** command and select a grid file in the Open 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

Data files are used to produce grid files, post data points on a map, or generate a residuals log. 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 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 15** can open .SRF files from previous **Surfer** versions v7 through v14. **Surfer 15** saves in **Surfer 11**, **Surfer 12**, **Surfer 13**, and **Surfer 14** .SRF format. For example, the *Surfer 14 Document .SRF* file type can be opened in **Surfer 14** or **Surfer 15**, but does not contain features that are new in **Surfer 15**. Previous versions of **Surfer** (e.g. **Surfer 14**) cannot open **Surfer 15** .SRF files.

Gridding Overview

A grid 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, shaded relief, grid values, vector, viewshed, watershed, 3D surface, and 3D wireframe map layers all require grids in **Surfer**.

What is Gridding?

Gridding is the process of taking irregularly 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 Z values from 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 fills in these holes by extrapolating or interpolating Z values at those locations where no data exists. 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.

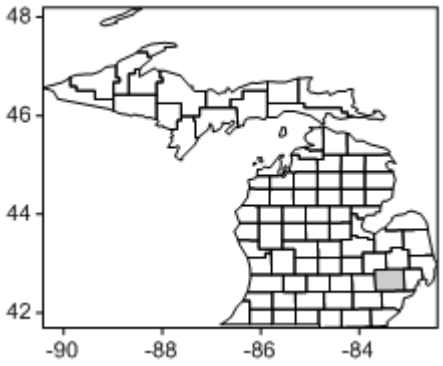
When your XYZ data is regularly spaced, meaning the distance between data points does not change in the X and Y directions, you may produce a grid file that uses the Z values directly and does not interpolate values for the grid nodes. See the *Producing a grid file from a regular array of XYZ data*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

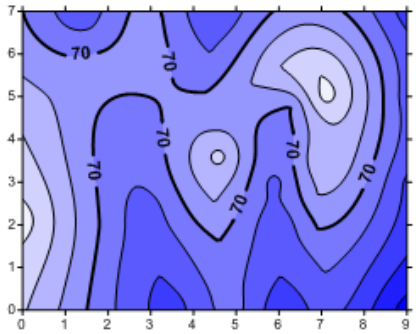
Several different map types can be created, modified, and displayed with **Surfer**. These map types include base, contour, post, classed post, 3D surface, 3D wireframe, color relief, shaded relief, grid values, watershed, 1-grid vector, and 2-grid vector maps. A brief description and example of each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



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

Base Map

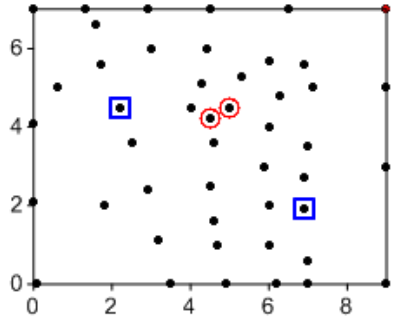
Base maps display boundaries on a map and can contain polygons, polylines, points, text, images, or metafiles. Base maps can be overlaid with other map layers to provide details such as roads, buildings, streams, city locations, areas of no data, and so on. Base maps can be produced from vector files, 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 contour map consisting of contour lines representing elevation.

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.



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.

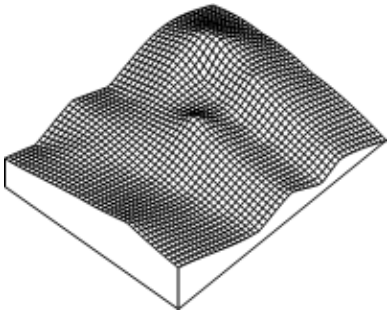
Post Map

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.

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.

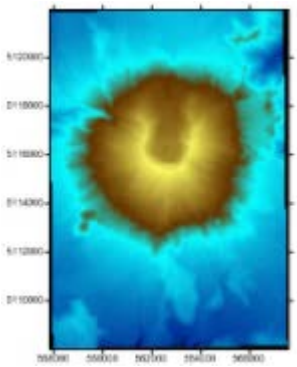
This is a 3D surface map of the Telluride, Colorado USGS SDTS grid file.



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

3D Wireframe Map

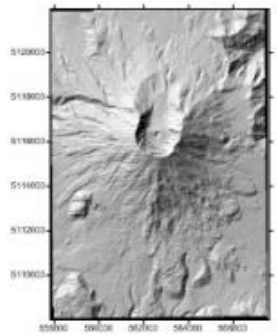
3D wireframe maps are three-dimensional representations of a grid file. Wireframes are created by connecting Z values along lines of constant X and Y.



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

Color Relief Map

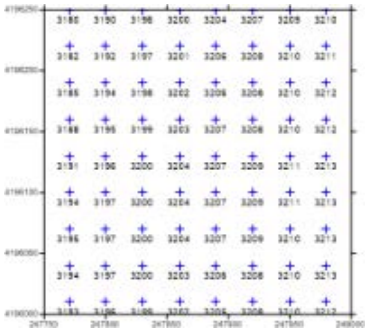
Color relief maps are raster images based on grid files. Color relief maps assign colors based on Z values from a grid file. NoData regions on the color relief map are shown as a separate color or as a transparent fill. Pixels can be interpolated to create a smooth image. Hill shading can be applied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



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

Shaded Relief Map

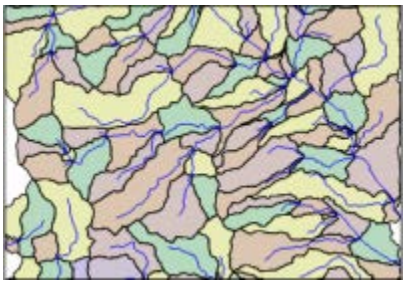
Shaded relief maps are raster images based on grid files. Shaded relief maps assign colors based on slope orientation relative to a light source. Surfer determines the orientation of each grid cell and calculates reflectance of a point light source on the grid surface. The light source can be thought of as the sun shining on a topographic surface.



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

Grid Values Map

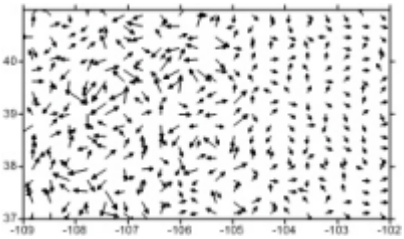
Grid values maps show symbols and labels at grid node locations across the map. The density of the labels and symbols is controlled in the X and Y directions independently. Symbol color can vary by value across a colormap, and symbols and labels can be displayed for only a specific range of values. Grid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

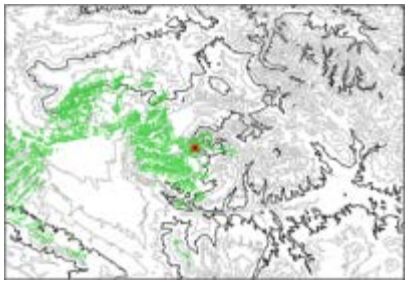
Vector Map

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.

Map Wizard

Click the **Home | Wizard | Map Wizard** command 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.

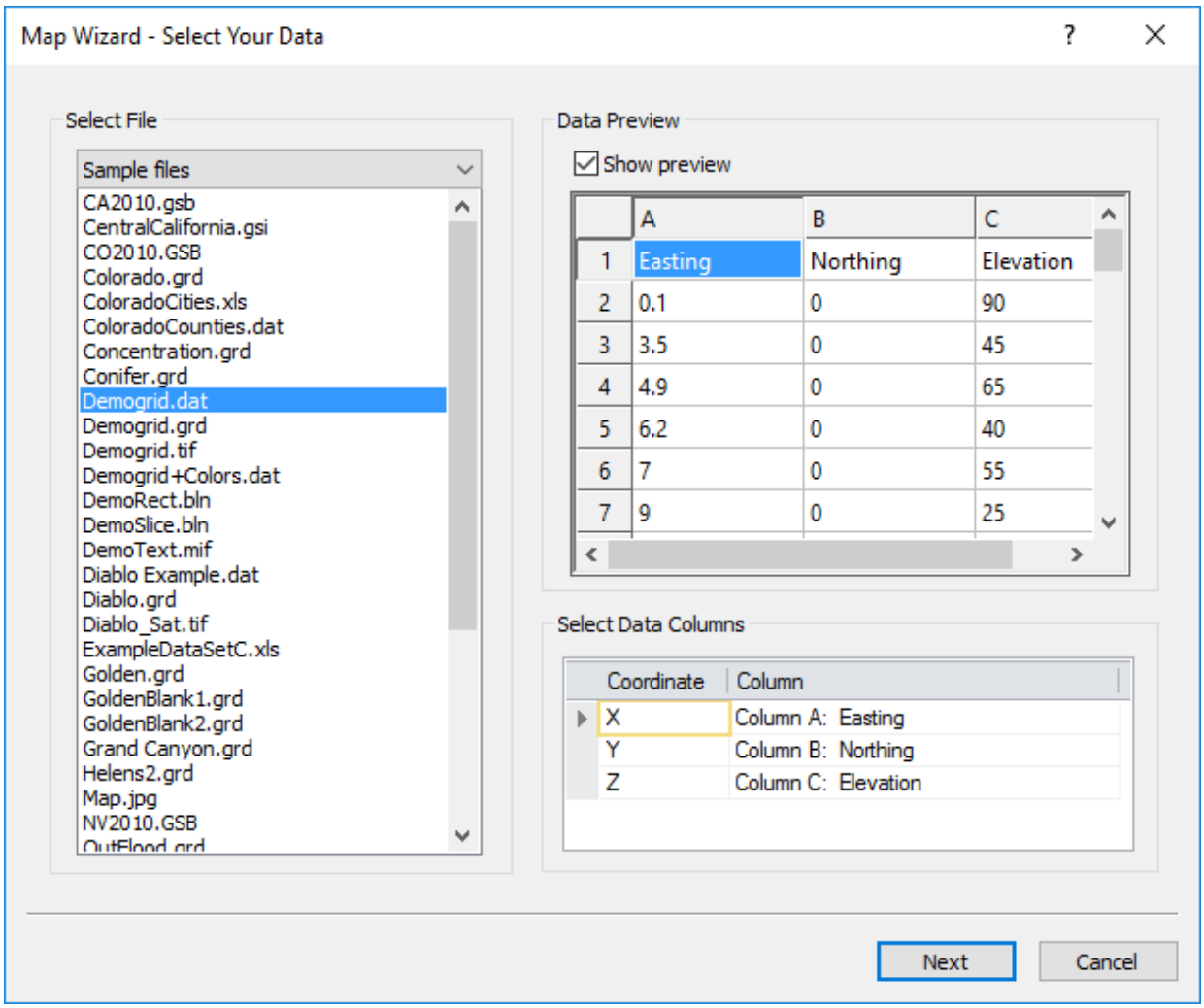
The **Map Wizard** consists of two or three pages, depending on the input data type. Click the following links for more information on each of the wizard pages.

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 sample files 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

Select *Browse* in 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.

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

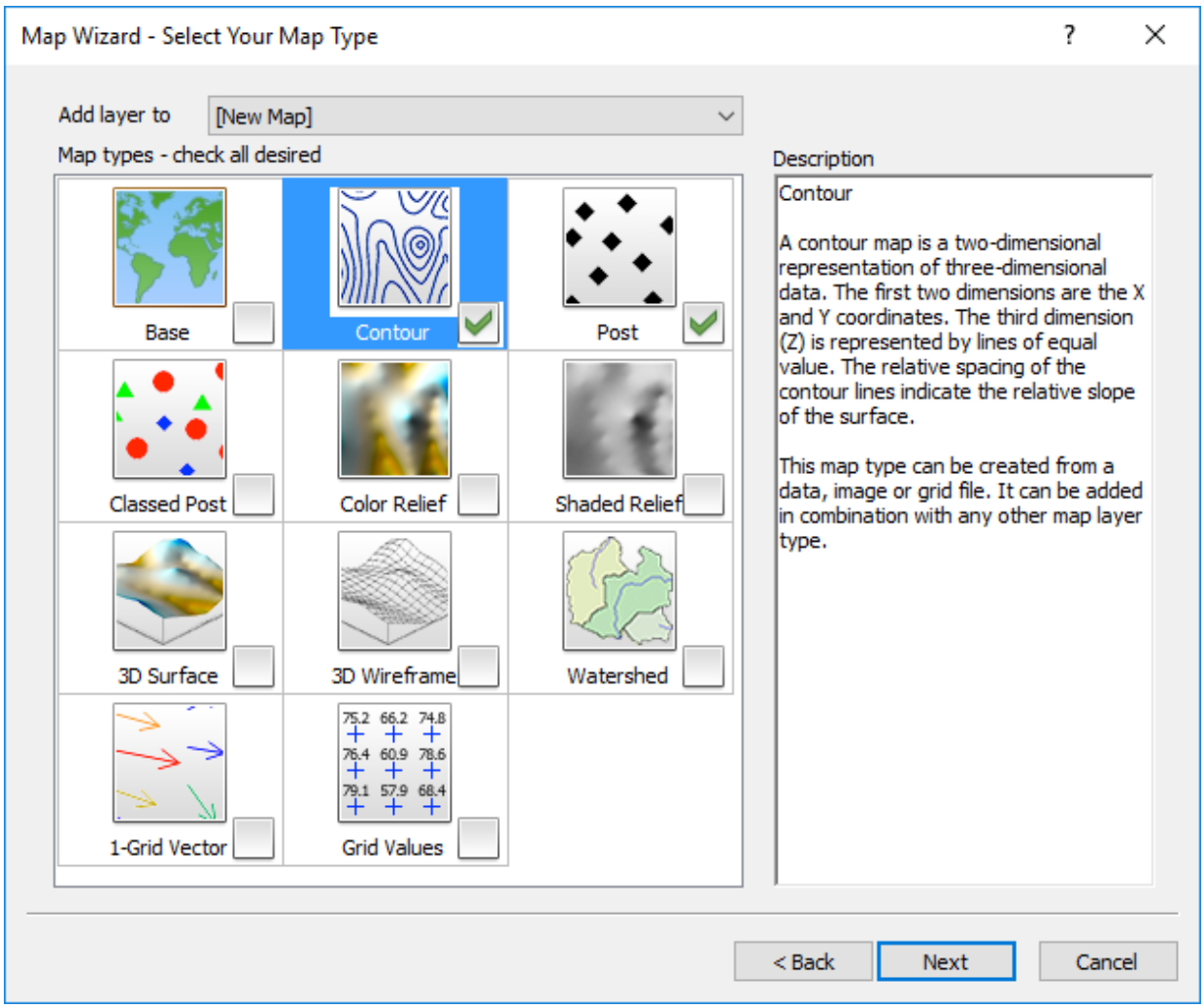
When a data file is selected in the *Select File* list, the *Select Data Columns* section is displayed. Specify the columns that contain the X, Y, and Z data in the *Select Data Columns* table. The *Coordinate* column indicates the row for the X, Y, and Z selections. The *Column* column includes the 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 can be used to create any of the map layer types: Base, Contour, Post, Classed Post, Color Relief, Shaded 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, Shaded Relief, 3D Surface, 3D Wireframe, Watershed, 1-Grid Vector, and Grid Values.
- Base map files can only be used to create a Base map. Click 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, color relief, or shaded 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

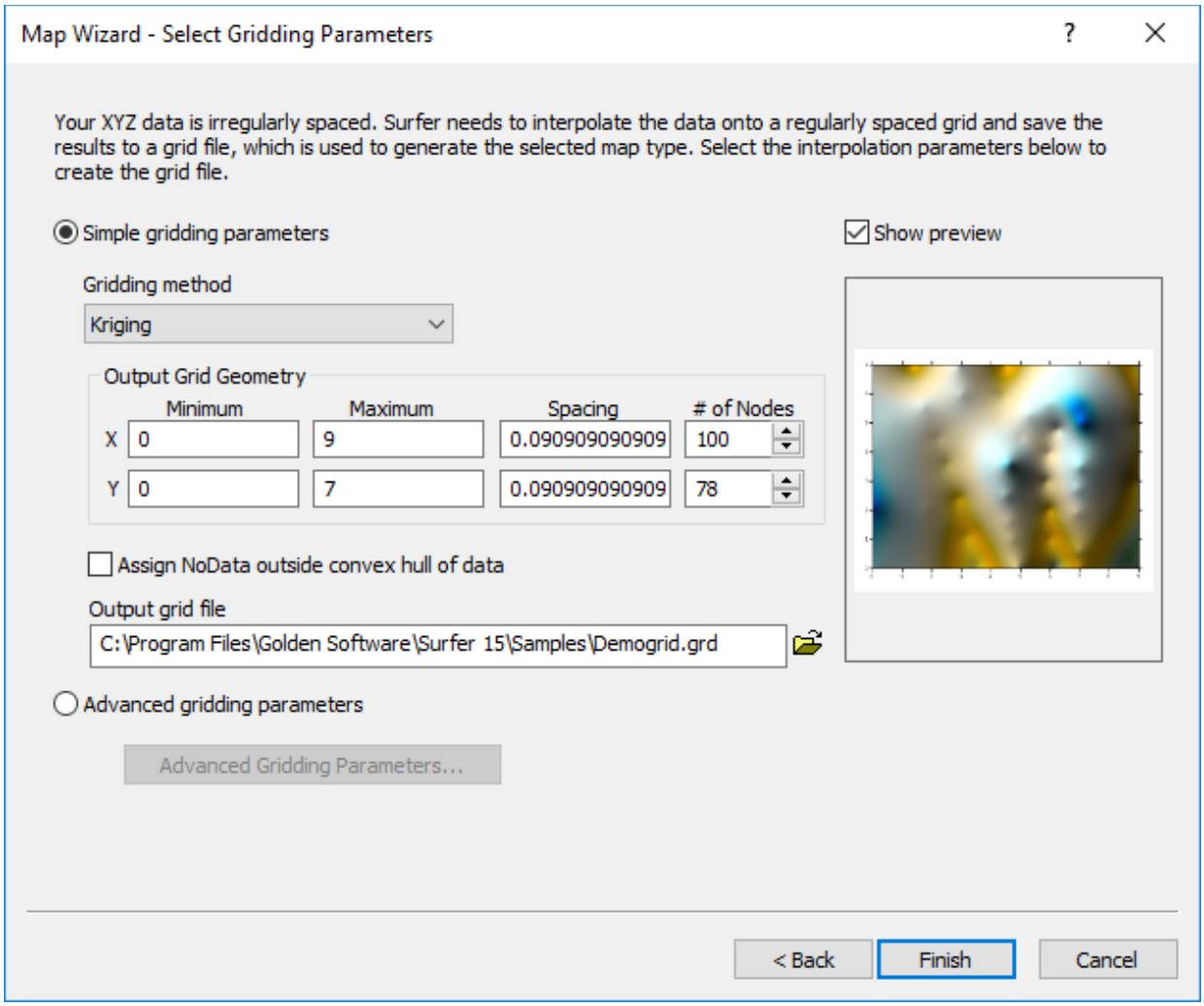
The *Description* section includes a brief explanation of the map type, as well as which data types are used to create the layer type and any conflicts with other layer types.

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 girdding 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.

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, 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.

Assign NoData Outside Convex Hull

Check the box next to the *Assign NoData outside convex hull of data* to automatically assign the NoData value 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

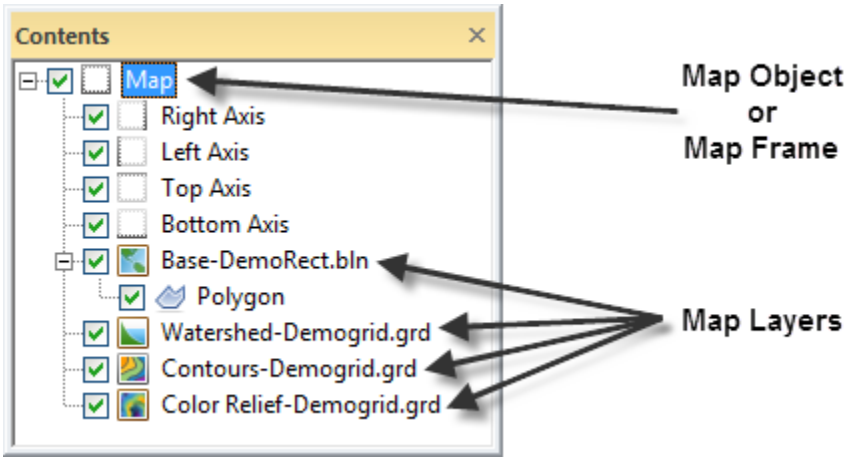
Click *Back* to return to the Select Your Map Type page. Click *Finish* to create the map or add the layers to the existing map. Click *Cancel* to close the dialog without creating a map.

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, shaded relief, vector, watershed, viewshed, or 3D surface layers to a single map. However, a map can contain only one 3D wireframe layer.

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.



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.

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 shaded relief maps, 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** command 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.

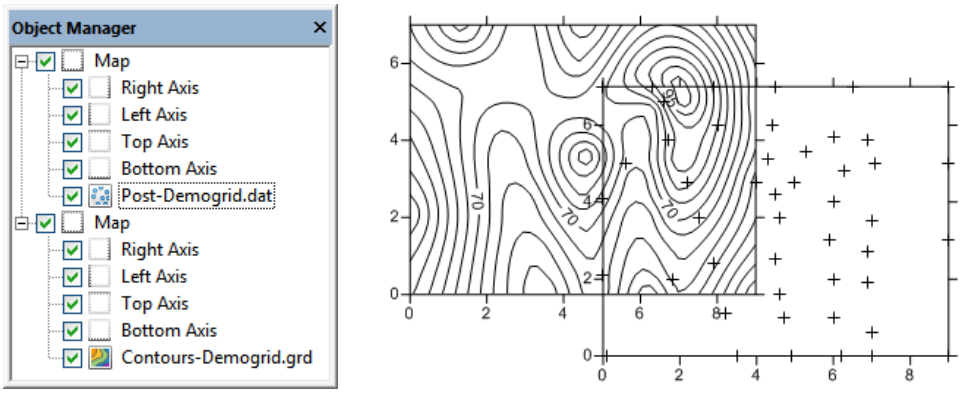
5. Click **Map Tools | Map Tools | Overlay Maps**. The two maps are combined onto a single map object with a single set of axes. The empty map object is automatically deleted.

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

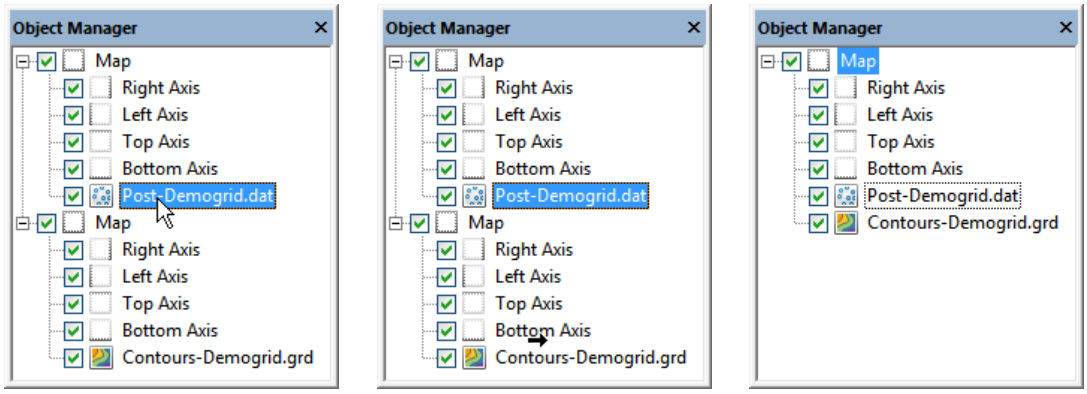
If two maps already exist, you can move (or overlay) a map layer from one map frame into the other map frame by dragging and dropping in the Contents window.

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.



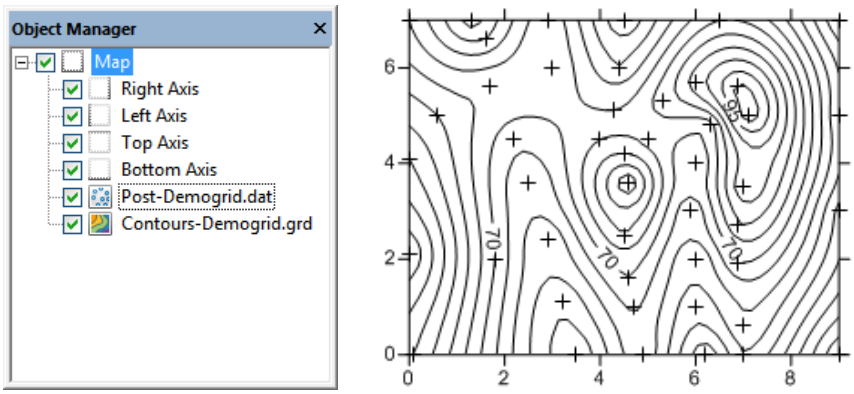
*The contour map layer and the post map layer are displayed in separate map objects in the **Contents** window and the plot window.*

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.



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).

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 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 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. Click on the map layer to select it.
3. In the Properties window, click on the Coordinate System tab.
4. 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*.
5. 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.
6. Click on the *Change* button next to *Coordinate System* to set the desired target coordinate system. When finished, click *OK*.
7. 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 projection 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, worksheet, or grid node editor
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
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. 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

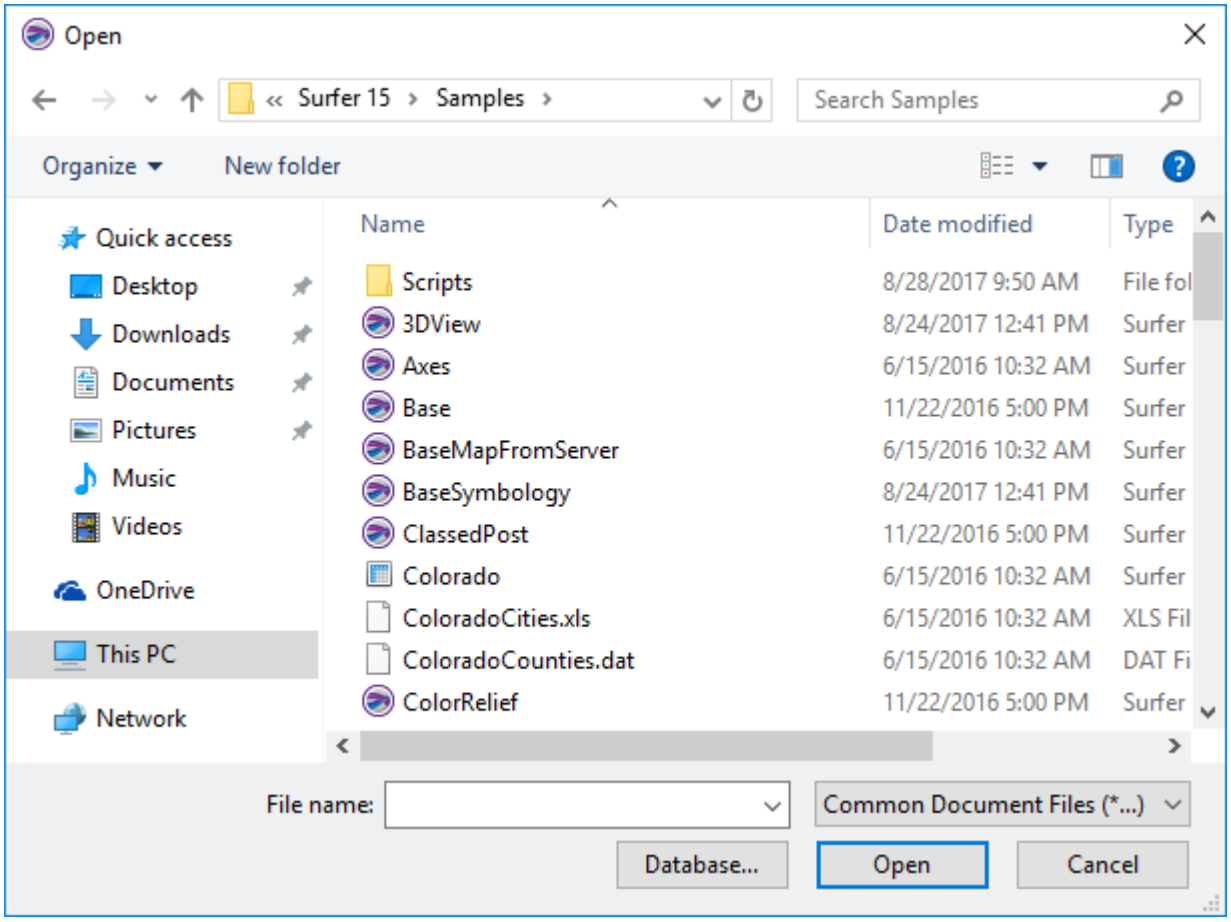
You can create a new plot document or worksheet with the **File | New** command. The worksheet document is used to manage data. Click the **File | New | Worksheet** command, or click the button in the Quick Access Toolbar, to create a new worksheet window.

Open

The **File | Open** command opens a file into a new window. A **Surfer** file .SRF opens in a plot document, a grid file .GRD opens into the grid editor, and data files open in the worksheet. You can also click on the button on the Quick Access Toolbar, or press CTRL + O on the keyboard to open files.

Open Dialog

Use the **File | Open** command in the plot document, worksheet document, or grid node editor to open the **Open** dialog.



*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 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* 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).

Surfer files open in the Plot Document, data files open in the Worksheet Document, and grid files open in the Grid Node Editor.

Import Database

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

File Types

Surfer Files open in a new plot window. These are .SRF files.

Data Files open in the worksheet. These include *.accdb, *.bln, *.bna, *.csv, *.dat, *.dbf, *.dxf, *.las, *.mdb, *.mid, *.sp1, *.seg, *.slk, *.txt, *.xls, *.xlsm, *.xlsx

Grid Files open in the grid node editor. These include *.adf, *.am, *.col, *.an1, *.an2, *.asc, *.aig, *.agr, *.grd, *.asi, *.fld, *.bil, *.bip, *.bsq, *.dic, *.dcm, *.ddf, *.dem, *.dos, *.dat, *.dt?, *.ers,

*.flt, *.?10g, *.grd, *.gxf, *.hdf, *.hdr, *.img, *.rst, *.info, *.lat, *.raw, *.bin, *.stk, *.tif, *.tiff, *.vtk

Image Files open in the grid node editor. These include *.an1, *.an2, *.bmp, *.dic, *.dcm, *.ecw, *.gif, *.jpg, *.jpeg, *.png, *.pnm, *.ppm, *.pgm, *.pbm, *.rgb, *.rgba, *.bw, *.sid, *.ras, *.sun, *.tga, *.tif, *.tiff, *.vtx, *.x, *.ximg

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.

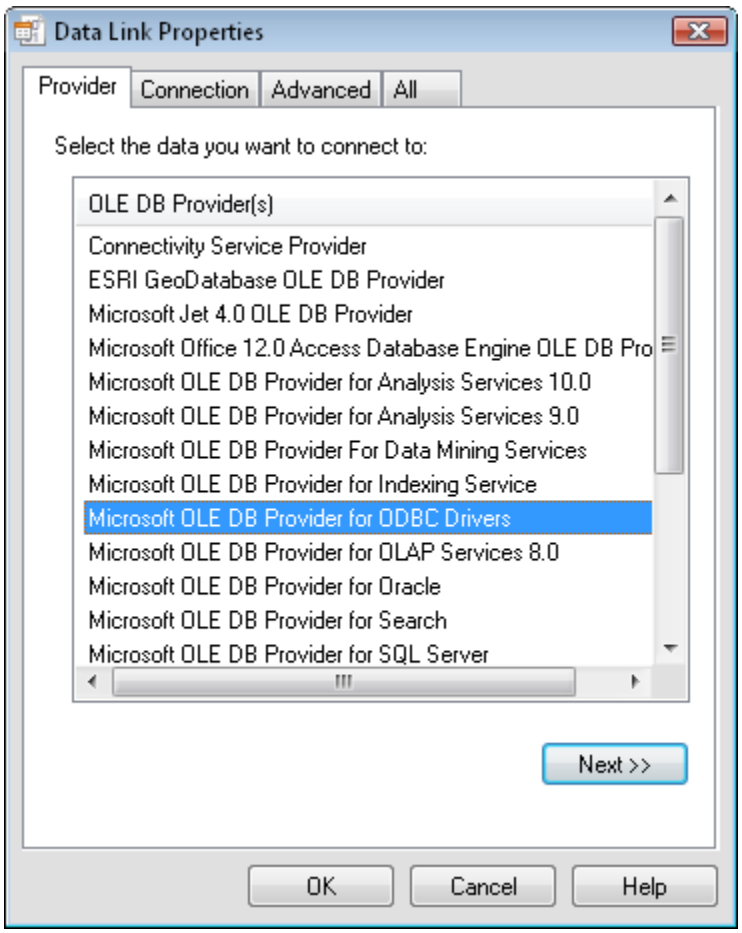
The data linking provides a method to link virtually any database supported by Microsoft via an OLE DB Provider, ODBC, or some other supported format. Because the data link provides access to many types of databases that vary by computer, and since this link is provided by Microsoft, only general information 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

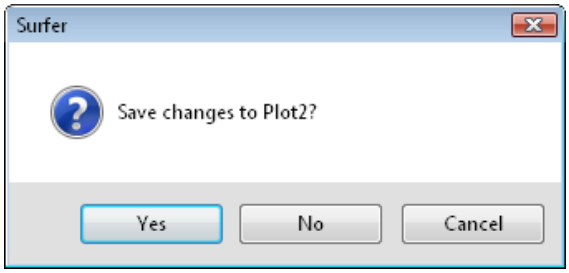
Use the **All** tab to view and edit all OLE DB initialization properties available for your OLE DB provider. Properties can vary depending on the OLE DB provider you are using. For more information about the initialization properties, refer to the documentation provided with each specific OLE DB provider.

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, 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.

Close All

Click the **File | Close All** command 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**.

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 the *Save as type* field is set to *Excel Spreadsheet (*.XLS)*, the file is saved as MYDATA.XLS in Excel format.

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 the *Save as type* field is set to *Comma Separated Variables (*. csv)*, the file is saved as MYDATA.ABC in the .CSV format.

Surfer File Format

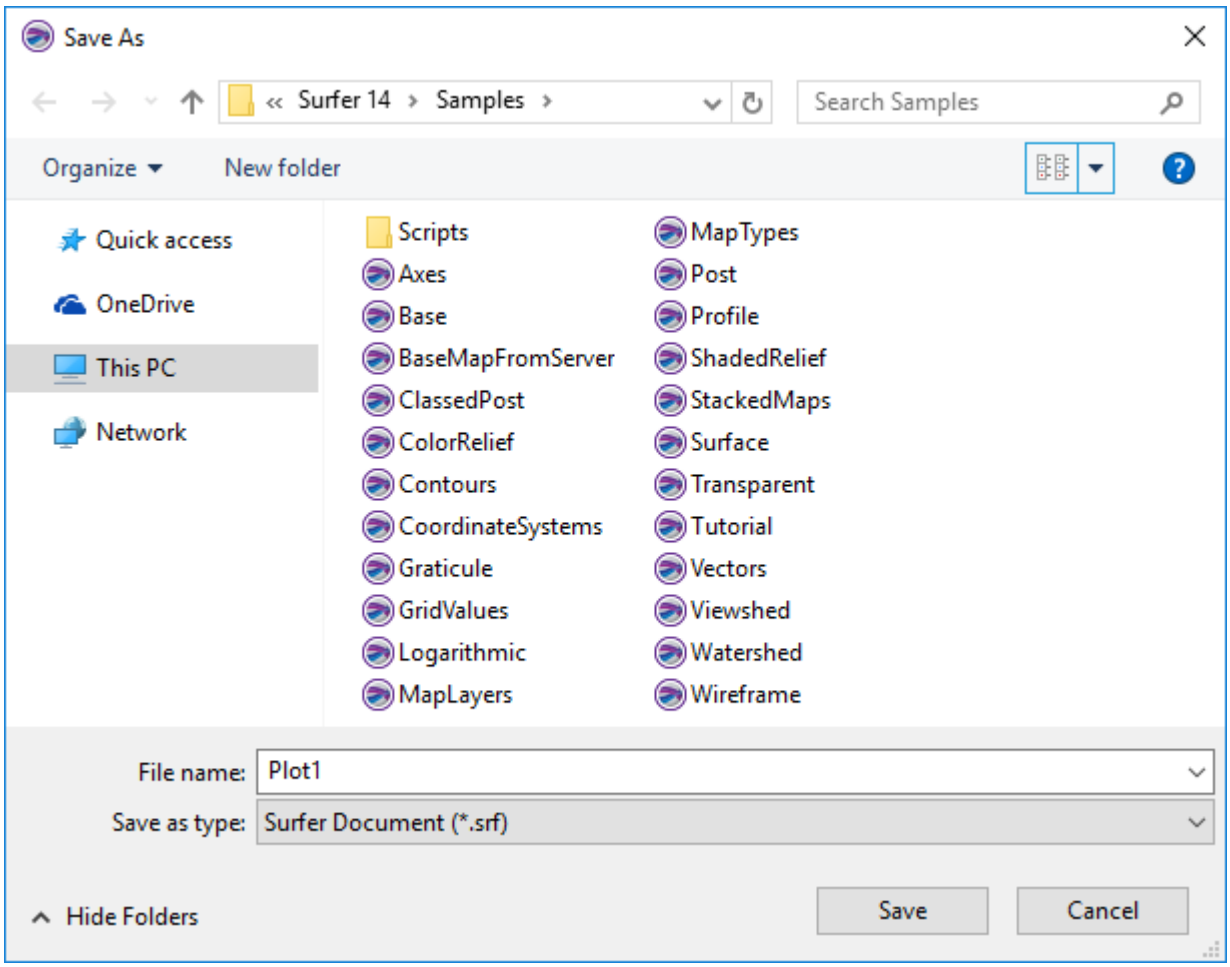
Surfer 15 can open .SRF files from previous versions of **Surfer 7, 8, 9, 10, 11**, **12**, **13**, and **14**. **Surfer 15** saves in **Surfer 11**, **Surfer 12**, **Surfer 13**, and **Surfer 14** .SRF format. For example, the *Surfer 14 Document .SRF* file type can be opened in **Surfer 14** or **Surfer 15**, but does not contain features that are new in **Surfer 15**. Previous versions of **Surfer** (e.g. **Surfer 14**) cannot open **Surfer 15** .SRF files.

Save As

The **File | Save As** command 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 Document (*.srf)* files created in Surfer 15 can only be opened in Surfer 15. *Surfer 11 Document (*.srf)*, *Surfer 12 Document (*.srf)*, *Surfer 13 Document (*.srf)*, and *Surfer 14 Document (*.srf)* files can be opened in previous versions of **Surfer**. For example *Surfer 11 Document (*.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 the *Save as type* field is set to *Excel Spreadsheet (*.XLS)*, the file is saved as MYDATA.XLS in Excel format.

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 the *Save as type* field is set to *Comma Separated Variables (*. csv)*, the file is saved as MYDATA.ABC in the .CSV format.

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 to check for the most recent version of **Surfer**. If there is an update available (e.g. **Surfer 15.0** to **Surfer 15.1**), you can follow the directions to download and install the free update. An update contains minor changes to the program. Normally, no 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 (e.g. 15.x) 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 updates once every seven days by default. This preference can be adjusted at any time using the **File | Options** command. Automatic updates allow the program to periodically check for an available update. Enabling automatic updates will allow your copy of **Surfer** to always automatically stay up-to-date.

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.

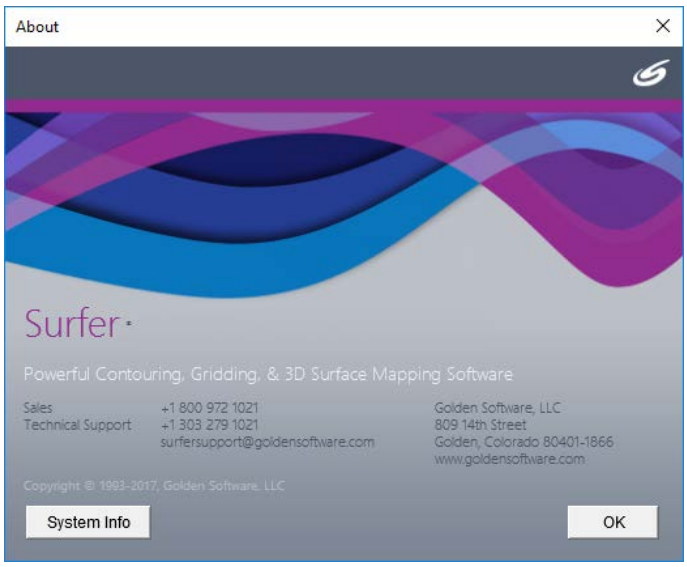
Upgrade

If your maintenance license is active, you can upgrade to the next major version (e.g. **Surfer 15** to **Surfer 16**) 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 to see detailed information about **Surfer**, such as: version number, copyright date, serial number, system information, and Golden Software, LLC contact information.

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

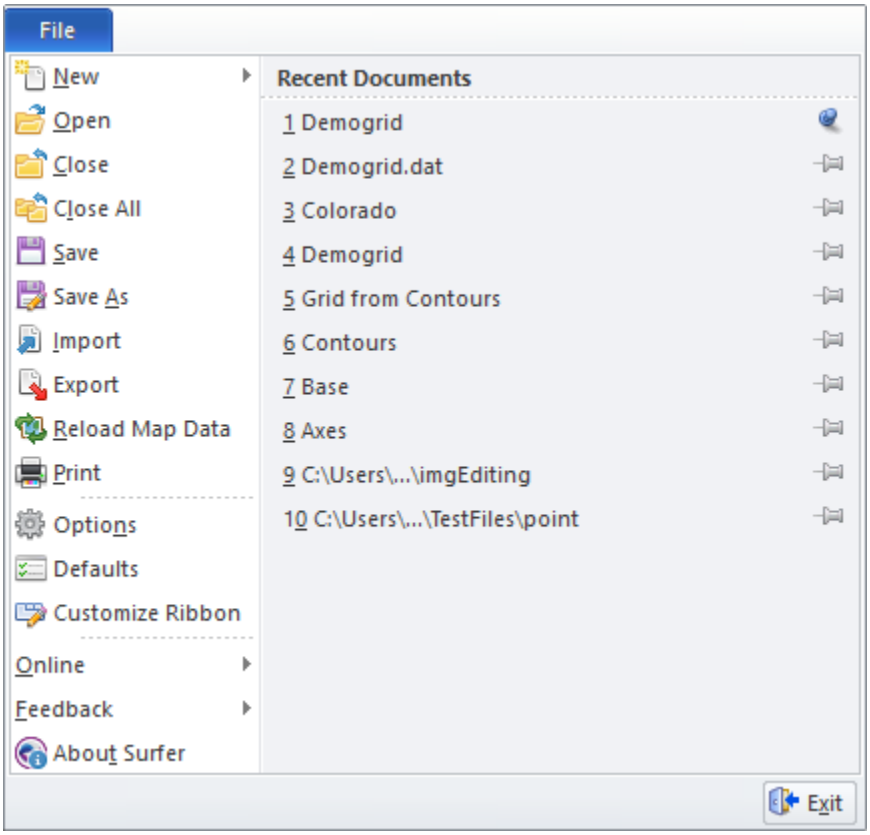
Click the link to open the Golden Software website in a new internet browser window.

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 to close **Surfer**. If changes have been made to any open documents there will be a prompt to save the changes.

Printing the Online Help

The online help topics may be printed. You can print part a single topic, a section of the Table of Contents, or all topics in the Table of Contents.

Printing One Topic

To print one help topic, click the topic you wish to print and click the *Print* button. If the **Contents** page is open in the help navigation pane, the **Print Topics** dialog appears. Select *Print the selected topic* and click *OK*.

Printing One Book

To print one help book, click the **Contents** tab in the help window navigation pane. Click the book you want to print, for example click the *Tutorial* book. Click the *Print* button, and the **Print Topics** dialog appears. Select *Print the selected heading and all subtopics* and click *OK*.

Printing the Online Help

To print all the topics in the help file *Table of Contents*, select the top-level book in the **Contents** navigation pane, *Surfer 15*. Click the *Print* button, and the **Print Topics** dialog appears. Select *Print the selected heading and all subtopics* and click *OK*. **WARNING**: Printing the entire help file takes hundreds of letter-sized sheets of paper and is very time consuming. There is no *Table of Contents* or *Index* printed with the file.

You can purchase the PDF *Surfer User's Guide* that includes all of the documentation for the program. The PDF user's guide is the online help rearranged in a book format and can be printed by the user, if desired. The *Surfer User's Guide* includes a *Table of Contents* and *Index*. The guide can be purchased at www.goldensoftware.com.

Welcome to Surfer Help

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

Getting Help

Within **Surfer**, the online help file is opened by clicking the help button in the upper right corner of the ribbon. You can also quickly search the help by typing a term in the command and help search above the ribbon and clicking *Search help file* in the results. Alternatively, press F1 at any time to open the help. You can navigate help using the **Contents**, **Index**, **Search**, and **Favorites** pages in the navigation pane two the left of the topic page.

- The **Contents** page displays the predefined table of contents. The table of contents has a variety of help books and help topic pages.
- The **Index** page displays and searches index words to find a help topic. If you do not find a topic with an index word, try a search on the **Search** page.
- The **Search** page offers advanced search options including phrases, wildcards, boolean, and nested searching.
- The **Favorites** page allows you add help pages to a custom list. This allows you to quickly find favorite help topics that you reference frequently.

Navigating the Online Help

The navigation pane shows the **Contents**, **Index**, **Search**, and **Favorites** pages. The navigation pane is displayed by default.

The navigation pane can be displayed with the

button and hidden with the

button.

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 pressing 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.

- Click the *Forums* button or the *Knowledge Base* button at the top of the online help to post a question on the forum or search the knowledge base.
- Use the **File | Feedback** commands to send an Information Request, Problem Report, or Suggestion by email.
- Search our website at www.goldensoftware.com or use the **File | Online** commands for additional help, including the Golden Software Home Page, Surfer Product Page, and Frequently Asked Questions.
- The Golden Software support website has a variety of resources including training videos, a support forum, a blog, a user image gallery, and a variety of downloads.
- The **Surfer** web help can be viewed in a browser window by navigating to surferhelp.goldensoftware.com.

Navigating the Web Help

The web help includes three layouts: desktop, tablet, and mobile. The layout automatically changes based on the screen resolution of the device you are using or the size of the browser window. Screen resolutions or browser windows less than approximately 960 pixels wide will display the mobile layout. Screen resolutions or browser windows less than approximately 1280 pixels wide will display the help in the tablet layout.

Desktop Layout

The web help is made up of three sections: the header and toolbar, the navigation pane, and the topic pane. The header and toolbar contain various commands and links. The navigation pane contains the Table of Contents and Index. The topic pane displays the information from the current help topic.

- Click the Golden Software logo in the header to navigate to the Golden Software home page: www.goldensoftware.com
- Click the button to print the contents from the topic pane. The header, toolbar, and navigation pane are excluded from the printed output.
- Click the **Surfer** logo to navigate to the Surfer Help start topic: Introduction to Surfer.
- Click *Hide* to hide the navigation pane and increase the topic pane area. Click *Show* to display the navigation pane.
- Use the *Search* field to locate a topic by keyword or title.
- Click **Contents** to view the Table of Contents in the navigation pane.
- Click **Index** to view the Index in the navigation pane. You can filter the Index by typing words or characters into the *Filter Keywords* field.

Tablet Layout

The tablet layout also includes a header and topic pane. The navigation pane and search function is replaced by a toolbar on the right or bottom of the window depending on the tablet orientation.

- Click the Golden Software logo in the header to navigate to the Golden Software home page: www.goldensoftware.com
- Click the button to print the contents from the topic pane. The header, toolbar, and navigation pane are excluded from the printed output.
- Click the **Surfer** logo to navigate to the Surfer Help start topic: Introduction to Surfer.
- Click to open the Table of Contents.
- Click to open the Index.
- Click to open the Search.

Mobile Layout

The mobile layout contains only the topic pane and a toolbar at the bottom of the window.

- Click to open the Table of Contents.
- Click to open the Index.
- Click to open the Search.

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 15\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

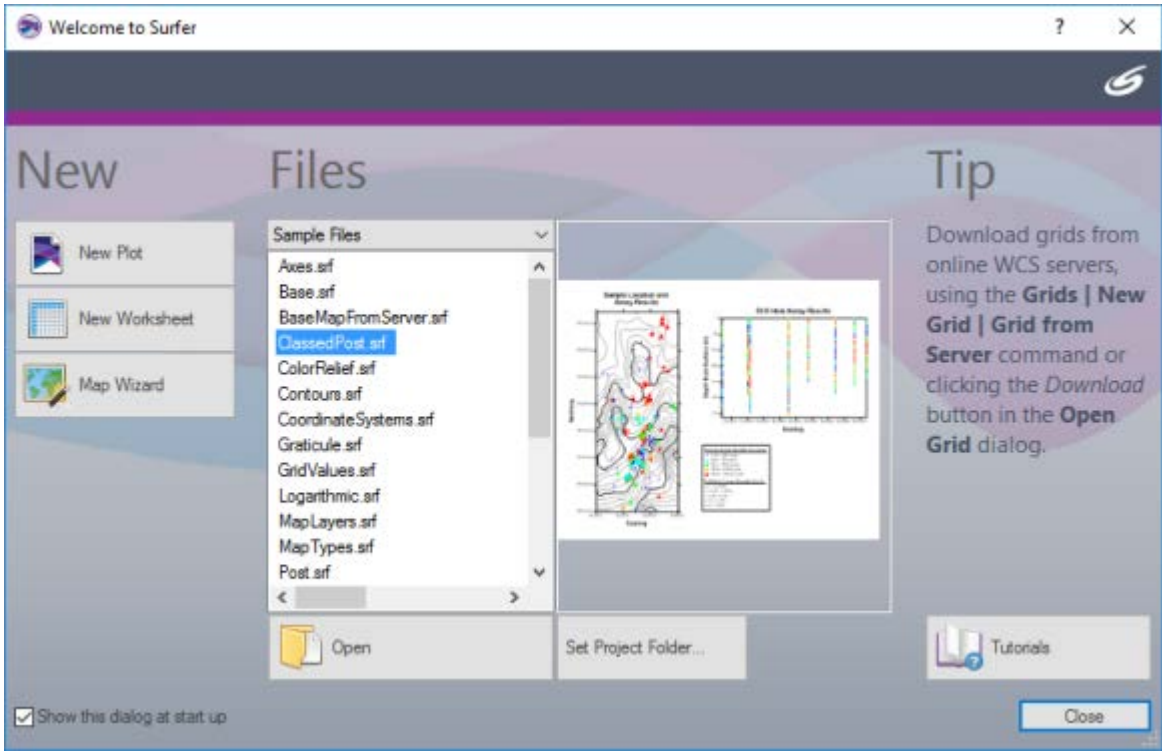
be located in the online help, other sources of **Surfer** help include our support forum, FAQs, knowledge base, and contacting our technical support engineers.

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, and Map Wizard commands, sample files, recent files, 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.

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 15**.

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.

Technical support is available Monday through Friday 8:00 AM to 5:00 PM Mountain Time, excluding major United States holidays. We respond to email and fax technical questions within one business day. 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, KB, or support forum, 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, 809 14th Street, Golden, Colorado, 80401-1866, 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.

New Features

This is an overview of some of **Surfer 15's** new features.

User Friendly

- Separated base layers into *base (raster)* and *base (vector)* layers for clarity
- Updated the default font for better appearance and readability
- The Export file format is now persistent
- The status bar section sizes are now persistent
- Changed the term "blank" to "NoData" for clarity
- Open the Map Wizard directly from the Welcome to Surfer dialog

Map Features

- Apply symbology to a base map:
  - Unique Values - specify line, fill, and symbol properties for unique attribute values
  - Unclassed Colors - colors from a color spectrum are applied to the features by numeric attribute value.
  - Unclassed Symbols - symbols are added for each feature and scaled proportionally by numeric attribute value.
  - Classed Colors - colors are applied to the features by classifying numeric attribute values.
  - Classed Symbols - symbols are added for each feature and classified by numeric attribute values.
  - Include the base layer symbology in a legend.
- Create a point cloud map
  - Color points in the point cloud layer by elevation, intensity, return number, and classification
  - Select, modify, and remove points in the point cloud layer
  - Export the point cloud layer to an LAS/LAZ file
  - Create a grid from the point cloud layer
- View maps in the 3D view

    - Pan, tilt, and zoom the map in 3D space
    - Walk along 3D surfaces
    - Create, view, and record fly-throughs
    - Copy snapshots of the 3D view to the clipboard
- Optimized and multithreaded color relief layer rendering for faster draw times when creating the layer or changing the view
- Export the Grid Slice *Output BLN* formatted as a cross section to plot the profile as a base map
- Select, rearrange, and drag and drop legend layers in the Layers for Legend dialog

Gridding Features

- Multithreading and algorithm optimizations for an average 5x, and up to 14x, faster gridding speed than Surfer v14
- Slice now creates a bln file that can be used to generate a profile plot as a base map

Drawing and Boundary Editing Features

- Create Delaunay triangulation diagrams from points
- Create Thiessen (Voronoi) polgyons from points

Worksheet Features

- Deselect single rows/columns in a multiple row or column selection

Import and Export Improvements

- Import Vector PDF files
- Import TIGER/Line .RT* Format files
- Import IDRISI Binary Vector .VCT files
- Import GeoJSON Data Interchange Format [.GEOJSON, .JSON] files
- Import S-57 Electronic Navigation Chart .000 files
- Import Microstation Design DGN v7 files
- Import MapInfo Table TAB vector data files
- Import and export spatial reference information in MapInfo TAB format

Automation

- Deprecated BaseLayer object and created RasterBaseLayer and VectorBaseLayer to match the user interface
- Extended the image properties to the RasterBaseLayer object

Projections, Coordinate Systems, and Datums

- Renamed and moved Regional/National | Europe | Europe ETRS89 / ETRS-TM[26-39] coordinate systems to UTM | Europe | ETRS89 / ETRS-TM[26-39]
- Renamed Europe UTM zone [28N-38N] to European Datum 1950 UTM zone [28N-38N]
- New Coordinate Systems
    - ETRS89 / ETRS-UTM zone [26N-39N]

Chapter 2 - 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. The Golden Software website contains a knowledge base of questions and answers, an interactive forum, 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	shows you how to begin a new Surfer session and open a new plot window.
Lesson 1 - Viewing and Creating Data	opens and edits an existing data file and creates a new data file.
Lesson 2 - Using the Map Wizard	creates a grid file, the basis for most map types in **Surfer**, and a map with contour, post, and color relief layers.
Lesson 3 - Changing Layer Properties	edits the contour, post, and color relief layer properties.
Lesson 4 - Modifying an Axis	edits the axis tick labels and axis title properties.
Lesson 5 - Creating a Profile	creates a profile line on the contour map and displays the profile.
Lesson 6 - Saving a Map	saves your map and all the information it contains to a **Surfer** .SRF file.
Lesson 7 - Creating a 3D Surface Map	creates and edits 3D surface map.
Lesson 8 - Adding Transparency, Color Scales, and Titles	changes the transparency of various objects, adds a color scale, and adds a map title.
Lesson 9 - Creating Maps from Different Coordinate Systems	loads multiple map layers from different coordinate systems and sets the target coordinate system for the entire map.

Advanced (optional) Lessons

Optional Advanced Tutorial Lessons are available to demonstrate additional features of **Surfer**.

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 15\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 15\Samples, by default.

Starting Surfer

To begin a **Surfer** session:

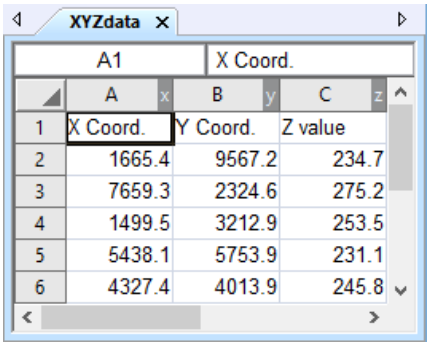
1. Navigate to the installation folder, which is C:\Program Files\Golden Software\Surfer 15 by default.
2. Double-click on the Surfer.exe application file.
3. The Welcome to Surfer dialog appears. Click *New Plot* to open a new blank plot window.
4. A new empty plot window opens in **Surfer**. This is the work area where you can produce grid files, maps, and modify grids.

If this is the first time that you have opened **Surfer**, you are prompted to license Surfer. Activate your Single-User product key, select a license server, or continue using the trial. Your product key is located in the download instructions email. You may also access your product key at your Golden Software My Account page.

If you have already been working with **Surfer**, open a new plot window before starting the tutorial. To open a new plot window, click the File | New | Plot command.

Lesson 1 - Viewing and Creating Data

An XYZ data file is a file containing at least three columns of data values. The first two columns are the X and Y coordinates for the data points. The third column is the Z value assigned to the XY point. Although it is not required, entering the X coordinate in column A, the Y coordinate in column B, and the Z value in column C is a good idea. **Surfer** looks for these coordinates in these columns by default. You can customize the default columns for XYZ data with the Assign XYZ Columns worksheet command. **Surfer** requires the use of decimal degree Latitude (Y) and Longitude (X) values when using Latitude and Longitude values.

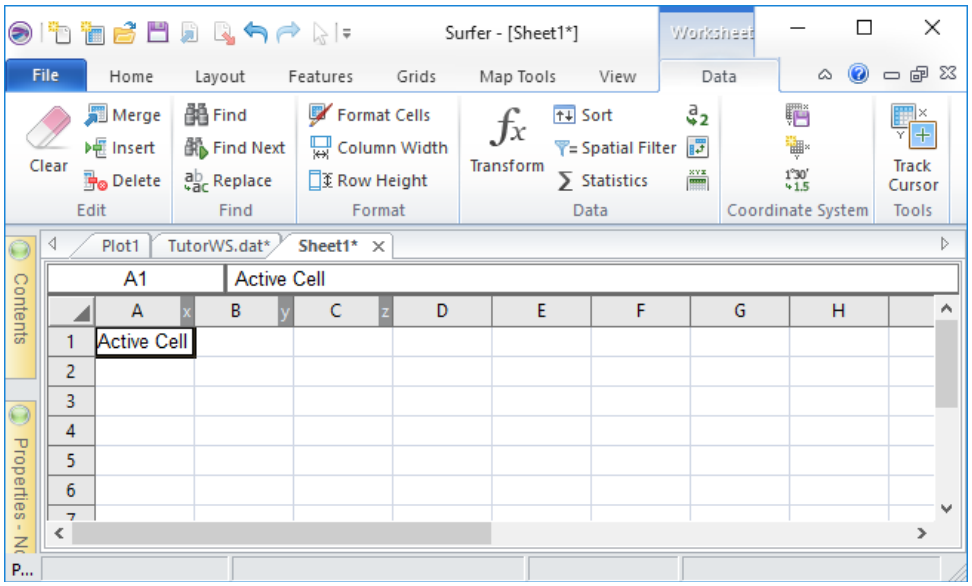


A simple XYZ data file. Notice that the X, Y, and Z data are placed in columns A, B, and C, respectively.

Creating a New Data File - Tutorial

The **Surfer** worksheet can also be used to create a new data file. To open a worksheet window and begin entering data:

1. Click the File | New | Worksheet command, click the on the quick access toolbar, or press CTRL+W on the keyboard. A new empty worksheet window is displayed.
2. Data is entered into the active cell. The active cell is selected by clicking on the cell or by using the arrow keys to move between cells. The active cell is indicated by a heavy border and the contents of the active cell are displayed in the active cell edit 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.
3. When a cell is active, enter a value or text, and the information is displayed in both the active cell and the active cell edit box.
4. The BACKSPACE and DELETE keys can be used to edit data as you type.
5. To preserve the typed data in the active cell, move to a new cell. Move to a new cell by clicking a new cell with the pointer, pressing one of the arrow keys, or pressing ENTER. Press the ESC key to cancel without entering the data.



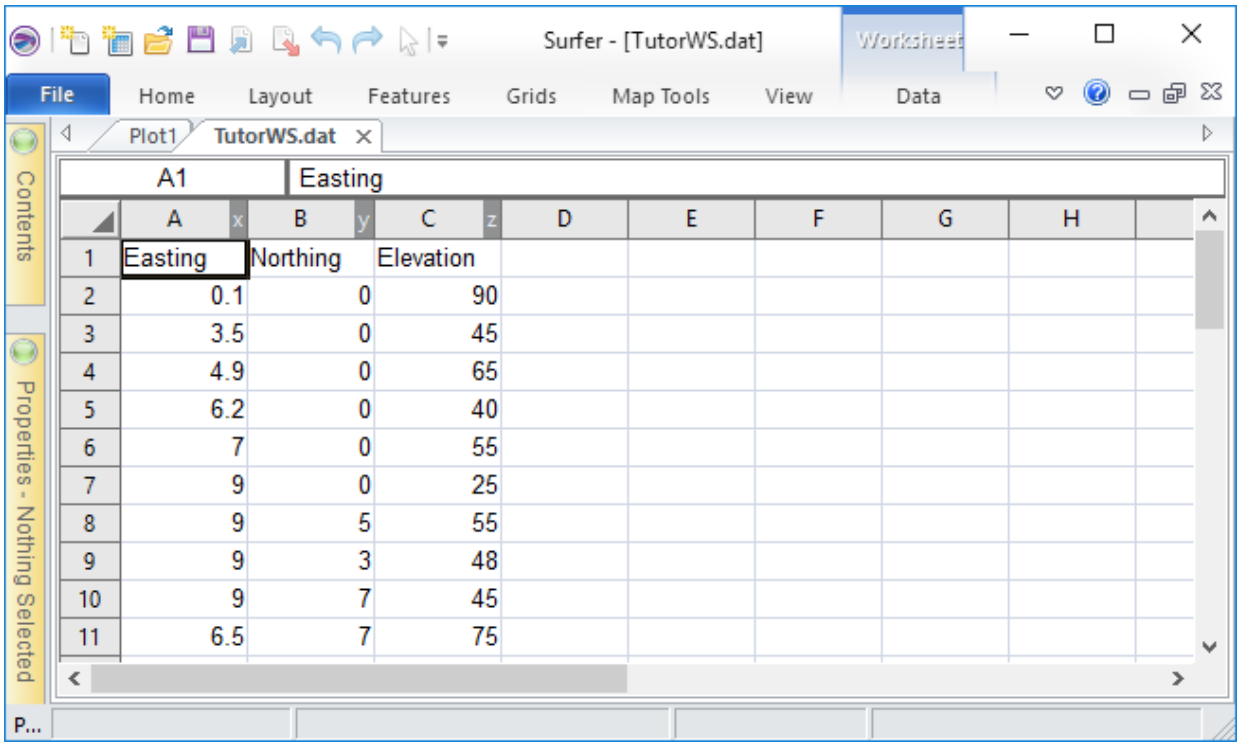
Data are entered into the active cell of the worksheet. Click on the text "A1" or "Active Cell" for the definition of the active cell, active cell location, and the active cell edit box.

Opening an Existing Data File - Tutorial

To look at an example of an XYZ data file, you can open any sample data file in a worksheet window:

1. Click the File | Open command, click the button on the quick access toolbar, or press CTRL+O on the keyboard to open the **Open** dialog.
2. If you are not in the *Samples* folder, browse to it. By default, the *Samples* folder is located in C:\Program Files\Golden Software\Surfer 15. In the list of files, click *TutorWS.dat*.
3. Click *Open* to display the file in the worksheet window.

Notice that the X coordinate (Easting) is in column A, the Y coordinate (Northing) is in column B, and the Z value (Elevation) is in column C. Although it is not required, row 1 contains header text, which is helpful in identifying the type of data in the column. When a header row exists, the information in the header row is used in the **Properties** window when selecting worksheet columns.



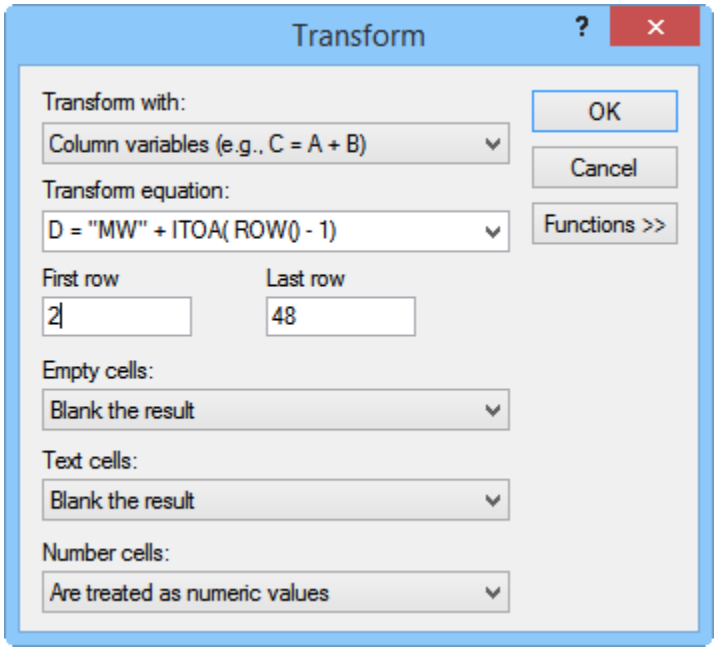
When a data file is displayed, the name of the file is shown in the title bar and in the worksheet tab. In this file, row 1 contains descriptive information about each column of data.

Adding New Data - Tutorial

To edit any value, click in the cell to select it. Type information and the existing value is overwritten. Data can be transformed, sorted, or filtered in this window. New columns can be added. For instance, an ID column can be added which labels each row with a unique identifier. To do this,

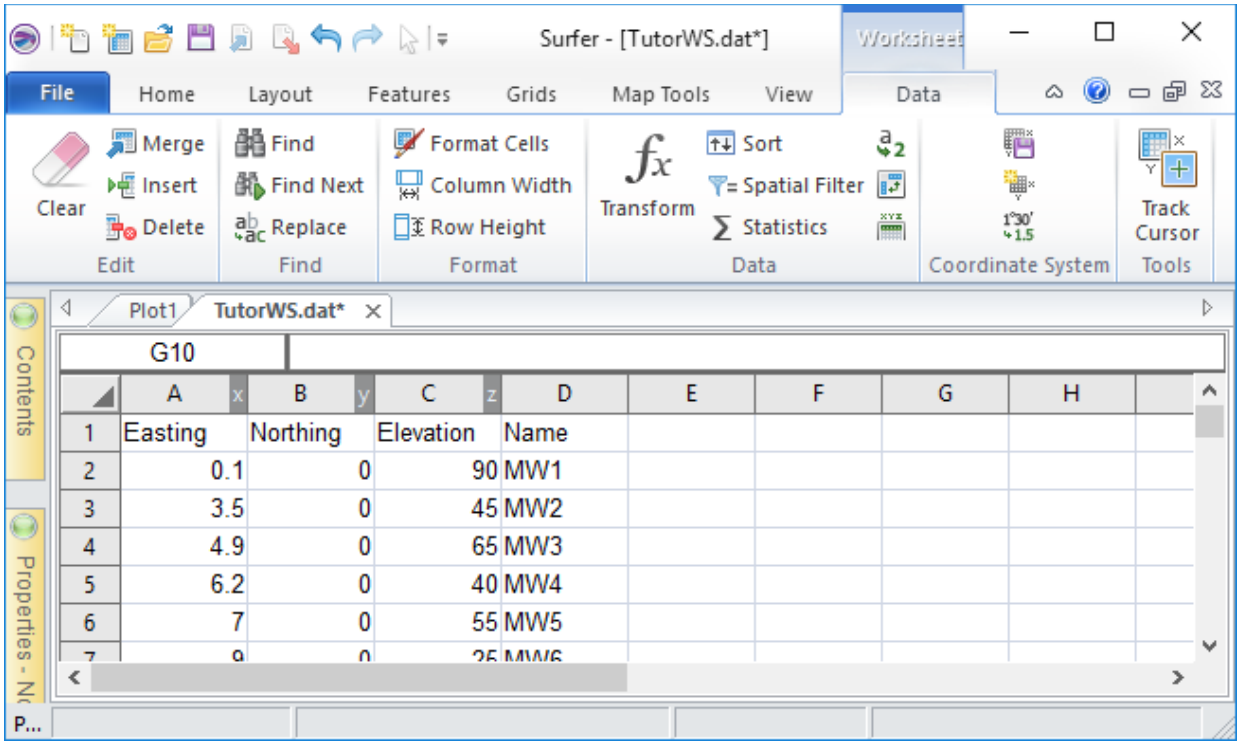
1. Click in cell D1.
2. Type the text *Name*.
3. Press ENTER to save the text and move the active cell to cell D2.
4. Click the Data | Data | Transform command.

5. In the **Transform** dialog, set the Transform with to Column variables (e.g., C = A + B).
6. Set the *Transform equation* to *D = "MW" + ITOA(ROW() - 1)*. This equation will use a prefix of "MW" before a number. The number is the row number minus 1 for each row. The ITOA function converts the ROW() -1 number to text.
7. Set the *First row* to 2.
8. Set the *Last row* to 48 (the last row in the worksheet).
9. Leave the *Empty cells*, *Text cells*, and *Number cells* set to the defaults.
10. Click *OK* and each row will have a unique identifier.



*Set the options in the **Transform** dialog as above to add a unique identifier to each row.*

The worksheet should now have a unique identifier column:



The new column contains a unique identifier for each row. This can be used for labels later in the tutorial.

Saving the Data File - Tutorial

When you have completed entering all of the data, the file can be saved.

1. Click the File | Save As command. The **Save As** dialog is displayed.
2. Navigate to the folder in which you wish to save the tutorial, for example the *Documents* folder.
3. In the *Save as type* list, choose the *DAT Data (*.dat)* option.
4. Type *Tutorial* into the *File name* box.
5. Click the *Save* button and the Data Export Options dialog opens.
6. Accept the defaults in the **Data Export Options** dialog by clicking *OK*.

The file is saved in the *Data .DAT* format as *Tutorial.dat*. The name of the data file appears in the title bar and on the worksheet tab.

Lesson 2 - Using the Map Wizard

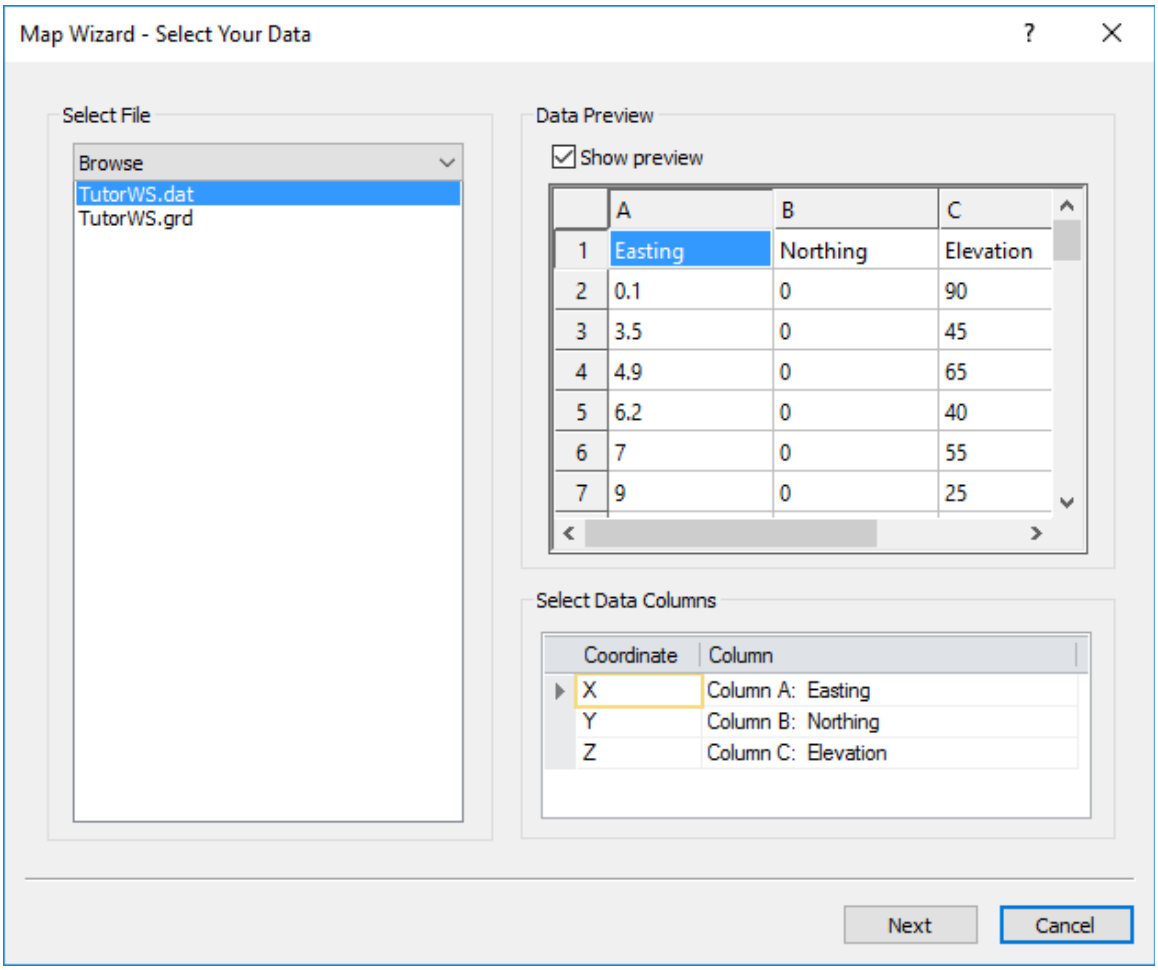
Now that we have saved the data file, we will use the Map Wizard to create a grid and a map with contour and post layers. The **Map Wizard** steps through the map creation process from raw data to a map with one or more layers. The **Map Wizard** is useful for creating multiple map types from a single data file. The **Map Wizard** can use a data, grid, or boundary file as an input file.

1. If you have the worksheet window open, click on the **Plot1** tab above the worksheet window. Alternatively, you can create a new plot window with the **File | New | Plot** command.
2. Click the **Home | Wizard | Map Wizard** command.

The **Map Wizard** opens to the first page, the Select Your Data page. The remaining topics in Lesson 2 will step through the pages of the Map Wizard.

Select Your Data - Tutorial

The first page in the Map Wizard is the **Select Your Data** page. Here you select the XYZ data, grid, vector data, or image file you wish to use to create your map.

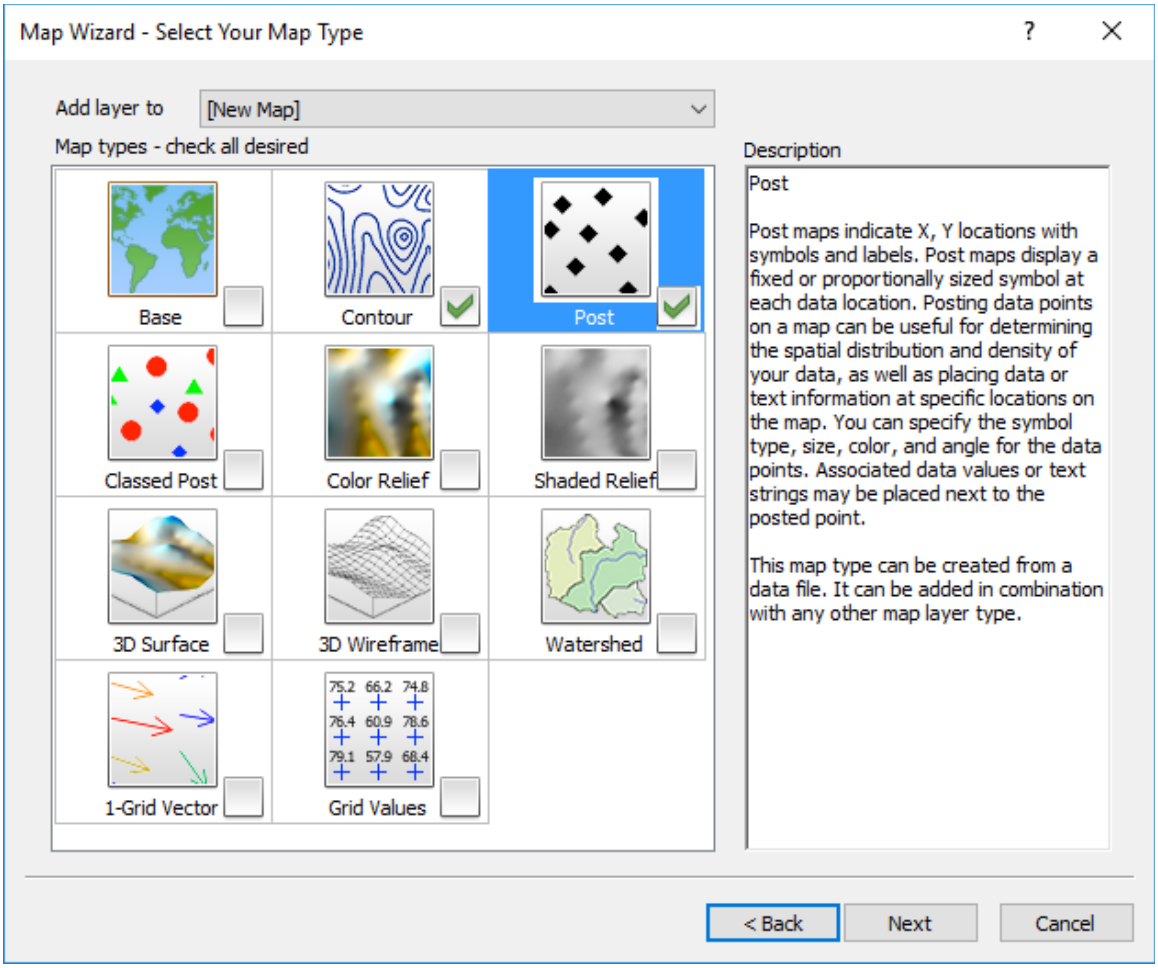


*Select the data file from Lesson 1 in the **Select Your Data** page.*

1. By default, the **Map Wizard** displays the sample files in the *Select File* list. Click *Sample files* and select *Browse* from the list. The Open dialog is displayed. You can also display *Recent files* and *Project files* in the *Select File* list.
2. In the **Open** dialog, navigate to the *Tutorial.dat* file you saved in *Lesson 1 - Saving the Data File*.
3. Select the *Tutorial.dat* file and click *Open*. The *Tutorial.dat* file is loaded in the *Data Preview* section. The column letters and header row information is displayed in the *Select Data Columns* list. By default the *X* coordinate is column A, the *Y* coordinate is column B, and the *Z* coordinate is column C. Any other valid input files in the folder are also displayed in the *Select File* list.
4. Click *Next* in the **Map Wizard**.

Select Your Map Type - Tutorial

Now that you have selected a data file and specified the data columns, we can select which map layers will be included in the map on the **Map Wizard - Select Your Map Type** page.



*Select the map layers you wish to create with the **Map Wizard**.*

XYZ data files are the most flexible input file type. All of the layers are available in the **Select Your Map Type** page after selecting an XYZ data file on the **Select Your Data** page. Some map types will be unavailable after choosing an image, vector, or grid file on the **Select Your Data** page. The data file type and the map type selections determine if a map is created after the **Select Your Map Type** page or if a grid must be created first.

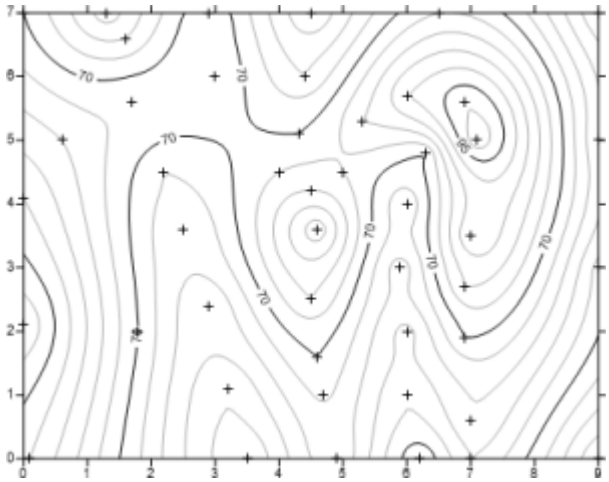
For this tutorial we will include a contour and post layer in our map:

1. Click the *Post* map in the *Map types - check all desired* list to select it. Notice a description is displayed in the *Description* field.
2. Click the *Contour* map in the *Map types - check all desired* list to select it. The *Finish* button changes to *Next*. This is because we must create a grid from the XYZ data file before we can create a contour map.
3. Click *Next*.

Select Gridding Parameters - Tutorial

Grid files are required to produce a grid-based map. Grid-based maps include contour, color relief, shaded relief, vector, viewshed, watershed, 3D wireframe, and 3D surface map layers. If necessary, grid files are created with the **Map Wizard**. Grid files can also be created at any time by using the Home | Grid Data | Grid Data command.

A grid must be created from the *Tutorial.dat* file to display a contour map. The **Map Wizard - Select Gridding Parameters** page controls the gridding options and output grid file name. The **Select Gridding Parameters** page displays a preview color relief map for you to quickly compare gridding methods. We will create a grid with the default gridding method and options.



A map is created with default contour and post layers.

1. Verify that the *Gridding method* is set to *Kriging*. If it is not, click the current gridding method and select *Kriging* from the list.
2. Verify that the Assign NoData outside convex hull of data option is not checked.
3. Verify that the *Output grid file* is named *Tutorial.grd* and in the desired directory, for example your *Documents* folder. If it is not, click and select the desired path for the created grid file.
4. Click *Finish*.

The grid is created and saved, and a map is created in the plot window with a contour and post layer. The map uses the default display properties. The **Map Wizard** is a useful tool for quickly creating maps and grids. However, it is not necessary to use the **Map Wizard**. Grids can be created with the **Grid Data** command, and maps and layers can be created with the **Home | New Map** and **Home | Add to Map | Layer** commands.

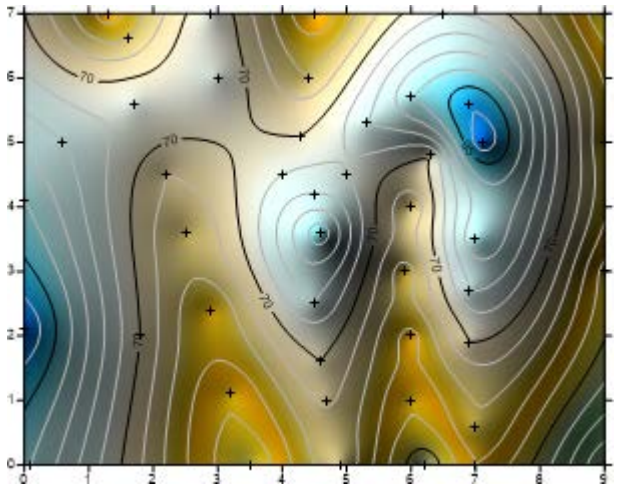
Adding a Color Relief Layer - Tutorial

Map layers allow you to add multiple maps to an existing map to create one map object displaying a variety of map types. The map uses a single set of axes and the map layers are positioned according to the target coordinate system. For example, if you have a contour map of weather data, you can add a post map layer displaying the location and station names of each data collection station.

Multiple map layers can be created at one time when using the **Map Wizard**. However, map layers can also be added to an existing map by selecting the map and using the **Home | Add to Map | Layer** command, by dragging an existing map layer from one map object to another, or by selecting all maps and using the **Map Tools | Map Tools | Overlay Maps** command. Now we will add a color relief layer to the map:

1. Click on the *Map* object in the Contents window, or click on the map in the plot window, to select it.
2. Click the **Home | Add to Map | Layer | Color Relief** command. The Open Grid dialog is displayed.

3. Navigate to the *Tutorial.grd* file you created in *Select Gridding Parameters* and select it.
4. Click *Open* to add the color relief layer to the map.

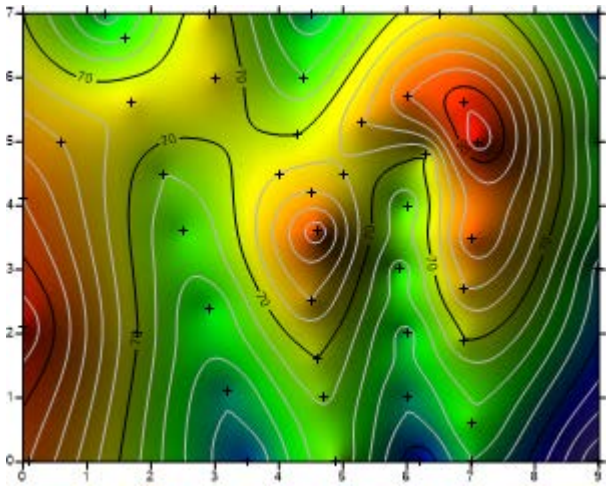


Now a color relief layer is also displayed in the map.

The color relief layer is added to the map and uses the default display properties. In Lesson 3, we will edit the appearance of the map by changing the color relief, contour, and post layer properties.

Lesson 3 - Changing Layer Properties

The map's appearance is mainly determined by the properties of the map layers. This lesson will demonstrate a few of the common properties for controlling the display of contour, post, and color relief layers. However, each map type has many properties and display options. A description and explanation is included for every property in the help.



This color relief layer uses the Rainbow colormap.

We will begin by changing the color relief layer's colors:

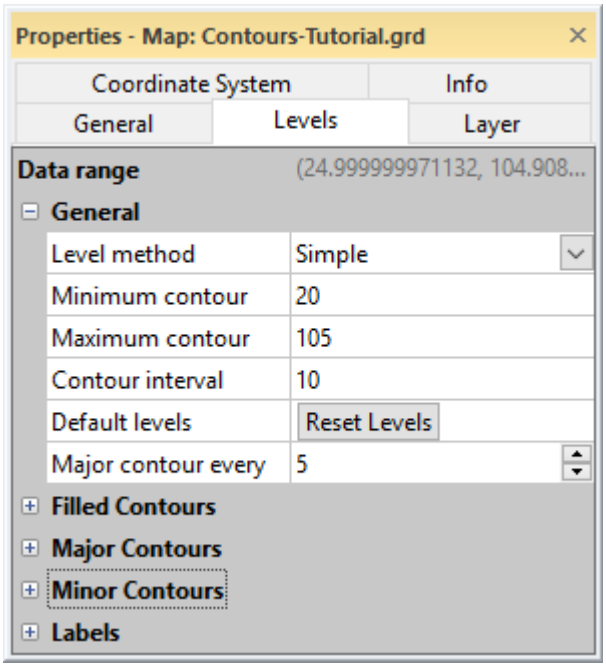
1. Click the *Color Relief-Tutorial.grd* layer in the Contents window to select it. When multiple layers are overlaid in a single map, it is often easier to select the desired layer in the

Contents window. When the color relief layer is selected, the color relief layer properties are displayed in the Properties window.
2. Click the **General** tab in the **Properties** window to display the General page.
3. If necessary, click the ⊞ button next to *General* to expand the *General* section.
4. The *Colors* property determines the colormap used in the color relief map. The default colormap is *Terrain*. Click *Terrain* and select *Rainbow* from the *Colors* list.

Now the color relief layer is using the *Rainbow* colormap. You can click the ... next to the *Colors* property to customize the colormap in the Colormap dialog.

Changing Contour Levels - Tutorial

You can easily modify any of the contour map features. For example, you might want to change the contour levels displayed on the map.

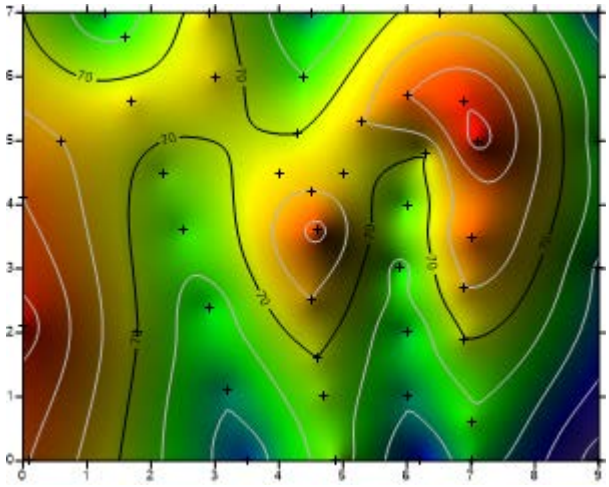


*Go to the **Levels** page to display the contour level properties.*

To change the contour levels:

1. Click on the *Contours-Tutorial.grd* object in the **Contents** window. When the contour layer is selected, the contour properties are displayed in the Properties window.
2. In the **Properties** window, click the Levels tab to display the contour levels and contour line properties for the map. In this example, the contour levels begin at Z = 20. This is displayed next to *Minimum contour*. The *Maximum contour* level is Z = 105.
3. To change the contour range, click in the box next to *Minimum contour* or *Maximum contour*. Highlight the current value and type a new value. The *Data range* of the grid file is displayed at the top of the **Levels** page, making selecting an appropriate range easier. For best results, select values for *Minimum contour* and *Maximum contour* that are in or near this *Data range*.
4. The *Contour interval*, or the frequency of contour lines, is five. This means that a contour line will be displayed every five Z units. We should see contour lines at 20, 25, 30, 35, etc. up to 105. Click in the *Contour interval* box, highlight the value 5, and type the value 10.

5. Press ENTER on the keyboard. The map automatically updates to show contour lines every 10 Z units. The minimum contour level is Z = 20, and the largest contour level is Z = 100.



The contour map is redrawn using new contour levels based on a contour interval of 10.

Changing Contour Line Properties - Tutorial

You can set any of the options in the list on the Levels page to customize the contour map. The *Major contour every* value allows the setting of two different line styles, the major and minor contour lines, for the contour map. By default, the major contour lines are black and labeled and the minor contour lines are gray and unlabeled. The number of minor contour lines and the line properties for both the major and minor contours can be changed.

Setting the Major Contour Value

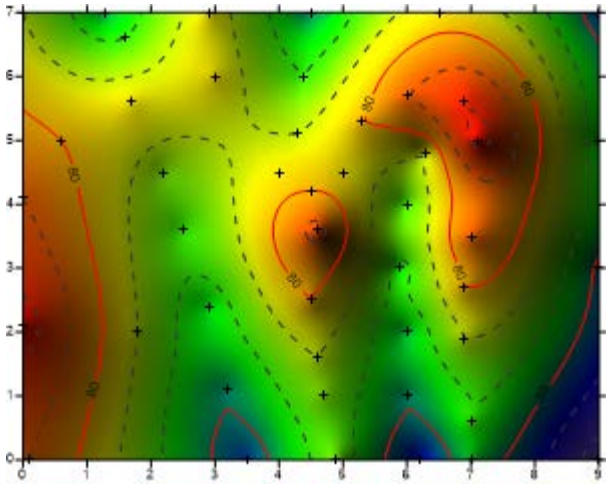
1. Highlight the number in the box next to *Major contour every* and type in a new value of 3.
2. Press ENTER on the keyboard and every third line is a major contour line.

Changing the Major Contour Line Properties

1. Click the ⊞ next to *Major Contours,* if the section is not already expanded.
2. Click the ⊞ next to *Line Properties* in the *Major Contours* section. The major line properties appear.
3. Click the *Black* color box next to *Color*. Select another color, such as *Red*, from the list. The map automatically updates.
4. Click the ▲▼ next to *Width* and change the value to 0.03 inches. Thick red lines now appear at the major contours.

Changing the Minor Contour Line Properties

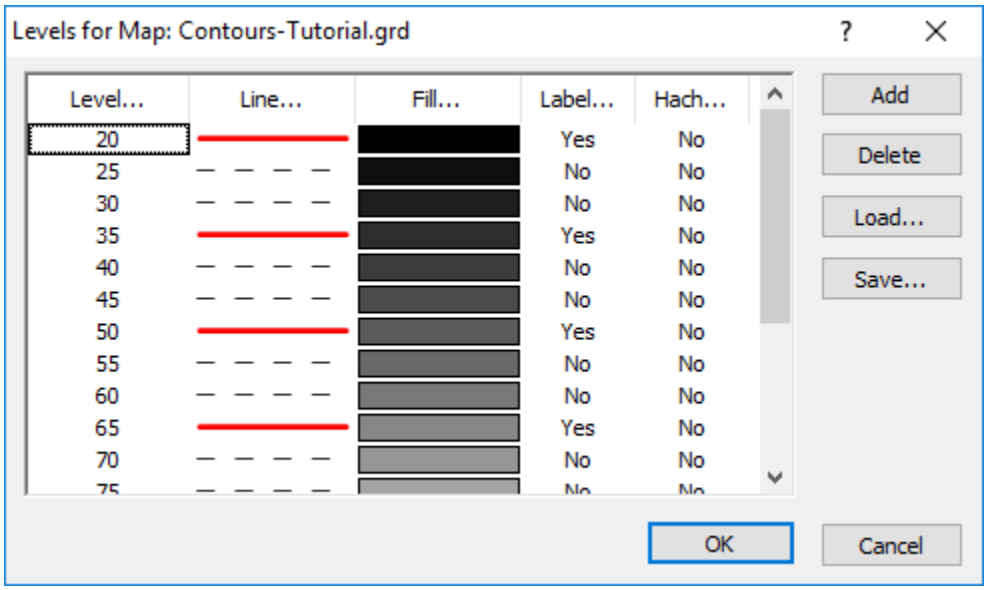
1. Click the ⊞ next to *Minor Contours,* if the section is not already expanded.
2. Click the ⊞ next to *Line Properties* in the *Minor Contours* section. The minor line properties appear.
3. Click the *30% Black* color box next to *Color*. Select another color, such as *80% Black*, from the list.
4. Click in the box next to *Style* and select a dashed line from the list. Dashed gray lines now appear at the minor contours.



The contour map should look similar to this example after changing the major and minor line properties.

Advanced Contour Level Properties - Tutorial

Contour map level properties can be set in one of three methods: *Simple*, *Logarithmic*, or *Advanced*. As seen is the previous topic, the *Simple* method is easy to use and quick to adjust. The *Logarithmic* method is very similar to the *Simple* method, but it uses a logarithmic scale rather than a linear scale. When using the *Advanced* method, each contour line is individually controlled.



*The **Levels for Map** dialog is used to adjust level properties with the Advanced level method.*

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.

The changes that can be made by clicking the **Levels for Map** dialog header buttons include the following:

- Set the minimum, maximum, and contour interval by clicking the *Level* button.
- Set the line properties for all lines to a uniform or gradational color and style by clicking the *Line* button.
- Set the colormap for the foreground and background color and the fill pattern between all contour lines by clicking the *Fill* button.
- Set the label properties for all contour labels or contour labels on a frequency basis by clicking the *Label* button.
- Set the hachure properties for all contours or on a frequency basis by clicking the *Hach* button.

Individual level changes that can be made include the following items:

- Set an individual level value by double-clicking on the level value to enter a new Z value.
- Set the individual line properties for a single level by double-clicking the line style for that level.
- Set the fill color or pattern for a single level by double-clicking on the fill pattern for that level.
- Set the label properties for a single contour label by double-clicking on the *Yes* or *No* under the *Label* column for that level.
- Set the hachure properties for a single contour level by double-clicking on the *Yes* or *No* under the *Hach* column for that level.

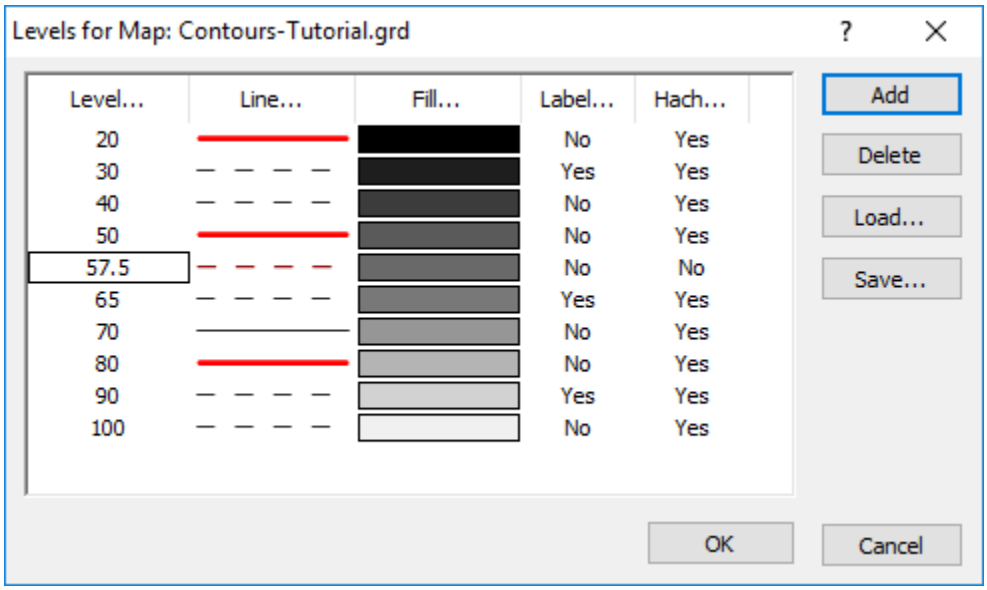
Now we will apply the *Advanced* level method and customize the contour levels with some bulk changes:

1. In the **Contents** window, click once on the *Contours-Tutorial.grd* contour layer to select it.
2. In the **Properties** window, click on the **Levels** tab.
3. Change the *Level method* by clicking on the word *Simple* next to *Level method* and selecting *Advanced* from the list.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced Levels for Map dialog.
5. Clicking the column header buttons makes bulk changes at regular intervals. Click on the *Label* button. The **Labels** dialog opens.
6. Change the *First* value to 2, the *Set* value to 1, and the *Skip* value to 2.
    - The *First* value tells **Surfer** which contour line to first change. This says to set the label format for the second contour line (Z=30).
    - The *Set* value tells **Surfer** how many lines to set with this style. This says to set only one line with the label format.
    - The *Skip* value tells **Surfer** how many lines to skip before setting the next contour line. This says to skip two contour lines. So, the Z=40 and Z=50 contours are not set. The next contour line Z=60 uses the label format. Z=70 and Z=80 are skipped. Z=90 is set. Z=100 is skipped.
7. Click the *Font* button. The **Font Properties** dialog opens.
8. Set the *Size (points)* to 14.
9. Set the Foreground color and opacity color to White.
10. Click *OK* in the **Font Properties** dialog.
11. Click *OK* in the **Labels** dialog. Notice how the label status is changed in the **Levels for Map** dialog.
12. Click on the *Hach* button. The **Hachures** dialog opens.
13. Set the *First* to 1, the *Set* to 1, and the *Skip* to 0.
    - The *First* value tells **Surfer** to set the hachure setting for the first contour line, Z=20.
    - The *Set* value tells **Surfer** to set only one contour line to the hachure style.
    - The *Skip* value tells **Surfer** how many contours to skip. In this case, no contours are skipped. This means that all of the contours will have the hachure style.
14. Check the *Hachure Closed Contours Only* box, if it is not already checked.
15. Change the Direction to Uphill.

16. Click *OK* in the **Hachures** dialog. This changes all of the items under *Hach* to Yes. All closed contours will have hachure marks.
17. Click *OK* in the **Levels for Map** dialog and the bulk changes are made to the contour map.

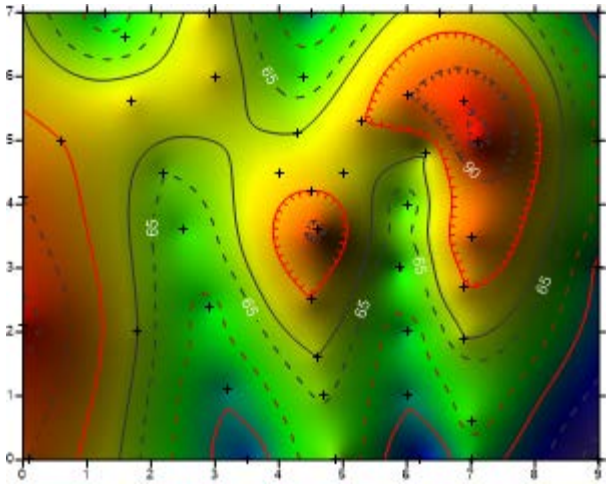
Now we will open the **Levels for Map** dialog again and set properties for individual contour levels:

1. In the **Contents** window, click once on the *Contours-Tutorial.grd* contour layer to select it.
2. In the **Properties** window, click on the **Levels** tab.
3. Make sure that the *Level method* is set to *Advanced*.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced **Levels for Map** dialog.
5. In the **Levels for Map** dialog, you can double-click an individual Z value in the list underneath the *Level* button to change the Z value for that particular contour level. Let's double-click on the number 60.
6. In the Z Level dialog, highlight the value 60 and type in 65.
7. Click *OK* in the **Z Level** dialog, and the contour line level changes to 65.
8. You can also double-click the line style for an individual level to modify the line properties for the selected level. This provides a way to emphasize individual contour levels on the map. Double-click on the line style next to the level 70.
9. In the Line Properties dialog, change the *Style* to a solid line by clicking on the dashed line and selecting the *Solid* line from the list.
10. Click *OK* in the **Line Properties** dialog.
11. Let's add a single contour line halfway between two existing values. Click on the number 65 under the *Level* column.
12. Click the *Add* button. The value 57.5 is added between the 50 and the 65.



*Use the **Levels for Map** dialog to make bulk and individual changes to contour levels.*

13. Click *OK* in the **Levels for Map** dialog and the individual settings are made to the contour map.

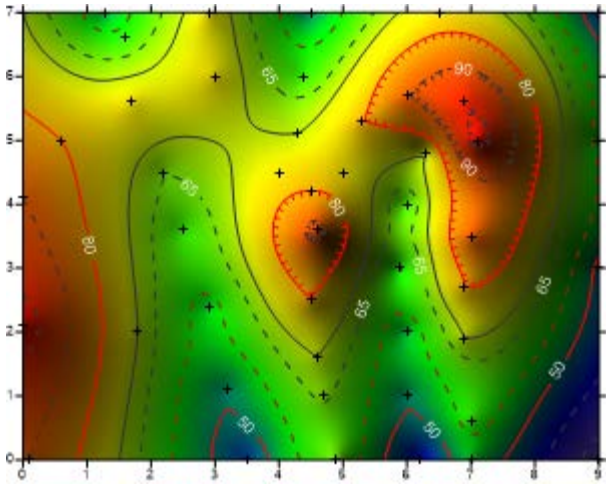


Now the advanced level properties have been applied to the contour map.

Adding, Deleting, and Moving Contour Labels - Tutorial

Contour label locations can be changed on an individual basis. Labels can be added, deleted, or moved. This section will demonstrate adding, deleting, and moving contour labels:

1. Select the contour layer by clicking the *Contours-Tutorial.grd* object in the **Contents** window.
2. Click the Map Tools | Edit Layer | Contour Labels command or right-click on the contour map and select **Edit Contour Labels**. The cursor changes to to indicate that you are in edit mode. Contour labels have rectangular boxes around them in edit mode.
3. To delete a label, click on the label and press the DELETE key on the keyboard. For example, left-click on one of the 65 labels and press the DELETE key on your keyboard.
4. To add a label, press and hold the CTRL key on the keyboard and left-click the location on the contour line where you want the new label to be located. The cursor changes to a black arrowhead with a plus sign + to indicate you are able to add a new label. Add several contour labels to the red lines.
5. To move a contour label, left-click on the label, hold down the left mouse button, and drag the label. Release the left mouse button to complete the label movement.
6. To duplicate a label, hold the CTRL key and then click and drag an existing label. The duplicate label will be dragged to a new location along the line.
7. To exit the *Edit Contour Labels* mode, press the ESC key, click the **Home | Selection | Select** command, or click the **Map Tools | Edit Layer | Contour Labels** command.



*Contour labels can be moved, added, or deleted with the **Map Tools | Edit Layer | Contour Labels** command.*

Exporting 3D Contours - Tutorial

When you have completed a contour map in the plot window, you can export the contour lines with associated Z values to an AutoCAD DXF, 2D SHP, 3D SHP, or TXT file. To export contour lines to a DXF, 2D or 3D SHP, or TXT file:

1. Select the contour map layer by clicking *Contours-Tutorial.grd* in the Contents window.
2. Click the Map Tools | Layer Tools | Export Contours command.
3. In the **Save As** dialog, type *Tutorial contours* in the *File name* box.
4. Select AutoCAD DXF File (*.dxf), 2D Esri Shapefile (*.shp), 3D Esri Shapefile (*.shp), or Text format (*.txt) in the Save as type list.
5. Click *Save* and the file is exported to the current directory. This creates a file titled *Tutorial contours.dxf*, *Tutorial contours.shp*, *or Tutorial contours.txt* depending on what file type you selected. Additional files may also be created that accompany the DXF or SHP file.

The contours are exported as polylines or polygons. The labels and gaps are removed. The exported file can be used in **Surfer** as a base map, or used in other applications. The File | Export command can also be used to export 2D or 3D contours. A comparison between the **Export Contours** and **Export** commands is available on the *Export Contours* help topic.

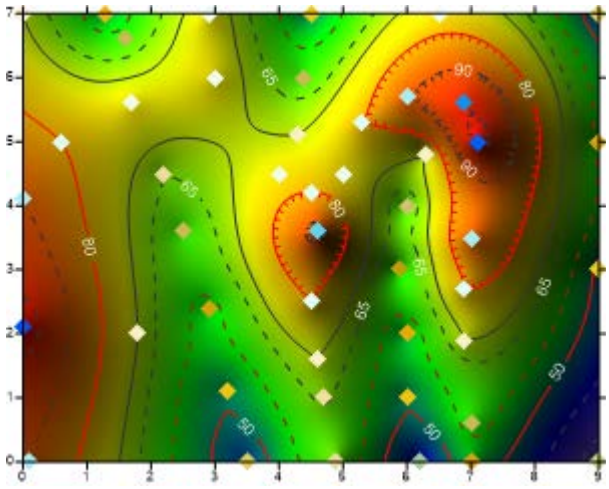
Changing the Post Layer Properties - Tutorial

Symbols in a post map can all be the same or can be selected with a worksheet column. Symbol sizes can all be the same or have proportional sizes. Symbol colors can all be the same or have color based on a column. Now we will edit the post map layer properties:

1. Click on the *Post-Tutorial.dat* layer in the **Contents** window.
2. In the **Properties** window, click on the Symbol tab.
3. Click the ⊞ next to *Symbol*, if the *Symbol* section is not already expanded.
4. Click the ⊞ next to *Symbol Properties* to open the Symbol Properties section.
5. Click the selected symbol next to the *Symbol* property. In the list, click on the filled diamond symbol (*Symbol set: GSI Default Symbols, Number: 6*) from the symbol palette.
6. Click the ⊞ next to *Symbol Size* to open the *Symbol Size* section.
7. Highlight the value next to the *Symbol size* option and type 0.15 in.
8. Press ENTER on the keyboard. The symbols update with the new symbol size.
9. Click the ⊞ next to *Symbol Color*.
10. To change the symbol colors based on a worksheet value, click on the *None* next to the *Color column* option and select *Column C: Elevation*.
11. Verify that the Color method is set to Numeric via colormap.

12. Click the colormap next to the *Symbol colors* and select the desired colormap, such as *Terrain*.

If the post map is not visible, ensure that the post layer is on top of the contour layer in the **Contents** window. The order the layers are listed in a map object is the order the map layers are drawn in the plot window. To move the post layer in the **Contents** window, left-click and drag the post layer above the other layers in the map object. Alternatively, select the post layer and click the **Layout | Arrange | Bring to Front | Bring Forward** command, or right-click the post layer and select **Order Objects | Move Forward**.



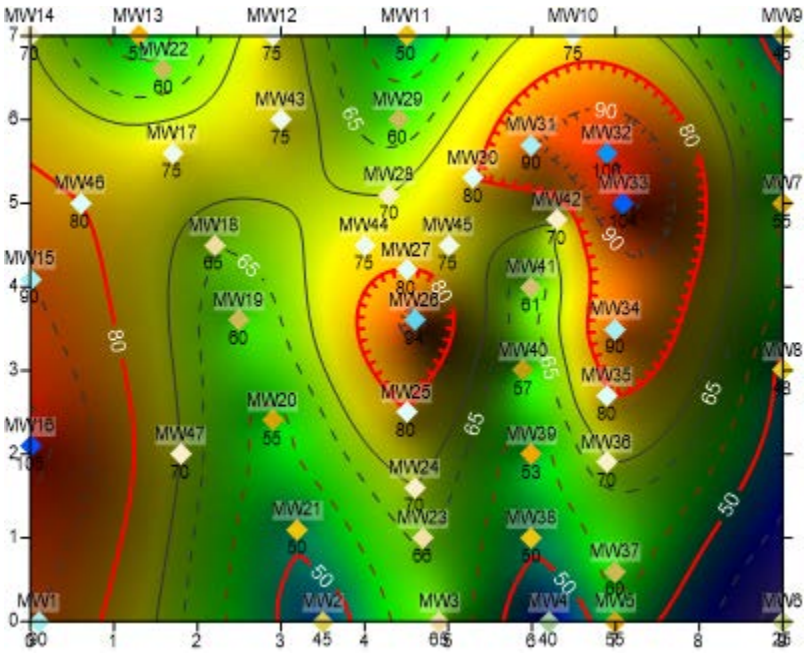
The updated post map is displayed overlaid on the contour layer and color relief layer.

Adding Labels to the Post Layer - Tutorial

You can add labels to the data points on post maps and classed post maps. Multiple labels can be added to display all of the information desired in the map. We will add labels showing the elevation and names for the data points:

1. Click on the *Post-Tutorial.dat* layer in the Contents window.
2. In the **Properties** window, click on the **Labels** tab.
3. Click the ⊞ next to *Label Set 1*, if the section is not already open.
4. Next to *Worksheet column*, click the word *None*. A list displaying all of the columns in *Tutorial.dat* are displayed. Select *Column C: Elevation* from the list.
5. For the *Position relative to symbol* option, click on the existing option and select *Below* from the list.
6. Click the *Add* button next to the *Add label set* option to add a second label to the post map.
7. Next to *Worksheet column*, click the word *None*. A list displaying all of the columns in *Tutorial.dat* are displayed. Select *Column D: Name* from the list.
8. For the *Position relative to symbol* option, click on the existing option and select *Above* from the list.
9. Click the ⊞ next to *Font Properties* to open the *Font Properties* section.
10. Change the *Background opacity* to *33%*. This places a semi-transparent white box around the names.

The post map layer is automatically redrawn with labels on each of the data points.

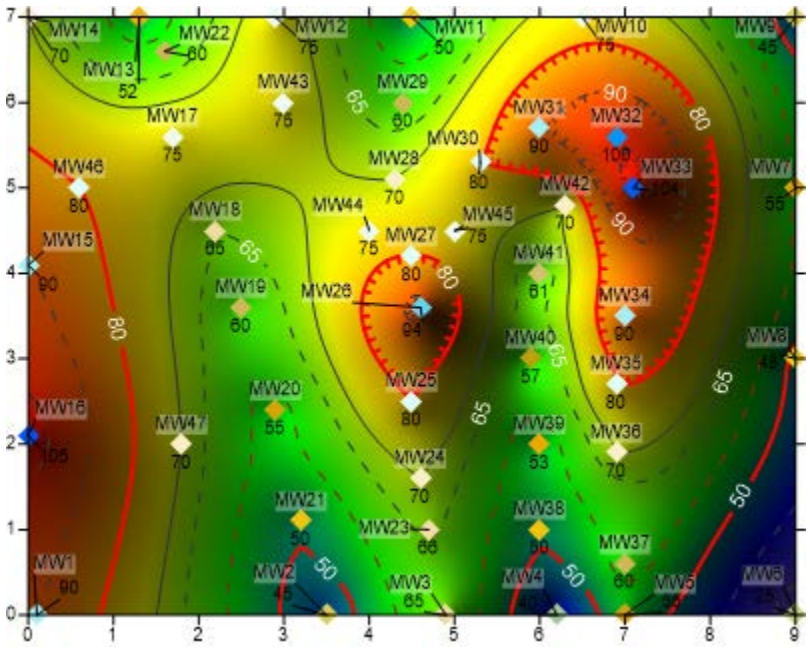


*Add labels to post maps in the **Properties** window on the **Labels** tab.*

Moving Individual Post Labels - Tutorial

You can move individual labels on post maps and classed post maps with the Map Tools | Edit Layer | Post Labels command. Alternatively, add labels, and then right-click the post map and select **Edit Post Labels** to enter edit mode. A customizable line is automatically added from the data point label to the actual X, Y data point location.

1. Select the *Post-Tutorial.dat* layer in the **Contents** window.
2. Click the Map Tools | Edit Layer | Post Labels command or right-click on the selected map and select **Edit Post Labels**. The cursor will change to to indicate you are now in post label editing mode.
3. Left-click on a label, hold the left mouse button down, and drag the label to a new location. With the left mouse button held down, the arrow keyboard keys can be used to nudge the label location. Release the left mouse button to place the label in the new location. A leader line will be added from the point location to the new label location by default. The leader line visibility and line properties are controlled on the **Labels** page in the **Properties** window when the post layer is selected.
4. Press the ESC key to exit the post label editing mode.



*Customize the post map labels with the **Edit Post Labels** command.*

Lesson 4 - Modifying an Axis

Every map is created with four map axes: the bottom, right, top, and left axes. 3D maps also have an additional Z axis. You can control the display of each axis independently of the other axes on the map. Additional left, right, top, bottom, or Z axes can be added to a map with the **Map Tools | Add to Map | Axis** commands. In this example, we will change the axis label spacing and add an axis title:

1. Move the cursor over one of the axis tick labels on the bottom X axis and left-click the mouse. In the status bar at the bottom of the plot window, the words "Map: Bottom Axis" are displayed. The *Bottom Axis* object is selected in the **Contents** window. This indicates that you have selected the bottom axis of the map. Additionally, blue circle handles appear at each end of the axis, and green square handles appear surrounding the entire map. This indicates that the axis is a "sub-object" of the entire map.
2. The bottom axis properties are displayed in the Properties window. Click on the General tab.
3. Click the ⊞ next to *Title* to open the *Title* section if it is not already open.
4. Click in the box next to *Title text*. Type *Bottom Axis* and press the ENTER key on the keyboard. This places a title on the selected axis. Alternatively, click the Σ button. Type the text in the Text Editor and click *OK*.
5. If you cannot see the axis title, click the View | Zoom | Selected command. The map automatically increases its size to fill the plot window.

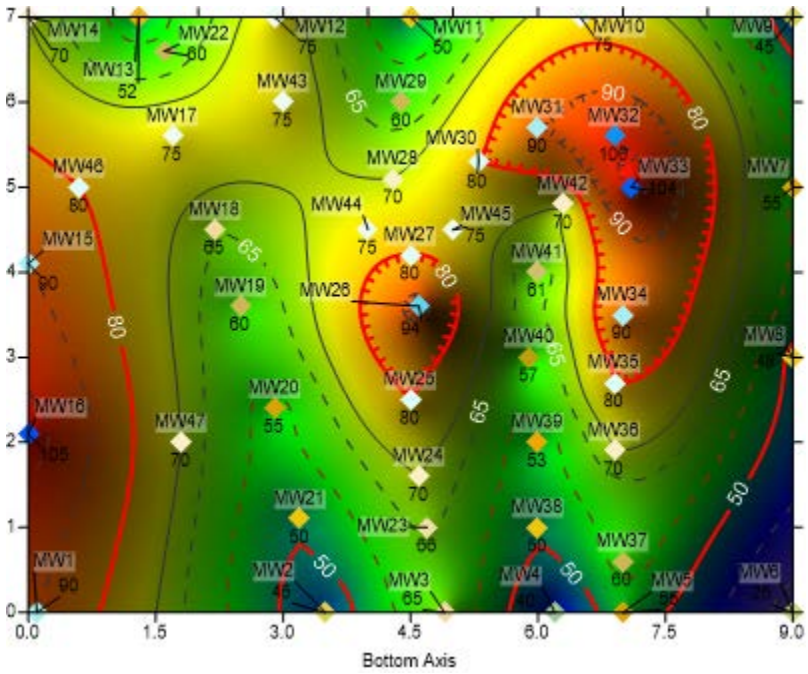
Changing the Tick Label Properties - Tutorial

All properties of the axis can be edited, including the tick label format and frequency. We will change both format and frequency in this example:

1. In the **Properties** window, click on the Scaling tab to display the axis scaling options.
2. In the *Major interval* field, highlight the value 1 and type the value 1.5.
3. Press ENTER on the keyboard to place 1.5 X map units between tick marks. This spacing automatically updates on the map axis.
4. Click on the **Labels** tab in the **Properties** window.
5. Click the ⊞ next to *Labels*, if it is not already open.
6. Click the ⊞ next to *Label Format* to open the Label Format section.

7. In the *Label Format* section, select *Fixed* for the *Type*.
8. Click in the box next to *Decimal digits.* Highlight the existing value and type the value 1.
9. Press ENTER on the keyboard. This indicates that only one digit follows the decimal point for the axis tick labels.

The map is updated immediately after every change, showing the axis tick spacing, labels, and the axis title.



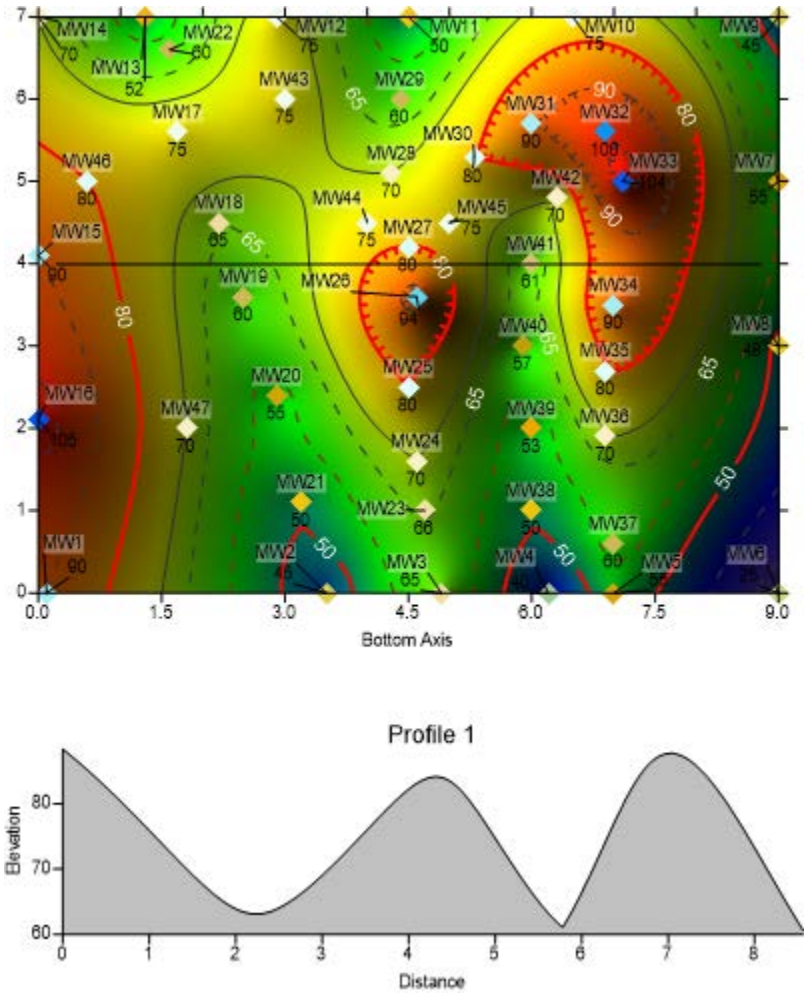
You can use the axis properties to change the tick mark and axis title properties.

Lesson 5 - Creating a Profile

The ability to slice a grid file in **Surfer** to create a file of data points along a specified line of section is a very powerful tool. The sliced data can be visually displayed as a profile in **Surfer**, or multiple profiles can be combined to display a cross section. However, the Map Tools | Add to Map | Profile command provides an excellent quick method to producing a profile from a grid-based map. The profile trace will be drawn directly on the map:

1. Click once on the *Contours-Tutorial.grd* contour layer to select it.
2. Click the **Map Tools | Add to Map | Profile** command. The cursor changes to a ⌐|¬ to indicate that you are in the drawing mode.
3. Click inside the contour map near the (0,4) and (9,4) coordinate locations. The exact coordinates of the cursor are displayed in the status bar for reference.
4. After the second point has been clicked, a line connects the points. Press ENTER on the keyboard to end drawing mode.
5. Click the **View | Zoom | Fit to Window** command to see the entire map and profile.

The *Base(vector)-Profile 1* layer is automatically added to the map and the profile graph is automatically created. The properties can be edited by clicking on the *Profile 1* object in the **Contents** window and adjusting the properties in the **Properties** window.



The profile trace is displayed on the map. The profile is displayed in a graph below the contour map.

Lesson 6 - Saving a Map

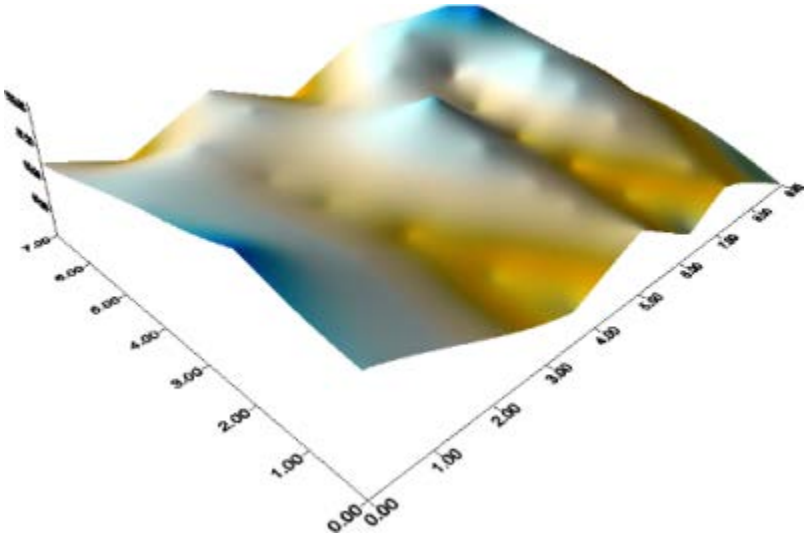
When you have completed the map in the plot window, you can save the map to a **Surfer** .SRF file. **Surfer** .SRF files contain all the information necessary to reproduce the project. When you save a map as a .SRF file, all the scaling, formatting, and map properties are preserved in the file. An asterisk (*) next to the file name in the title bar and tab indicates the file has been modified and the modifications have not yet been saved.

1. Click the File | Save command or click the button on the quick access toolbar. The **Save As** dialog is displayed because the map has not been previously saved. Set the *Save in* directory to any directory on your computer.
2. In the *File name* box, type *Tutorial*.
3. Make sure that the Save as type is set to Surfer 15 Document (*.srf).
4. Click *Save* and the file is saved to the current directory with a .SRF extension. The saved map remains open and the title bar changes to reflect the name change. There is no longer an asterisk next to the file name.

If desired, the *Save as type* can be set to *Surfer 11 Document (*.srf)*, *Surfer 12 Document (*.srf)*, *Surfer 13 Document (*.srf)*, or *Surfer 14 Document (*.srf)* if the file is to be shared with users using **Surfer 11**, **Surfer 12**, **Surfer 13**, or **Surfer 14**. After selecting the format, click *Yes* in the dialog. Any **Surfer 15** specific features are lost when saving to a previous **Surfer** version format.

Lesson 7 - Creating a 3D Surface Map

Surfaces are three-dimensional shaded renderings of a grid file. Surfaces provide an impressive visual interpretation of data. Surfaces can be layered with other surfaces, so that the surfaces will intersect with each other. Surfaces can also have layers of other map types, excluding 3D wireframes. Surfaces allow you to generate an elevation model of your area of interest and then add layers of data on the top of the surface. You can control the color, lighting, overlay blending, and wire mesh grid of a 3D surface.



The 3D surface map shows the grid with a 3D aspect and color filled areas.

For example, if you have location (X, Y) and temperature (Z) data for a region and you have the same location (X, Y) and corresponding elevation (Z) data for the area, you could create a grid file with the Z variable being elevation and a grid file with the Z variable being temperature. You could create a 3D surface of the elevation grid to represent topography, then add a contour map of the temperature variation. You could continue to add map layers, such as a classed post map layer with the temperature collection stations that have different symbols depending on the elevation.

We are going to use the same grid file you used to create the tutorial contour map. The 3D surface map will provide a new perspective to the contour map you have already created. Although we are going to create this map in a new plot window, the surface map could easily be added to the existing plot window.

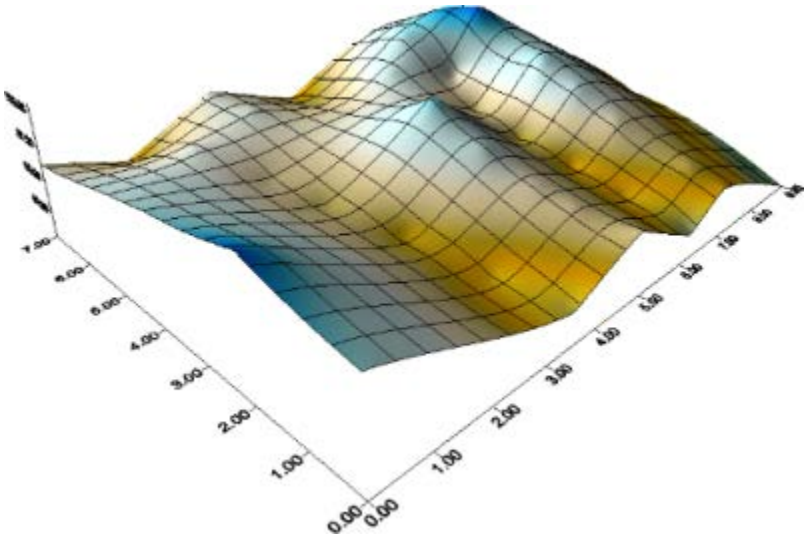
1. Click the **File | New | Plot** command or click the button on the quick access toolbar to open a plot document.
2. Click the Home | New Map | 3D Surface command.
3. In the **Open** dialog, select the grid file *Tutorial.grd* from the list of files. The *Tutorial.grd* file was created in Lesson 2 - Using the Map Wizard.
4. Click *Open* and the 3D surface is created using the default settings.

Adding a Mesh - Tutorial

Mesh lines can be applied to surfaces. 3D surface maps have more capability than 3D wireframe maps. 3D surfaces can be combined with more map types, and the surface map limits can be changed. Adding mesh lines to a 3D surface map simulates a 3D wireframe map. We will add a surface mesh to the map:

1. Click once on *3D Surface-Tutorial.grd* in the **Contents** window to select it. The 3D surface properties are displayed in the **Properties** window.
2. Click the Mesh tab.
3. Check the box next to the *Draw lines* option in both the *Lines of Constant X* and *Lines of Constant Y* sections.
4. Change the *Frequency* in both the *Lines of Constant X* section and *Lines of Constant Y* section to 5.

The mesh is automatically added to the selected 3D surface.



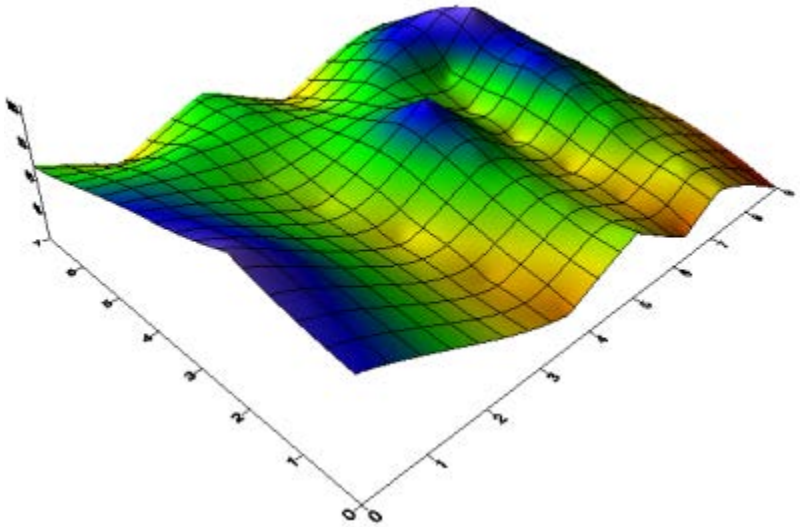
The mesh lines indicate lines of constant X and Y on the 3D surface.

Changing the 3D Surface Layer Colors - Tutorial

Changing color schemes on 3D surfaces is similar to changing colors on other map types such as color relief maps or contour maps. A colormap is used to load previously defined color schemes or to create your own color schemes. In this example, we will use a modified *Rainbow* colormap:

1. Click on the *3D Surface-Tutorial.grd* layer in the **Contents** window to select it.
2. In the **Properties** window, click on the General tab.
3. Click the ⊞ next to *Material Color* to open the section if it is not already open.
4. Click the ··· button to the right of the selected colormap for *Upper*. The **Colormap** dialog opens.
5. In the **Colormap** dialog, select the *Rainbow* colormap from the *Presets* list. The *Presets* list contains a variety of predefined color schemes.
6. The *Rainbow* preset has six nodes that range from purple to red. You can add, remove, apply opacity, customize the nodes, or accept the default selections. To reverse the color order, click the *Reverse* button.
7. Click *OK* in the **Colormap** dialog to update the surface map properties with your color changes.

You can continue to experiment with the colors by selecting other color spectrums from the list next to *Upper*. Or, click the ··· button to the right of the colormap and make changes in the **Colormap** dialog. You can experiment with selecting custom node locations and colors.



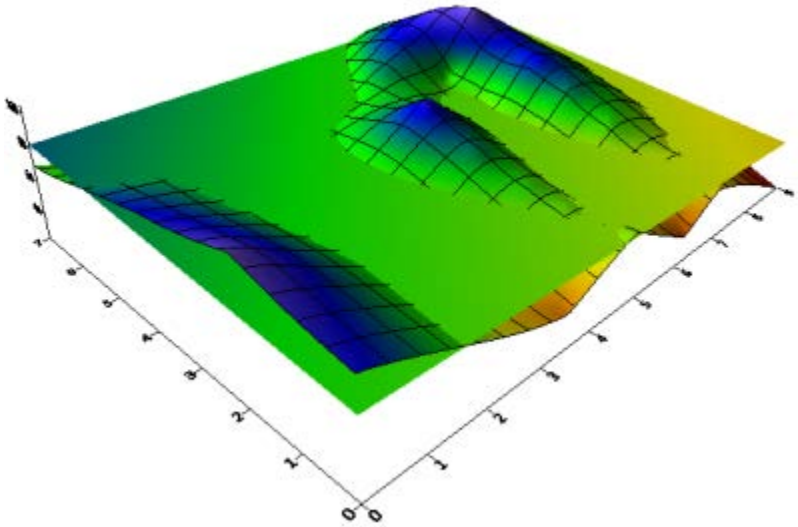
This is a 3D surface map with a mesh displayed at a frequency of five. The 3D surface map is using the reverse of the Rainbow color spectrum.

Adding a Surface Map Layer - Tutorial

You can add additional map layers to the 3D surface with the **Home | Add to Map | Layer** command. All map layers, except other 3D surfaces, are converted into a type of image known as a texture map. This texture map is then applied to the surface by stretching it and shrinking it as necessary. When these maps are added to the surface map, you have a choice on how to treat the texture map. You can use the colors from overlays only, from the surface only, or blend colors from the overlays and surface. For example, you could create a color filled contour map, add the contour map and surface, and then use the colors from the contour map only. A 3D wireframe layer cannot be added to a 3D surface map.

When multiple 3D surfaces of differing elevations are added, the surfaces can intersect and overlap each other. If a surface map is added to another surface map with the **Home | Add to Map | Layer | 3D Surface** command and the two maps are adjacent to each other in the X or Y direction, the surfaces are drawn side-by-side. In this example, we will add a planar layer to the surface you just created:

1. Click on the *3D Surface-Tutorial.grd* layer in the **Contents** window.
2. Click the **Home | Add to Map | Layer | 3D Surface** command, or right-click the surface map and select **Add to Map | 3D Surface**.
3. In the **Open Grid** dialog, select the planar grid, *TutorPl.grd* from **Surfer's** *Samples* directory. If you are not in the *Samples* folder, browse to it. By default, the *Samples* folder is located in C:\Program Files\Golden Software\Surfer 15.
4. Click *Open* and the new surface map layer is added using the default settings.
5. Click on the *3D Surface-TutorPl.grd* surface map layer in the **Contents** window.
6. In the **Properties** window, click on the **General** tab.
7. Click the ⊞ next to *Material Color* to open the *Material Color* section.
8. Click the ··· next to *Upper* to open the **Colormap** dialog.
9. Select *Rainbow* in the *Presets* list.
10. Click *Reverse* to match the colormap in the *3D Surface-Tutorial.grd* layer.
11. In the *Data to Color Mapping* section, type *25* in the *Minimum* field, and type *104.9* in the *Maximum* field. Now the surfaces use the same colormap mapped to similar data values.



You can overlay two or more 3D surfaces. Depending on each surface's XYZ ranges, the surfaces may overlap or intersect each other. This example shows the intersection of the Tutorial.grd and TutorPl.grd sample files.

If you wish to save your map, click the **File | Save** command. We will create a new plot in the next lesson.

Lesson 8 - Adding Transparency, Color Scales, and Titles

The opacity of a map, image, text, line, fill, symbol, or entire layer can be customized in **Surfer**. Opacity is the amount of light that is obscured by an object. An object can be made semi-transparent by adjusting the opacity value. Reducing the opacity of an object makes other objects visible through the less than 100% opaque object. An *Opacity* value of 0% means that the object is invisible. An *Opacity* value of 100% means that the object is fully opaque. Setting the opacity is useful when creating a semi-transparent map layer. For example, you may want to display a semi-transparent contour map layer over a satellite image base map layer so that both map layers can be seen. Being able to set the Opacity of entire layers is especially useful when you have multiple layers with filled objects and you need to see all of the layers.

Creating a Filled Contour Map

First, we will create a filled contour map:

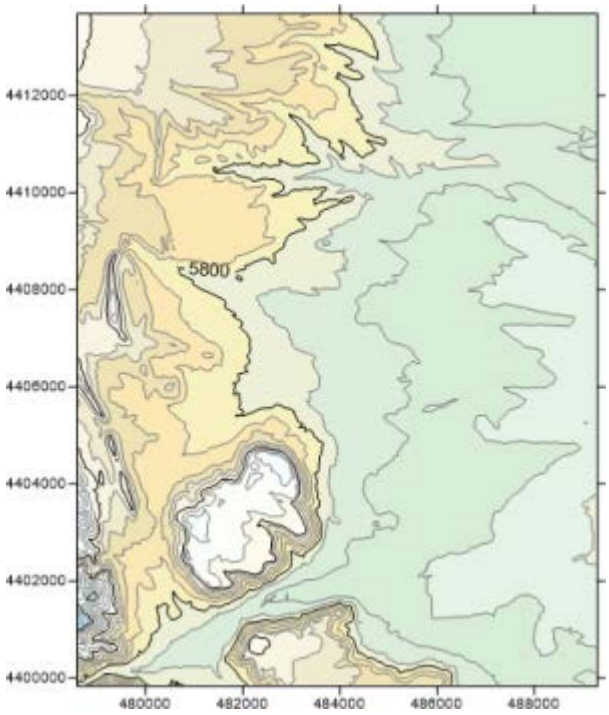
1. Click the **File | New | Plot** command or click the button on the quick access toolbar. A new empty plot window is displayed.
2. Click the **Home | New Map | Contour** command.
3. Select the grid file *Golden.grd* from the list of files in the **Open Grid** dialog. By default, the *Samples* folder is located in C:\Program Files\Golden Software\Surfer 15.
4. Click *Open.* The map is created using the default settings. Some settings are persistent while **Surfer** is open. If you have completed Lesson 3 in the same session, the map created in this step will have uphill hachures and white-text contour labels.
5. Click on the contour map layer to select it.
6. In the **Properties** window, click on the **Levels** tab.
7. Set the *Level method* to *Simple*, if it is not already *Simple*.
8. Click the next to *Filled Contours* to open the *Filled Contours* section, if it is not already open.

9. Check the box next to *Fill contours* to fill the contours.
10. Click the ⊞ next to *Labels* to open the *Labels* section, if it is not already open.
11. Click the ⊞ next to *Font properties* to open the *Font properties* section.
12. If the *Foreground color* is not *Black*, click the current color and select *Black* from the color palette.

Adding Transparency to Map Layers - Tutorial

You can adjust the *Opacity* value of a map layer or of individual contour fill, polygon fill, text, lines, or symbols when the appropriate object is selected. Adjusting the *Opacity* for an entire layer may be useful when you have multiple map layers and need to make one or more layers semi-transparent to best represent your data. For this example, we will adjust the opacity for the contour fill while keeping the contour lines and labels completely opaque.

1. Click on the *Contours-Golden.grd* layer in the **Contents** window to select it.
2. In the **Properties** window, click on the **Levels** tab.
3. Click the ... button next to *Fill colors*. The Colormap dialog opens.
4. Click the current selection in the *Presets* list and select *Terrain*.
5. Verify the *Apply opacity to ALL nodes* check box is checked. If it is not checked, click the check box.
6. Highlight the existing *100%* value next to the *Opacity* option and type *30*.
7. Click *OK* in the **Colormap** dialog. The *Terrain* colormap and 30% opacity setting is applied to the contour layer's *Fill colors*. Notice *Custom* is displayed in the *Fill colors* field.



The contour map is displayed with a partially transparent fill color.

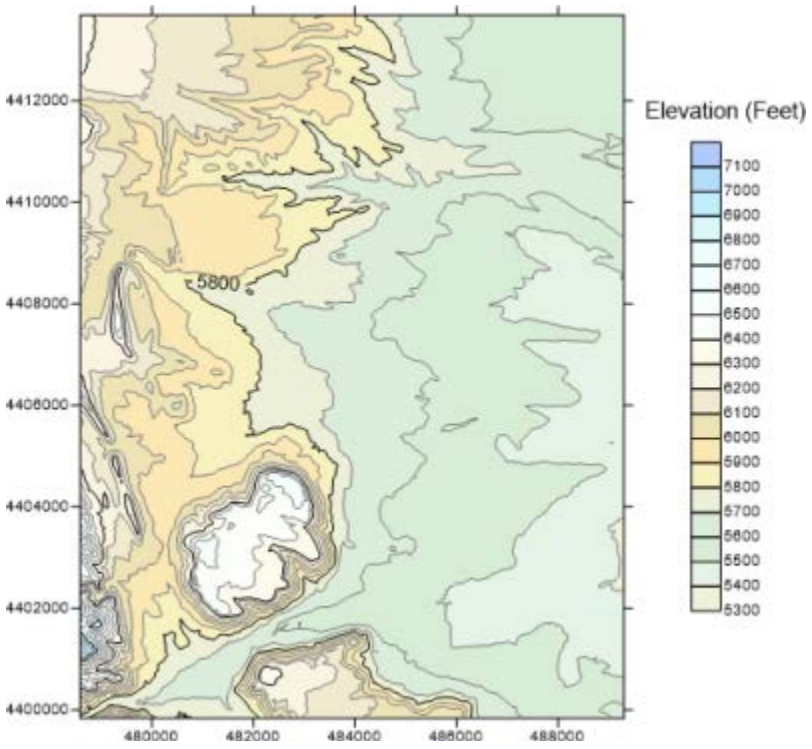
Adding and Editing a Color Scale - Tutorial

Color scales are legends that show the fill colors. Color scales are available for contour, 3D wireframe, 3D surface, color relief, and vector maps. The color scale displays the colors assigned to levels in a filled contour map or 3D wireframe, the colors used in a color relief map or 3D surface,

and the fill assigned to vector symbols. Let's add a color scale and color scale title to the contour map

1. Click on the *Contours-Golden.grd* contour layer to select it. The properties are displayed in the **Properties** window.
2. Click on the Levels tab in the **Properties** window.
3. Click the ⊞ next to *Filled Contours* to open the *Filled Contours* section, if it is not already open.
4. Select the box next to *Color scale*. A default color scale is created. A new *Color Scale* object is added to the **Contents** window.
5. Click on *Color Scale* in the **Contents** window to select it.
6. In the **Properties** window, click on the **General** tab to edit the color scale properties.
7. Click the ⊞ next to *Title* to open the *Title* section, if it is not already open.
8. Click in the empty box next to *Title text.*
9. In the Title text field, type Elevation (feet).
10. Press ENTER. The title is added with the default settings.
11. Change the title position by clicking the current selection next to *Position*. Select *Top* from the *Position* list.

Notice the color scale title moves to the top of the color scale, and the text orientation automatically changes to horizontal. The color scale has the same opacity as the contour layer when transparency is applied to the contour layer *Fill colors* colormap.



The map and color scale object are shown in this image.

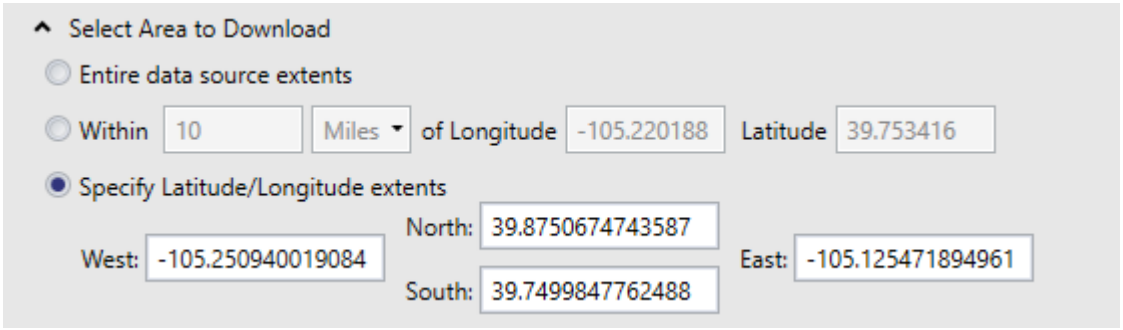
Downloading an Online Base Map Layer - Tutorial

A base map layer can be added below the existing semi-transparent contour layer to enhance the map's appearance. To add a base map layer from an online server,

1. Click anywhere on the map to select it.
2. Click the **Home | Add to Map | Layer | Base from Server** command to download an image base map from an online server. The Download Online Maps dialog is displayed.

Surfer can download base layers from WMS, OSM, and WFS servers. **Surfer** can also download grids from WCS servers.

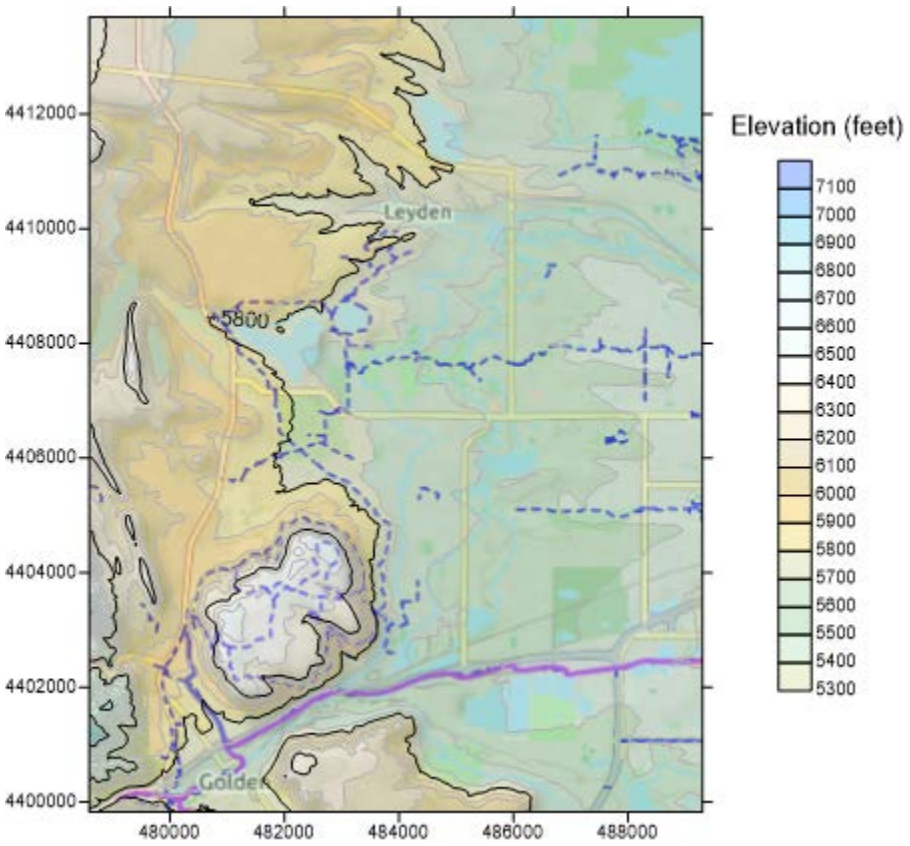
3. In the **Download Online Maps** dialog, click the ▷ next to *OpenStreetMaps Imagery*.
4. Click the *OpenCycleMap* server in the *OpenStreetMaps Imagery* category. A preview is displayed in the preview section.
5. Notice *Specify Latitude/Longitude Extents* is selected, and the values are set to the boundaries of the *Map*. When the map coordinate system is a geographic or projected system and the **Home | Add to Map | Layer | Base from Server** command is used, the area to download will be automatically set to the map extents.



The Specify Latitude/Longitude extents values are automatically filled with the extents of the selected map.

6. For the tutorial, we will use the default setting in the *Select Image Resolution to Download* section.
7. Click *OK* and the base layer downloads. The base layer is automatically placed behind the contour layer. If a **Surfer Warning** dialog appears prompting you to adjust the map limits, click *No*.

Increasing the image resolution will increase the download size when retrieving layers with the **Download Online Maps** dialog. With some servers such as the *OpenCycleMap* server, increasing the resolution will also return a different layer than the one displayed in the preview. Feel free to experiment with different resolutions by repeating steps 2 through 9 and selecting a higher or lower resolution in step 7. You will need to hide previous base layers to view the new one. In the **Contents** window, clear or select the check box next to the *Contours-Golden.grd* or *Base(raster)-OpenCycleMap* layers to toggle the visibility of the maps on and off.



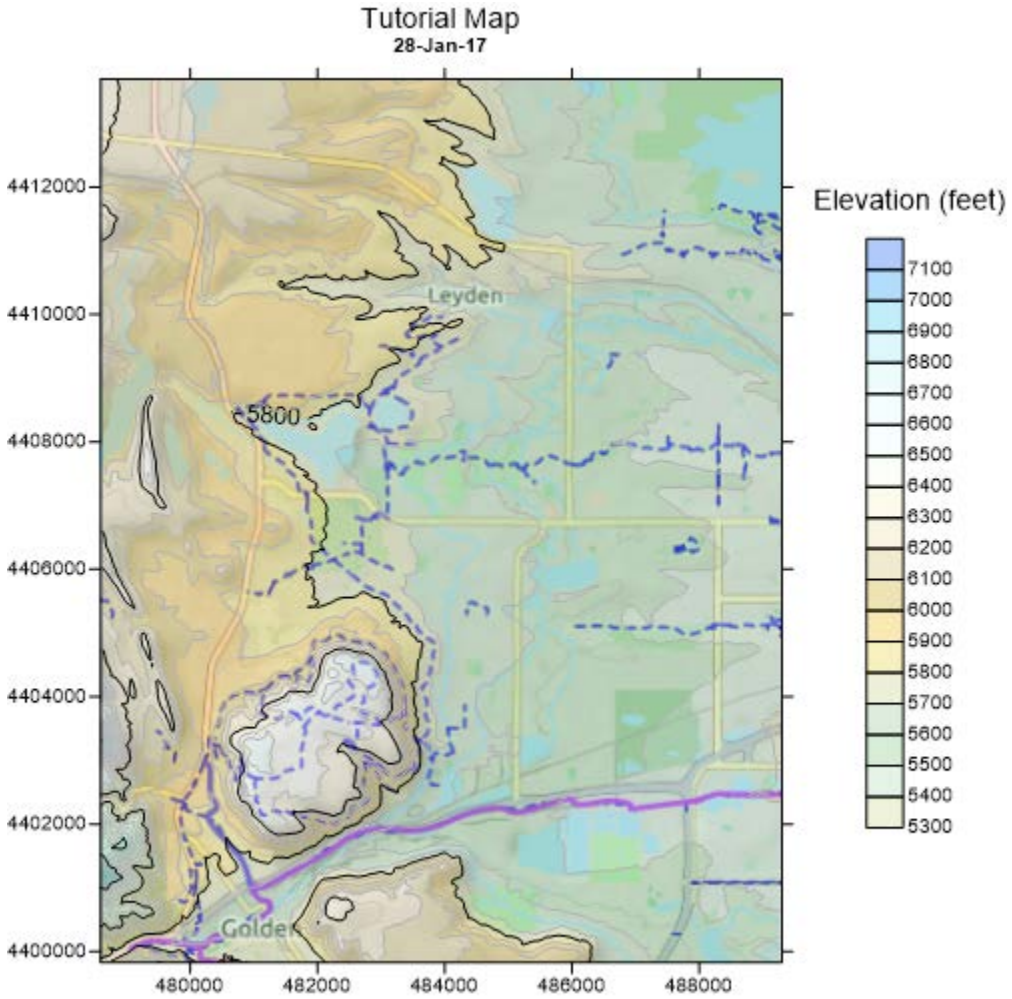
The base map is visible behind the partially transparent contour map.

Adding a Map Title - Tutorial

Adding a title to a map is a great way to stay organized and create publication quality maps.

1. Click once on the *Top Axis* in the **Contents** window to select it.
2. In the **Properties** window, click on the General tab.
3. Click the ⊞ next to *Title*, if the section is not already open.
4. In the box next to *Title text*, click the Σ button to open the Text Editor. The **Text Editor** provides more control over the text appearance than the **Properties** window.
5. Type *Tutorial Map* and press the ENTER key on the keyboard.
6. On the second line, we will use a dynamic predefined math text instruction to insert the current date. Click the button.
7. In the Date/Time Format Builder dialog, select the desired date/time format in the *Predefined date/time formats* list. For instance, select *dd-MMM-yy*.
8. Click *Insert* next to the selected date/time format in the *Predefined date/time formats* list. Notice the format in the *Date/Time format* field updates to the selected format.
9. Click *OK* in the **Date/Time Format Builder** dialog. Today's date is added to the **Text Editor**.
10. Select the date in the **Text Editor** by double-clicking the date or by clicking and dragging across the date.
11. Click the **B** button to make the highlighted text bold.
12. Select the *Tutorial Map* text.
13. Change the *Size (points)* to 14. The size is located immediately to the right of the font name.
14. Click the center justification button to center the text.
15. Click *OK* to close the **Text Editor**.

The map is automatically updated with the new map title. Save the project if you wish. We will open a new plot window in the next lesson.



This map contains a semi-transparent contour layer on top of a base layer. A color scale and title were added to the map.

Lesson 9 - Creating Maps from Different Coordinate Systems

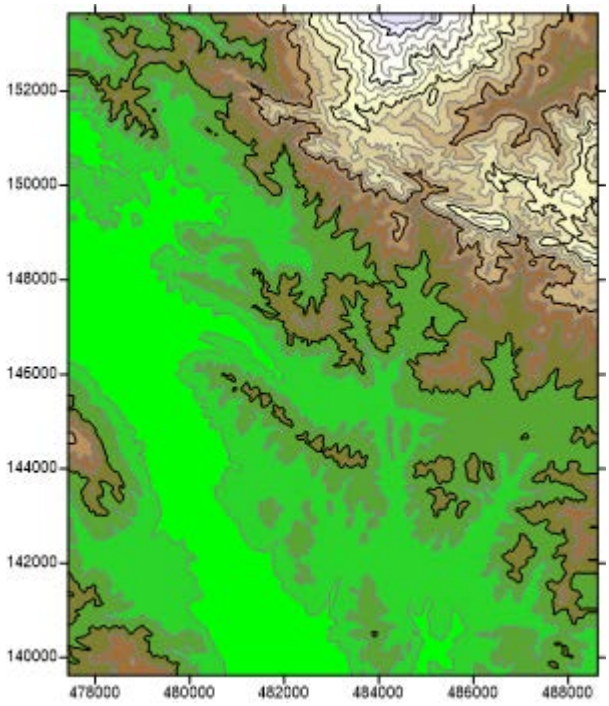
Map layers from different coordinate systems can be created in the same map frame. **Surfer** converts the source coordinate system for each map layer to the target coordinate system for the entire map. The axes display the target coordinate system. A coordinate system is method of defining how a file's 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 lat/long coordinate system, or referenced to a known projection and datum.

Creating the First Map - Tutorial

First, we will create a map with a defined coordinate system in **Surfer**:

1. Click the **File | New | Plot** command to open a new plot window.
2. Click **Home | New Map | Contour** to create the map with a contour layer.

3. In the **Open Grid** dialog, click on the *Diablo.grd* file from **Surfer's** *Samples* folder. If you are not in the *Samples* folder, browse to it. By default, the *Samples* folder is located in C:\Program Files\Golden Software\Surfer 15.
4. Click *Open*. The contour map is created.
5. Click on *Contours-Diablo.grd* in the **Contents** window to select the contour layer.
6. Click the Levels tab in the **Properties** window. Set the interval, fill, and contour lines properties for the contour map using the methods described in Lesson 3 and Lesson 8.
7. In the **Properties** window, click on the Coordinate System tab. Note that the contour map layer was imported with a coordinate system already specified. This map layer is in the *State Plane 1927 - California III (Meters)* coordinate system.



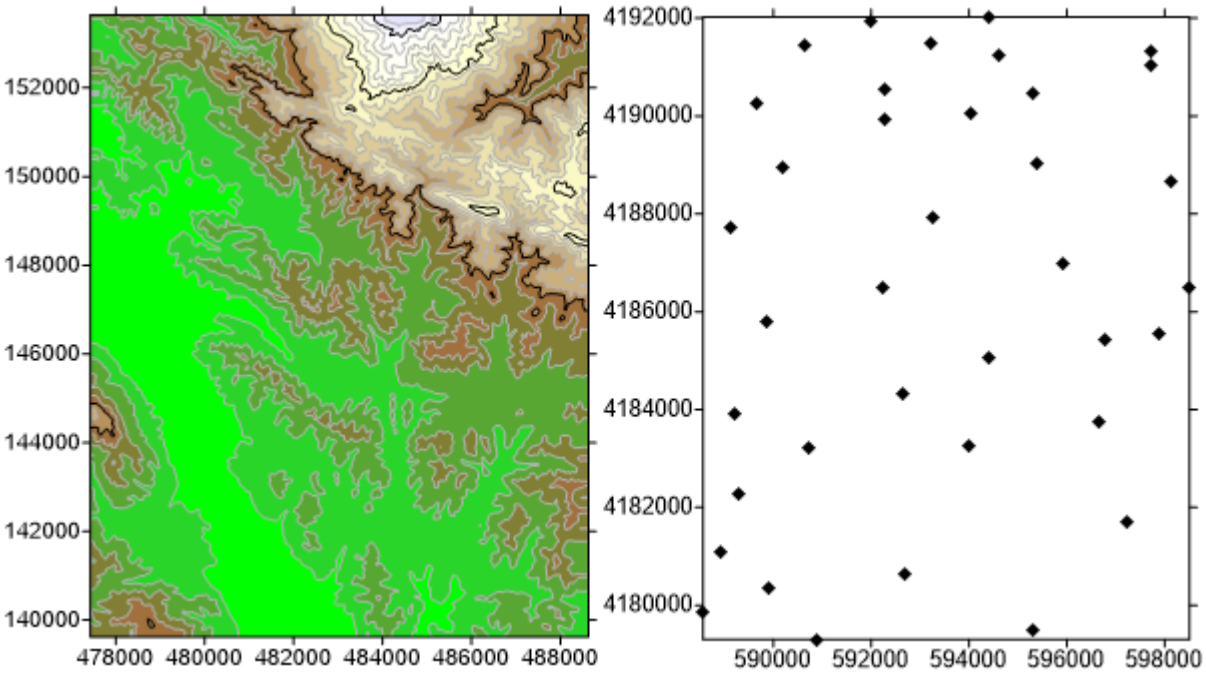
The first map layer is created with a predefined coordinate system.

Adding a Post Map Layer - Tutorial

Maps can be created without predefined coordinate systems and assigned the correct coordinate system in the layer properties.

1. Create a new post map with the **Home | New Map | Post** command.
2. In the **Open Data** dialog, select the *Diablo Example.dat* file in the **Surfer** *Samples* directory. If you are not in the *Samples* folder, browse to it. By default, the *Samples* folder is located in C:\Program Files\Golden Software\Surfer 15.
3. Click *Open*.
4. Click on the *Map* in the **Contents** window that contains the post map to select it.
5. Click and drag the map in the plot window to move the post map. Move the post map until the two maps are side by side. Note that the axes on the two maps have very different coordinates.
6. Click on *Post-Diablo Example.dat* in the **Contents** window to select the post layer.
7. In the **Properties** window, click on the **Coordinate System** tab. Note that the post map does not have a predefined coordinate system.
8. Click the *Set* button to define the coordinate system for the post map. The Assign Coordinate System dialog is displayed. Since we know this coordinate system, we can set it.

9. We can use the search bar to reduce the number of projections listed in the **Assign Coordinate System** dialog, since we know the map coordinate system. In the *Search for text or EPSG code* box, type *UTM Zone 10N*.
10. Press ENTER or click the button.
11. In the **Assign Coordinate System** dialog, click the ⊞ next to *Predefined* to open the *Predefined* section.
12. Click the ⊞ next to *Projected Systems* to open the *Projected Systems* section.
13. Click the ⊞ next to *UTM* to open the *UTM* section.
14. Click the ⊞ next to *North America* to open the *North America* section.
15. Click on the *North America NAD27 UTM Zone 10N* coordinate system to select it.
16. Click *OK*. On the **Coordinate System** tab, the post layer shows the defined coordinate system next to *Name*.



The two maps are displayed side by side with very different coordinates displayed on the axes.

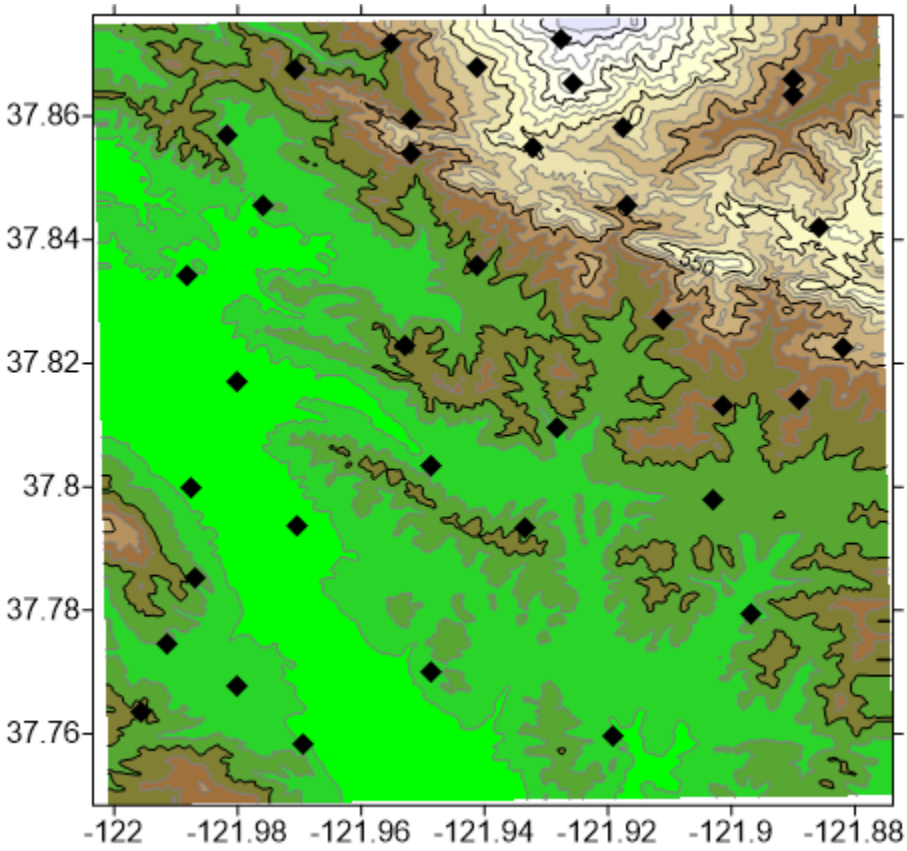
17. In the **Contents** window, click and drag the *Post-Diablo Example.dat* layer into the *Map* just above the *Contours-Diablo.grd* map layer. The two map layers are now overlaid. You can see the posted symbols are located on the contour lines, despite the different coordinate systems.

Setting the Target Coordinate System for the Map - Tutorial

The target coordinate system is the system displayed in the plot and on the map axes. Once all map layers are defined, the target coordinate system can be changed to any desired coordinate system.

1. Click on the *Map* object in the **Contents** window.
2. In the **Properties** window, click on the Coordinate System tab.
3. Click the *Change* button.
4. In the **Assign Coordinate System** dialog, click the ⊞ next to *Predefined* to open the *Predefined* section.
5. Click the ⊞ next to *Geographic (lat/lon)* to open the *Geographic (lat/lon)* section.
6. Click on *World Geodetic System 1984* near the bottom of the list to select it.
7. Click *OK*.

The map now has a different coordinate system than either the contour or post map layers on the **Coordinate System** page. Notice that the axes are now showing latitude and longitude values as well. In the above section, we did not use the search function 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.



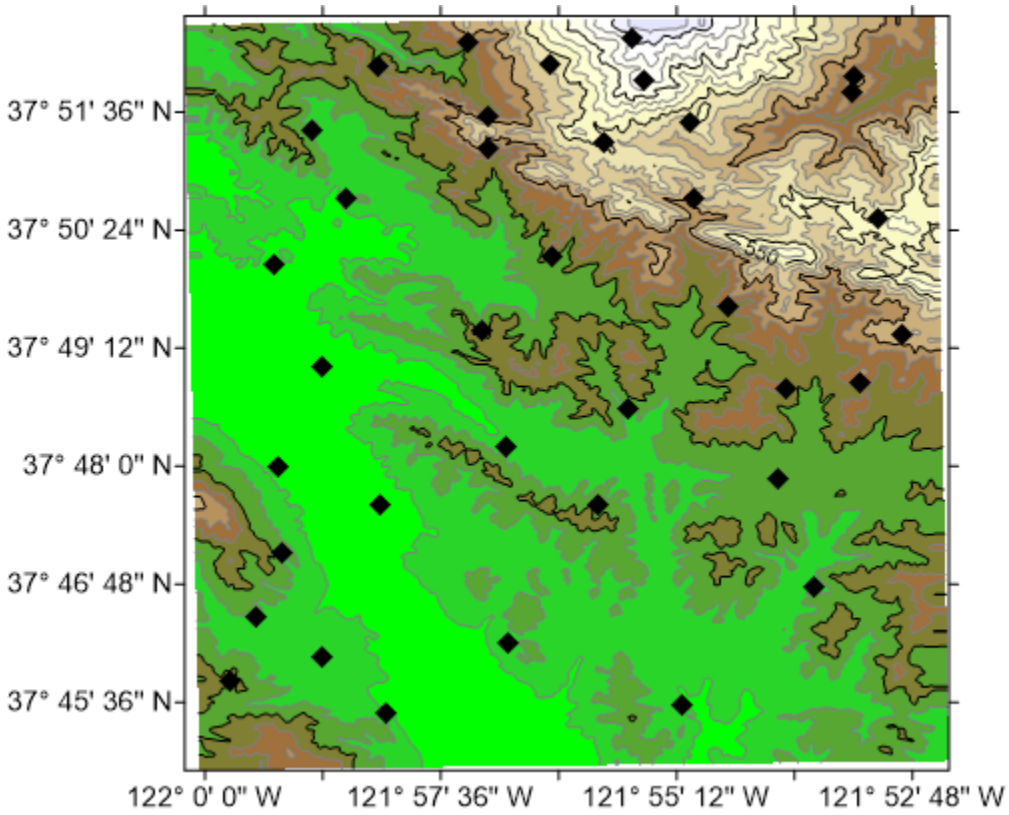
The map axes now display latitude and longitude coordinates.

Changing the Axis Label Format - Tutorial

The axis labels can be displayed in a variety of number formats. We will change the axis labels to Degrees, Minutes, Seconds format.

1. Click on the *Left Axis* object in the **Contents** window.
2. In the **Properties** window, click on the **General** tab to view the General page.
3. Click the ⊞ next to *Labels* if the *Labels* section is not already open.
4. Click the ⊞ next to *Label Format* to view the *Label Format* properties.
5. Click the current selection next to *Type* and select *DMS (Lat/long)* from the list.
6. Click on the *Bottom Axis* object in the **Contents** window.
7. Repeat steps 2 through 5 for the *Bottom Axis*.

The axis labels are now in Degrees, Minutes, Seconds format. Many additional edits can be made to the map. You can continue to experiment with the various coordinate systems or editing any portion of the map layers.



The final map contains two overlaid layers, each with different source coordinate systems. The axis labels are in Degrees, Minutes, Seconds format.

Tutorial Complete

Congratulations! You have completed the **Surfer** tutorial. The remaining tutorial lessons are optional advanced lessons. It is recommended that you complete the optional lessons, because these lessons provide additional information about how **Surfer** works. If you have questions, try looking for answers in the online help, quick start guide, online knowledge base, and interactive forum. If you find you still have questions, do not hesitate to contact Golden Software's technical support team.

Optional Advanced Lessons

By completing lessons 1 through 9 of the tutorial, you now have a basic understanding of **Surfer** and how to create and customize basic maps.

The remaining tutorial lessons are optional advanced lessons.

Lesson 10 - Customizing the User Interface shows you how to modify the quick access toolbar, ribbon, and keyboard shortcuts to improve your efficiency in **Surfer**.

Lesson 11 - Overlaying Map Layers shows you the three methods to overlay map layers.

Lesson 12 - Assign NoData Values to a Grid File shows you how to create a blanking file and use the **Grids | Edit | Assign NoData** command to create a grid file with an irregular boundary.

Lesson 10 - Customizing the User Interface

The ribbon interface is efficient and easy to use by default. However as you develop workflows and processes, you may find you can make **Surfer** even easier to use by customizing the quick access toolbar, ribbon, or keyboard commands.

Customize the Quick Access Toolbar

If you use a command frequently, you may want to add the command button to the quick access toolbar. This can easily be accomplished in **Surfer**. In this example, we'll add a separator and three commands.

1. Click the **File | Customize Ribbon** command to open the Customize dialog, or right-click the ribbon and select **Customize Quick Access Toolbar**.
2. Click the **Quick Access Toolbar** tab on the left side of the **Customize** dialog.
3. Click the current selection in the *Choose commands from* field, and select *Features*.
4. Click the <*Separator*> at the top of the *Commands* list on the left side of the dialog.
5. Click *Select* at the bottom of the quick access toolbar list on the right side of the dialog. Any additions from the *Commands* list will be placed below the item selected in the quick access toolbar list.
6. Click *Add* to add a separator to the end of the quick access toolbar.
7. Scroll down the *Commands* list and click *Polyline*.
8. Click *Add* to add the **Home | Insert | Polyline** command to the quick access toolbar.
9. Repeat steps 7 and 8 and add the **Polygon** and **Reshape** commands to the quick access toolbar.
10. Click *OK* in the **Customize** dialog to update the quick access toolbar with the new changes.

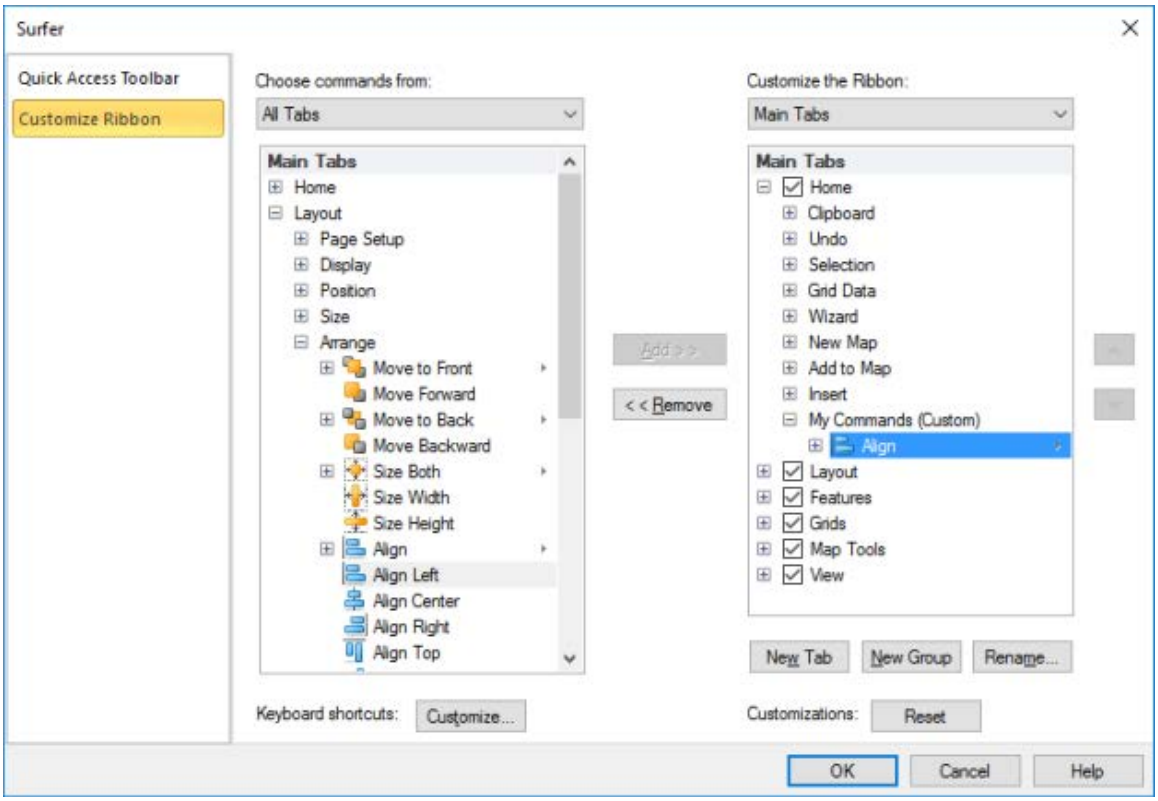
*Now the quick access toolbar includes the **Polyline**, **Polygon**, and **Reshape** commands after the default commands.*

Customize the Ribbon

The ribbon can be modified by adding new tabs, adding new command groups, adding commands to tabs or groups, and rearranging the tabs, groups, and commands.

1. Open the **Customize** dialog by clicking the **File | Customize Ribbon** command.
2. Click the **Ribbon** tab on the left side of the **Customize** dialog.
3. Select *Main Tabs* in the *Customize the Ribbon* field, if it is not already selected.
4. Click the ⊞ next to *Home* to expand the tab.
5. Click the *Insert* group at the bottom of the *Home* list. We will insert a new group below the *Insert* command group.
6. Click *New Group*. Commands must be added to a custom command group. Default command tabs and groups cannot be modified. However, default command groups can be removed and default command tabs can be hidden if desired.
7. Click *Rename* to rename the new command group. The **Rename** dialog is displayed.
8. Type a name for the command group, such as *My Commands*.
9. Press ENTER or click *OK* in the **Rename** dialog.
10. Select All Tabs in the Choose commands from field.
11. Click the ⊞ next to *Layout* to expand the *Layout* tab commands.
12. Click the ⊞ next to *Arrange* to expand the *Arrange* command group.
13. Click Align in the Choose commands from list.
14. Click *Add* to add the **Align** subgroup to the *My Commands (Custom)* command group.
15. Click *OK* in the **Customize** dialog to add your commands to the ribbon.

Notice on the **Home** tab, there is now a **Home | My Commands** group. You can use the **File | Customize Ribbon** command to tailor your **Surfer** experience to fit you.



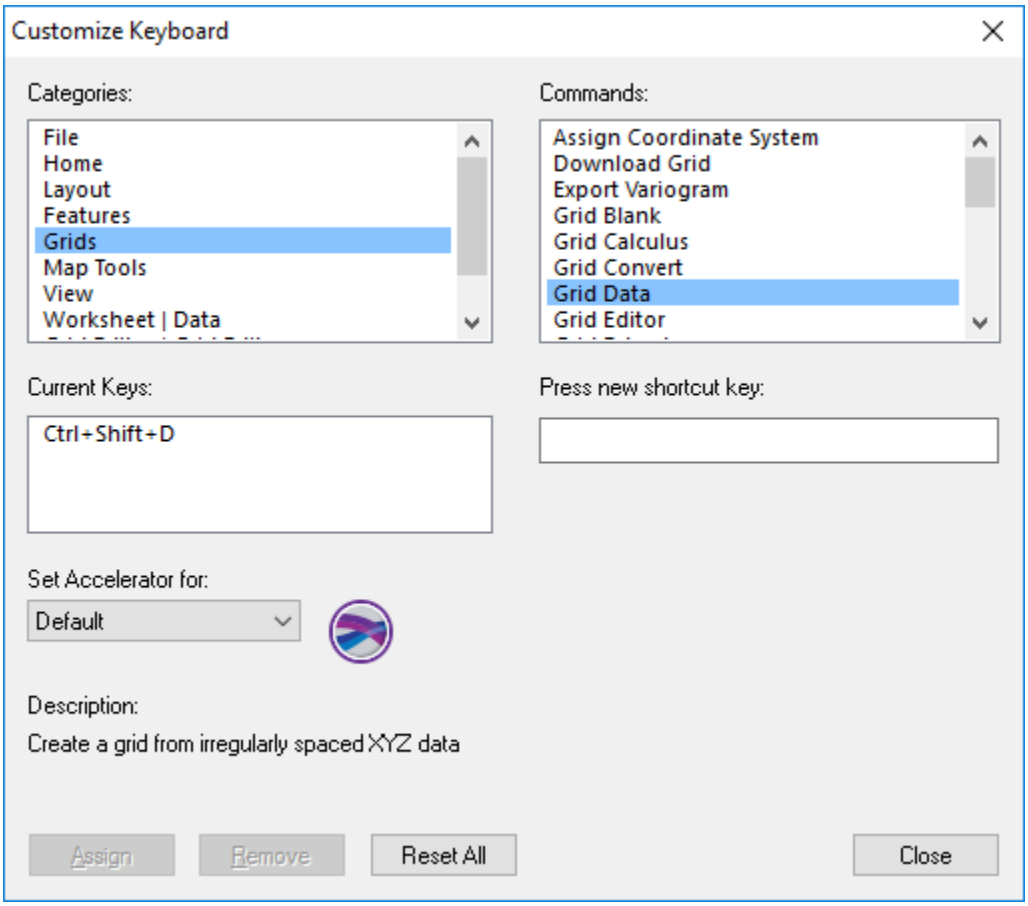
Customize the ribbon to display your own tabs and command groups.

Creating Keyboard Shortcuts

There are often times where you may use a command often enough to merit creating a custom keyboard shortcut. This can easily be accomplished in **Surfer**. In this example, we will add a keyboard shortcut to the Grid Data command.

1. Click the File | Customize Ribbon command to open the **Customize** dialog.
2. Click *Customize* next to *Keyboard shortcuts* near the bottom of the **Customize** dialog. The Customize Keyboard dialog is displayed.
3. Select *Grids* in the *Categories* list.
4. Select *Grid Data* in the *Commands* list. Notice nothing is displayed in the *Current Keys* field. This means there are no keyboard shortcuts assigned to the **Grid Data** command.
5. Click in the box next below *Press New Shortcut Key*.
6. Press CTRL + SHIFT + D on the keyboard. The shortcut will appear automatically in the *Press New Shortcut Key* box.
    - If no other command has this keyboard shortcut, *[Unassigned]* will be displayed below *Assigned to*. If the shortcut is not assigned to another command, click the *Assign* button. The shortcut is added to the *Current Keys* list.
    - If another command has the keyboard shortcut, the command is listed below *Assigned to*. If this is the case, the *Assign* button is grayed out. Select a different shortcut key for the command. Each shortcut key can be assigned to only one command.
7. Click the *Assign* button at the bottom of the **Customize Keyboard** dialog.
8. Click *Close* in the **Customize Keyboard** dialog.
9. Click *OK* in the **Customize** dialog.

10. In the plot window, press the CTRL + SHIFT + D command on the keyboard. The Grid | Data command is executed, and the **Open Data** dialog opens. Click *Cancel* in the **Open Data** dialog.



Custom keyboard shortcuts can be added to any command.

Removing the Customizations

You may wish to revert the quick access toolbar, ribbon, and keyboard shortcuts back to their default state.

1. Click the **File | Customize Ribbon** command.
2. Click *Reset* next to *Customizations* to clear the ribbon customizations. Click *Yes* in the **Surfer** warning dialog.
3. Click the **Quick Access Toolbar** tab in the left side of the **Customize** dialog.
4. Click *Reset* below the quick access toolbar command list. Click *OK* in the **Surfer** warning message.
5. Click *Customize* next to *Keyboard shortcuts* to open the **Customize Keyboard** message.
6. Click *Reset All* in the **Customize Keyboard** dialog. Click *Yes* in the **Surfer** warning message.
7. Click *Close* in the **Customize Keyboard** dialog.
8. Click *OK* in the **Customize** dialog.

Now all the tutorial customizations have been removed. This process can be used to remove customizations in the future should you wish to revert to the default state.

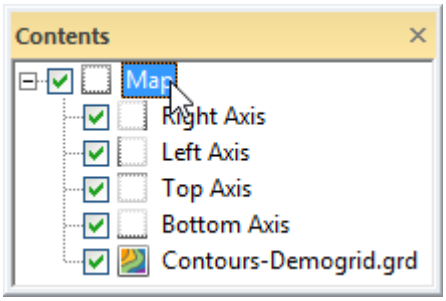
Lesson 11 - Overlaying Map Layers

Surfer 15 has three methods of overlaying map layers onto a single map object. You can drag a map layer from one map object to another map object in the **Contents** window, you can select a map and click the **Home | Add to Map | Layer** command to add a map layer, or you can select multiple map objects and use the **Map Tools | Map Tools | Overlay Maps** command.

This tutorial will cover all three methods that are available to overlay map layers in **Surfer**. This tutorial will also cover combining maps from different **Surfer .SRF** files.

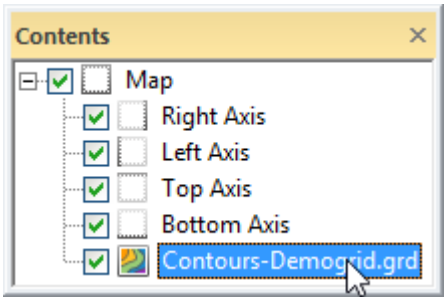
Before we start, it is important to understand the difference between a map object and a map layer. The **Contents** window is the easiest place to see the difference between a map object and a map layer.

A map object is listed in the **Contents** window as *Map*. A map object consists of axes and an optional map layer or map layers. Click on the *Map* object to open the map properties in the Properties window, where the View, Scale, Limits, Frame, Coordinate System, and Info are controlled.



*Click the Map object to display properties for the entire map in the **Properties** window.*

A map layer is listed in the **Contents** window as the map type name (i.e. *Contours*). A single map layer or multiple map layers can be part of a map object. Click on the map layer (i.e. *Contours*) to open the properties for the selected map type (i.e. contour map properties) in the **Properties** window. The specific properties related to the map type are controlled separately from the entire map properties.



*Click Contours to open the properties for the contour map in the **Properties** window.*

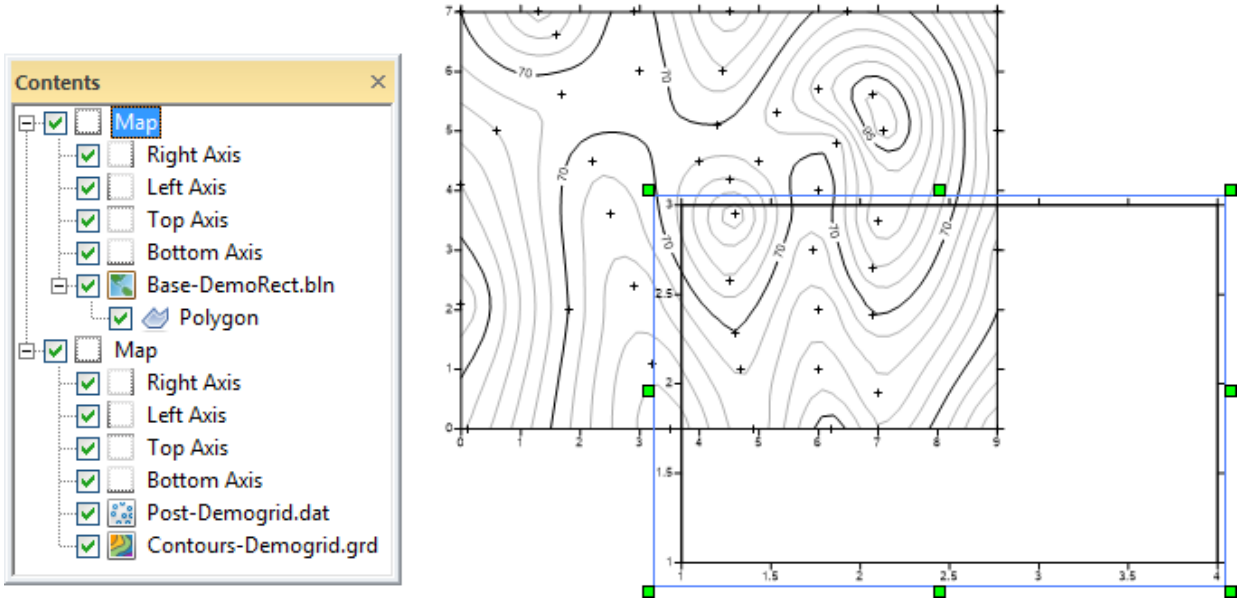
Method 1: Adding a Map Layer to an Existing Map Frame

1. Create a new map with a **Home | New Map** command. Click **Home | New Map | Contour** to create a contour map.
2. Select a grid file in the Open Grid dialog. Select the *Demogrid.grd* sample file and click *Open*.
3. Select the map and click the **Home | Add to Map | Layer** command to add a map layer. Select the map layer type to add to your existing map. Select the contour map and use the **Home | Add to Map | Post** command to add a post map layer to the contour map.
4. Select a data file in the Open Data dialog. Select the *Demogrid.dat* sample file and click *Open*.
5. 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: Combining Two Existing Maps in the Contents Window

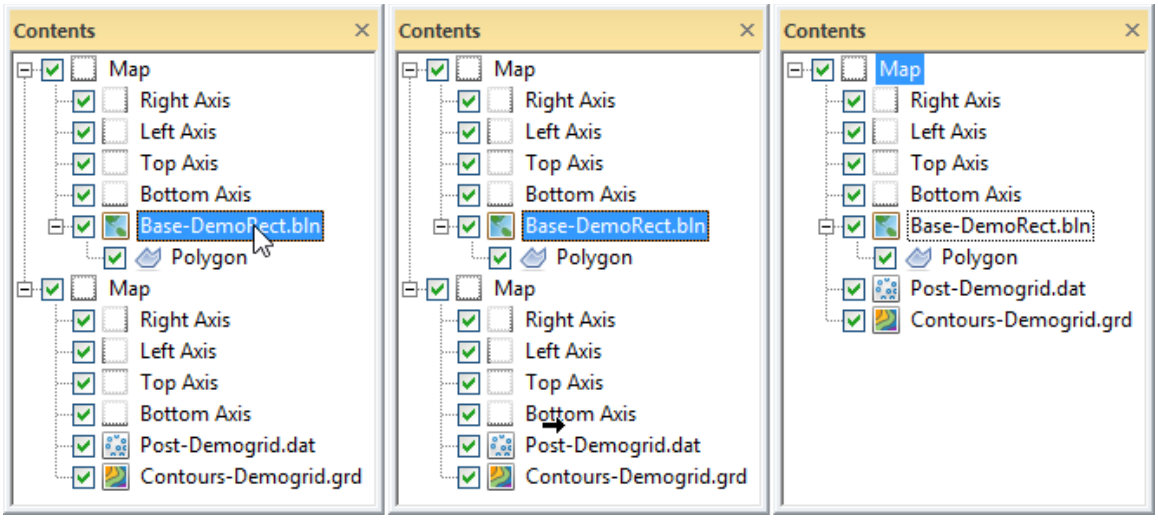
If two maps already exist, you can move (or overlay) a map layer from one map frame into the other map frame by dragging and dropping in the Contents window.

1. Create a new base map with a **Home | New Map** command. Click the **Home | New Map | Base** command.
2. Select the *DemoRect.bln* sample file in the Import dialog and click *Open*.
3. Note that each map has an independent set of axes.



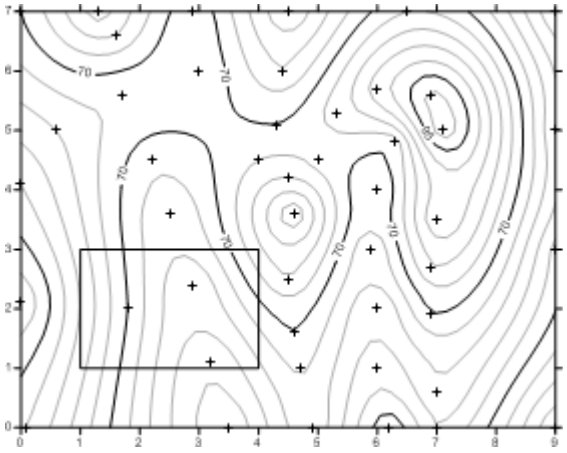
*The contour and post layers are in one map. The base layer is in a separate map. This is reflected in the **Contents** window and the plot window.*

4. Select the base map layer in the **Contents** window and drag it to the map object that includes the contour and post layers. 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.



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

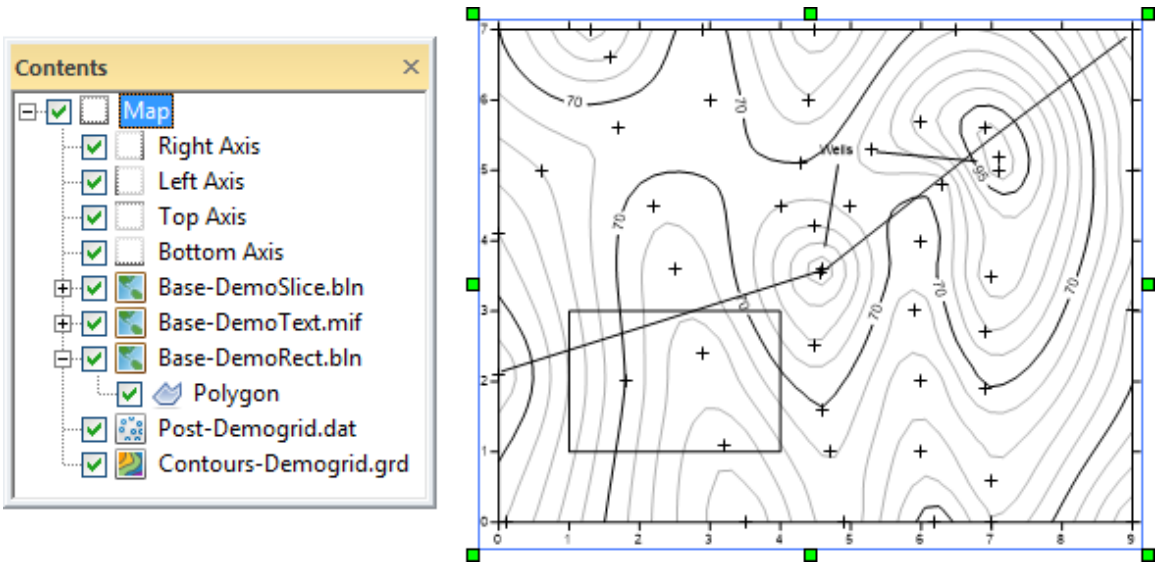
5. If an empty map frame exists, select the empty map frame and press DELETE on the keyboard to remove the empty map frame. The end result is a single map object with three map layers: a post map layer, contour map layer, and base map layer. Additional map layers can be added with the **Home | Add to Map | Layer** commands.



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

Method 3: Overlaying Two Existing Maps

1. Create a new base map with a **Home | New Map** command. Click the **Home | New Map | Base** command.
2. Select the *DemoSlice.bln* sample file in the Import dialog and click *Open*.
3. Add another base layer to the map by clicking the **Home | Add to Map | Layer | Base** command.
4. Select the *DemoText.mif* sample file in the Import dialog and click *Open*.
5. Note that each map has an independent set of axes. Both maps have multiple layers.
6. Click **Home | Selection | Select All** to select both map objects.
7. 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.



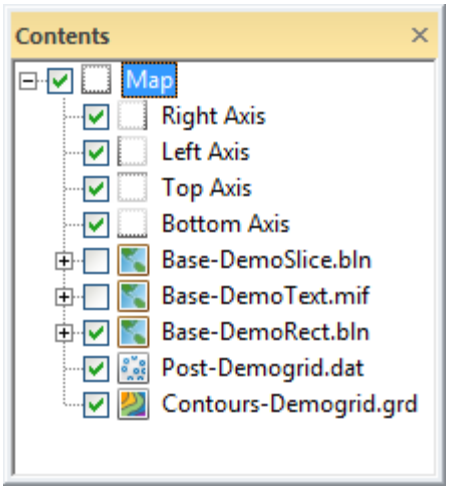
Now all five layers are in one Map object.

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

Lesson 12 - Assign NoData Values to a Grid File

Surfer creates grid files that are always rectangular or square. When you need to have a grid file where the contour lines do not cover a rectangular or square area, NoData values will need to be assigned to the grid. The **Grids | Edit | Assign NoData** command combines vector objects, e.g. polygons, with a rectangular grid file. The result is a new grid file where the contours stop at the boundary of the polygon or polygons.

In the previous lesson (Overlaying Map Layers) in methods 1-3, you created a map with a contour map layer, a post map layer, and a three base map layers. The first base map displays the rectangular area of interest, while the contour map displays a larger area than we need to display. Using the **Assign NoData** command, we will create a new grid file that has everything outside the base map rectangle blanked. Click on the plot window that contains the overlaid map layers from the previous lesson. Click on the ☑ next to the *Base-DemoText.mif* and *Base-DemoSlice.bln* layers. This will turn the display of these base maps off.



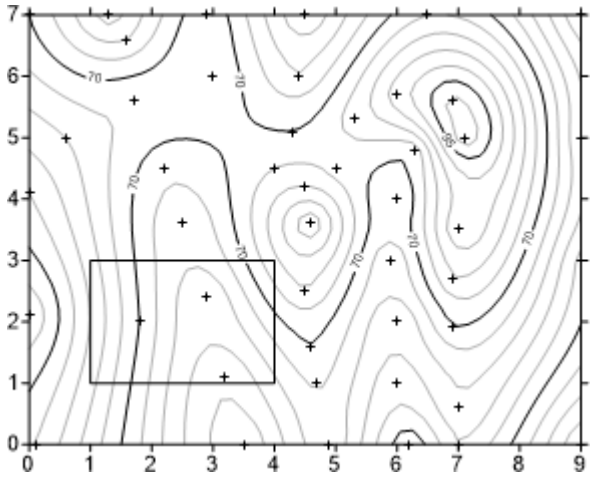
Uncheck the upper two base maps so only the contour, post, and original base map are displayed.

Now we will assign NoData values to the grid outside of the rectangle in the *Base-DemoRect.bln* layer. Note that it is not necessary to hide the other base layers. However, it may help you determine the extents of the base layer you use for assigning the NoData values.

1. First, open the *DemoRect.bln* file in the worksheet by clicking **File | Open**.
2. Select the *DemoRect.bln* sample file in the **Open** dialog and click *Open*. Notice the blanking flag in cell B1 is 0. This means the blanking flag is set to assign NoData values outside the polygon.
3. Click the plot window tab to return to the plot window.
4. In the plot window, click the Grids | Edit | Assign NoData command.
5. Click <*None*> in the *Input Grid* section and select the *Contours-Demogrid.grd* layer from the list.
6. Click <*None*> in the *NoData Polygon Boundary* section and select *Base-DemoRect.bln* from the list. Notice the *NoData Outside* option is automatically selected.
7. Click the in the *Output Grid* field. Select a location and file name such as your Documents folder. Type *Demogrid_NoData.grd* in the *File name* field.
8. Click *Save* in the **Save Grid As** dialog.
9. Click *[New Map]* next to *Add grid as layer to* and select *Map* from the list.
10. If the *New layer type* is not color relief, click the current selection next to *New layer type* and select *Color Relief* from the list.
11. Click *OK*.
12. Click the *Color Relief - Demogrid_NoData.grd* layer in the **Contents** window.
13. In the Properties window, click on the General tab.
14. In the *Hill Shading* section, clear the *Enable hill shading* check box.
15. In the *NoData* section, type 0% in the *Opacity* field or click and drag the slider all the way to the left.

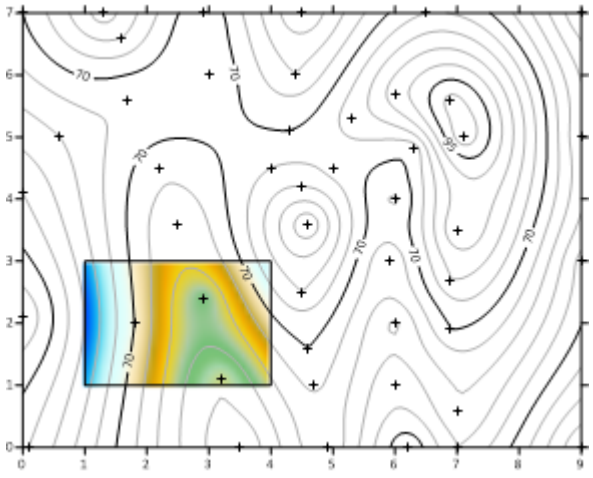
The new grid is used to display a color relief layer within the blanking file boundary.

Before Assign NoData:



The blanking file is displayed as a base map layer on the contour map.

After Assign NoData:



*This is a color relief layer created from the **Assign NoData** output grid file.*

Tutorial Complete

Congratulations! You have completed the advanced **Surfer** tutorial lessons.

Chapter 3 - Data Files and the Worksheet

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 (i.e. **Grid | Data**). If you want to view or alter the data in a data file, you can use the **File | Open** 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.

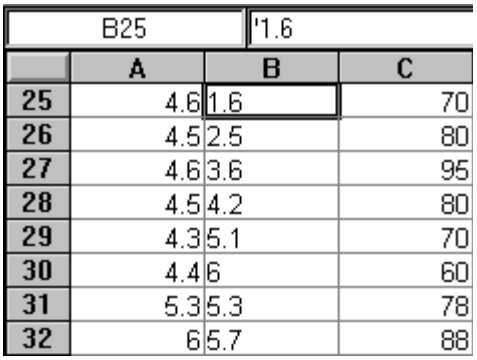
Data as Numbers or Text

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

- File | Import in the plot document
- Data | Edit | Merge in the worksheet document
- File | Open 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:

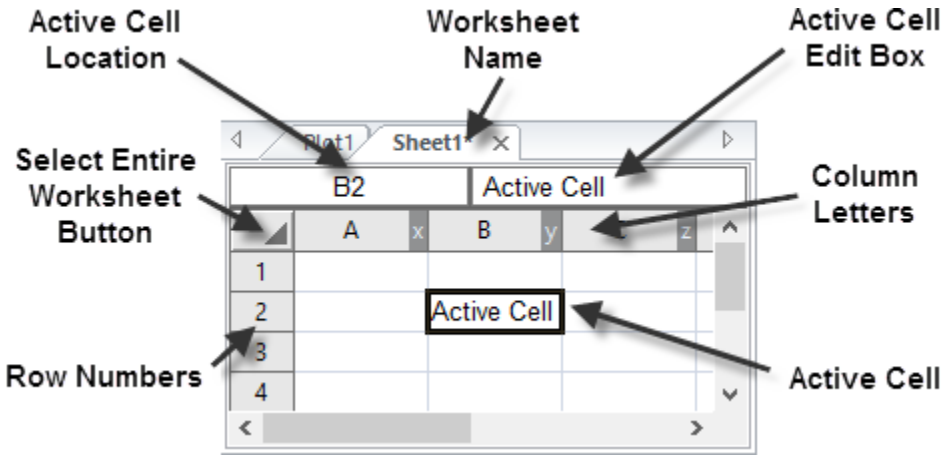
- Click the File | Open command in the plot document, grid node editor, or worksheet document and then select a data file.
- Click the button in the toolbar. In the Open dialog, select a data file.
- Select the File | Open or File | Import command in the worksheet and then select a data file.
- If there 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.

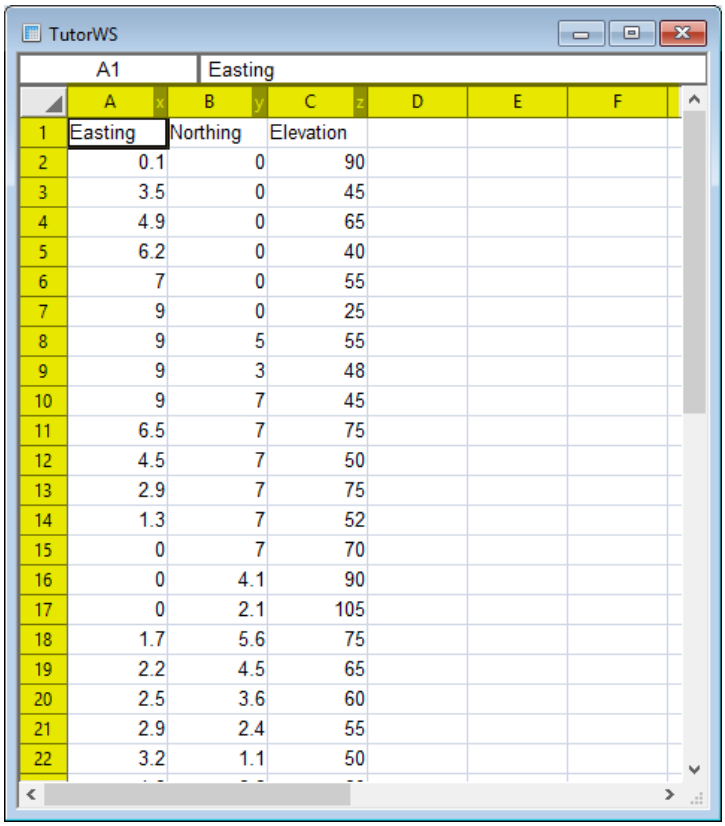


The components of a worksheet window shown above are described in the table below.

Component Name	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.



	A	B	C	D	E	F
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			
11	6.5	7	75			
12	4.5	7	50			
13	2.9	7	75			
14	1.3	7	52			
15	0	7	70			
16	0	4.1	90			
17	0	2.1	105			
18	1.7	5.6	75			
19	2.2	4.5	65			
20	2.5	3.6	60			
21	2.9	2.4	55			
22	3.2	1.1	50			

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

Active Cell

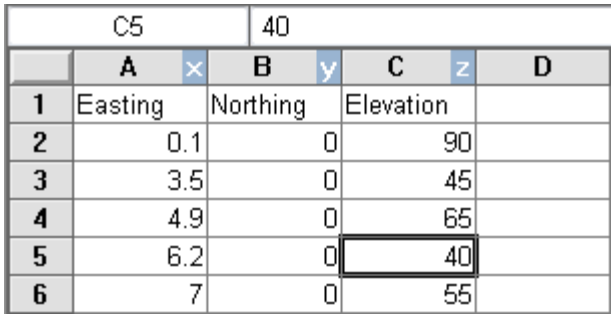
The active cell is displayed with a heavy border surrounding the cell. The contents of this cell are displayed in the cell edit box. You can enter or edit data in the active cell. To edit existing data, activate the desired cell and press the F2 key or highlight the information in the cell edit box.

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	B	C	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.

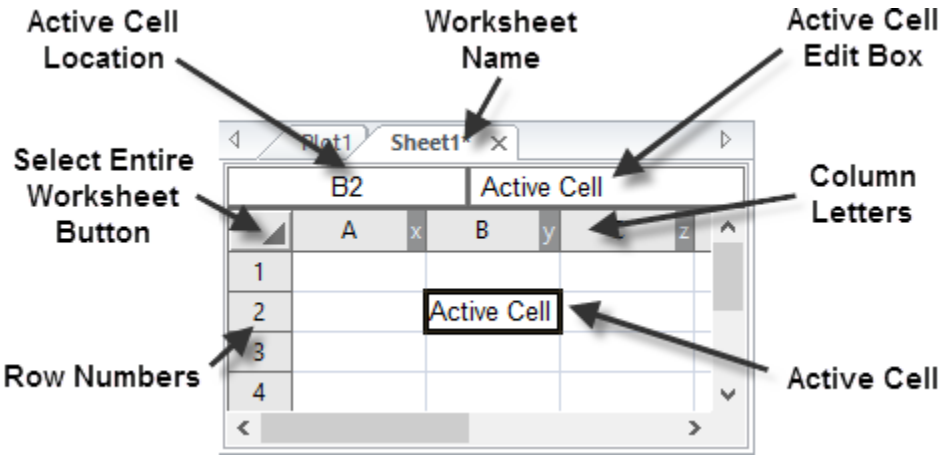
Special Key Functions when Editing the Active Cell:

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 File | Open 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 worksheet.
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+LEFT ARROW moves the active cell to the first column in the current row.
CTRL+RIGHT ARROW	The CTRL+RIGHT ARROW behavior depends on the position of the active cell. If the active cell is to the left of the first occupied column in the current row, it moves the active 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 occupied column in the current row, it moves the active cell to the last occupied column. Otherwise, CTRL+RIGHT ARROW moves the active cell to the last column in the current row.
CTRL+UP ARROW	The CTRL+UP ARROW behavior depends on the position of the active cell. If the active cell is below the bottom occupied row in the current column, it moves the active cell to the bottom occupied row in the current column. If the active 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 occupied row in the current column. Otherwise, CTRL+UP ARROW moves the active cell to the first row in the current column.
CTRL+DOWN ARROW	The CTRL+DOWN ARROW behavior depends on the position 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, CTRL+DOWN ARROW moves the active cell to the last row in the current column.
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 selection.

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 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 File | Import 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 **File | Import** 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.

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 rectangular 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.

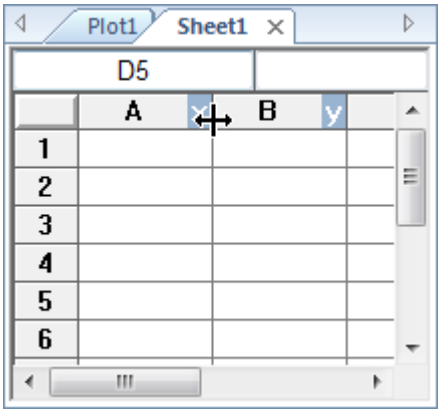
Selecting a Column or Row Dividing Line

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

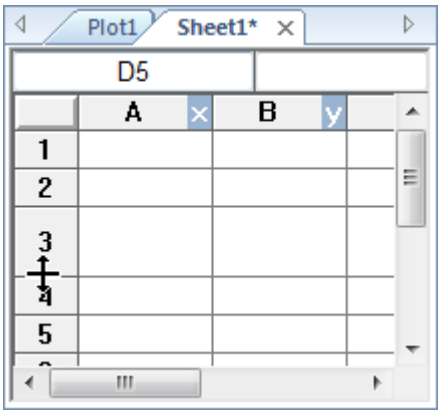
dividing line, the cursor must be within appoximately a characther's width of the dividing line and it must be on the label bar.

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.

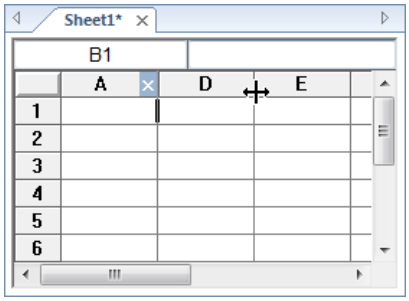
Displaying Hidden Columns or Rows

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 E. Select that vertical dividing line with the cursor slightly to the right of the dividing line. This will select the line for column D. (If the cursor is to the left of the dividing line, then the dividing line for column A will be selected.) Drag the vertical dividing line to the right to unhide column D. Repeat for columns C and B.



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 primary commands when viewing a worksheet window are located on the **Data** tab. 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, reproject data, and calculate mathematical transformations

Paste

Click the **Home | Clipboard | Paste** command 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.

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.

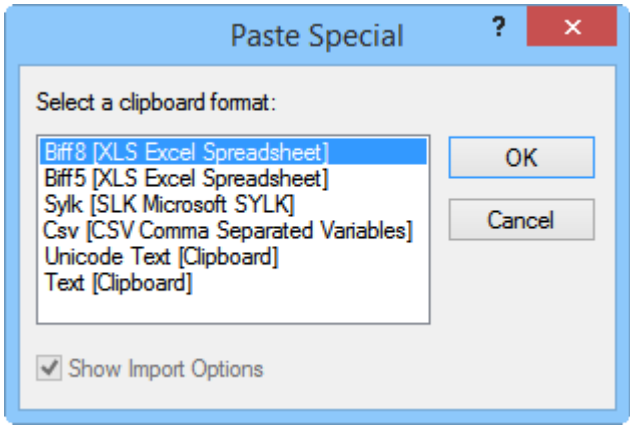
The multiple paste operation will not be performed for an entire row, entire column, or across multiple selections. When an entire row or column is selected, only the first cell in the row or column will receive the pasted content. If you attempt to paste in multiple selections an error message will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

Paste Special

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.

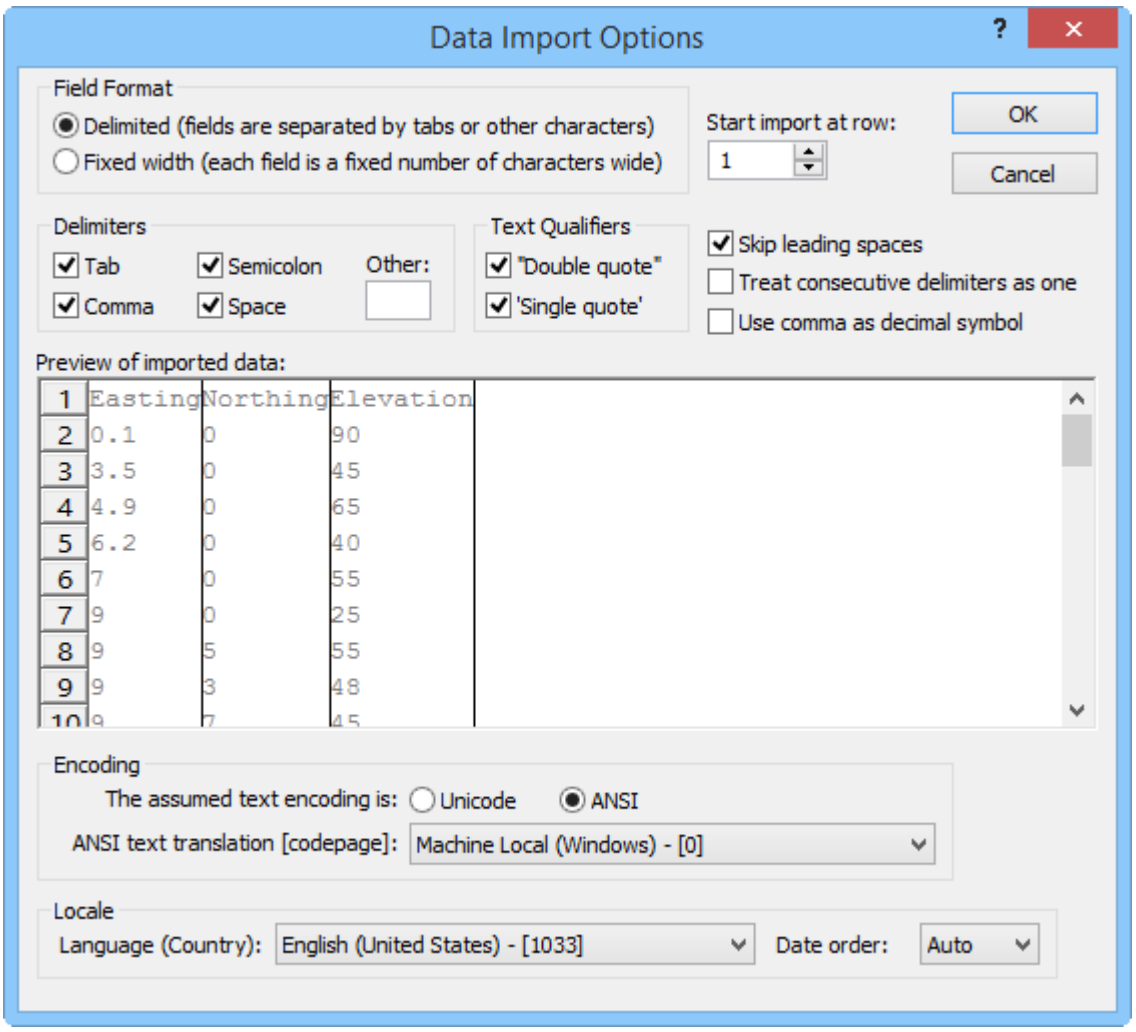
Show Import Options

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*).

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 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 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:

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

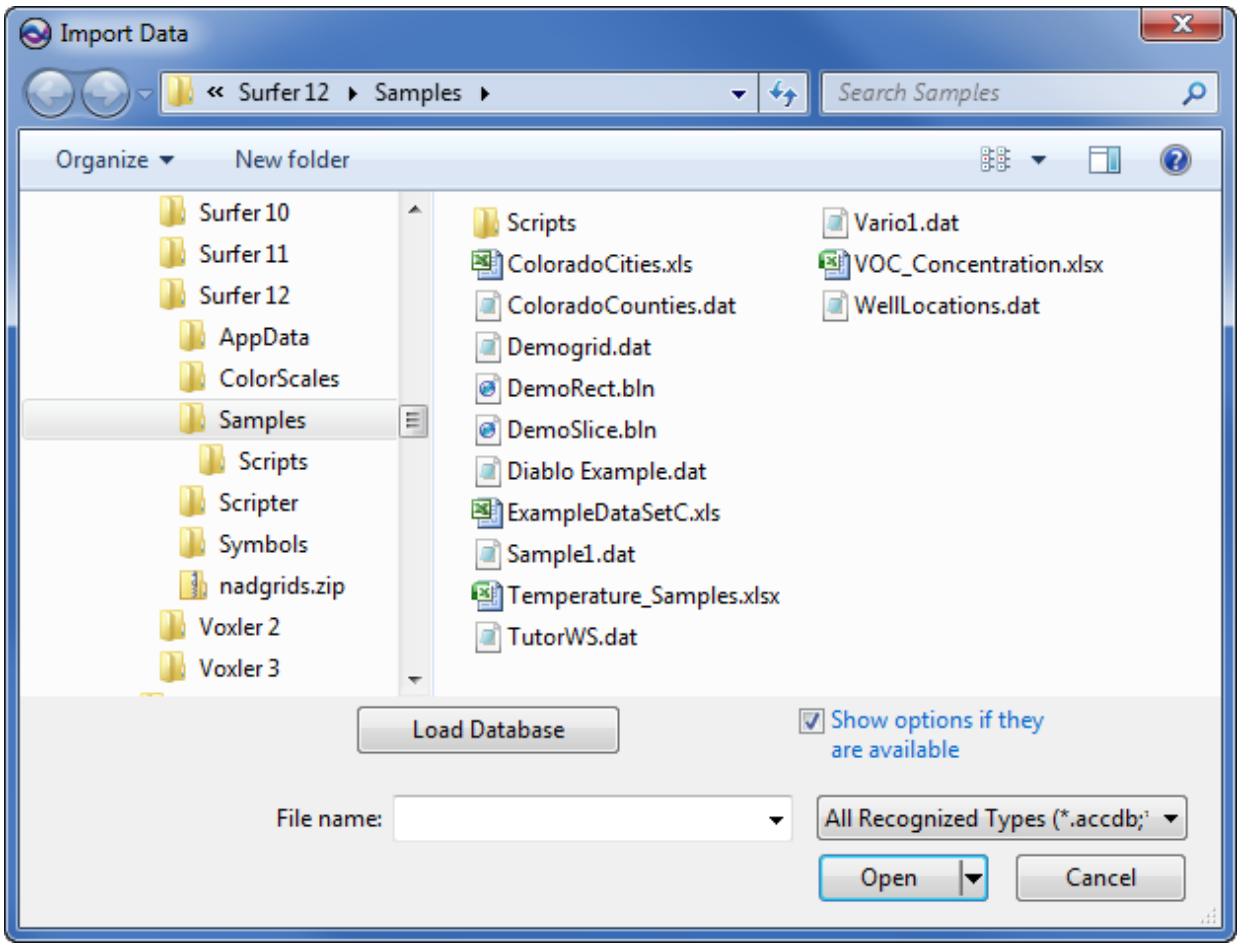
Click the **Data | Edit | Clear** command, or press the DELETE key on the keyboard, to remove data from selected worksheet cells. The cells become empty when the data are removed. To shift the data from unselected cells into the selected cell locations, use the Delete command instead.

Merge

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 in the worksheet 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.

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.

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

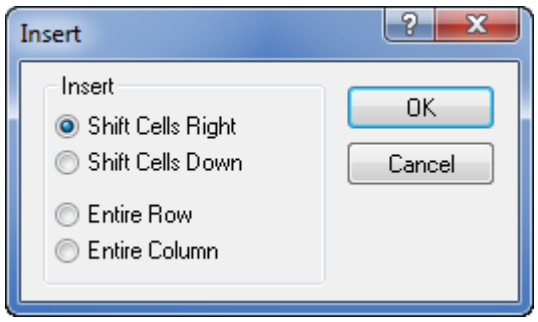
If *Show options if they are available* is checked, then opening .TXT files or ASCII text files with unsupported file extensions will bring up the Data Import Options dialog where you can specify the

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

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 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.

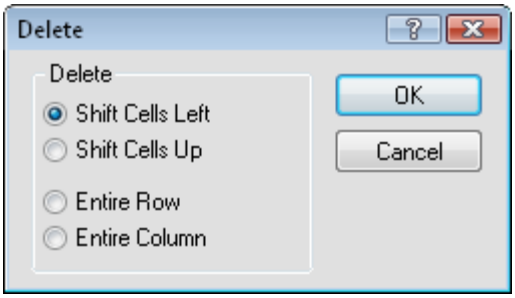
Entire Row or Entire Column

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

Delete

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 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.

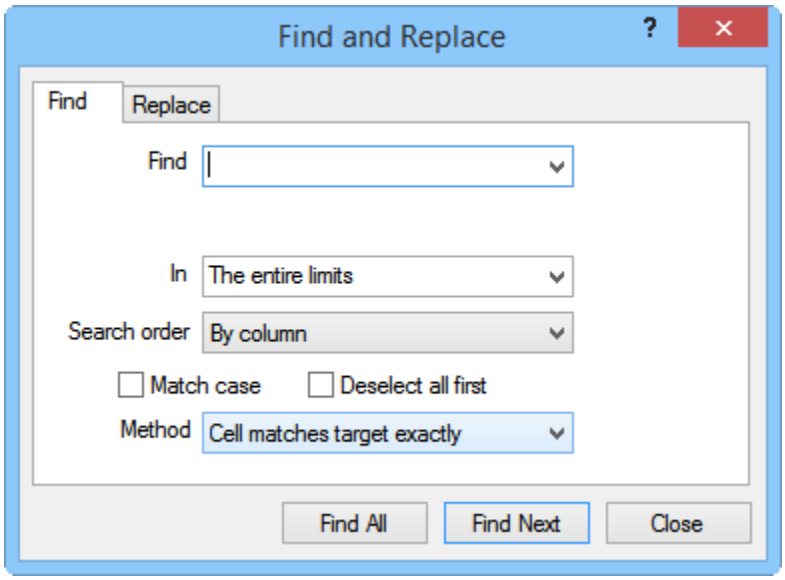
Find

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, 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.

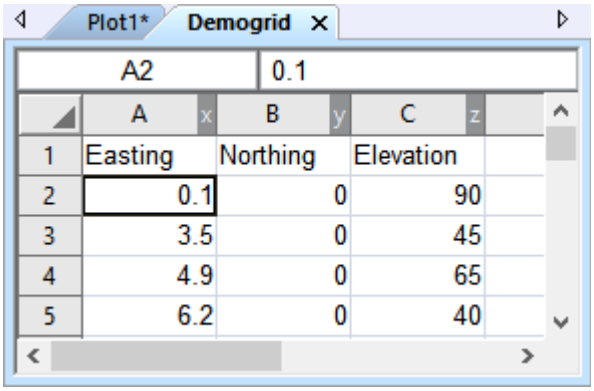
In

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

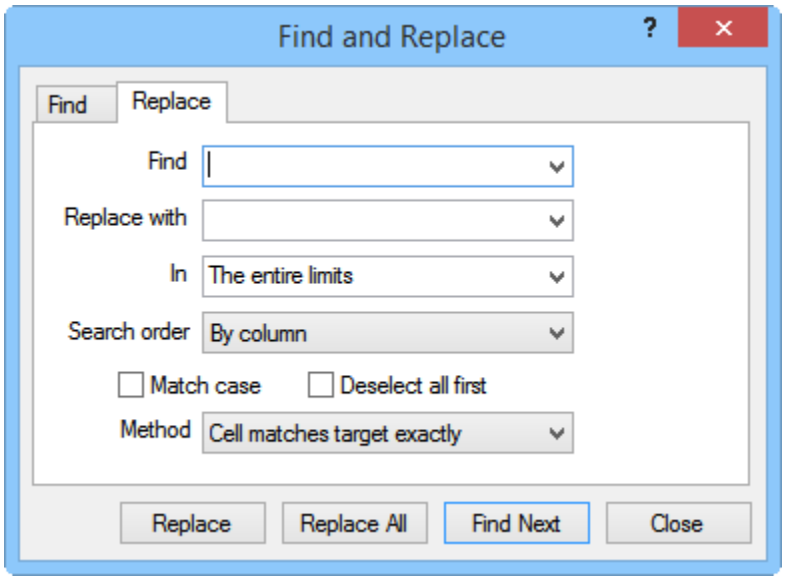
Click the *Find Next* button to find the next occurrence of the characters specified in the *Find* box. This allows you to meet the criteria one at a time. The next instance of the *Find* criteria will be highlighted.

Close Button

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

Replace Page

Clicking the **Data | Find | Replace** command 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.

Find Next

Clicking the **Data | Find | Find Next** command 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.

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. 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

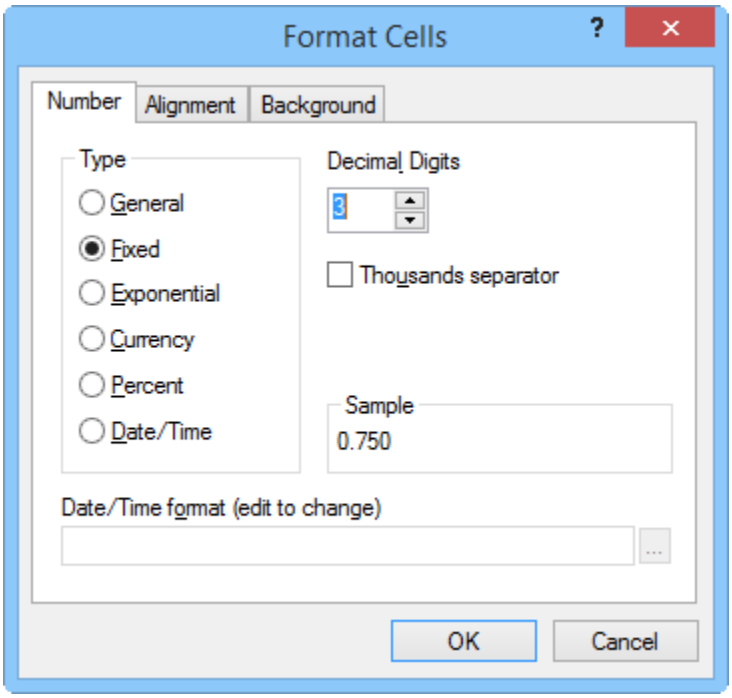
cell edit box. For example, an ASCII data file might contain the digits "8123" (digits surrounded by quotes), '8123 (digits preceded with an apostrophe), numbers with letters, or numbers with symbols (i.e. a blackslash "\"). These "numbers" are read as text and not as a number. The Transform command 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 the *Decimal Digits* box.

- *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

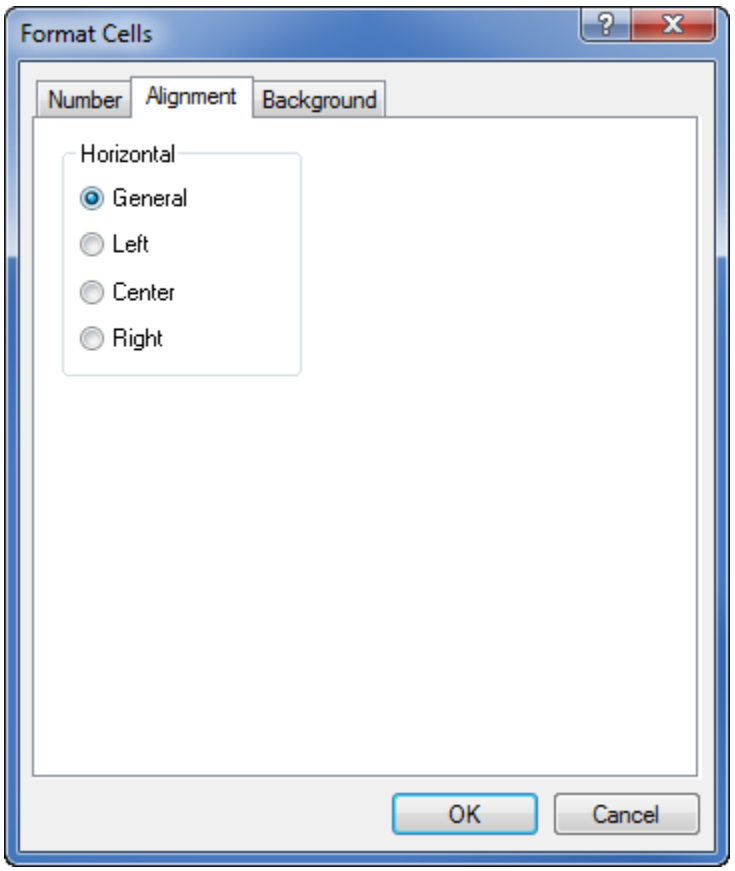
When the *Type* is set to *Date/Time*, the *Date/Time format* option becomes available. Type the desired format into the *Date/Time format* field, or click the button to insert a date/time format with the Date/Time Format Builder dialog. Available formats are made of combinations of day, month, year, 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

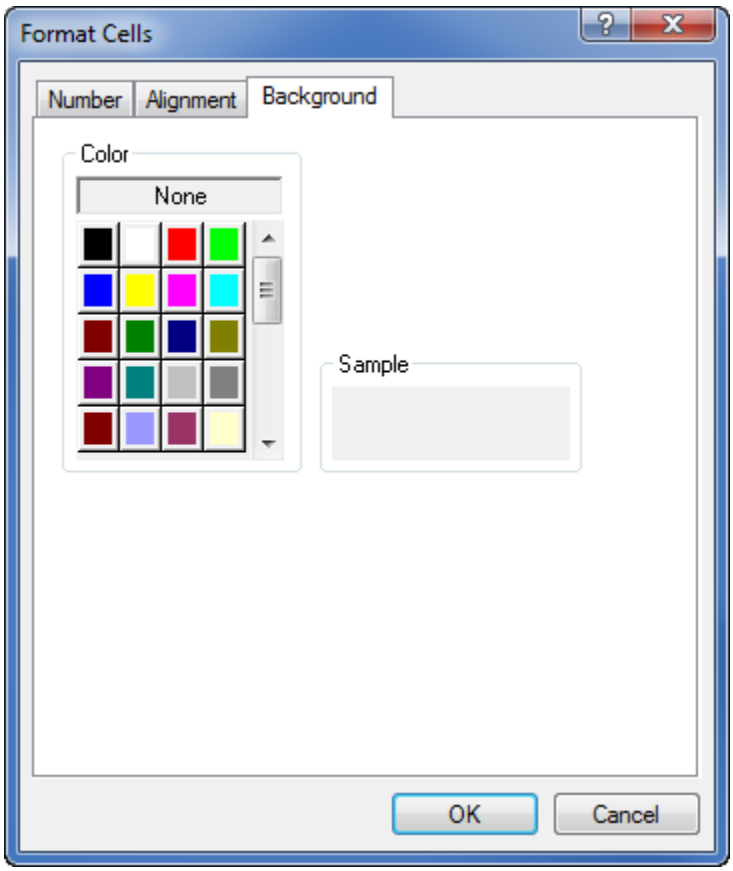
Center aligns text and numbers with the center of the cell.

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

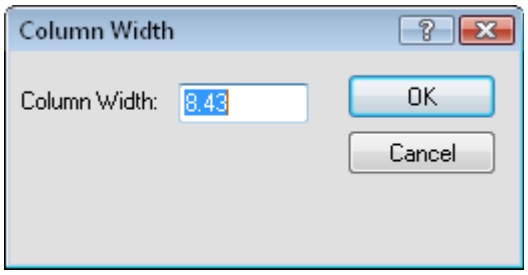
A sample of the color is displayed in the *Sample* box.

Column Width

You can change the column width of selected cells by clicking the **Data | Format | Column Width** command or by 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

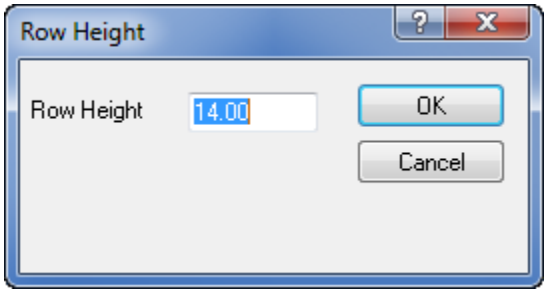
To display hidden columns, press and hold the left mouse button at the right edge of the hidden column and move the cursor to the right to widen the column.

Row Height

You can change the **Row Height** of selected cells by clicking the **Data | Format | Row Height** command or 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

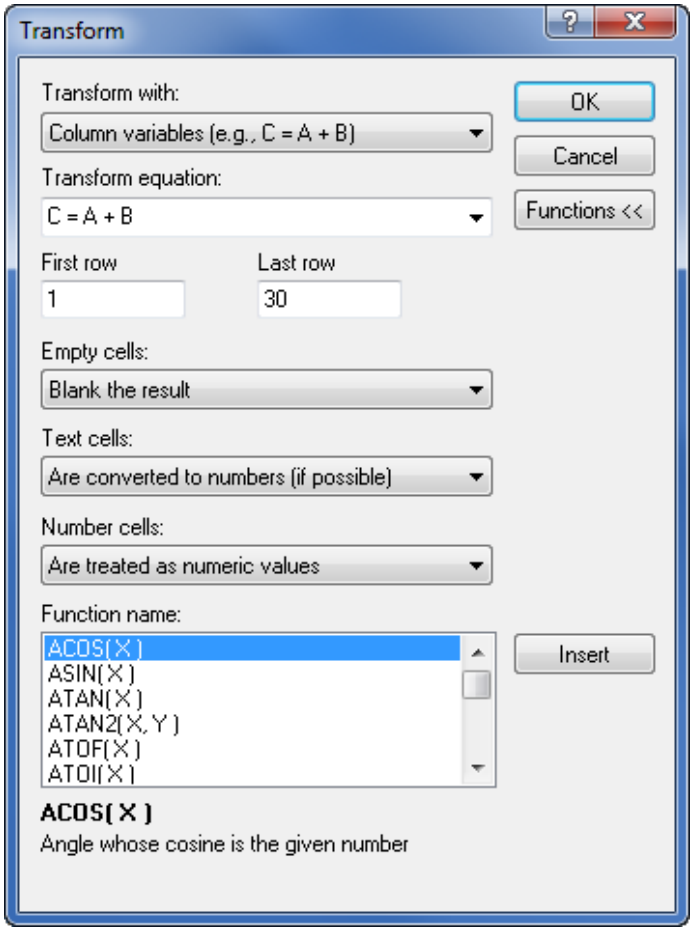
You can hide a row by moving the cursor up until the next dividing line is reached.

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

Click the **Data | Data | Transform** command 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.

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

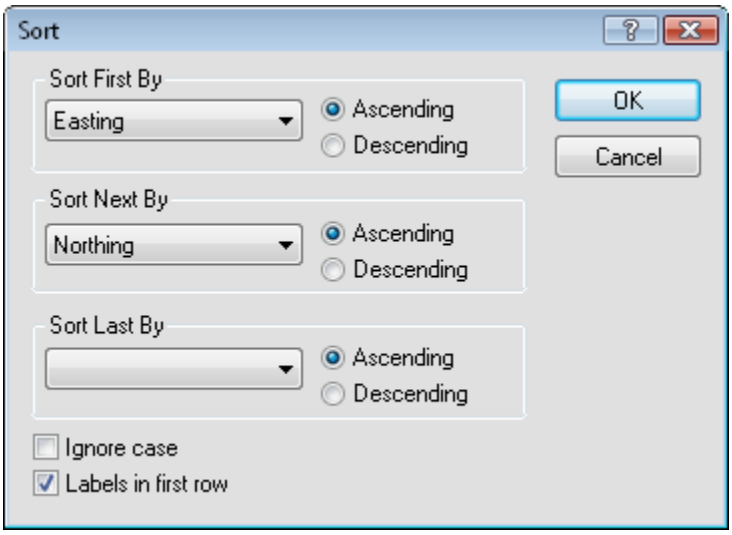
The **Data | Data | Sort** command arranges data according to rank in user-specified sort columns. Sorting rank is based on numbers, ASCII characters, and punctuation. Sort numeric data, text, or mixed columns. Sorting specifications are made in the **Sort** dialog.

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 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

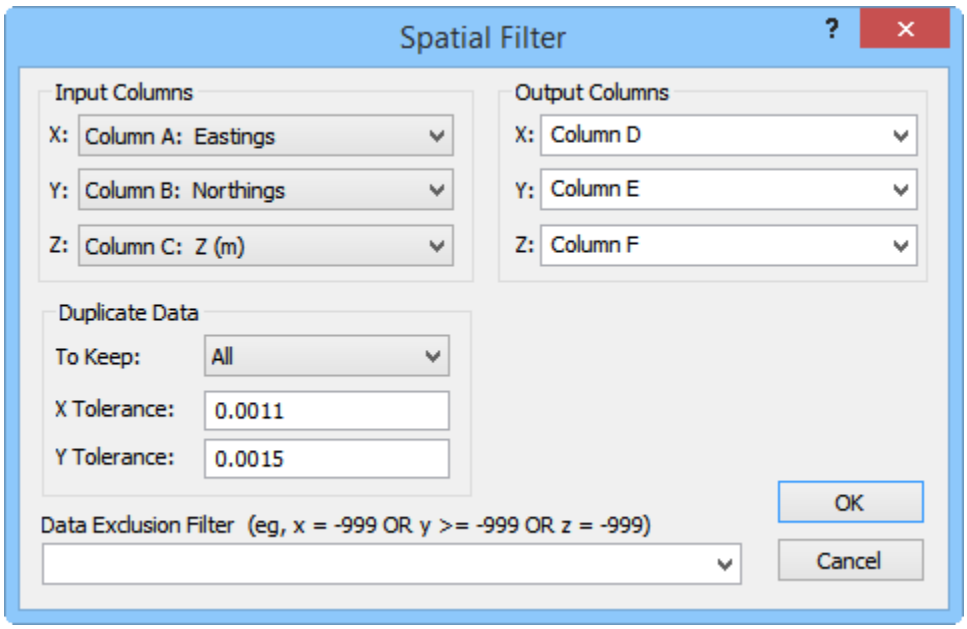
The data set may contain text identifying the data in the column (header information) in Row 1. In this case, click the *Labels in first row* option to exclude the label row from the sort process.

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 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, true_value, false_value)

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.

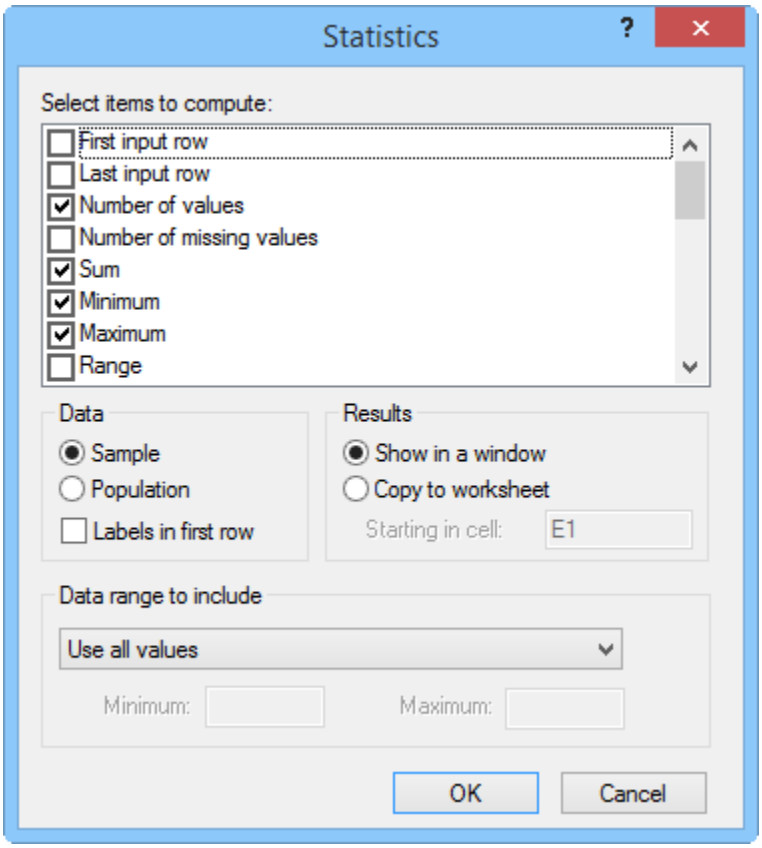
Consider a second case where data contains a numerical identifier in column D. When the value in this column is equal to -999, the data point is considered inaccurate and should not be used when gridding. To grid only those data where column D is not equal to -999, exclude column D with the *Data Exclusion 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

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 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 (25th percentile)* 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 (75th percentile)* 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

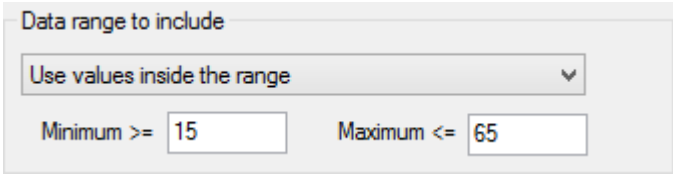
The *Data range to include* contains options to limit the values where the statistics are calculated. Available options are *Use all values*, *Use values inside the range*, *Use values outside the range*, and *Use all values except*.

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

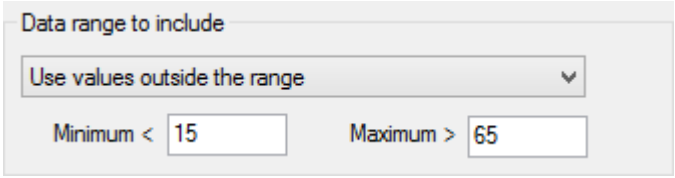
When the *Data range to include* is set to *Use values inside the range*, the *Minimum* >= and *Maximum* <= options are available. Type in the data values that bracket the range of values where the statistics should be calculated. For instance, if the *Minimum* >= is set to 15 and the *Maximum* <= is 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

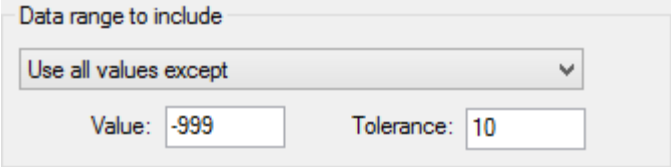
When the *Data range to include* is set to *Use values outside the range*, the *Minimum* < and *Maximum* > options are available. Type in the data values that bracket the range of values where the statistics should be calculated. For instance, if the *Minimum* < is set to 15 and the *Maximum* > is set 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.

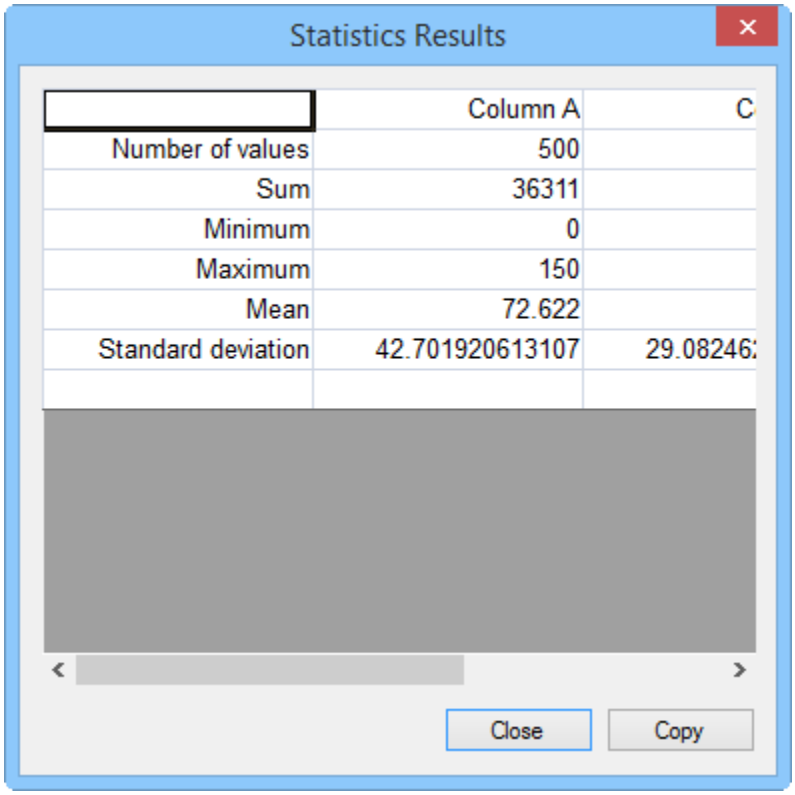


Only the values that are outside the range Value-Tolerance to Value+Tolerance are included in the calculated statistics.

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 CI and the sample mean plus CI is expected to include the true mean of the underlying population 95% of the time (for the 95% confidence interval) or 99% of the time (for the 99% confidence interval). This formula assumes that the data set is sufficiently large for the central limit theorem to apply.

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

$t_{v,\alpha}$ = the value of the Student's t distribution with v degrees of freedom such that difference between the cumulative probability function evaluated at $t_{v,\alpha}$ and - $t_{v,\alpha}$ is equal to 1- α.

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
x_i	= i^{th} data value

Coefficient of Kurtosis

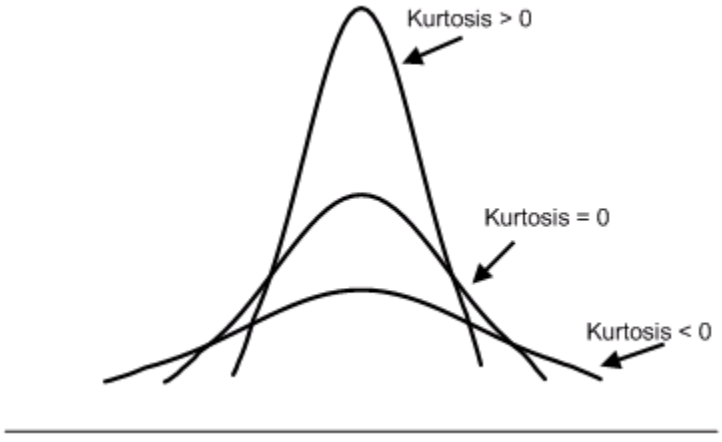
Kurtosis is a measure of how sharp the data peak is. Traditionally the value of this coefficient is compared to a value of 0.0, which is the coefficient of kurtosis for a normal distribution (i.e. the bell-shaped curve). A value greater than 0 indicates a peaked distribution and a value less than 0 indicates a flat distribution. Without a very large sample size, the use of this coefficient is of questionable value.

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
x_i	= i^{th} data value

Coefficient of Skewness

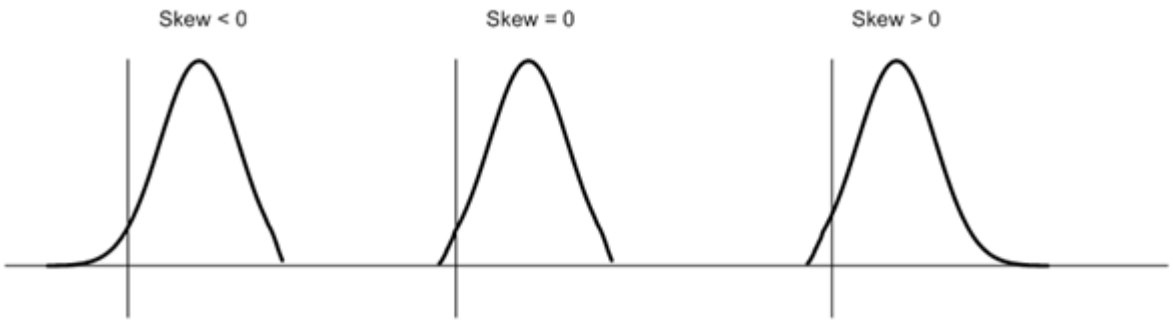
Skew is a measure of asymmetry in the distribution. A positive skew indicates a longer tail to the right, while a negative skew indicates a longer tail to the left. A perfectly symmetric distribution, like the normal distribution, has a skew equal to 0. For small data sets this measure is unreliable.

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
x_i	= i^{th} data 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

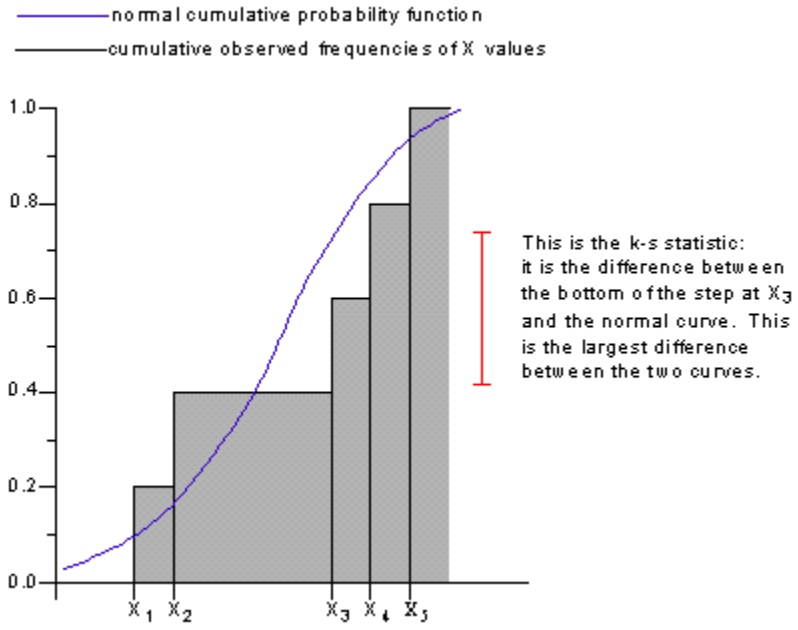
The critical value of K-S statistic at 90%, 95%, or 99% significance level are indicators of normal distributions. For example, if a sample collected from a population has a normal frequency distribution, the K-S statistic for that sample is less than the critical value 90, 95, or 99 percent of the time. 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
x_i	= i^{th} data 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

x_i = i^{th} data 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. *Biometrika*, 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 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	B	C	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	B	C	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 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

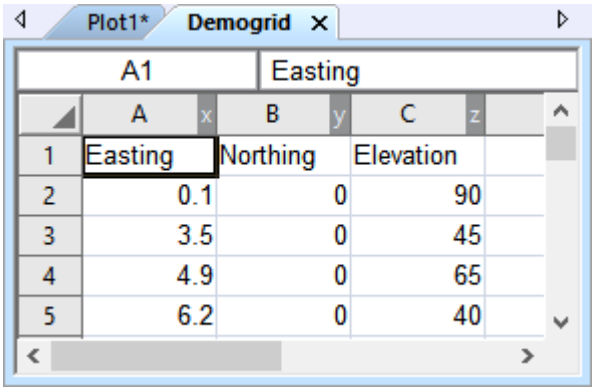
Categories A, B, and C are now displayed with each category in a separate column.

Assign XYZ Columns

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.

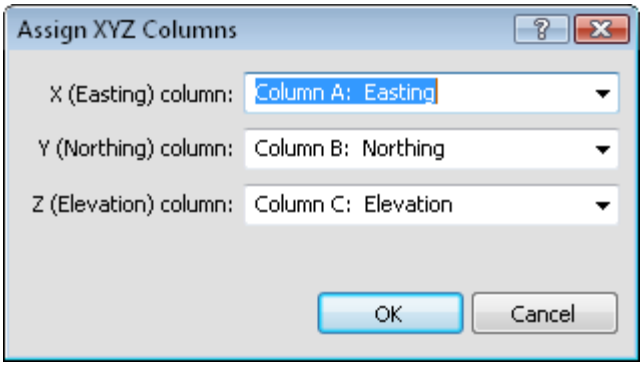
Indicators in the worksheet column headers display the current columns. When the coordinates are changed in the **Assign XYZ Columns** dialog, the X, Y, and Z indicators move to the specified columns. If there is no actual data in the user assigned columns, **Surfer** will assign columns that contain 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.*

Assign XYZ Columns Dialog

The **Data | Data | Assign XYZ Columns** command 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

The **Data | Coordinate System | Assign Coordinate System** command 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.

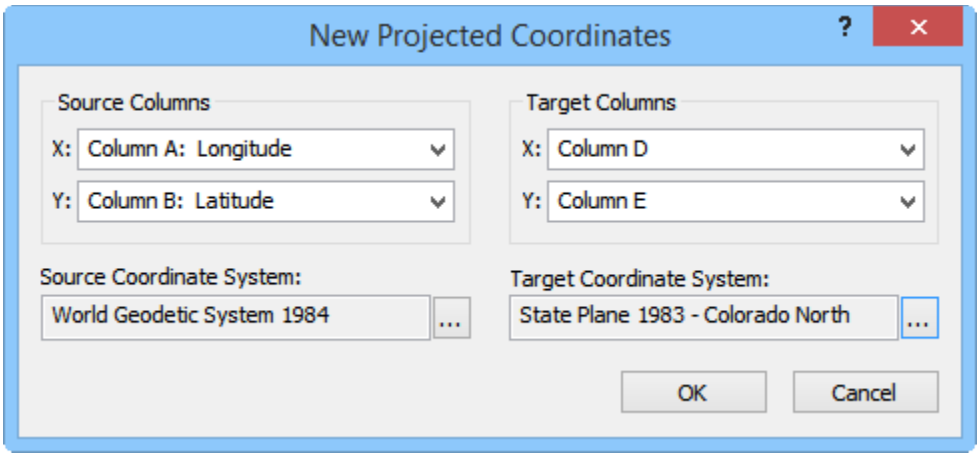
When a .GRD file is created using the Grids | New Grid | Grid Data command, the .GSR2 file for the data is read. The projection information can be saved with the grid file using the Spatial References options. It is recommended to check the *GS Reference (Version 2) file* if you intend to use the grid 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

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 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

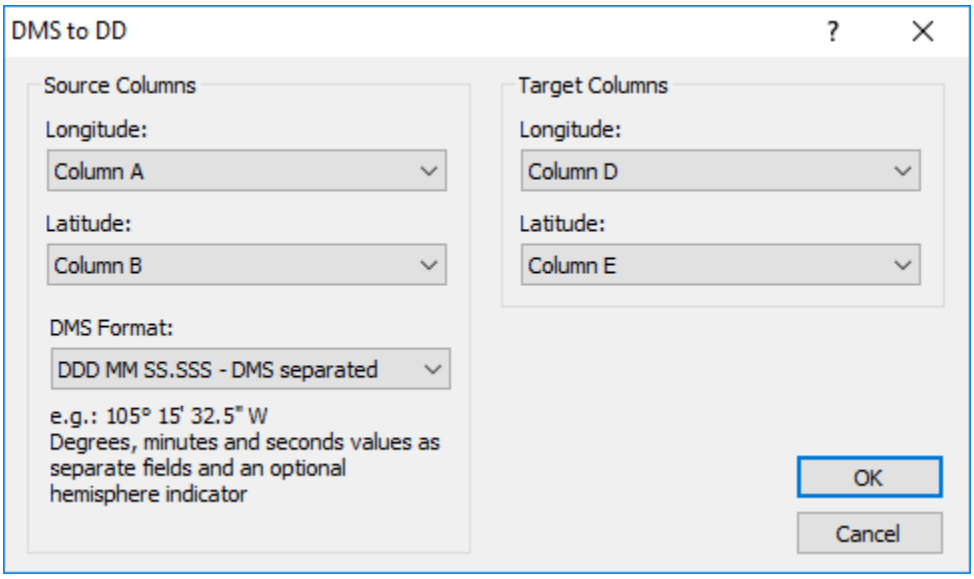
Click *OK* to write the new projected X and Y coordinates to the *Target Columns*. Click *Cancel* to close the **New Projected Coordinates** dialog without making any changes to the worksheet.

DMS to DD

Click the **Data | Coordinate System | DMS to DD** command 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 degress.

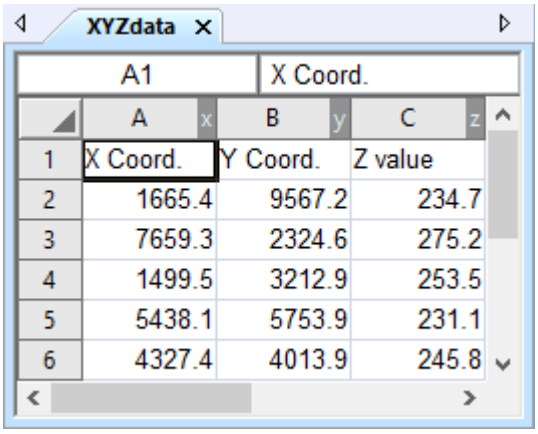
Chapter 4 - Creating Grid Files

Contour maps, color relief maps, shaded 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

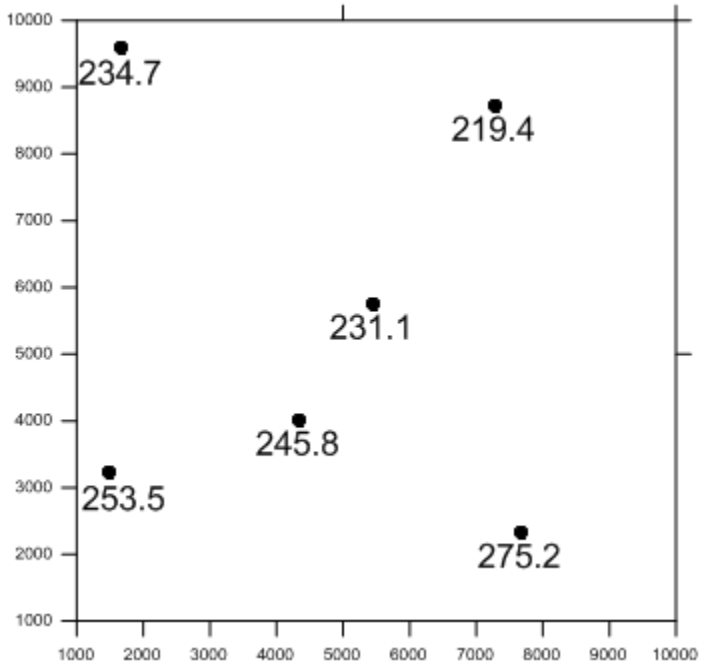
Consider the scenario of producing a contour map of water table depth given well data collected over a region. The well locations are not regularly spaced over the area of interest. If you provide **Surfer** with the locations of the wells (the XY coordinates) and the depth to the water table (the Z 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.



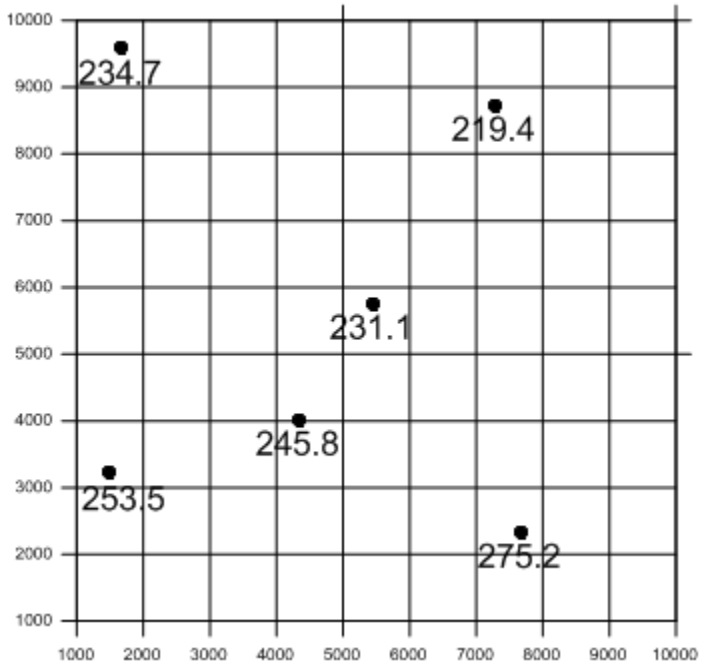
This is the XYZ data file that defines the well locations and water table depth at each location.

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.



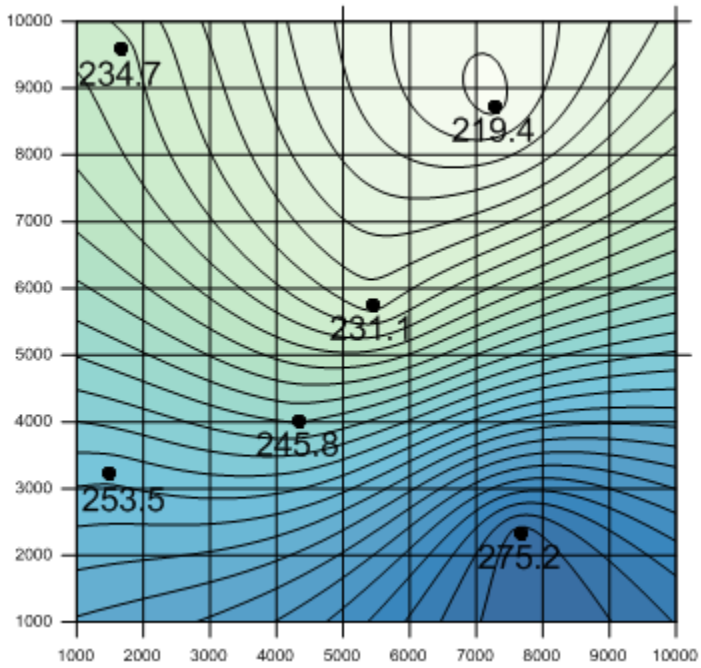
This is a post map displaying the data locations. The well locations are irregularly spaced over the map in this example. There are many "holes" where no data exists.

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

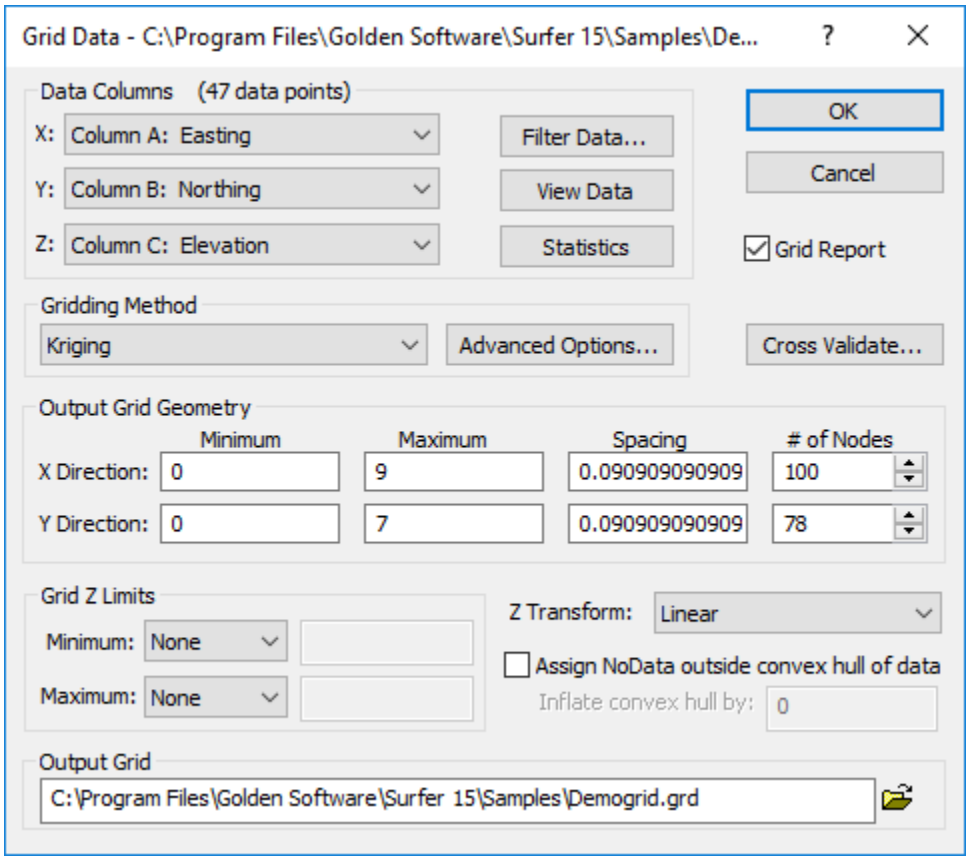
Grid files are necessary in **Surfer** to create grid-based maps types. Data files are typically randomly spaced files, and this data must be converted into an evenly spaced grid before using many of **Surfer's** features. Grid files are produced from XYZ data using the **Home | Grid Data | Grid Data** 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. Refer to the tutorial for more information on data and gridding data.

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 Open Data dialog. Select a data file and click *Open*. The **Grid Data** dialog is displayed.



*Gridding options are set in the **Grid Data** dialog.*

Data Columns

Individually specify the columns for the X data, the Y data, and the Z data in the *Data Columns* section. **Surfer** defaults to *X: Column A*, *Y: Column B*, and *Z: Column C*. However, your data can be in any three columns. Click the down arrow on each box and select the appropriate column for each variable. If the data file was selected from the *Open worksheets* list in the **Open Data** dialog, assigned XYZ columns (if any) will populate the appropriate columns in the *Data Columns* group. Columns containing dates or numbers can be selected.

Note: 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.

Filter Data

You can filter the data before gridding based on a predefined filter or based on a user-defined equation by clicking the Filter Data button.

View Data

If you are unsure of which columns contain your XYZ data, click the *View Data* button to see the data file in a worksheet format. If you get an *Insufficient data (3 or more XYZ triplets required)* error, use *View Data* to check the layout of the data. One common reason for this warning is that the data is not numeric or date/time format. After clicking *View Data*, make sure that all three columns of data are right aligned. If one of the columns is left aligned, the data are text, not numbers. You can use the data view to determine the appropriate columns for the X, Y, and Z values.

Statistics

Click the *Statistics* button to display statistics based on the selected X, Y, and Z columns.

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.

Gridding Method and Advanced Options

Surfer has several different gridding methods. These gridding methods define the way in which the XYZ data are interpolated when producing a grid file. Choose the *Gridding Method* and gridding options (*Advanced Options* button) in the *Gridding Method* group. Refer to the gridding methods help topics for more information on the options.

Cross Validate

Click the *Cross Validate* button to perform cross validation on your data. Cross validation is an objective way of assessing the gridding parameters for your data set. Cross validation is always performed on the linear Z values, not the transformed Z values.

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.

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.

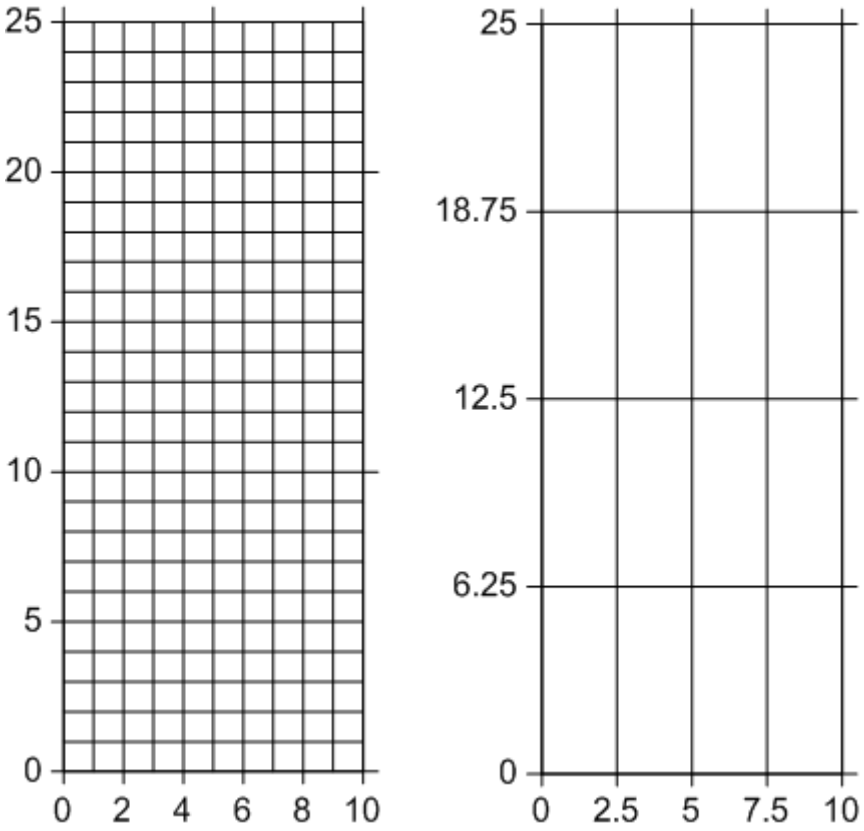
Some examples of the amount of memory needed to grid large files:

- 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 an 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

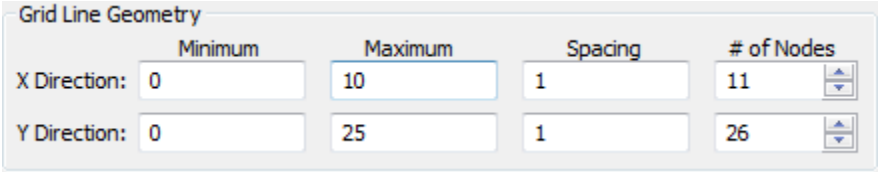
Consider these examples. The data range from 0 to 25 in the Y dimension and 0 to 10 in the X dimension. The two examples use different numbers of grid nodes, or grid spacing, during gridding.



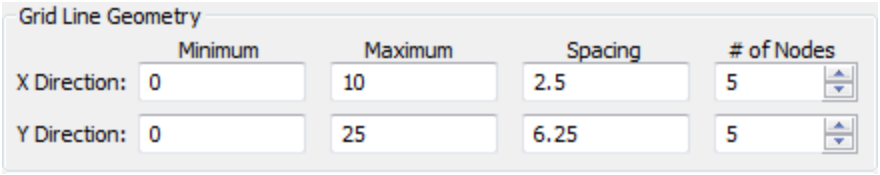
Two different Grid Line Geometry examples are shown here. These are based on the same data file. The coordinates range from zero to 10 in the X direction and zero to 25 in the Y dimension.

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*.

Convex Hull of 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. So, areas inside the convex hull without data are still gridded.

Assign NoData Outside Convex Hull

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

Inflate Convex Hull

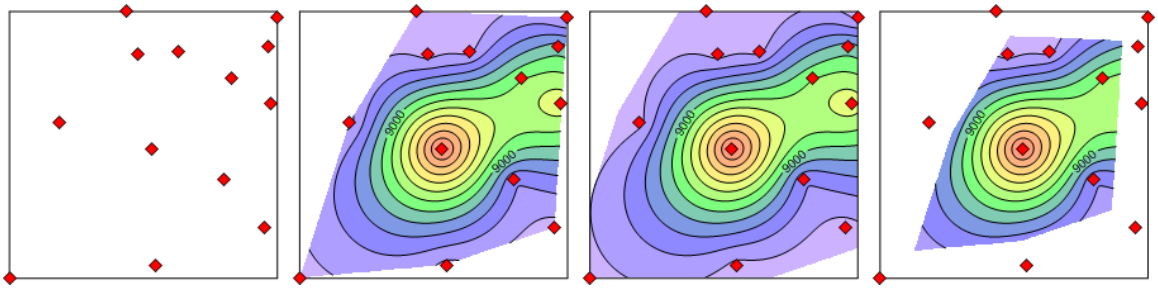
The *Inflate convex hull by* option expands or contracts the convex hull. When set to zero, the boundary connects the outside data points exactly. When set to a positive value, the area assigned the NoData value is moved outside the convex hull boundary by the number of map units specified. When set to a negative value, the area assigned the NoData value is moved inside the convex hull boundary by the number of map units specified.

To change the value, highlight the existing value and type the desired value. 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.



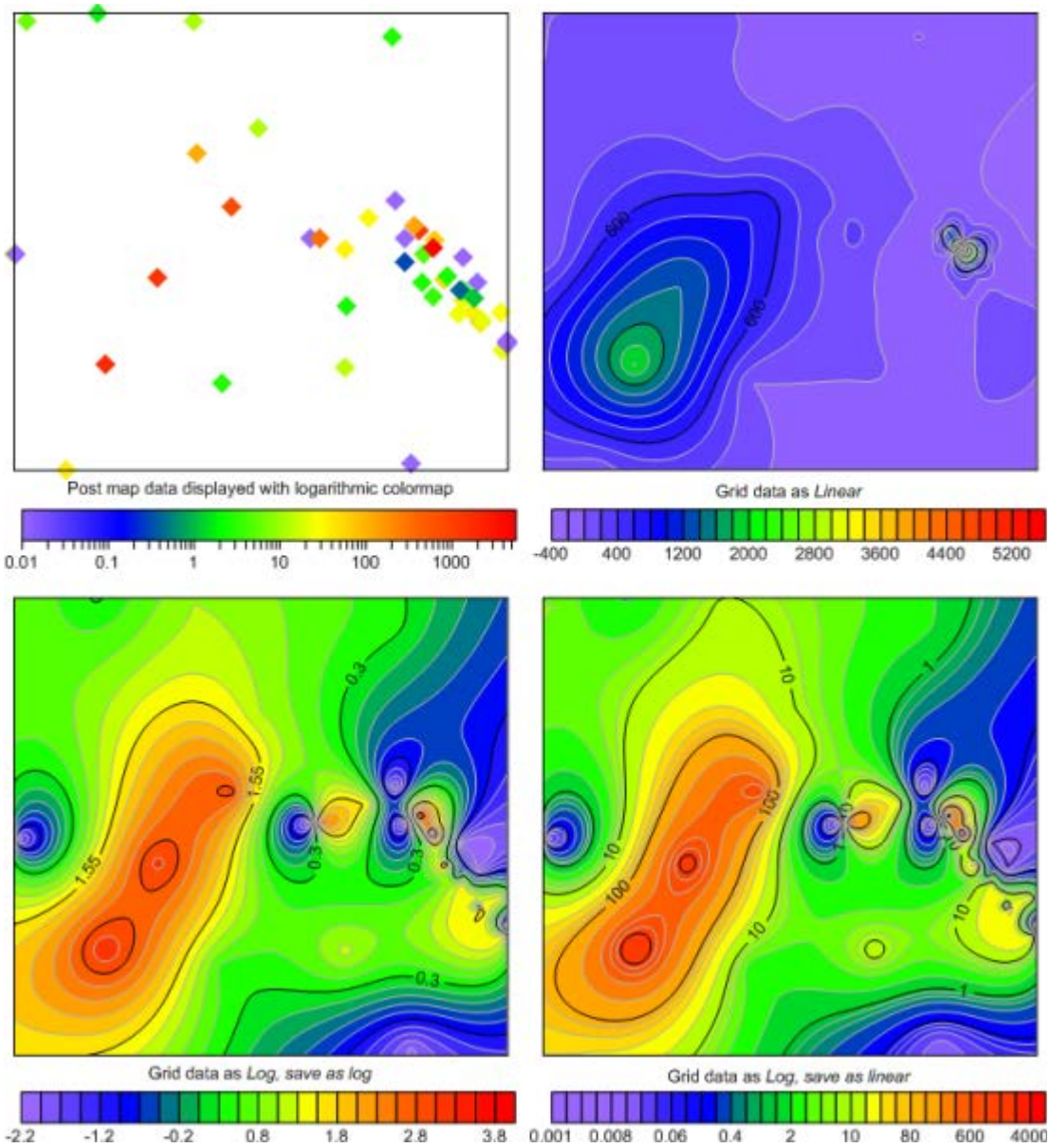
This data has an area near the top left corner with no data. When the Assign NoData outside convex hull of data option is selected, no contours appear in this area. Setting the Inflate convex hull by value to a positive value (3rd image) creates a buffer around the outside of the data points. Setting the Inflate convex hull by value to a negative value (last image) brings the contours inside the convex hull of data.

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.

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.

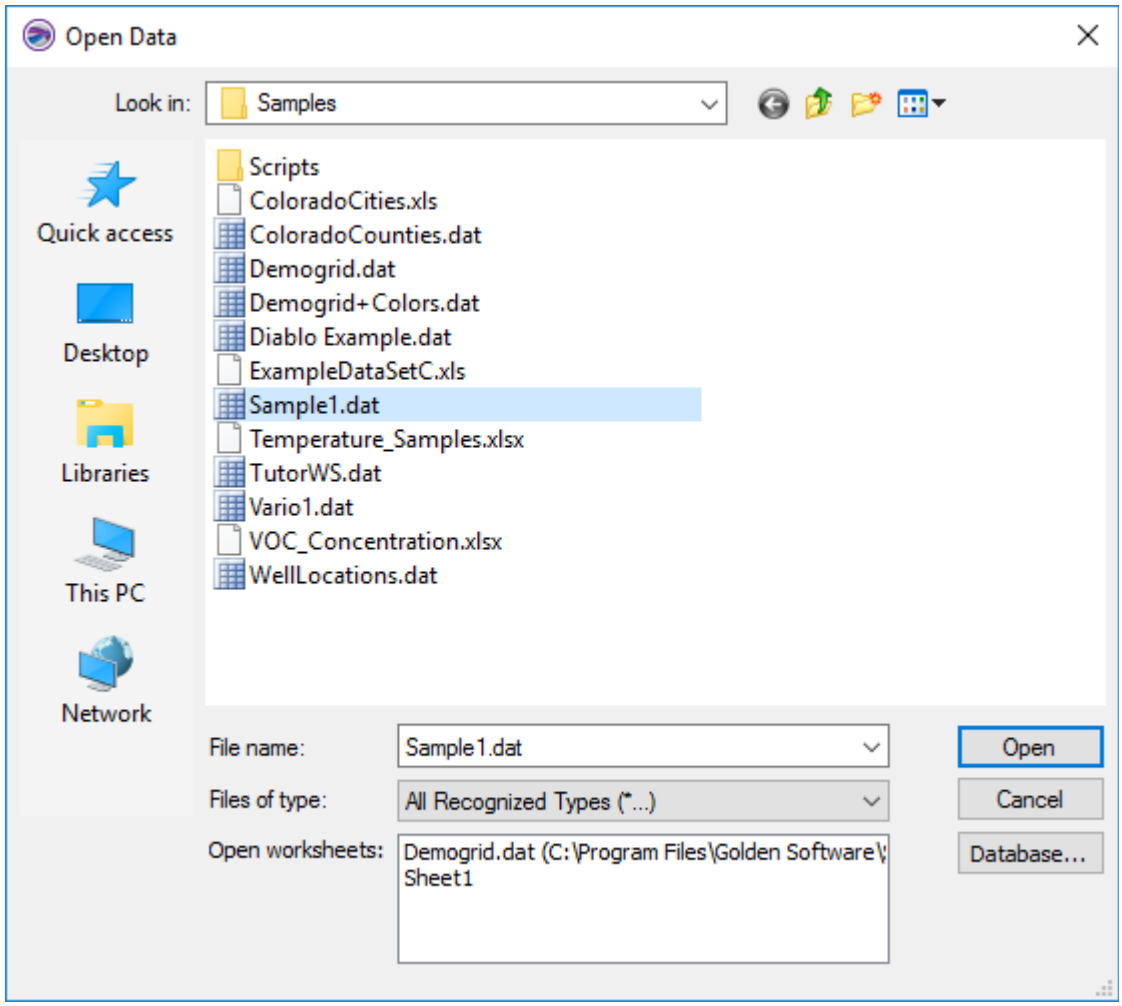
To Create a Grid File from an XYZ Data File

1. Create an XYZ data file. The data must be organized in columns: all X data in one column, all Y data in another column, and all Z data in a third column.
2. Click the **Grids | New Grid | Grid Data** command to display the **Open Data** dialog.

3. Specify the name of the XYZ data file, and click *Open*. To link to a database instead of a data file, click the *Database* button.
4. In the **Grid Data** dialog, specify the parameters for the type of grid file you want to produce.
5. Click *OK* and the grid file is created. During gridding, the status bar at the bottom of the **Surfer** window provides you with information about the progress of the gridding process.

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.

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

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.

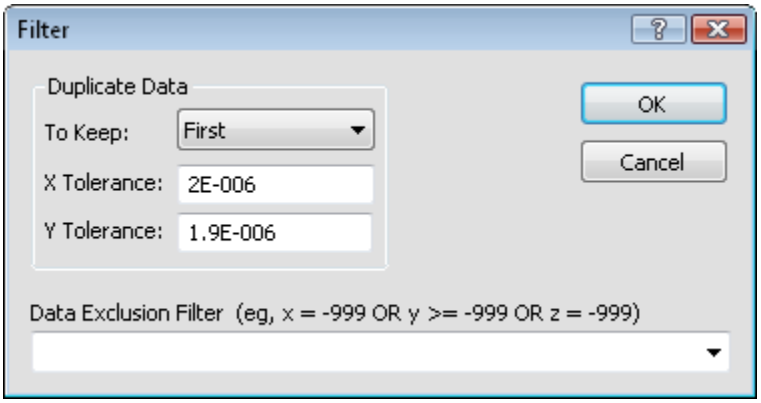
Load Database

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

Data Filters

You can filter the data before gridding in the worksheet using the Data | Data | Spatial Filter command. Or, you can filter data directly in the Grid Data dialog based on a predefined filter or based on a user-defined equation. The **Filter** dialog provides methods for filtering data. Filtering is also available on the **Data** page in the New Variogram dialog. You can filter data by using one of the predefined data filtering methods (*To Keep* list), or you can generate a filtering method with the *Data Exclusion Filter*. When filtering data, the *Data Exclusion Filter* is applied first, then the *Duplicate Data* methods are applied next.

To open the **Filter** dialog, click the **Grids | New Grid | Grid Data** command. Select a data file and click *Open* in the **Open Data** dialog. In the **Grid Data** dialog, click the *Filter Data* button to open the **Filter** dialog.



*Filter data before gridding in the **Filter** dialog.*

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 and Y coordinates. The Z values may vary for these X and Y coordinates. Duplicate data present a problem for a number of gridding methods, including *Kriging*, *Natural Neighbor*, *Radial Basis Function*, and *Triangulation with Linear Interpolation*. Filtering the data reduces the duplicate points to allow a better gridding fit.

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. A report containing the retained and deleted points can be generated.

All	Do not delete any duplicates. This option is not available for Kriging, Nearest Neighbor, Natural Neighbor, Radial Basis Function, and Triangulation with Linear Interpolation.
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 data point 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 data point 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 data point 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. The *Tolerance* values specify how close two points are located before **Surfer** considers the points duplicates. 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 points B and C to be "duplicates," but for points 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, true_value, false_value)

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.

Cross Validate

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**.

The 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.

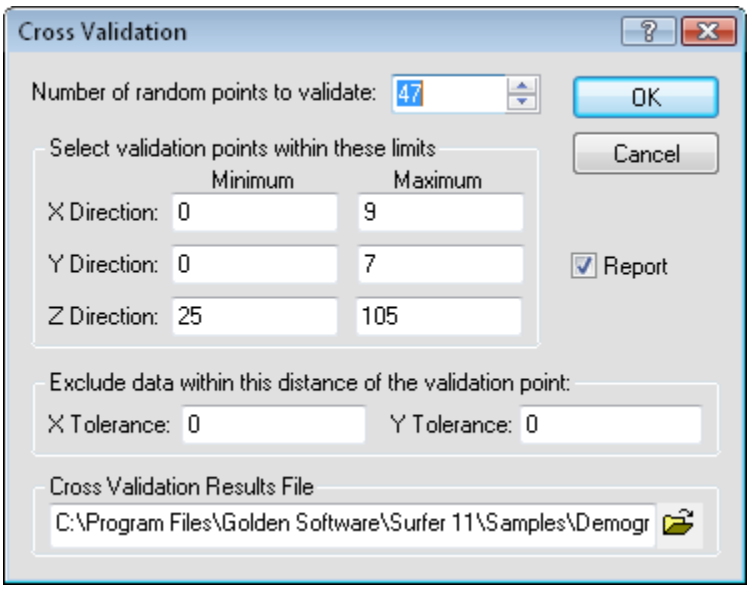
Using Cross Validation

To obtain cross validation information:

1. Click Grids | New Grid | Grid Data.
2. Select a data file in the **Open Data** dialog and click *Open*.
3. Click the *Cross Validate* button in the **Grid Data** dialog.
4. Select the number of random points, the validation limits, and exclusion limits you wish to use with cross validation.
5. Click the *Report* box to generate a Cross Validation Report.
6. Set the results path and file name in the *Cross Validation Results File* field.

The Cross Validation Dialog

Click the *Cross Validate* button in the **Grid Data** dialog to open the **Cross Validation** dialog.



*Click the Cross Validate button in the **Grid Data** dialog to open the **Cross Validation** dialog.*

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.

Select Validation Points Within These Limits

Set the *X Direction*, *Y Direction*, and *Z Direction* values for *Minimum* and *Maximum* in the S*elect validation points within these limits* 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 the *Z Direction* are useful to exclude cross validation at known anomalous observation locations.

Exclude Data Within this Distance of the Validation Point

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 Results File

Set a path and file name in the *Cross Validation Results File* field for the cross validation results. 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 in **Surfer**.

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 in Hydrogeology.* Cambridge University Press, Cambridge, 249 pp.

Olea, R. (1999) *Geostatistics for Engineers and Earth Scientists*. Kluwer Academic Publishers, Boston, 303 pp.

Reports

You can gather information about your data by

- clicking the *Statistics* button in the Grid Data dialog to obtain a Data Statistics Report,
- checking the *Grid Report* option in the **Grid Data** dialog to create a Gridding Report,
- checking the *Generate Report* option on the **General** page of the New Variogram dialog to create a Variogram Grid Report,
- clicking the *Display Statistics* button on the **Statistics** page in the Variogram Properties and clicking *Report* to create a Variogram Grid Report,
- clicking the button for a map layer in the **Properties** window to generate a Grid Information report,
- clicking the Grids | Info | Grid Info command to obtain a Grid Information report,
- clicking the Options | Grid Info command in the **Grid Editor** to create a Grid Information report,
- or checking the *Report* option in the Cross Validation dialog to create a Cross Validation Report.

All reports contain the name of the report and the name of the data file. For example, a Data Statistics Report for the file demogrid.dat will be named *DataStatisticsReport-Demogrid*. The Grid Data Report will be named *GridDataReport-Demogrid*.

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 above. All reports contain similar information and differences are noted below.

Report Types

There are 5 types of gridding reports. Each report contains different information. Some of the information is similar among the reports.

Data Statistics Report

Contains Time Stamp, Data Source, Filtered Data Counts, Exclusion Filtering, Duplicate Filtering, Breakline Filtering, Data Counts, Univariate Statistics, Inter-Variable Covariance, Inter-Variable Correlation, Inter-Variable Rank Correlation, Principal Component Analysis, Planar Regression: Z = AX+BY+C, and Nearest Neighbor Statistics. See below for a list of the information contained in each section. The data statistics always refer to the pre-transformed Z values, even when the Log, save as log or Log, save as linear option is selected when gridding.

Gridding Report

Contains Time Stamp, Data Source, Filtered Data Counts, Exclusion Filtering, Duplicate Filtering, Breakline Filtering, Z Data Transform, Data Counts, Univariate Statistics, Inter-Variable Covariance, Inter-Variable Correlation, Inter-Variable Rank Correlation, Principal Component Analysis, Planar Regression: Z = AX+BY+C, Nearest Neighbor Statistics, Gridding Rules, and Output Grid. See below for a list of the information contained in each section.

Grid Information Report

Contains *Time Stamp, Grid Information, Grid Geometry,* and *Univariate Grid Statistics.* See below for a list of the information contained in each section.

Cross Validation Report

ContainsTime Stamp, Data Source, Gridding Rules, Data Counts at Validation Points, Univariate Statistics, Univariate Cross-Validation Statistics, Residual Regression at Validation Points: R = AX+BY+C, Inter-Variable Correlation at Validation Points, and Rank Correlation at Validation Points. See below for a list of the information contained in each section.

Variogram Grid Report

Contains Data Source, Variogram Grid, Data Counts, Univariate Statistics, Inter-Variable Correlation, Inter-Variable Covariance, Planar Regression: Z = AX+BY+C, Nearest Neighbor Statistics, Exclusion Filtering, and Duplicate Filtering. See below for a list of the information contained in each section.

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 Filter string
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 $\gamma_2 = \left(\frac{1}{N\,\sigma^4}\sum_{i=1}^{N}(x_i-\mu)^4\right)-3$
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	$\text{MeanSquare} = \frac{1}{N}\sum_{i=1}^{N}X_i^2$

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 S_i = 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 S_i = 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

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.

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** page appears in the advanced options dialog. However, in some cases a few of the search options, such as *Anisotropy*, are available on the gridding method's **General** page. 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.

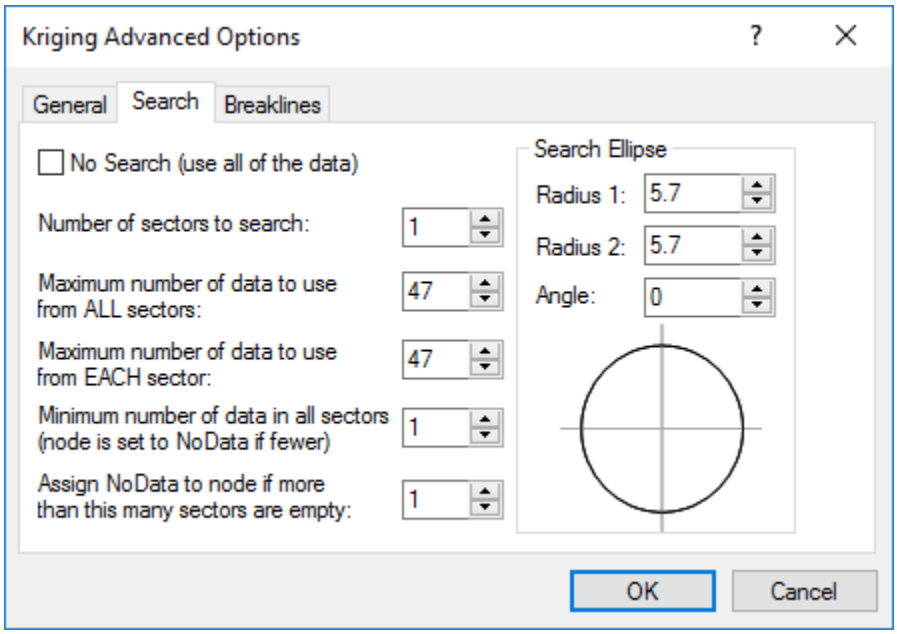
Search Rules

Search rules define the number of points included in interpolating a grid node value. Uncheck the *No Search (use all of the 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 *Minimum number of 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.

The Search Page

To access the **Search** page, if available with the gridding method, click the *Advanced Options* button in the Grid Data dialog.



*Specify the search options, if available, for the selected gridding method on the **Search** page.*

No Search

The No Search (use all of the data) option tells **Surfer** to use all data when interpolating each grid node. Clear the *No Search* 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 *Maximum number of 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 *Maximum number of data to use from EACH sector* value specifies the number of points to be used from each sector.

Minimum Number of Data in All Sectors (node is set to NoData if fewer)

The *Minimum number of data in all sectors (node is 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. 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.

Assign NoData to Node if More Than This Many Sectors are Empty

Assign NoData to node if more than this many sectors are empty assures that if more empty sectors than this 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*. The default values are calculated by taking half the distance of the diagonal.

Range 1 and 2

Some gridding methods have a *Range 1* and *Range 2* value instead of *Radius 1* and *Radius 2*. In these gridding methods, the *Range 1* and *Range 2* values perform a similar function to *Radius 1* and *Radius 2*. The default value is calculated similar to the method used for *Radius 1* and *Radius 2*.

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.

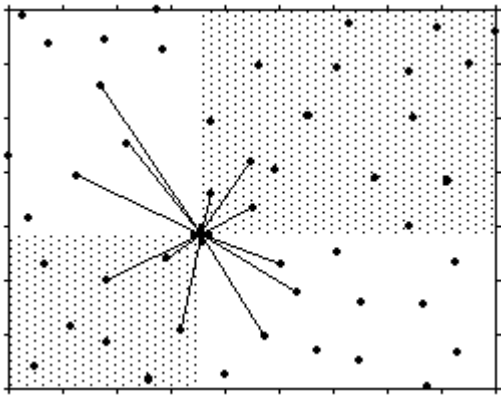
No Search

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

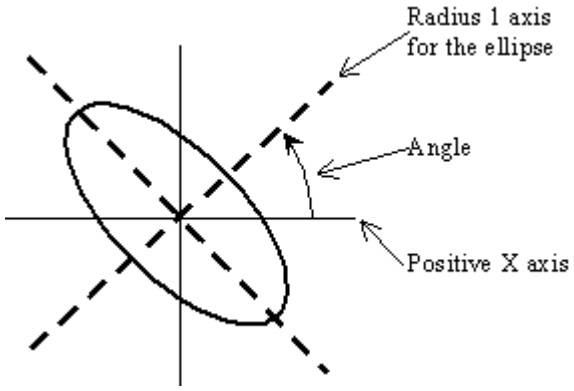
Radius 1 and *Radius 2* are positive values indicating the distance in data units. The *Radius 1* and *Radius 2* values are the length that is searched in the direction indicated by the *Angle*. The default values are calculated by taking half the distance of the diagonal.

Range 1 and 2

Some gridding methods have a *Range 1* and *Range 2* value instead of *Radius 1* and *Radius 2*. In these gridding methods, the *Range 1* and *Range 2* values perform a similar function to *Radius 1* and *Radius 2*. The default value is calculated similar to the method used for *Radius 1* and *Radius 2*.

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

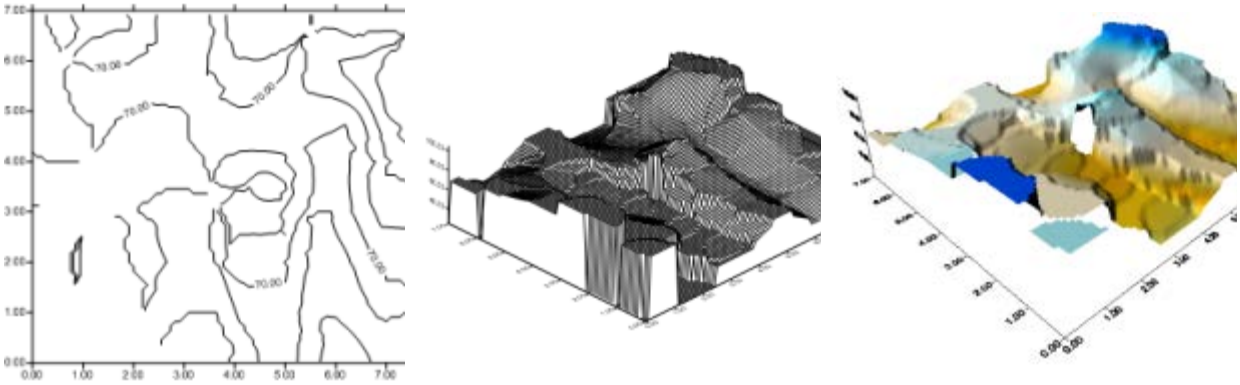
Radius 1 = 1

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.

Anisotropy

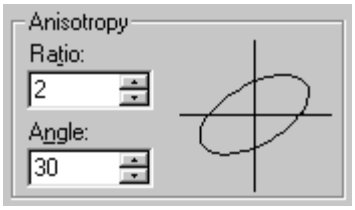
Natural phenomena are created by physical processes. Often these physical processes have preferred orientations. For example, at the mouth of a river the coarse material settles out fastest, while the finer material takes longer to settle. Thus, the closer one is to the shoreline the coarser the sediments 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.

For each different gridding method, anisotropy might be specified in a slightly different manner. In general, to set the anisotropy properties, click the Grids | New Grid | Grid Data command. Select the data file and click *Open*. In the **Grid Data** dialog, click the *Advanced Options* button.

The anisotropy settings include a *Ratio* and an *Angle* setting.



The anisotropy settings have an axis ratio and angle.

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

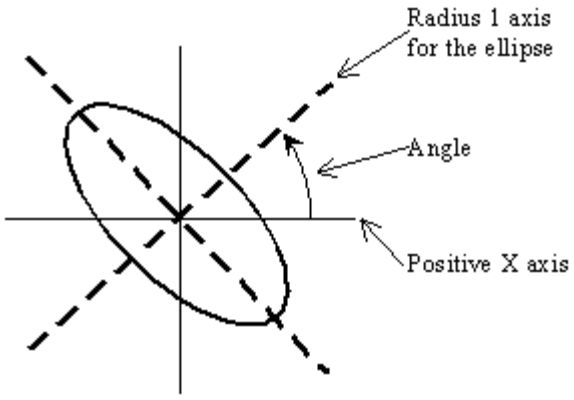
Angle = 0

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.

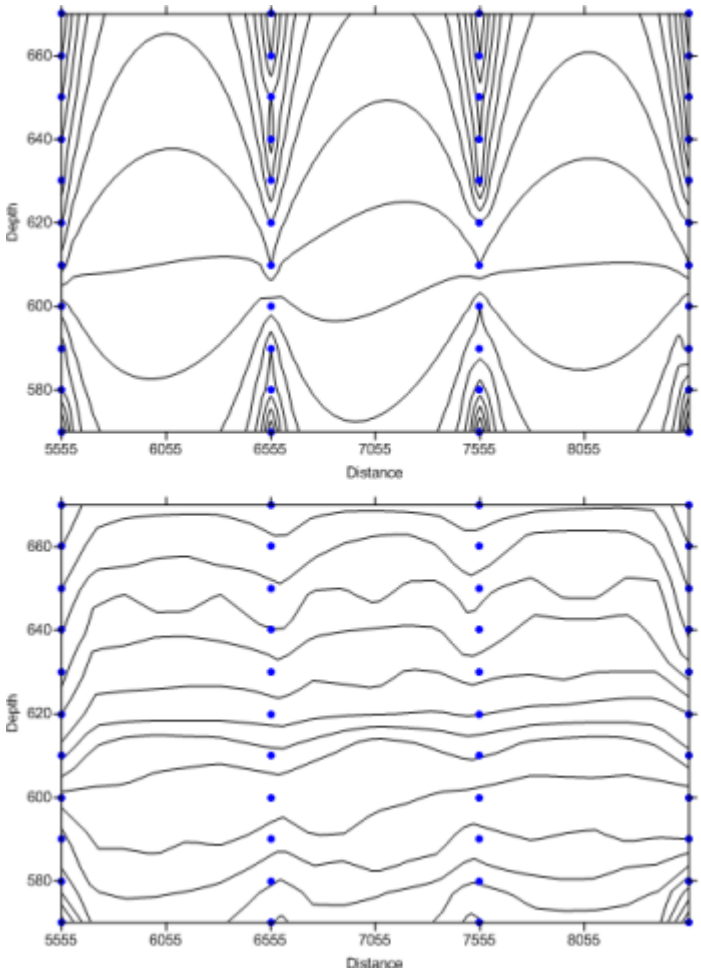
Example 2: Isotherm Map of Average Daily Temperature

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.



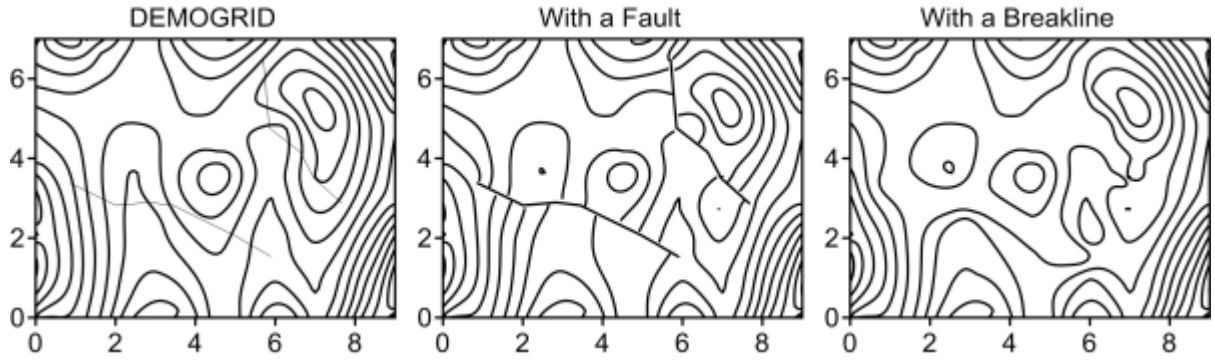
In the oceanographic survey described here, the contour lines cluster around the data points when an anisotropy ratio is not employed. In the lower contour map, an anisotropy ratio results in contour lines that are a more accurate representation of the data.

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 click the **Breaklines and Faults** tab. 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:

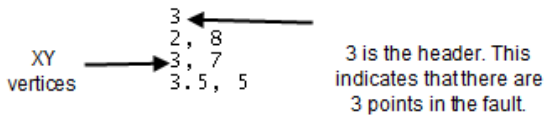
- Inverse Distance to a Power
- Minimum Curvature
- Nearest Neighbor
- Data Metrics

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

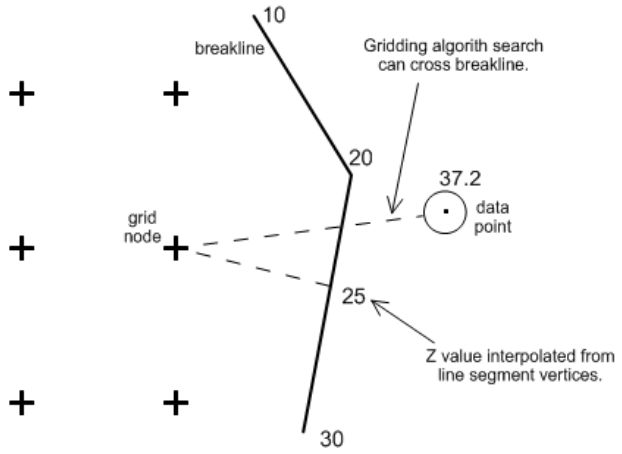
An example of a fault .BLN format is:



A fault is defined by X and Y values in a .BLN file.

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.

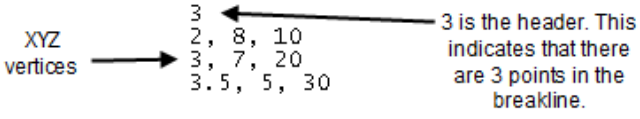
The following gridding methods support breaklines:

- Inverse Distance to a Power
- Kriging
- Minimum Curvature
- Nearest Neighbor
- Radial Basis Function
- Moving Average
- Data Metrics
- Local Polynomial

You can create a blanking file to define a breakline in the **Surfer** worksheet or any text editor. Enter a header containing the number of vertices in the breakline, followed by the (X, Y, Z) coordinates of each vertex, one per line. These blanking files do not require the assign NoData inside (1) or outside (0) flag. The blanking flag is ignored in a breakline file.

Breakline Example

An example of a breakline .BLN format is:



A breakline file contains X, Y, and Z values in the .BLN file.

Specify Advanced Gridding Options

Advanced gridding options can be specified in the Grid Data dialog. To specify advanced gridding options:

1. Select the Grids | New Grid | Grid Data command.
2. In the **Open** dialog, select a data file and then click the *Open* button.
3. Set the *Gridding Method* to the desired method.
4. Click the *Advanced Options* button.
5. In the dialog that appears, set any advanced options. See below for some information about the options for each gridding method. After setting the options, click *OK*.
6. Click *OK* in the **Grid Data** dialog and the grid is created.

For specific advanced options for each gridding method, refer to the gridding method for more information.

Inverse Distance to a Power

Kriging

Minimum Curvature

Modified Shepard's Method

Natural Neighbor

Nearest Neighbor

Polynomial Regression

Radial Basis Function

Triangulation with Linear Interpolation

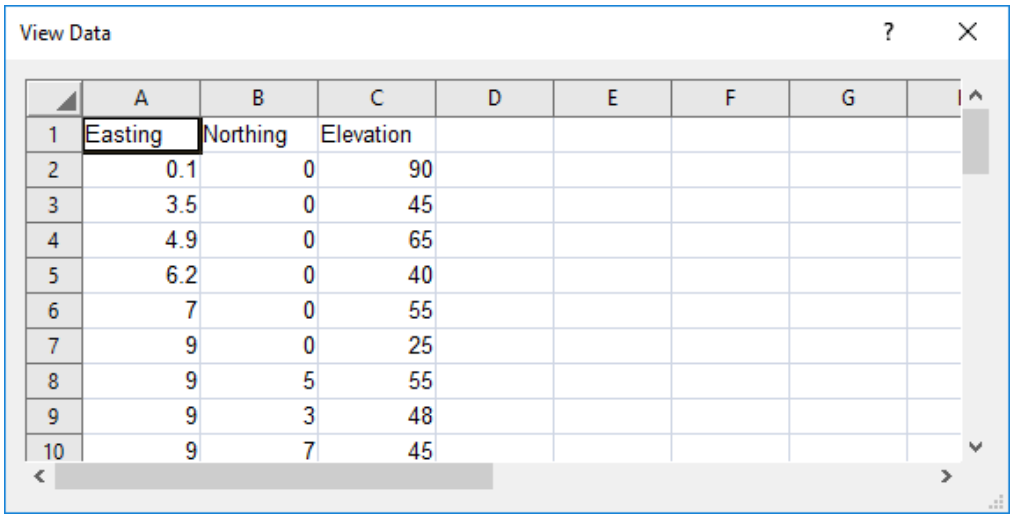
Moving Average

Data Metrics

Local Polynomial

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** dialog to 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.*

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:

- Inverse Distance to a Power
- Kriging
- 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 Power* does 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.

- 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.

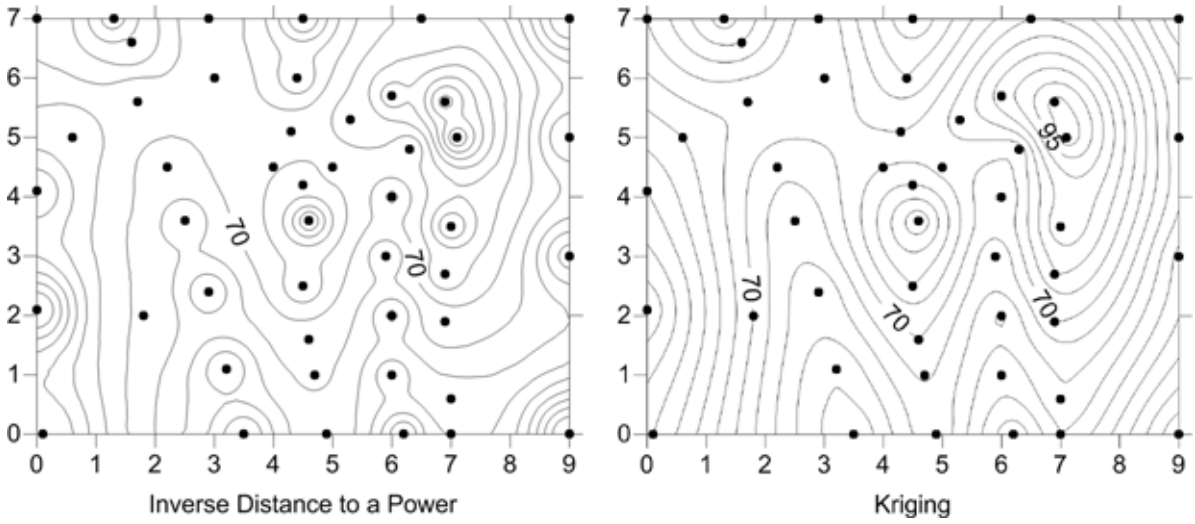
- Ten or fewer points are not enough to define more than a general trend in your data. Triangulation with Linear Interpolation and Moving Average are not effective with few points. As with most data sets, Kriging and Radial Basis Function methods will produce the best representation of your data in this 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.

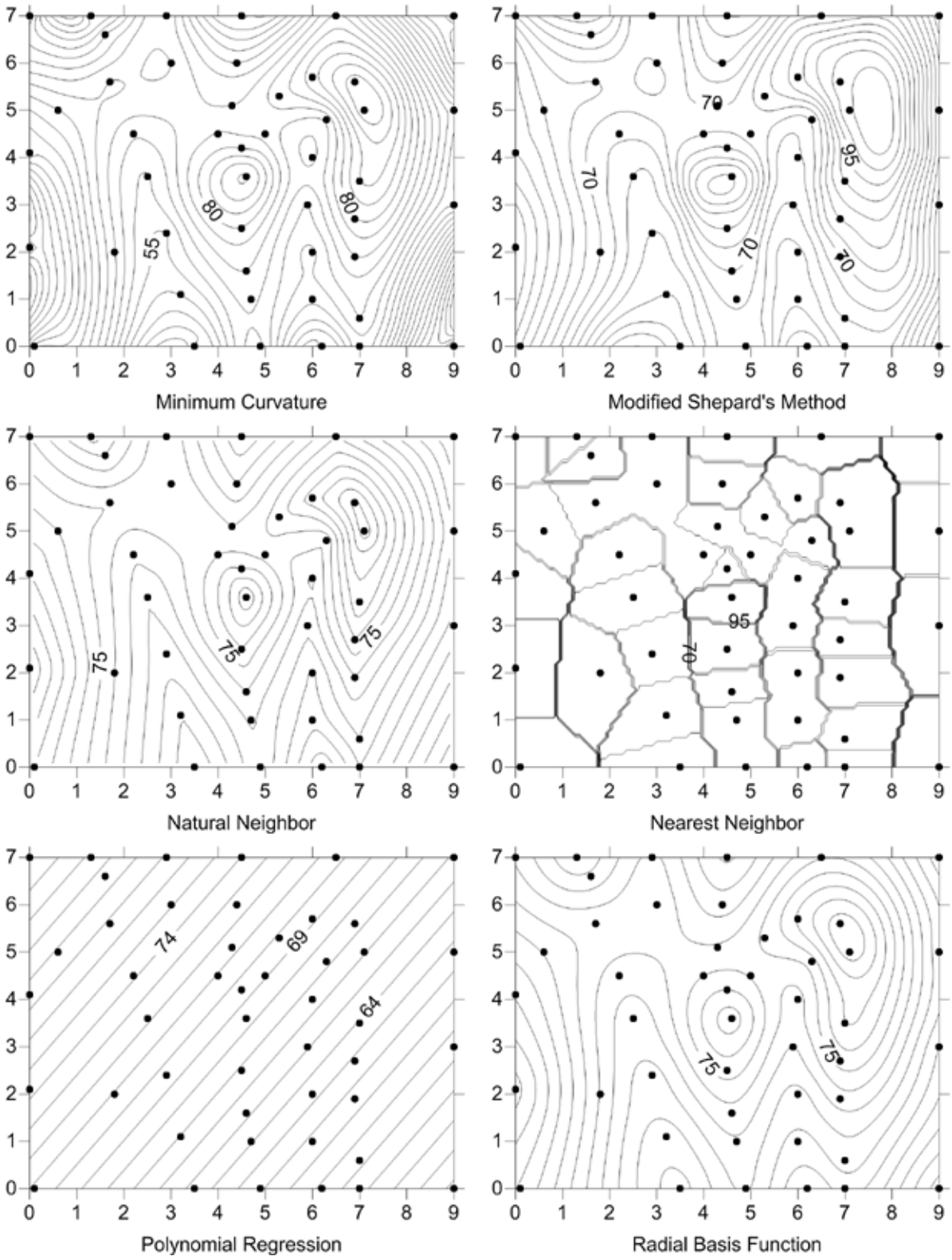
- With moderate-sized data sets (from 250 to 1000 observations), Triangulation with Linear Interpolation is fast and creates a good representation of your data. Although Kriging or Radial Basis Function generate the grids more slowly, they also produce good data representations.
- For large data sets (>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.

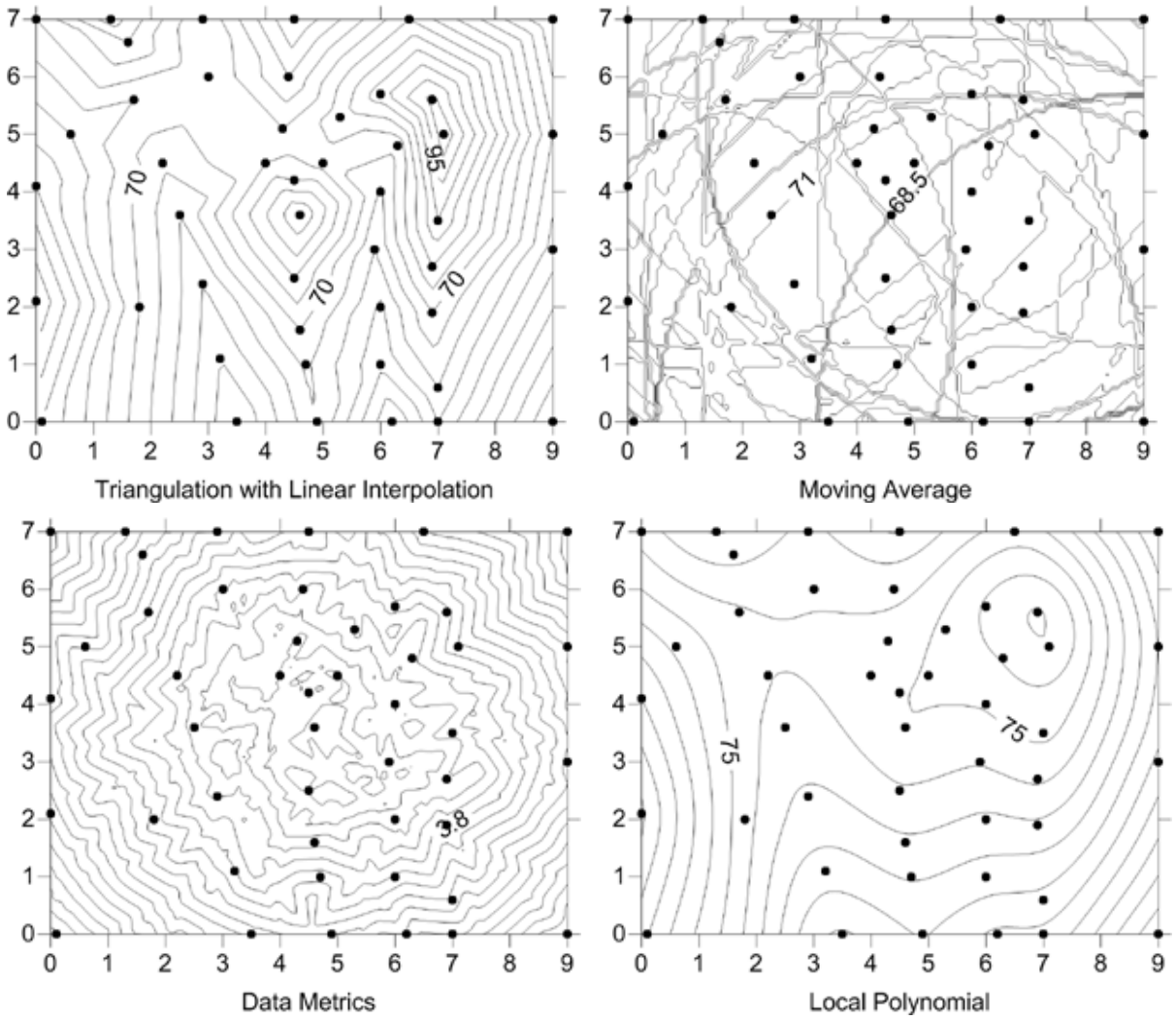
Gridding Method Comparison

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.




Minimum Curvature
Modified Shepard's Method
Natural Neighbor
Nearest Neighbor
Polynomial Regression
Radial Basis Function



This is a comparison of the 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.

The Results of Each Gridding Method with *Demogrid.dat*:

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

G_j = interpolated grid node value at node *j*;

N = number of points used to interpolate at each node;

Z_i Z value at the i^{th} point

w_{ij} weight associated with the i^{th} data value when computing G_j

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.

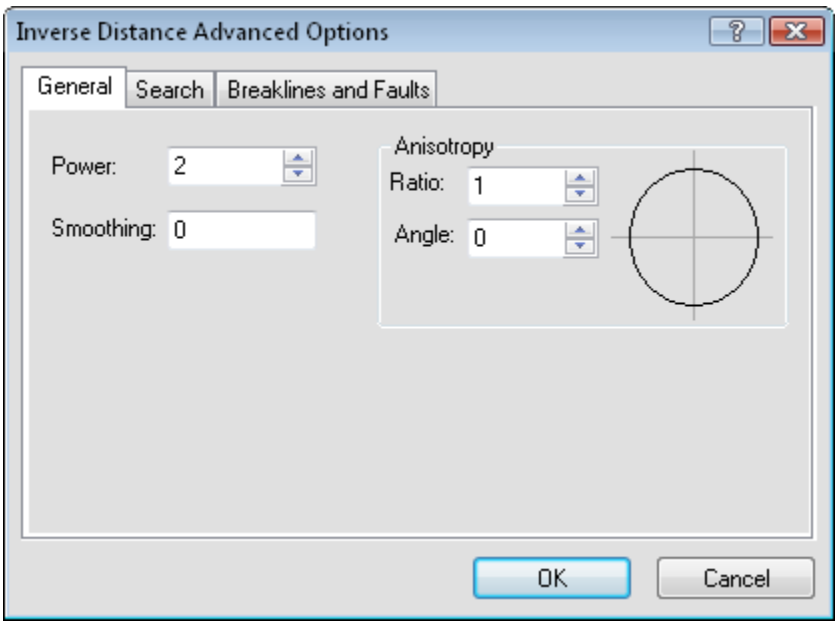
Normally, *Inverse Distance to a Power* behaves as an exact interpolator. When calculating a grid node, the weights assigned to the data points are fractions, and the sum of all the weights are equal to 1.0. When a particular observation is coincident with a grid node, the distance between that observation 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 during *Inverse Distance to a Power* to 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 Advanced Options Dialog

In the Grid Data dialog, specify *Inverse Distance to a Power* as the *Gridding Method* and click the *Advanced Options* to open the **Inverse Distance Advanced Options** dialog.



*The **General** page in the **Inverse Distance Advanced Options** dialog allows you to specify the parameters to use when gridding with the inverse distance to a power gridding method.*

General Page

Specify the *Power, Smoothing,* and *Anisotropy* on the **General** page.

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.

Smoothing

The *Smoothing* 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.

Anisotropy

Set *Anisotropy* settings if needed. For more information about anisotropy options, see Anisotropy.

Search Page

Specify *search rules*. For more information about search rules, see Search.

Breaklines and Faults Page

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:

h_{ij}	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";
Z_i	are the neighboring points;
d_{ij}	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.

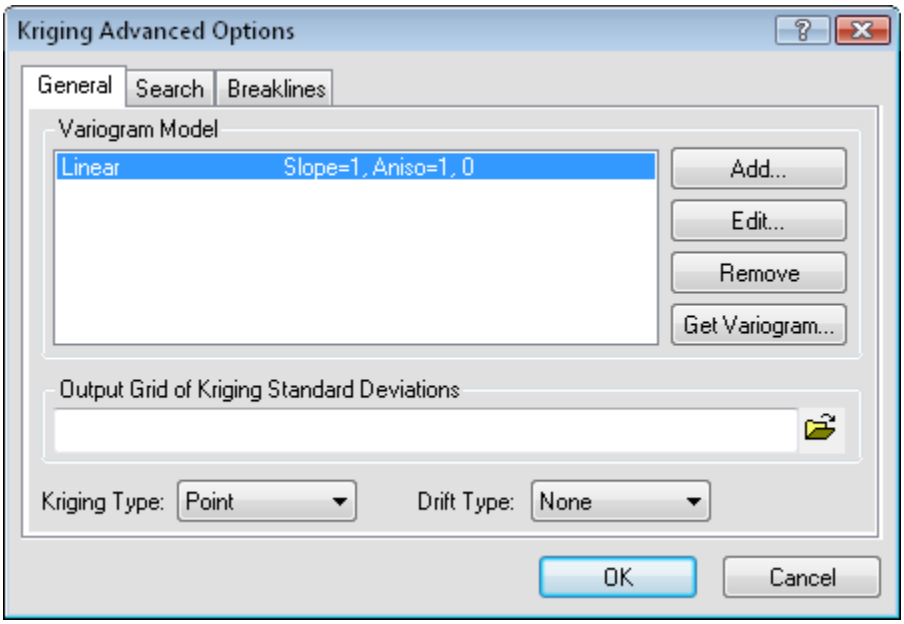
Kriging

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 Advanced Options Dialog

In the Grid Data dialog, specify *Kriging* as the *Gridding Method* and click the *Advanced Options* button to open the **Kriging Advanced Options** dialog.



*Specify the variogram parameters, Kriging standard deviations file, Kriging type, and drift type in the **Kriging Advanced Options** dialog.*

Variogram Model

An important component of the Kriging gridding method is the variogram. The default linear variogram provides a reasonable grid in most circumstances, as long as a nugget effect is not used. A variogram-modeling feature is included in **Surfer** to assist you in selecting the appropriate variogram model for your data. Do not change the variogram components without modeling the variogram first. If you do not understand variograms or variogram modeling, use the default linear variogram with no nugget effect. When in doubt, use the default linear variogram.

Add

Click the *Add* button to open the Variogram Components dialog where you can add additional variogram components. See variograms for more information on the variogram components.

Edit

Click the *Edit* button to display the Variogram Components dialog for the currently selected component. You can edit various parts of the variogram model from this dialog. Once you are finished editing the components, click *OK* to return to the **Kriging Advanced Options** dialog.

Remove

To remove a component from the variogram model, highlight the component in the *Variogram Model* group, and then click the *Remove* button.

Get Variogram

If you have modeled your data with **Surfer's** variogram modeling feature (Grids | New Grid | Variogram | New Variogram), click the *Get Variogram* button to use the variogram results. The *Get Variogram* button extracts the current variogram model from the variogram modeling subsystem and inserts it into the *Kriging* variogram model. You must have the variogram open in the current plot window to use the *Get Variogram* option.

Output Grid of Kriging Standard Deviations

Click the button to enter a file name into the *Output Grid of Kriging Standard Deviations* box to produce an estimation standard deviation grid. If this box is empty, then the estimation standard deviation grid is not created.

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 is not allowed with the *No Search* option on the Search page.

Kriging Type

Choose *Point* or *Block* Kriging from the *Kriging Type* list box.

Surfer includes two *Kriging* types: *Point* Kriging (which is the only method supported by **Surfer 6**), and *Block* Kriging. A detailed discussion of the two methods can be found in Isaaks and Srivastava (1989, Chapters 12 and 13). *Ordinary* (no drift) and *Universal Kriging* (linear or quadratic drift) algorithms can be applied to both *Kriging* types.

Isaaks, E. H., and Srivastava, R. M. (1989), *An Introduction to Applied Geostatistics*, Oxford University Press, New York, 561 pp.

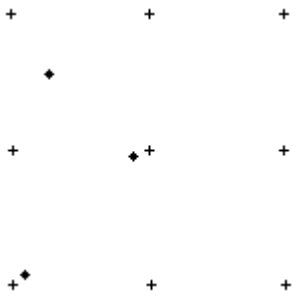
Both *Point* Kriging and *Block* Kriging generate an interpolated grid. *Point* Kriging estimates the values of the points at the grid nodes. *Block* Kriging estimates the average value of the rectangular blocks centered on the grid nodes. The blocks are the size and shape of a grid cell. Since *Block* 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 Abramowitz and Stegun (1972).

Abramowitz, M., and Stegun, I. (1972), *Handbook of Mathematical Functions*, Dover Publications, New York.

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.

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.

Search Page

The Search page allows you to specify search rules.

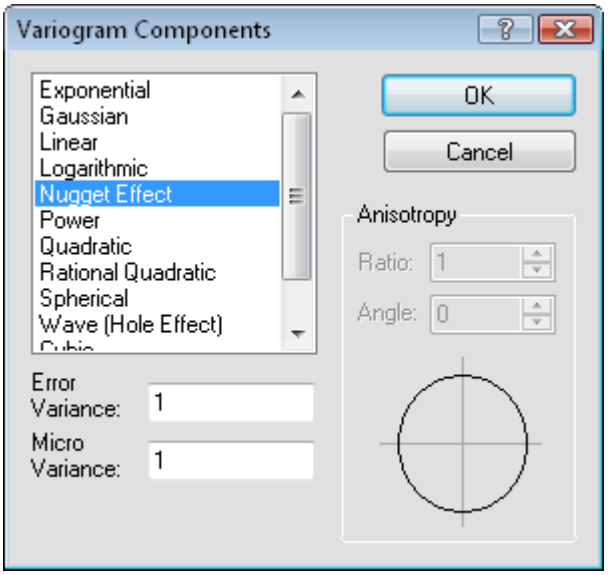
Breaklines Page

The Breaklines page is used to add breaklines to the gridding process. Faults are not supported with *Kriging*.

Variogram Components

Click the *Add* button in the Kriging Advanced 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, anisotropy=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.

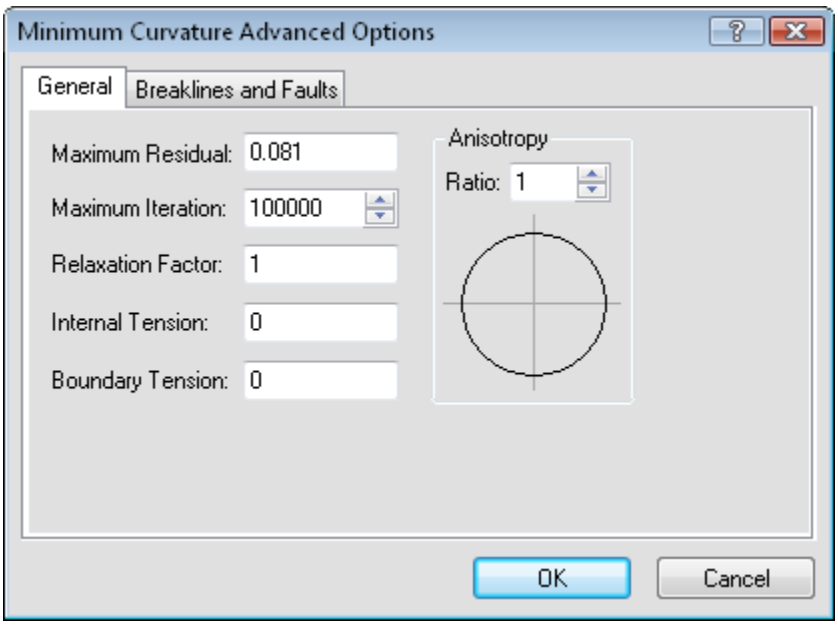
Minimum Curvature

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 Advanced Options Dialog

In the Grid Data dialog, specify *Minimum Curvature* as the *Gridding Method* and click the *Advanced Options* button to open the **Minimum Curvature Advanced Options** dialog.



*Click the Advanced Options button in the **Grid Data** dialog to set the minimum curvature options.*

General Page

You can control the convergence criteria for *Minimum Curvature* on the **General** page:

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

You can set *Anisotropy* parameters in this dialog. For more information about anisotropy options see Anisotropy.

Breaklines and Faults Page

The Breaklines and Faults page is used to add breaklines and faults to the gridding process.

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: $$(1 - T_b)\frac{\partial^2 Z}{\partial n^2} + (T_b)\frac{\partial Z}{\partial n} = 0$$

On the edges: $$\frac{\partial(\nabla^2 Z)}{\partial n} = 0$$

At the corners: $$\frac{\partial^2 Z}{\partial x \partial y} = 0$$

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., Flannery, B.P., Teukolsky, S.A., and Vetterling, W.T. (1988), *Numerical Recipes in C*, Cambridge University Press.

Smith, W. H. F., and Wessel, P. (1990), Gridding with Continuous Curvature Splines in Tension, *Geophysics*, v. 55, n. 3, p. 293-305.

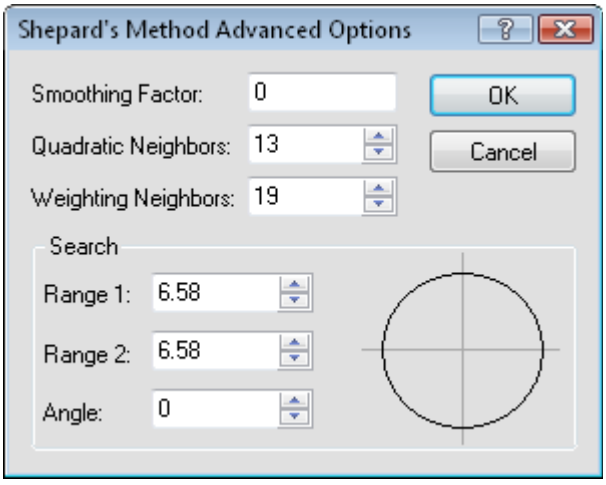
Modified Shepard's Method

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.)

Shepard's Method Advanced 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 **Shepard's Method Advanced Options** dialog.



*Set the options, including search, and smoothing factor, in the **Shepard's Method Advanced 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.

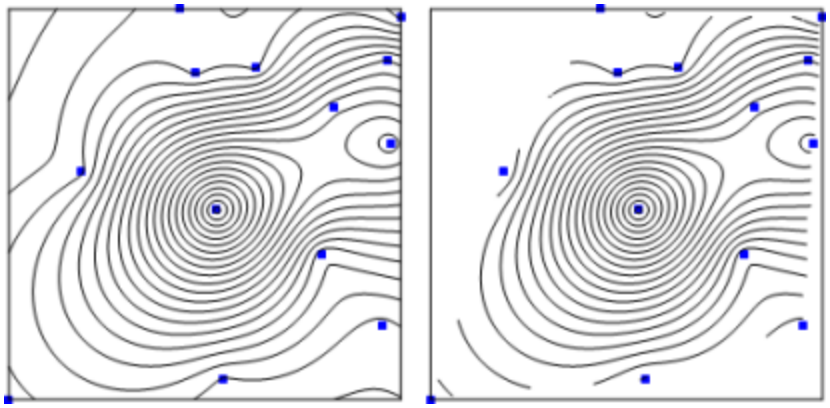
Renka, R. J. (1988), Multivariate Interpolation of Large Sets of Scattered Data, *ACM Transaction on Mathematical Software*, v. 14, n. 2, p. 139-148.

Shepard, D. (1968), A two dimensional interpolation function for irregularly spaced data, *Proc. 23rd Nat. Conf. ACM*, 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."

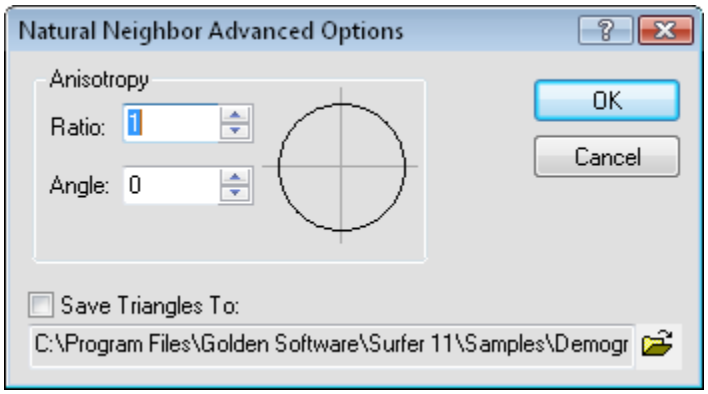
The *Natural Neighbor* method does not extrapolate contours beyond the convex hull of the data locations (i.e. the outline of the Thiessen polygons).



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 **Natural Neighbor Advanced Options** dialog.*

Anisotropy

For more information about anisotropy options see Anisotropy.

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 **Natural Neighbor Advanced Options** dialog.

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, though **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

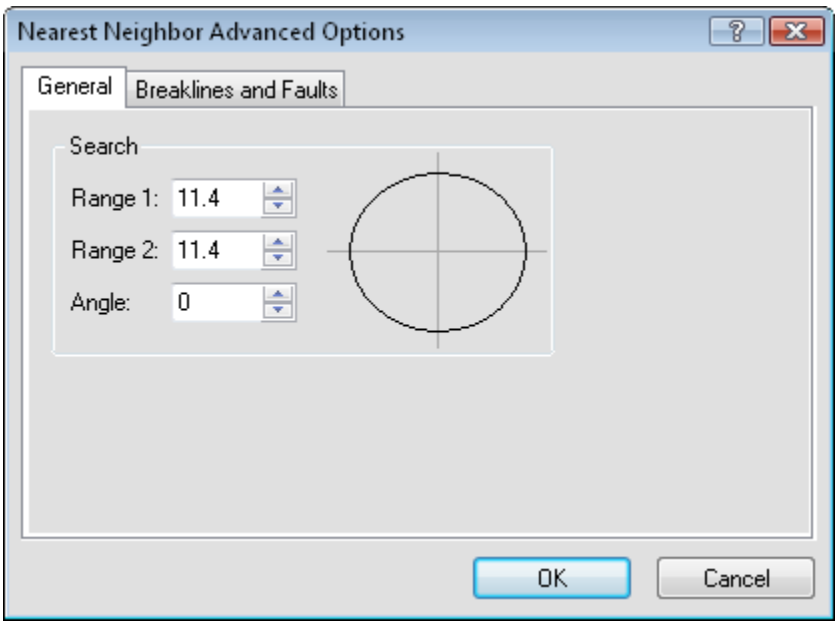
The *Nearest Neighbor* gridding method assigns the value of the nearest point to each grid node. This method is useful when data are already evenly spaced, but need to be converted to a **Surfer** grid file. Alternatively, in cases where the data are nearly on a grid with only a few missing values, this 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 Advanced Options Dialog

In the Grid Data dialog, specify *Nearest Neighbor* as the *Gridding Method* and click the *Advanced Options* button to open the **Nearest Neighbor Advanced Options** dialog.



*Set the search parameters on the **General** page in the **Nearest Neighbor Advanced Options** dialog.*

General Page

On the **General** page, set the search range and angle. For more information about search options, see Search.

Breaklines and Faults Page

The Breaklines and Faults page is used to add *breaklines* and *faults* to the gridding process.

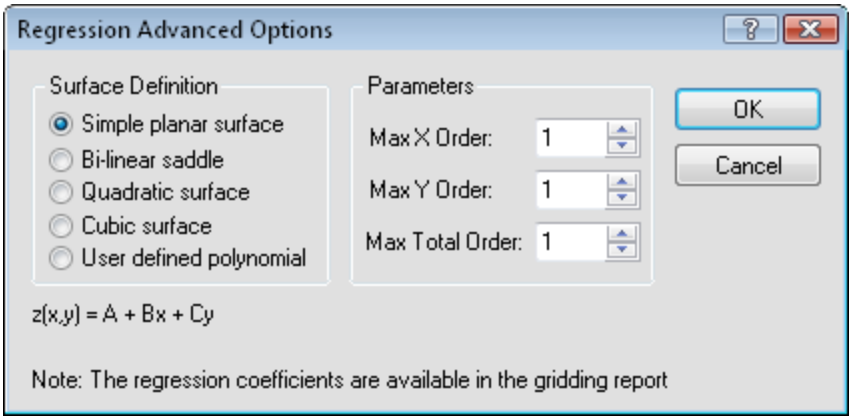
Polynomial Regression

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.

Regression Advanced Options Dialog

In the Grid Data dialog, specify *Polynomial Regression* as the *Gridding Method* and click the *Advanced Options* button to open the **Regression Advanced Options** dialog.



Specify the Surface Definition and Parameters in the ***Regression Advanced 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

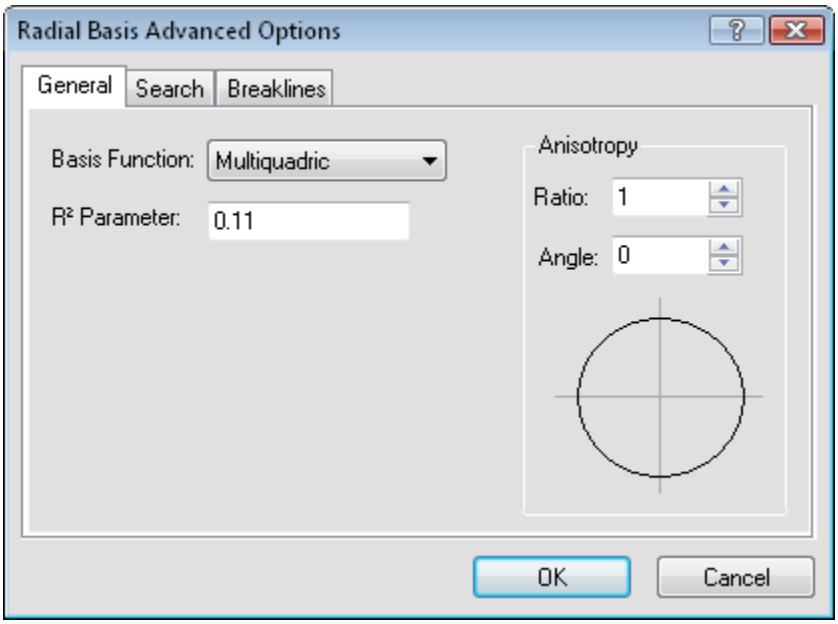
Draper, N., and Smith, H. (1981), *Applied Regression Analysis*, second edition, Wiley-Interscience, 709 pp.

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 Advanced Options Dialog

In the Grid Data dialog, specify *Radial Basis Function* as the *Gridding Method* and click the *Advanced Options* to open the **Radial Basis Advanced Options** dialog.



Set the type of Basis Function and Anisotropy in the Radial Basis Advanced Options dialog.

The General Page

The **General** page controls the *Anisotropy, Basis Function,* and R^2 *Parameter* options.

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 **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)

Anisotropy

You can set *Anisotropy* parameters in this dialog. For more information about anisotropy options see Anisotropy.

Search Page

The Search page allows you to specify search rules.

Breaklines Page

The Breaklines page 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 R^2 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

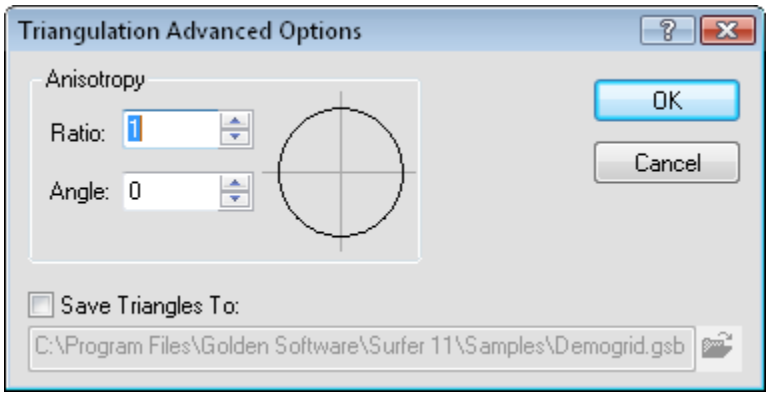
The *Triangulation with Linear Interpolation* method in **Surfer** uses the optimal Delaunay triangulation. The algorithm creates triangles by drawing lines between data points. The original points are connected in such a way that no triangle edges are intersected by other triangles. The result is a 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 Advanced Options Dialog

In the Grid Data dialog, specify *Triangulation with Linear Interpolation* as the *Gridding Method* and click the *Advanced Options* button to open the **Triangulation Advanced Options** dialog.



*Set the anisotropy options and save Delaunay triangles in the **Triangulation Advanced Options** dialog.*

Anisotropy

For more information about anisotropy options see Anisotropy.

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 **Triangulation Advanced Options** dialog.

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

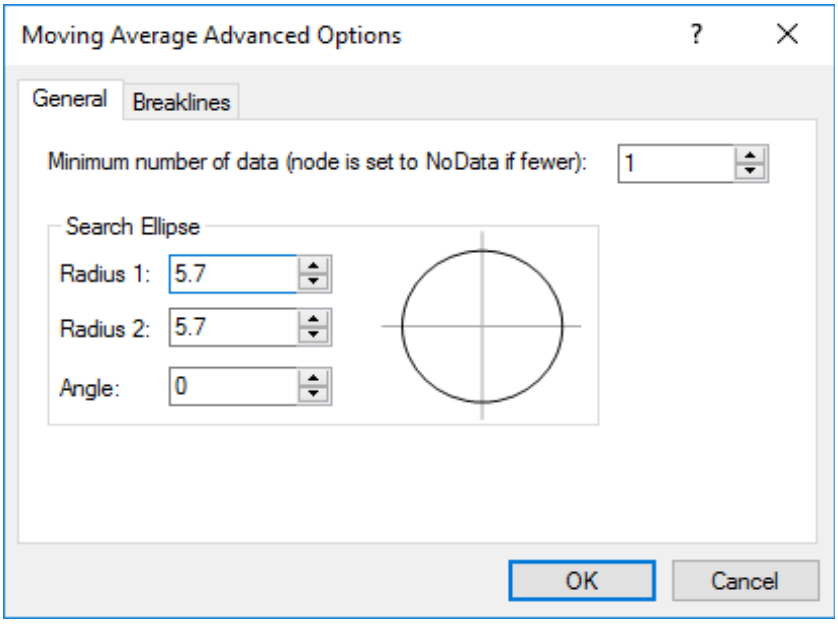
The *Moving Average* gridding method assigns values to grid nodes by averaging the data within the grid node's search ellipse.

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 Advanced Options Dialog

In the Grid Data dialog, specify *Moving Average* as the *Gridding Method* and click the *Advanced Options* button to open the **Moving Average Advanced Options** dialog.



*Set the NoData options and the search ellipse in the **Moving Average Advanced Options** dialog.*

General Page

Specify the minimum number of data options and search ellipse on the **General** page.

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 Ellipse

To use *Moving Average*, define a search ellipse and specify the minimum number of data to use. For each grid node, the neighboring data are identified by centering the search ellipse on the node. The output grid node value is set equal to the arithmetic average of the identified neighboring data. If there are fewer than the specified minimum number of data within the neighborhood, the grid node is assigned the NoData value.

Search ellipses are specified by defining the ellipse radii and the angle for the ellipse. *Radius 1* and *Radius 2* are positive values indicating the distance in data units. *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.

Breaklines Page

The Breaklines page is used to add breaklines to the gridding process. Faults are not supported with *Moving Average.*

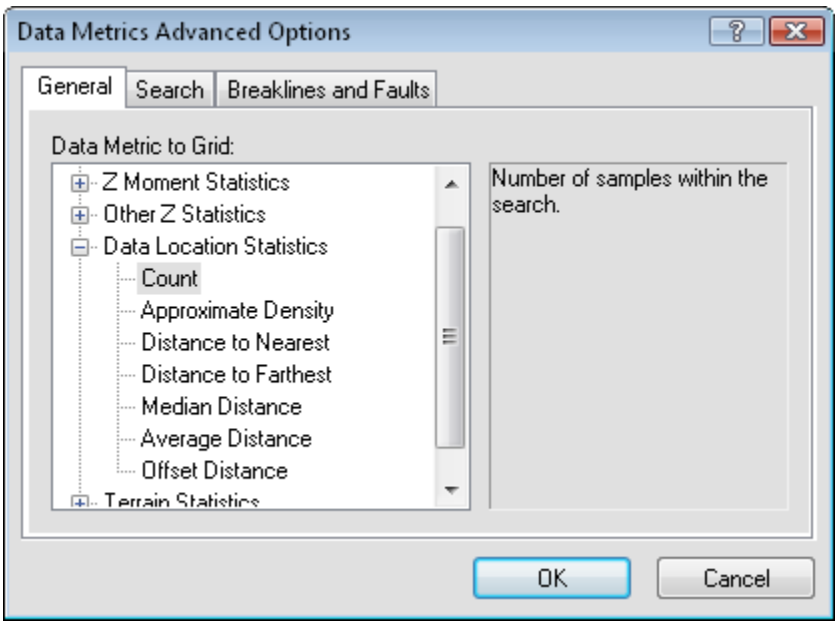
Data Metrics

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 Advanced Options Dialog

In the Grid Data dialog, specify *Data Metrics* as the *Gridding Method* and click the *Advanced Options* to open the **Data Metrics Advanced Options** dialog.



*A description of the metric is listed in the right side of the **General** page when you click on each metric.*

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 Page

The Search page allows you to specify search rules.

Breaklines and Faults Page

The Breaklines and Faults page is used to add *breaklines* and *faults* to the gridding process.

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.

$$\{z_{[1]}, z_{[2]}, \cdots, z_{[n]}\}$$

where $z_{[1]} \le z_{[2]} \le \cdots \le z_{[n]}$. 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 $d_i = \left\lvert z_i - z_{\left[\frac{n}{2}\right]} \right\rvert$ 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 over-stated.	$Z(r,c) = \frac{n}{\pi R_{[n]}^2}$
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.

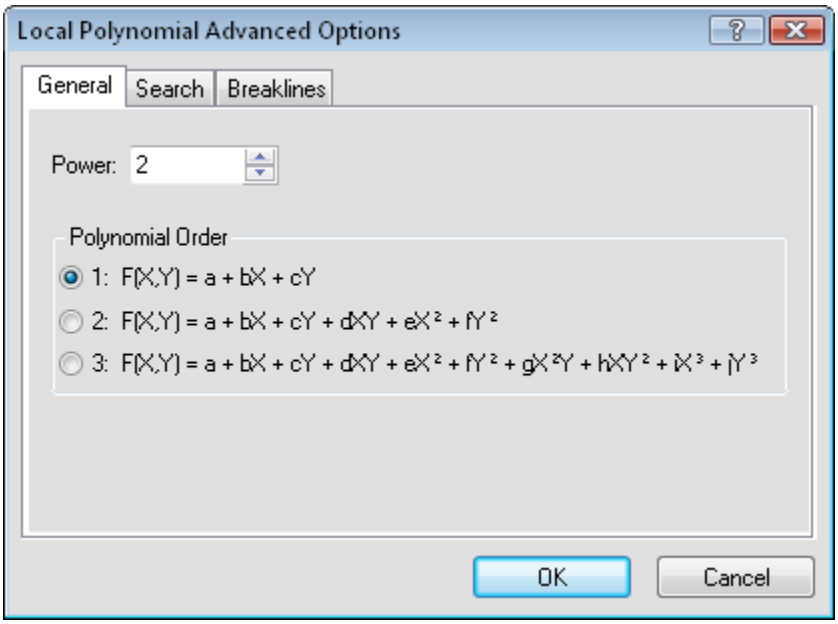
Data Metric	Definition	Equation
Terrain Slope	each nodal value is the terrain slope of the least-squares-fit plane of the data selected by the specified sector search centered at that node	$Z(r,c) = RadiansToDegrees(atan(hypot(A,B)))$
Terrain Aspect	each nodal value is the terrain aspect, as an angle 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 Advanced Options Dialog

In the Grid Data dialog, specify *Local Polynomial* as the *Gridding Method* and click the *Advanced Options* button to open the **Local Polynomial Advanced Options** dialog.



*Choose a Power and Polynomial Order in the **Local Polynomial Advanced Options** dialog.*

General Page

On the **General** page, set the *Power* to a number between 0 and 20, then select a *Polynomial Order*, 1, 2, or 3.

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 Page

The Search page allows you to specify search rules.

Breaklines Page

The Breaklines page 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 T_{XX}, T_{XY}, T_{YX}, and T_{YY} by

$$T_{XX} = \frac{\cos(\phi)}{R_1}$$
$$T_{XY} = \frac{\sin(\phi)}{R_1}$$
$$T_{YX} = \frac{-\sin(\phi)}{R_2}$$
$$T_{YY} = \frac{\cos(\phi)}{R_2}$$

where

ϕ	is the angle of the search ellipse
R_1	is search radius 1
R_2	is search radius 2

Define A_{XX}, A_{XY}, and A_{YY} by

$$A_{XX} = T_{XX}^2 + T_{YX}^2$$
$$A_{XY} = 2\left(T_{XX}T_{XY} + T_{YX}T_{YY}\right)$$
$$A_{YY} = T_{YY}^2 + T_{XY}^2$$

Note that these values (A_{XX}, A_{XY}, and A_{YY}) are function of the search ellipse parameters only. They are the same for all data and for all grid nodes.

Next, consider a datum at location (X_i, Y_i) and a grid node at location (X_0, Y_0). Let

$$dX = X_i - X_0$$
$$dY = Y_i - Y_0$$

then

$$R_i = \sqrt{A_{XX}dX^2 + A_{XY}dXdY + A_{YY}dY^2}$$

and finally,

$$W_i = (1 - R_i)^p$$

where W_i is the weight for datum i and *p* is the specified power.

Let the collection of neighboring data be enumerated as

$$\{(x_i, y_i, z_i) \text{ for } i = 1, 2, \ldots, N\}$$

The local least squares parameters are computed by minimizing the weighted sum of the squared residuals:

$$Minimize \sum_{i=1}^{N} W_i [F(x_i, y_i) - z_i]^2$$

Grid from Server

Click the **Grids | New Grid | Grid from Server** command 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.

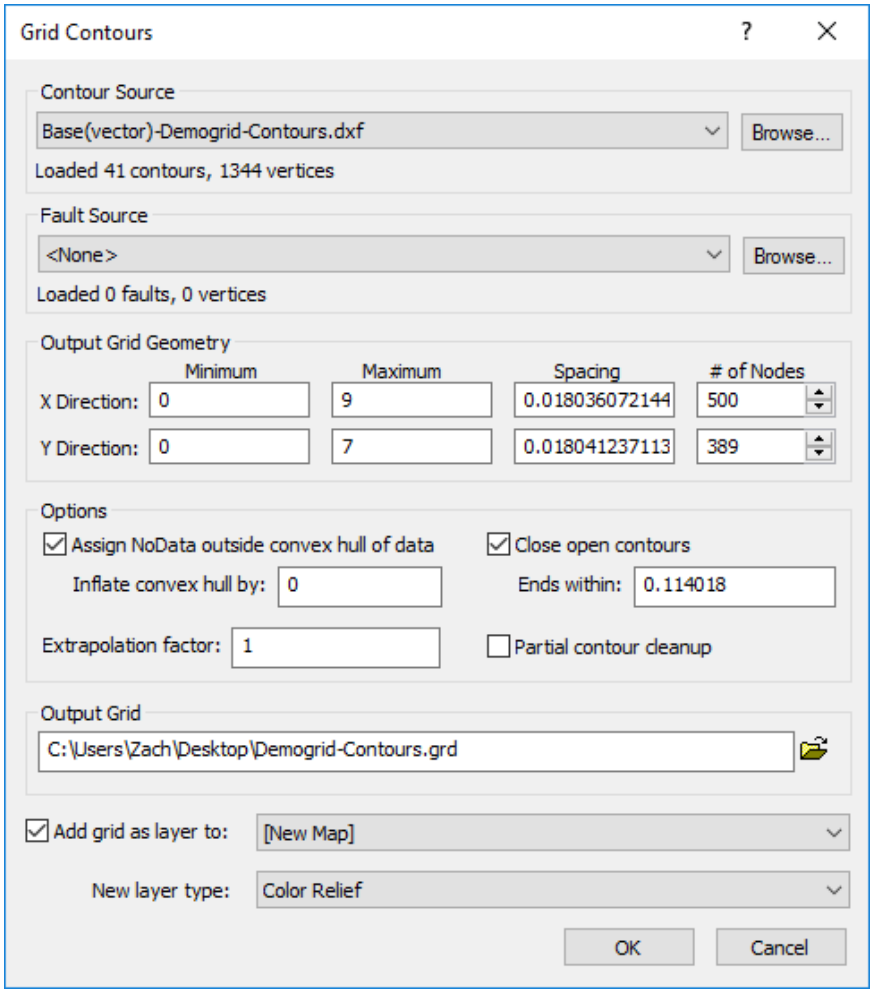
- When selecting a WCS server, data values are downloaded to be used as a grid file. **Surfer** comes with one WCS server by default, under the *Terrain Data* category, called *SRTM Global 90m (GMU)*.
- When selecting a WFS server, vector data is downloaded with points, polylines, and/or polygons representing features. WFS servers cannot be used with the **Grid from Server** command.

Grid from Contours

Click the **Grids | New Grid | Grid from Contours** command 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 or 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.

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.

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, 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 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, control over extrapolation in peaks and basins, contour closing, and contour cleanup.

Convex Hull Options

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. So, 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.

Assign NoData Outside Convex Hull

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

Inflate Convex Hull

The *Inflate convex hull by* option expands or contracts the convex hull. When set to zero, the boundary connects the outside data points exactly. When set to a positive value, the area assigned NoData is moved outside the convex hull boundary by the number of grid units specified. When set to a negative value, the area assigned NoData is moved inside the convex hull boundary by the number of grid units specified. To change the value, highlight the existing value and type the

desired value. Values are in horizontal (X) map units. When the value is set to too high of a positive value, the contours will extend all the way to the minimum and maximum X and Y limits of the grid. When the value is set to too low of a negative value, the entire grid will be assigned NoData, resulting in no grid file being created.

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 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.

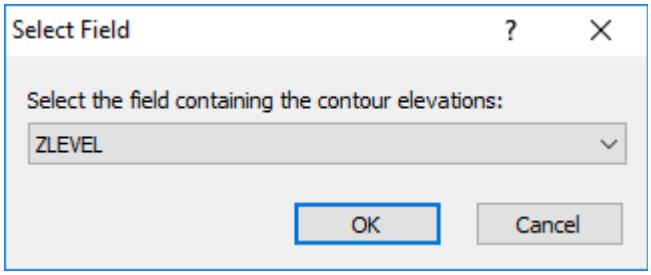
Note: If you are saving the grid file in the DEM grid format, clear the *Add grid as layer* check box and add the map or layer with a **Home | New Map** or **Home | Add to Map | Layer** command.

Grid from Contours References

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.*

Click the current attribute in the *Select the field containing the contour elevations* field and select the desired attribute from the list.

Click *OK* when you have selected the desired field.

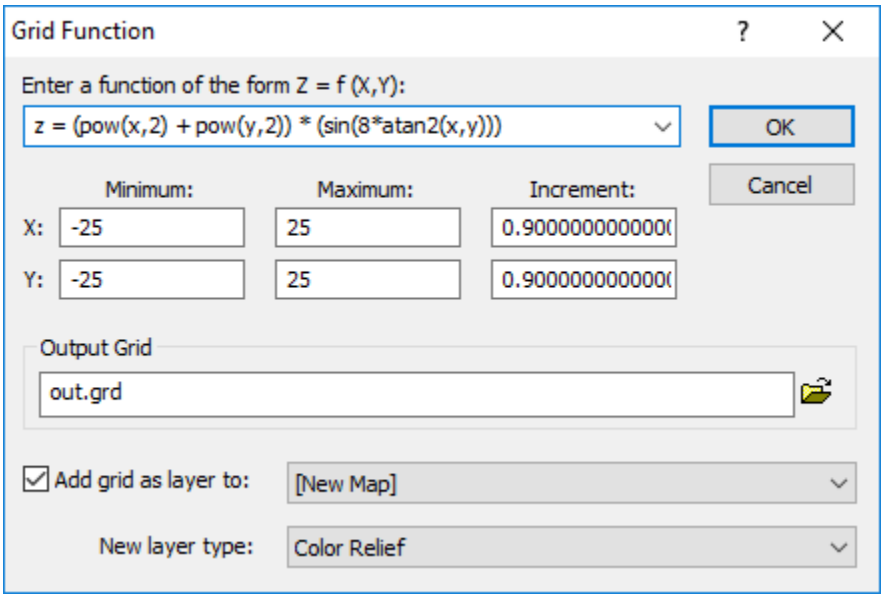
Click *Cancel* to return to the Grid Contours dialog without selecting the elevation field. If you click *Cancel* the map layer or vector file will not be loaded.

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 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

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*.

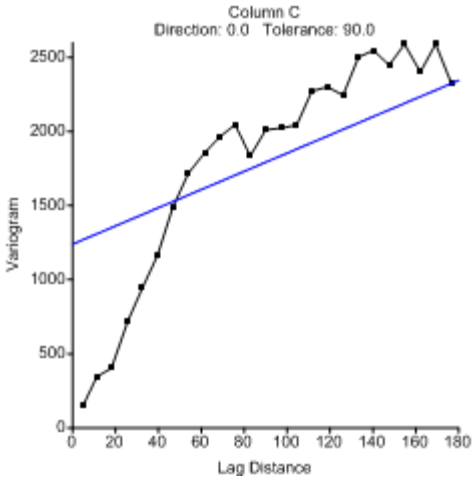
The Variogram

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 $\gamma(\theta,h)$). 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
The 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.

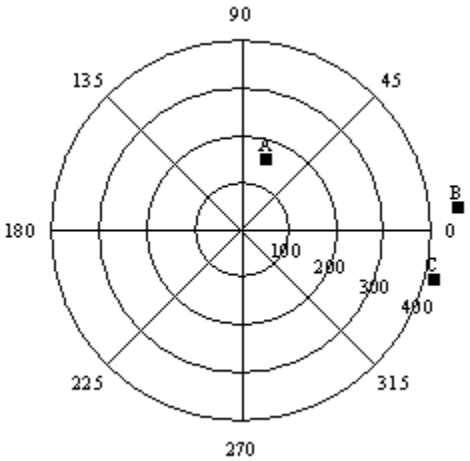
The Variogram Grid

Users familiar with GeoEAS or VarioWin® should be familiar with pair comparison files .PCF. **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*

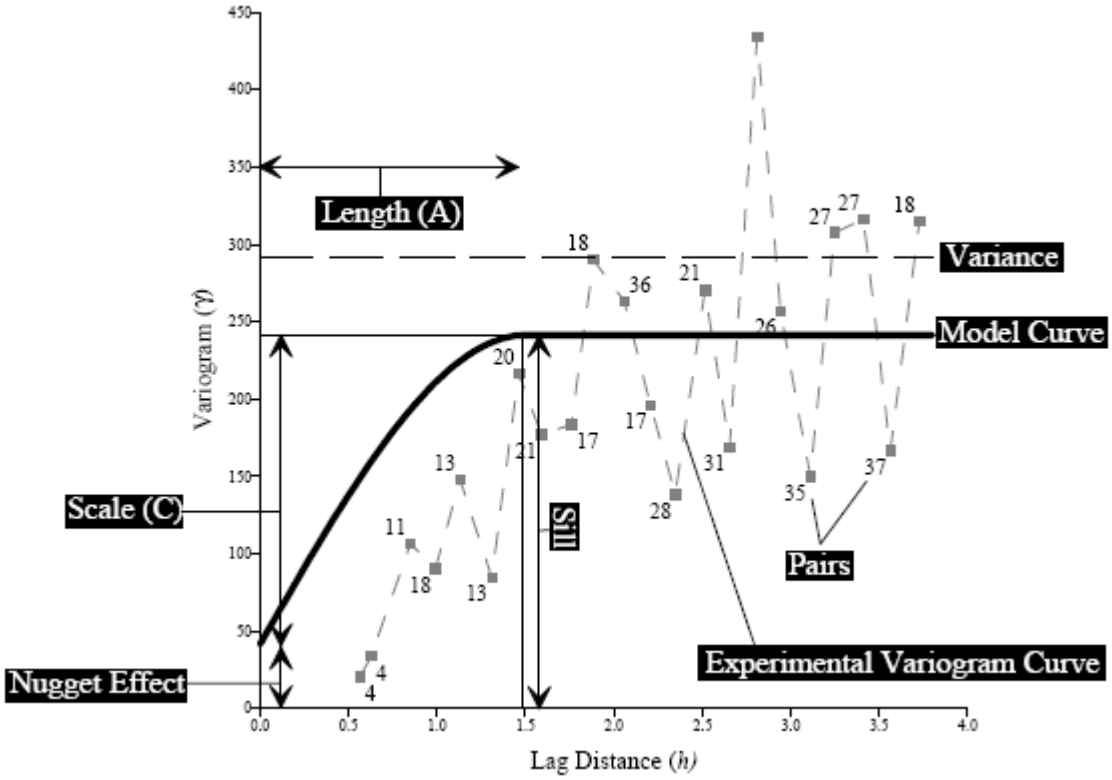
The separation angle and separation distance for each pair are calculated using the previous equations.

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).

Scale

The *Scale (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.

Length

The *Length* is the horizontal range of the variogram. (Some variogram models do not have a length 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.

$$\text{var} = \frac{\sum_{i=1}^{N} z_i^2}{N} - \left(\frac{\sum_{i=1}^{N} z_i}{N} \right)^2$$

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.

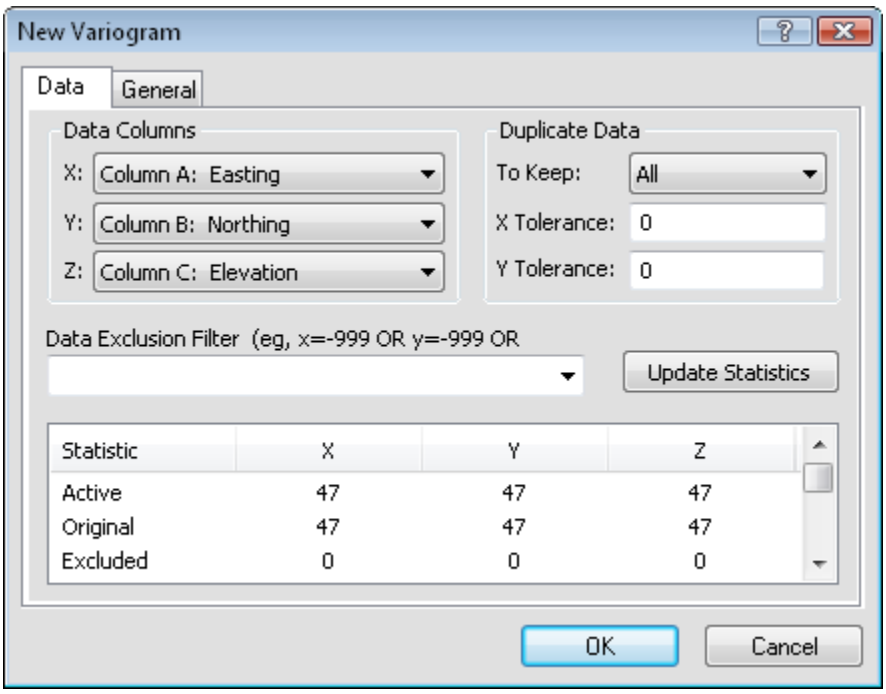
New Variogram

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 The Variogram Grid for more information on variogram grids.

The New Variogram Dialog

Click the **Grids | New Grid | Variogram | New Variogram** command 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.*

Data Page

Select data for a new variogram on the Data 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.

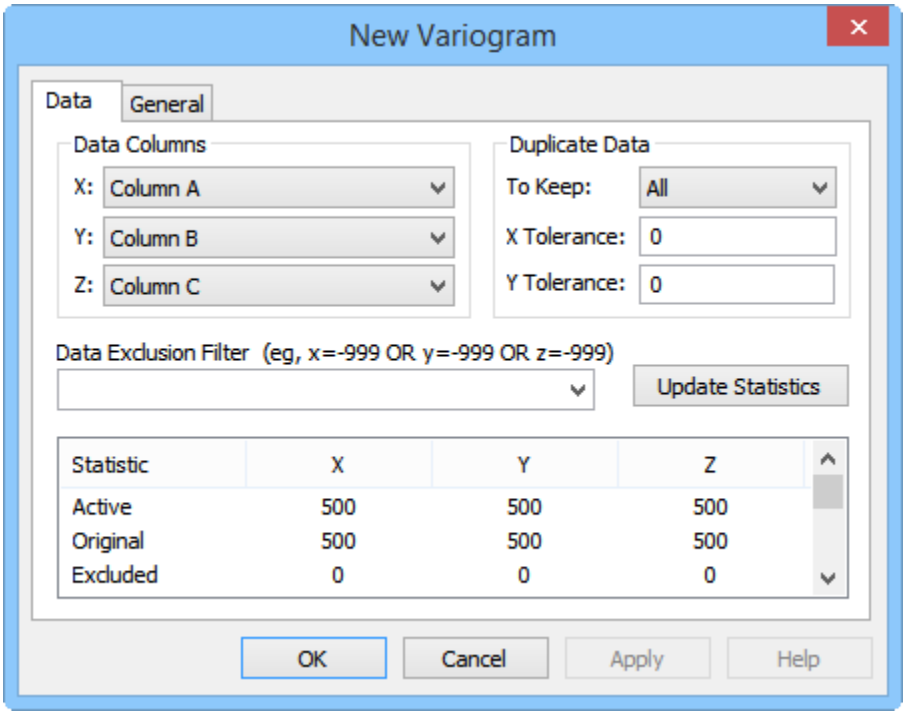
1. Click **Grids | New Grid | Variogram | New Variogram**.
2. Select a data file in the **Open Data** dialog and click *Open*.
3. Set the data columns to use on the Data page in the **New Variogram** dialog.
4. Select the variogram grid settings on the General page.
5. Click *OK* to create the variogram.
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.

Data Page - New Variogram

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.

Data Page

The **Data** 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.*

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.

Statistics

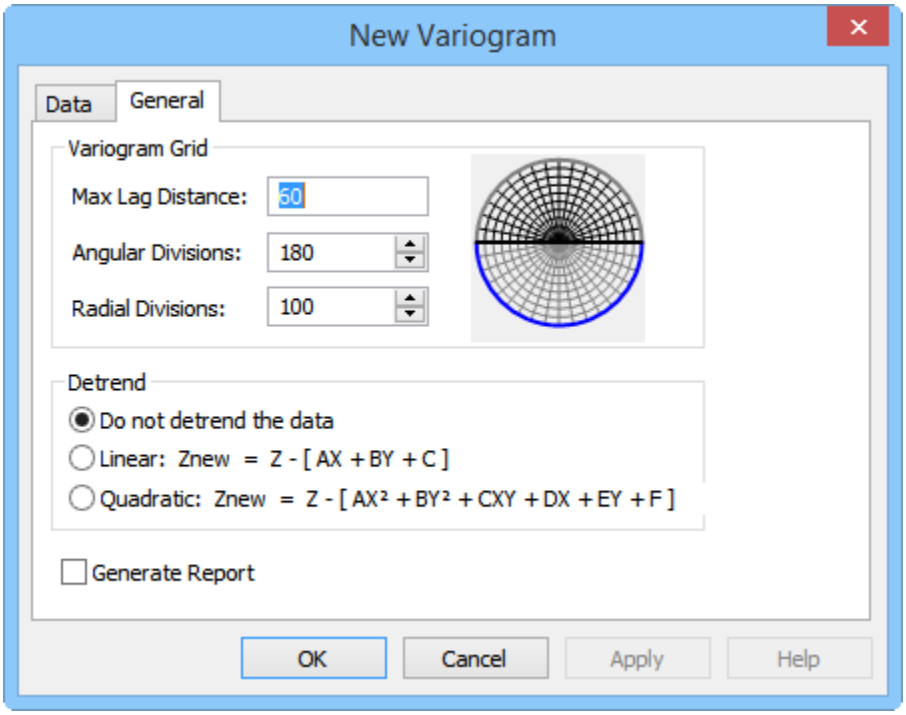
The bottom of the dialog displays statistics about the data. If you change any of the parameters in the dialog, click the *Update Statistics* button to display statistics based on the changed information.

General Page - New Variogram

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.

The 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.

Detrend

The *Detrend* group contains three options for detrending the data before creating a variogram. The *Detrend* 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.

- *Do not detrend the data* should be used most of the time - it should be used all of the time by novices.
- *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 advanced options in the **Grid Data** dialog. You must manually change the **Kriging Advanced Options** dialog *Drift type* to match the variogram *Detrend* type if you wish to use the variogram model to create a grid with the Kriging algorithm.

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

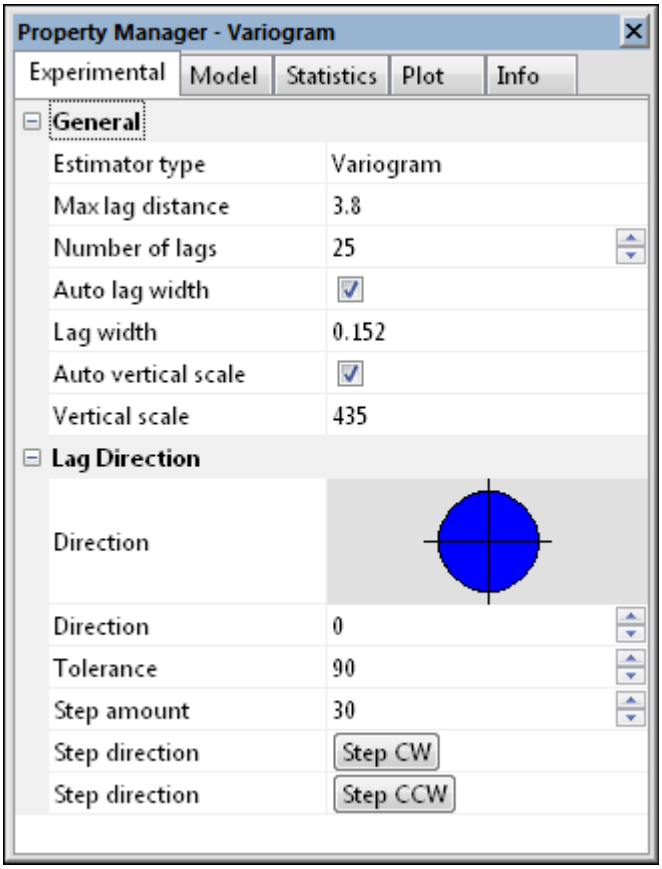
Once the variogram grid has been created through the with Grids | New Grid | Variogram | New Variogram, the properties of the variogram can be changed in the Properties window. To change

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.

The variogram properties contains four pages: **Experimental, Model, Statistics,** and **Plot**.

Experimental Page

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: the *variogram*, *standardized variogram*, *autocovariance*, and *autocorrelation*. When in doubt use the default classical *variogram*.

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 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 the *Lag Direction* group.

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 CW* or *Step CCW* 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 i^{th} pair, and

Zt_i be the observed value of the tail observation of the i^{th} pair.

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:

$$\gamma = \frac{(Zhh - 2Zht + Ztt)}{2N}$$

(see Pannatier, 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.

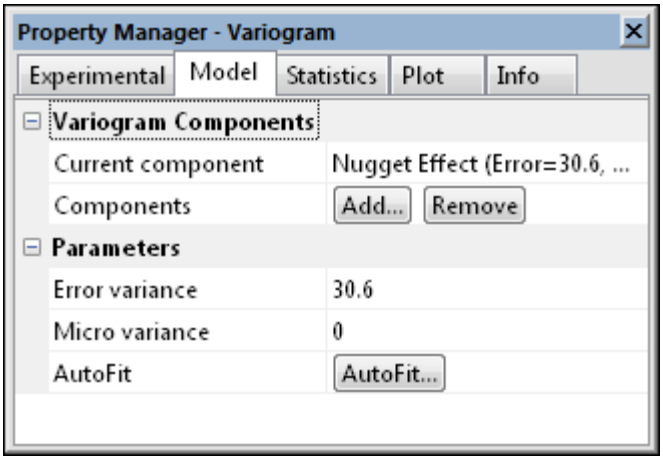
For example, consider the case where the *Max lag distance* is 100, the *Number of Lags* is five, and the *Lag Width* is 40. The resulting lag intervals would be 0 to 40, 10 to 50, 30 to 70, 50 to 90, and 60 to 100. Notice these intervals overlap. Overlapping lag intervals causes a moving average type 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 );
}
```

Model Page

The **Model** page in the **Variogram Properties** dialog 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.

Scale (C)

With the exception of the *Linear, Logarithmic,* and *Power* variogram models (which do not have a sill), the *Scale* 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 *Scale (C)* parameters. In many situations, the variogram model sill is approximately equal to the variance of the observed data (see Barnes, 1991).

Length (A)

The *Length (A)* parameters define how rapidly the variogram components change with increasing separation distance. The *Length (A)* parameter for a variogram component is used to scale the physical separation distance. For the *Spherical* and *Quadratic* variogram functions, the *Length (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

$[\Delta x \quad \Delta y]$ is the separation vector (in map coordinates), and

A is the component's *Length* parameter.

The anisotropically rescaled relative separation distance for the variogram equations is computed by the following matrix equation

$$h = \sqrt{\begin{bmatrix}\Delta x & \Delta y\end{bmatrix}\begin{bmatrix}\cos\theta & -\sin\theta \\ \sin\theta & \cos\theta\end{bmatrix}\begin{bmatrix}(1/A)^2 & 0 \\ 0 & (\rho/A)^2\end{bmatrix}\begin{bmatrix}\cos\theta & \sin\theta \\ -\sin\theta & \cos\theta\end{bmatrix}\begin{bmatrix}\Delta x \\ \Delta y\end{bmatrix}}$$

where

$[\Delta x \quad \Delta y]$ is the separation vector (in map coordinates),

A is the component's *Length* parameter.

θ is the anisotropy *Angle*, and

ρ is the anisotropy *Ratio*.

With a *Linear* variogram model, the slope is given by the *Scale* 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

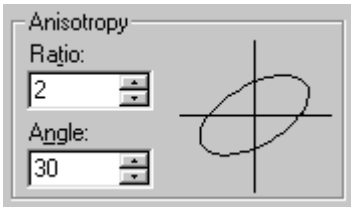
Natural phenomena are created by physical processes. Often these physical processes have preferred orientations. For example, at the mouth of a river the coarse material settles out fastest, while the finer material takes longer to settle. Thus, the closer one is to the shoreline the coarser the sediments 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.

For each different gridding method, anisotropy might be specified in a slightly different manner. In general, to set the anisotropy properties, click the Grids | New Grid | Grid Data command. Select the data file and click *Open*. In the **Grid Data** dialog, click the *Advanced Options* button.

The anisotropy settings include a *Ratio* and an *Angle* setting.



The anisotropy settings have an axis ratio and angle.

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

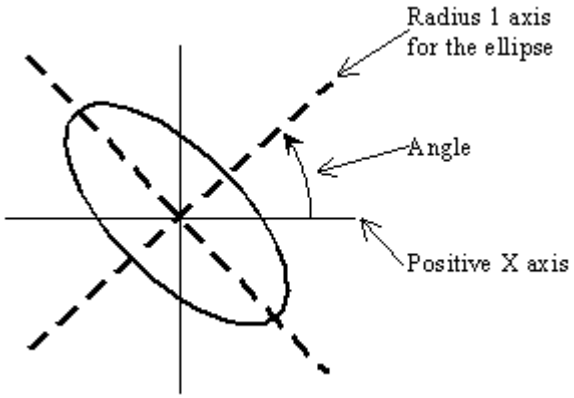
Angle = 0

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

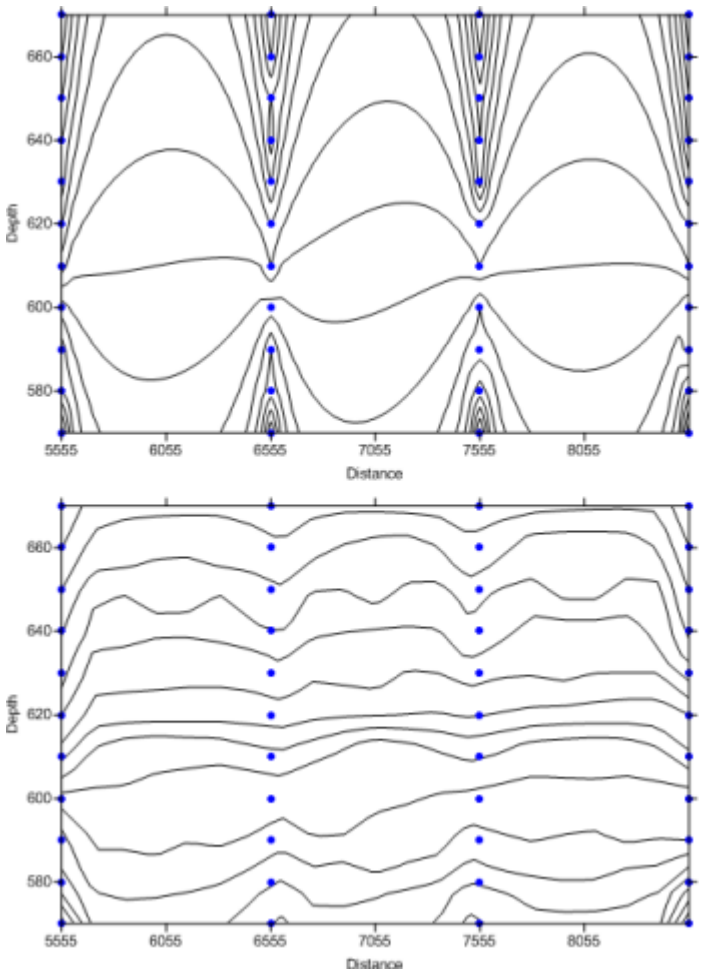
common scale, because one is distance and the other is time. One unit of X does not equal one unit of Y. While the resulting map can be displayed with changes in scaling, it may be necessary to apply anisotropy as well.

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

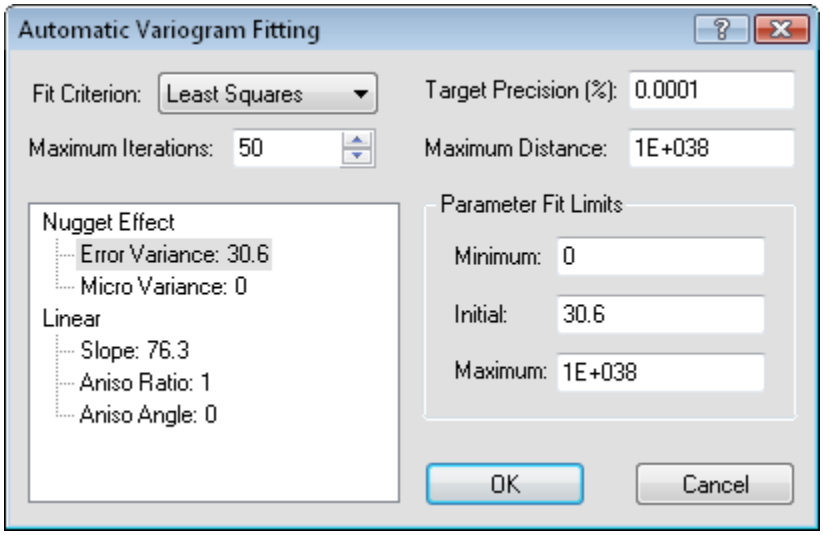
AutoFit takes a user-specified variogram model and an initial set of parameters, and attempts to find a better set of parameter values. For example, if you have selected a spherical variogram model with a nugget effect, the AutoFit tool can be used to determine the "best" fitting anisotropy parameters.

The *AutoFit* button on the Model page in the Variogram Properties 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

```
OldObjective - NewObjective < OldObjective * TargetPrecision/100
```

Note that the target precision is given as a percentage.

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.

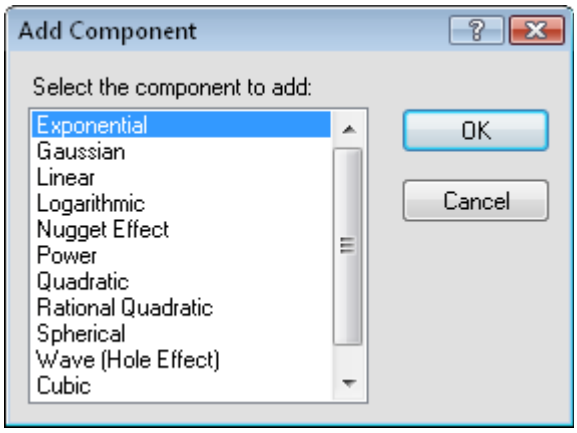
Maximum

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.

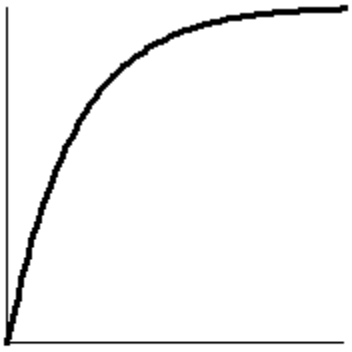
In the **Variogram Properties** in the Properties window, click on the Model tab. Click the *Add* button to open the **Add Component** dialog. Select one of the variogram model components and click the *OK* button.



*Select a variogram model component in the **Add Component** dialog.*

Variogram Model Components

The **variogram model** components available in **Surfer** are:

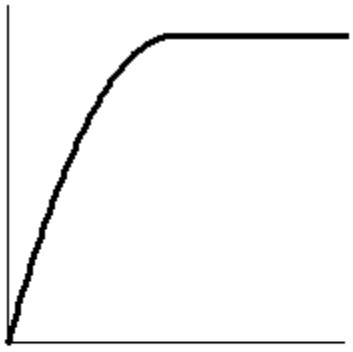


Exponential Model
Cressie (1991, p. 61)

$$\gamma(h) = C\left[1 - e^{-h}\right]$$

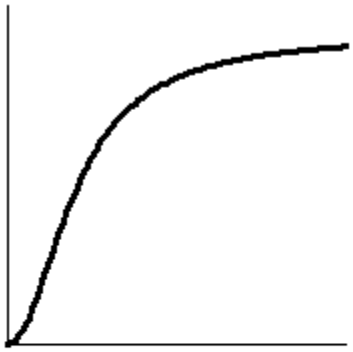
Gaussian Model
Pannatier (1996, p. 50)

$$\gamma(h) = C\left[1 - e^{-h^2}\right]$$



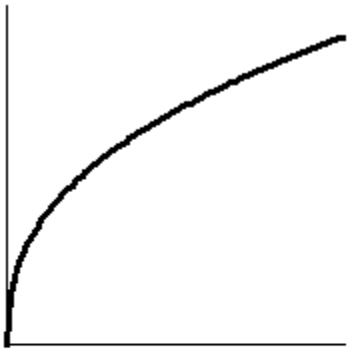
Quadratic Model
Alfaro (1980, p. 31)

$$\gamma(h) = \begin{cases} C\left[2h - h^2\right] & h < 1 \\ C & h \geq 1 \end{cases}$$



Rational Quadratic Model
Cressie (1991, p. 61)

$$\gamma(h) = C\left[\frac{h^2}{1+h^2}\right]$$



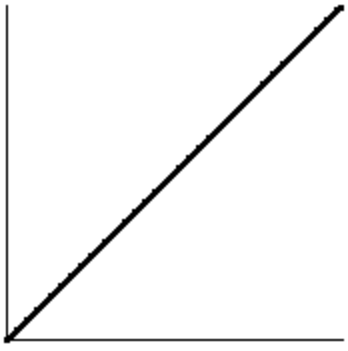
$0 < h < 1$

$1 < h < 2$

Power Models
Pannatier (1996, p. 51)

$$\gamma(h) = C\left[h^n\right]$$

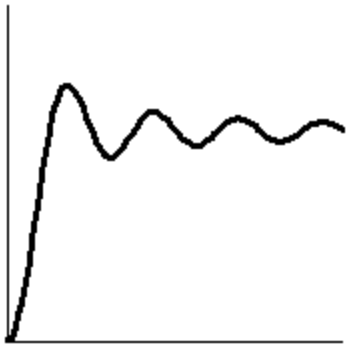
where $0 < n < 2$
when n=1, it is a Linear Model



Linear Model
Kitanidis (1997, p. 61)

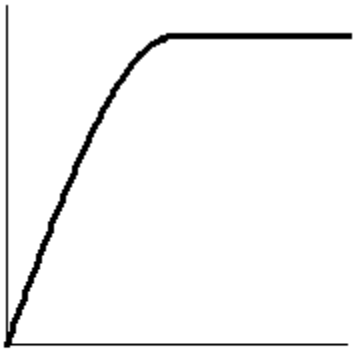
$$\gamma(h) = C(h)$$

The Linear Model is a special case of the Power Model where the power is equal to one



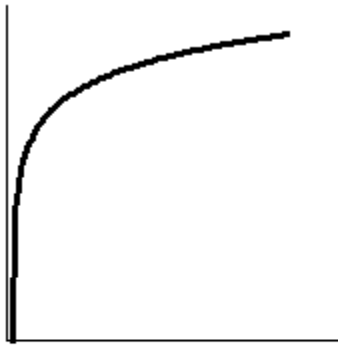
Wave (Hole Effect) Model
Cressie (1991, p. 62)

$$\gamma(h) = C\left[1 - \frac{\sin h}{h}\right]$$



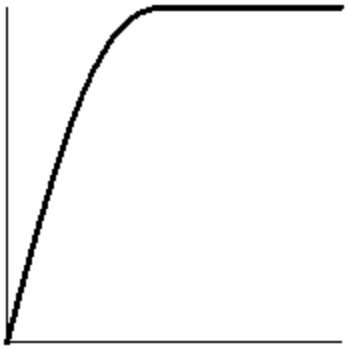
Spherical Model
Pannatier (1996, p. 48)

$$\gamma(h) = \begin{cases} C\left[1.5h - 0.5h^3\right] & h < 1 \\ C & h \geq 1 \end{cases}$$



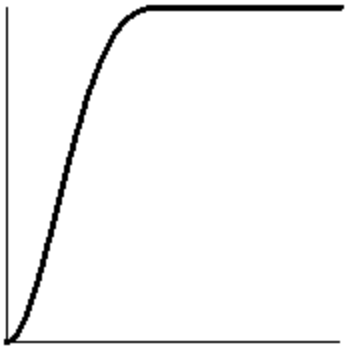
Logarithmic Model
Kitanidis (1997, p. 61)

$$\gamma(h) = C\left[\log_e(h)\right] \quad h > 0$$



Pentaspherical Model
Olea (1999, p. 79)

$$\gamma(h) = C\left(1.875h - 1.25h^3 + 0.375h^5\right)$$

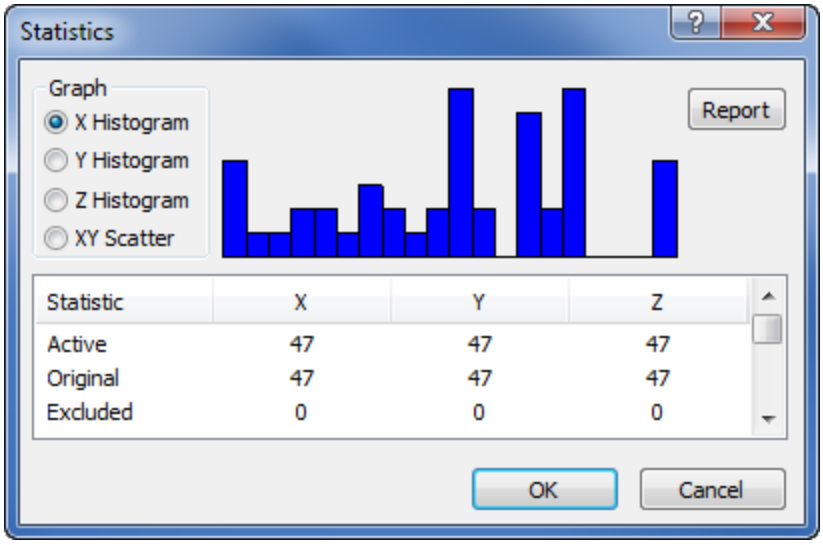


Cubic Model
Olea (1999, p. 79)

$$\gamma(h) = C\left(7h^2 - 8.75h^3 + 3.5h^5 - 0.75h^7\right)$$

Statistics Page

Clicking the *Display Statistics* button on the **Statistics** page in the Properties window, shows histograms and scatter plots of the data, as well as providing statistics on the data. To open the **Statistics** dialog, click on the existing variogram. In the **Properties** window, click on the **Statistics** tab. Click the *Display Statistics* button and the **Statistics** dialog opens.



*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

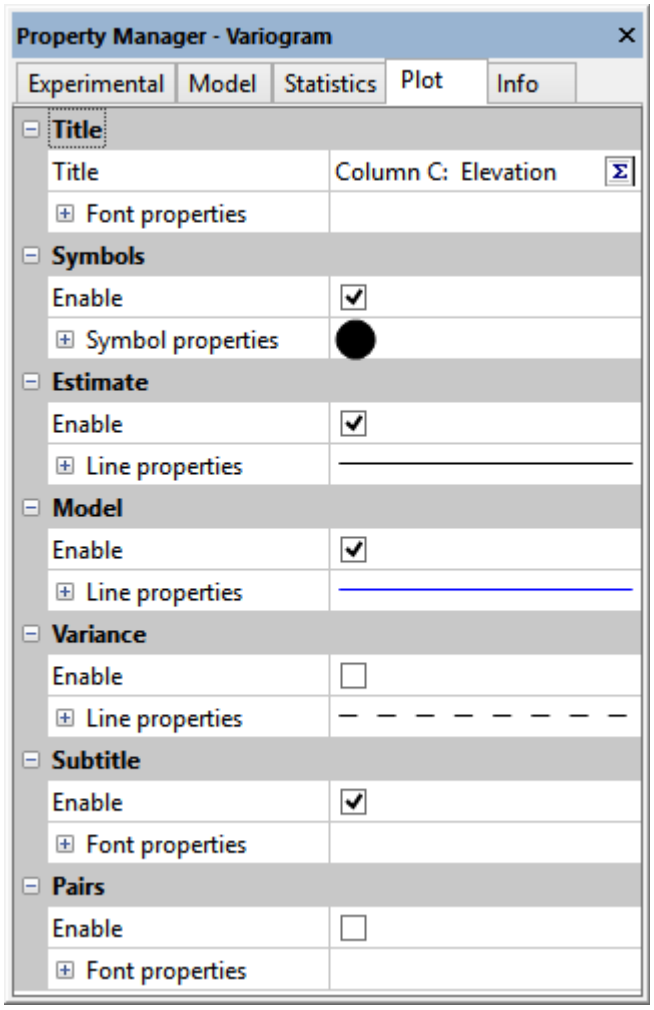
Click the *Report* button to open the statistics and variogram information in a report window. The statistics can be printed if they are opened in the report window.

Statistics

You can use the scroll bar on the right of the statistics box to view the statistics.

Plot Page

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.

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:

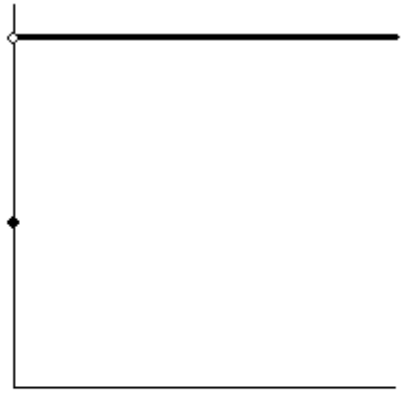
Nugget Effect = Error Variance + Micro Variance.

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

$$\gamma(h) = \begin{cases} Error\ Variance & h = 0 \\ Error\ Variace + Micro\ Variance & h > 0 \end{cases}$$

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. 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 - γ (h)) in column B, and the number of pairs in column C.

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 *Advanced Options* button to open the Kriging Advanced 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., Flannery, B.P., Teukolsky, S.A., and Vetterling, W.T. (1988), *Numerical Recipes in C*, Cambridge University Press.

Zimmerman, D. L., and M. B. Zimmerman (1991), A Comparison of Spatial Semivariogram Estimators and Corresponding Ordinary Kriging Predictors, *Technometrics*, v. 33, n. 1, p. 77-91.

Variogram Tutorial

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.

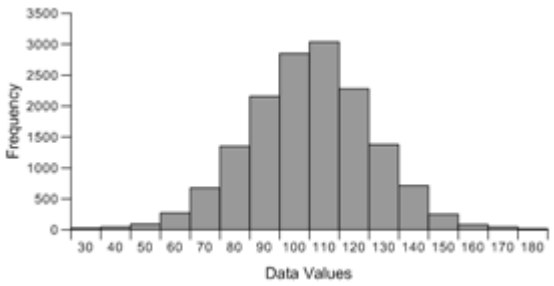


*Figure 1.1 Data Set **A** Histogram*

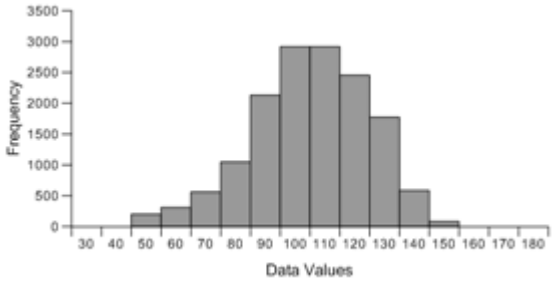


*Figure 1.2 Data Set **B** Histogram*

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.

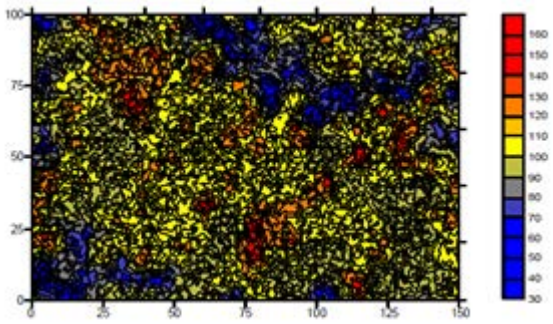


*Figure 1.3 Data Set **A** Contour Plot*

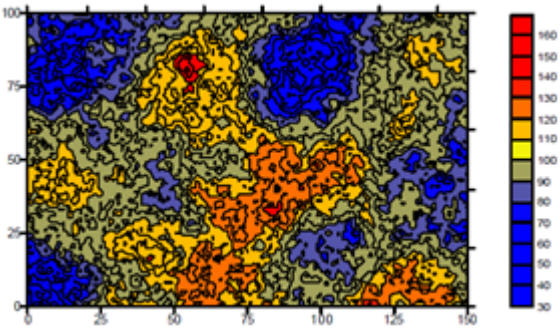


*Figure 1.4 Data Set **B** Contour Plot*

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.

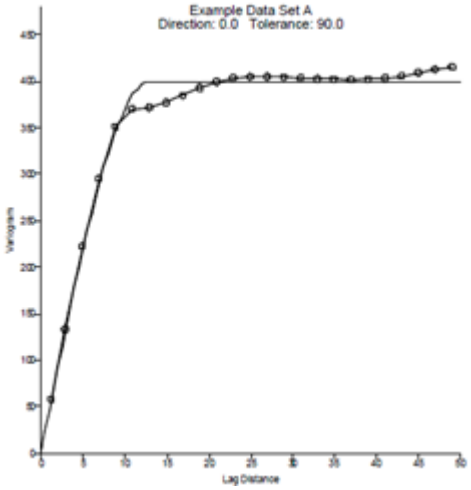


*Figure 1.5 Data Set **A** Variogram and Model*

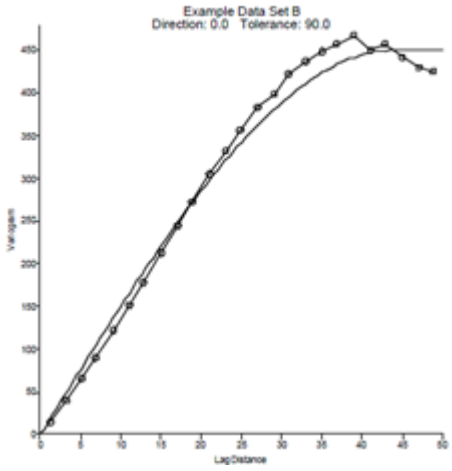


*Figure 1.6 Data Set **B** Variogram and Model*

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 **ε [ ]** is the statistical expectation operator. Note that the variogram, **γ( )**, is a function of the separation between points **(Δx, Δy)**, 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), ... (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.

The 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*.

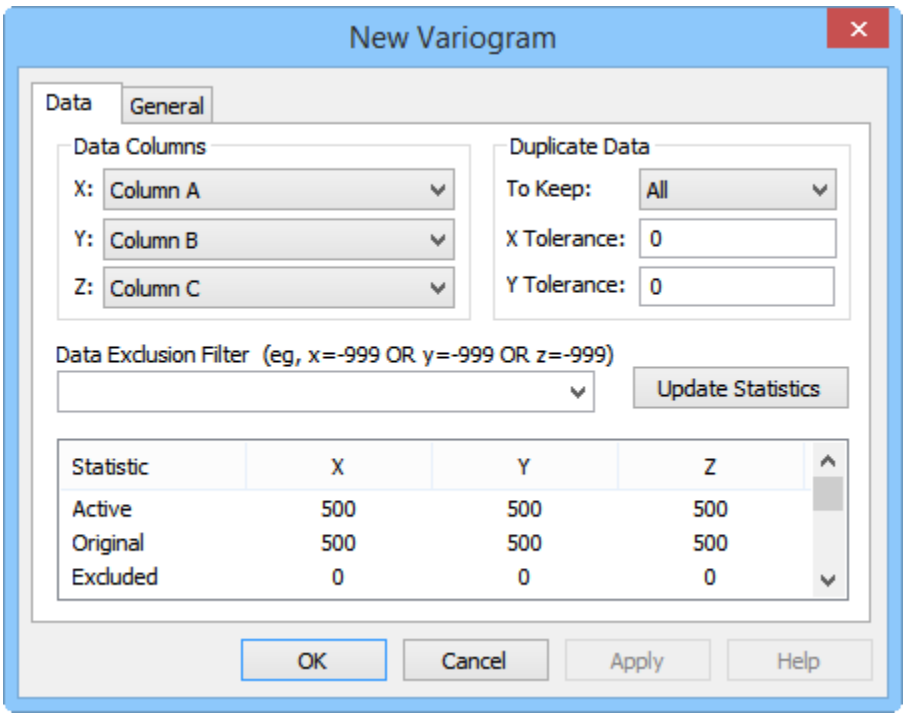


*Figure 4.1 Choose the **Grid | Variogram | New Variogram** menu to display the **Data** tab of the **New Variogram** dialog box.*

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*.

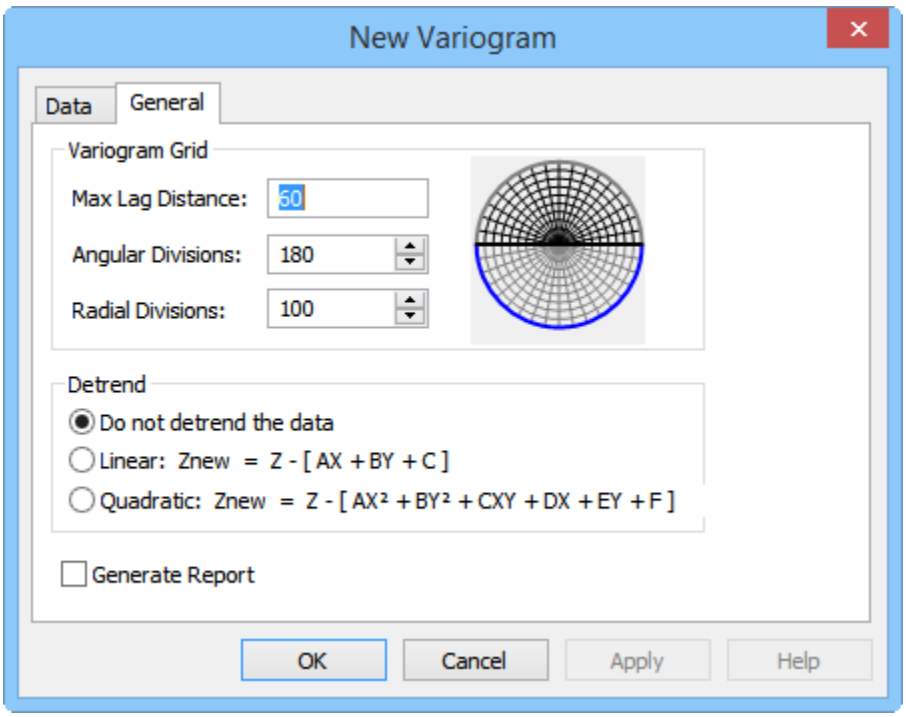


*Figure 4.2 Click the **General** tab in the **New Variogram** dialog box to display the Variogram Grid, Detrend, and Generate Report options.*

Without changing any of the settings, click *OK*. Figure 4.3 is displayed.

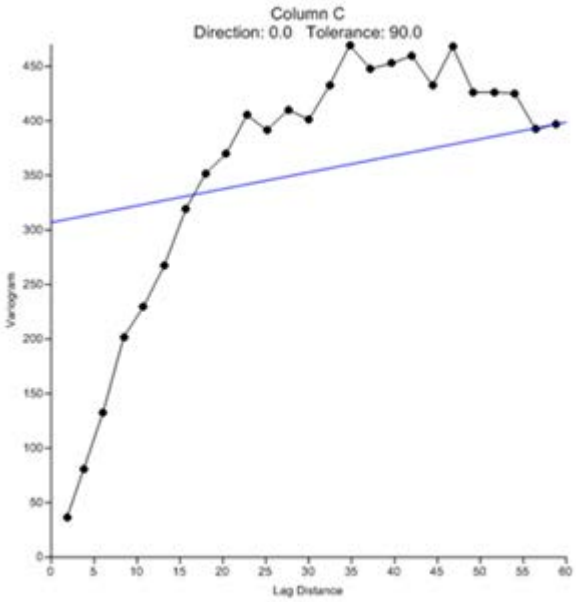


Figure 4.3 Resulting variogram with default variogram settings using ExampleDataSetC.xls.

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.

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.

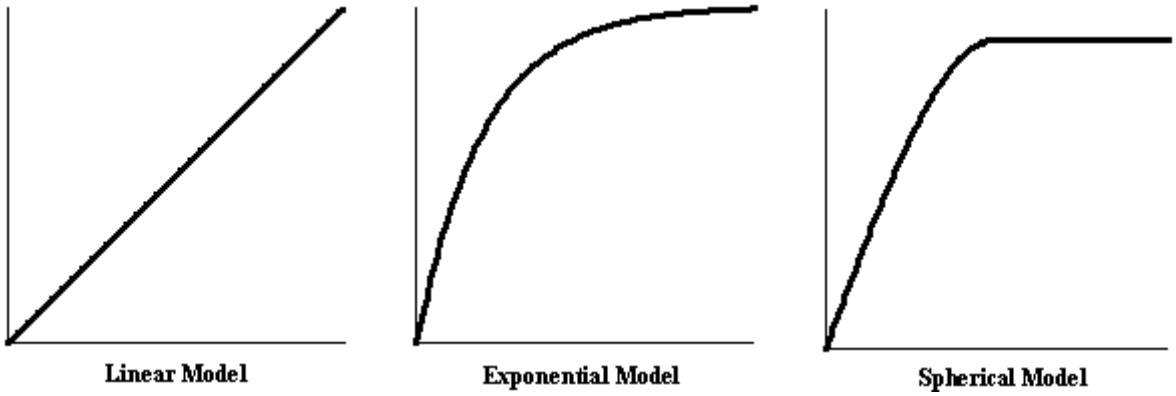


Figure 5.1 Variogram Models

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.

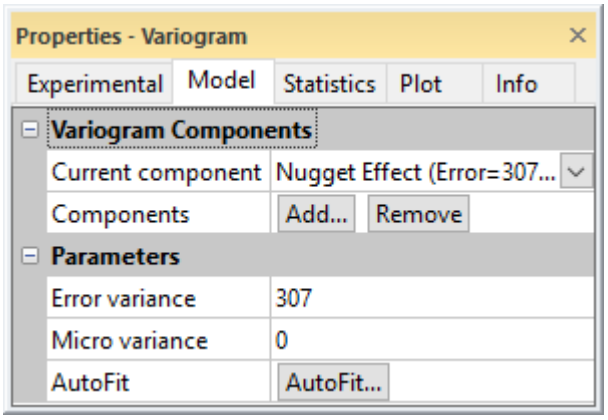


*Figure 5.2 The **Model** tab of the Variogram properties in the **Properties** window.*

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*.

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.

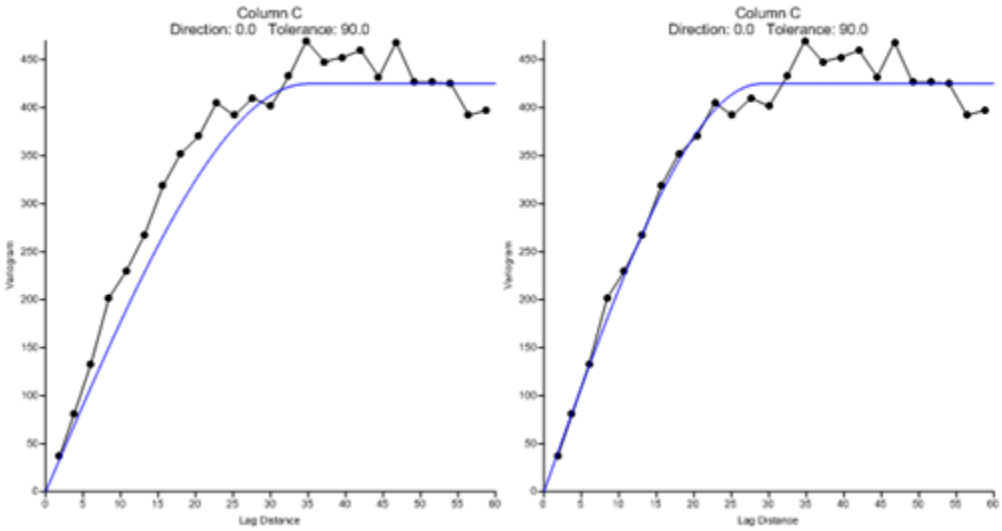


Figure 5.3 Variogram model with initial assumptions Left: Scale = 425, Length (A) = 35. Right: Scale =425. Length (A) = 29.

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.

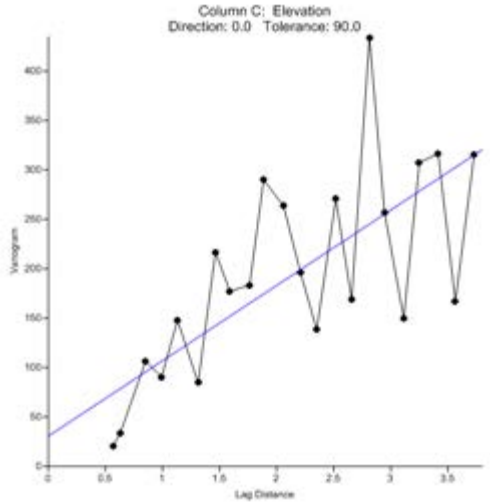


Figure 5.4 Linear Variogram model with nugget effect for data set Demogrid.dat.

In **Surfer** the nugget effect is partitioned into two sub-components: the *error variance* and the *micro variance*. Both of these sub-components are nonnegative,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.

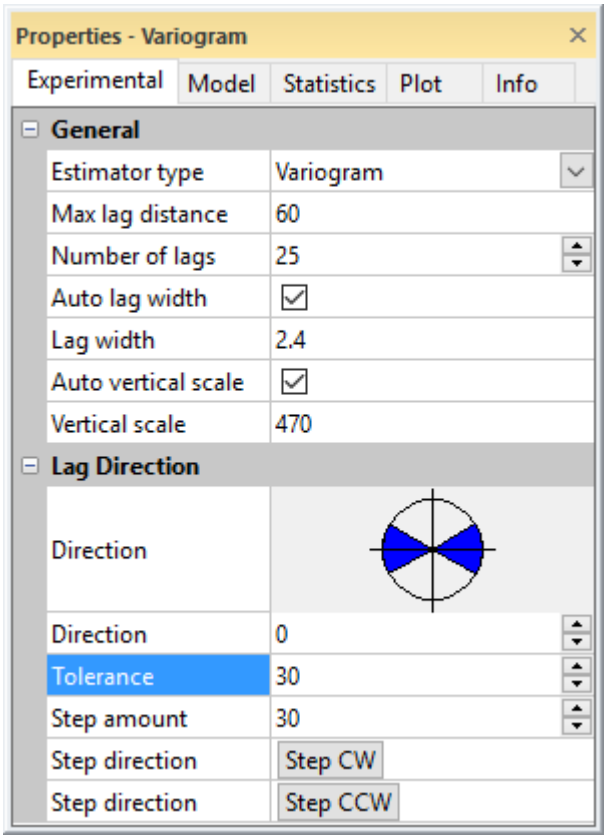


*Figure 6.1 Set the Tolerance to 30 on the **Experimental** tab of the Variogram properties in the **Properties** window.*

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.

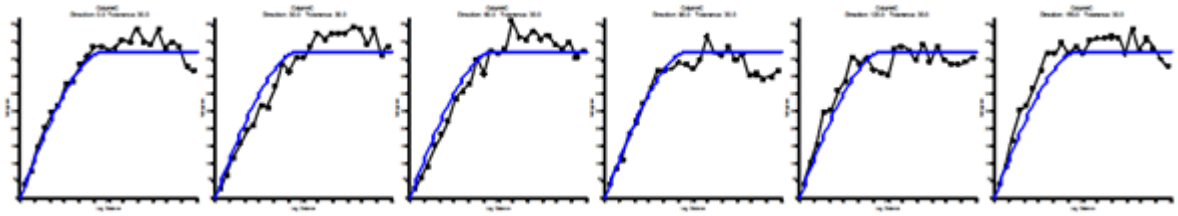


Figure 6.2 Directional variogram plots with Tolerance set to 30°. From left to right: Direction = 0°, 30°, 60°, 90°, 120°, 150°.

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.

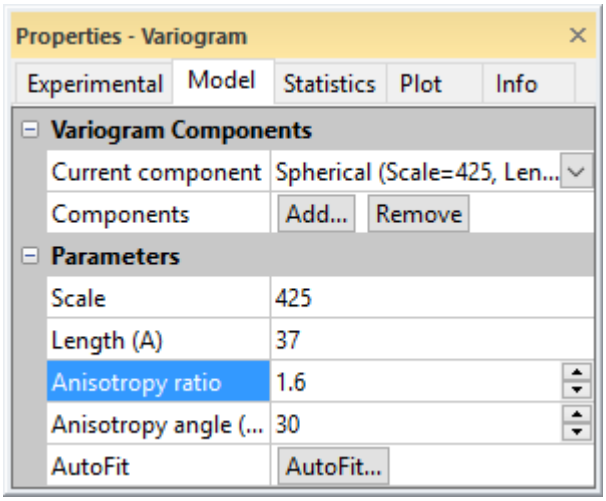


Figure 6.3 Change the variogram model properties to add anisotropy.

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.

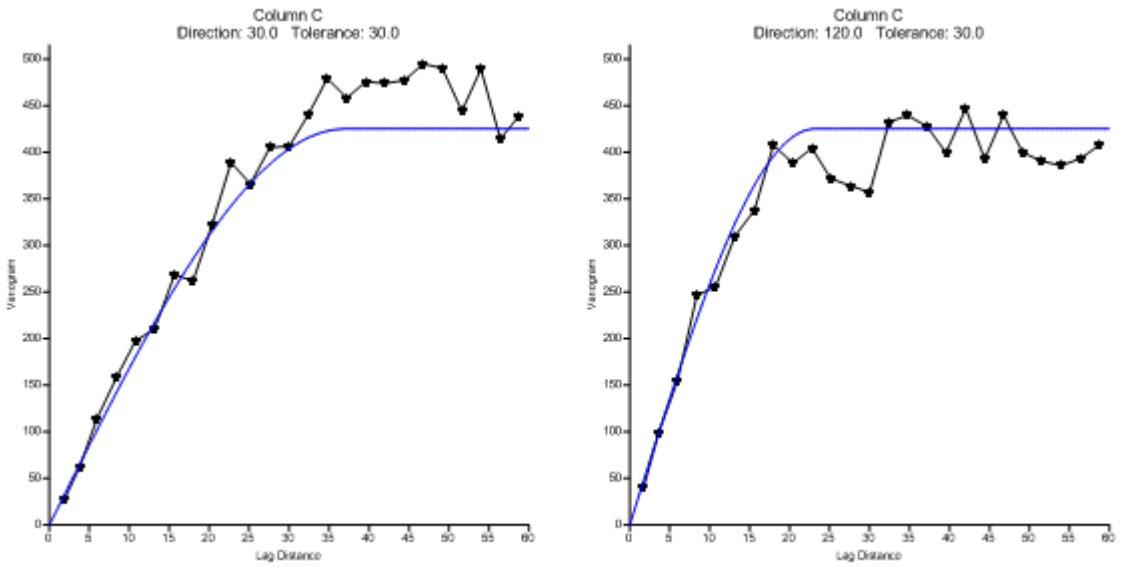


Figure 6.4 Directional variograms with anisotropy settings. Left: Direction = 30°, Right: Direction = 120° Anisotropy Ratio = 1.6, Angle = 30°.

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°.

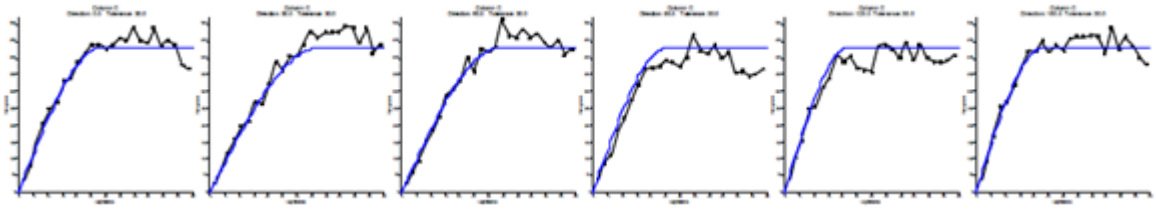


Figure 6.5 Directional Variogram plots with Anisotropy Ratio = 1.75 and Anisotropy Angle = 38.49°. From left to right: Direction = 0°, 30°, 60°, 90°, 120°, 150°.

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 $2\gamma( )$ was given the name variogram. The function of practical interest was $\gamma( )$ which was given the name semi-variogram since it is one-half the variogram function. Since only the function $\gamma( )$ is used in kriging, the prefix semi- is regularly dropped, and the function $\gamma( )$ 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 surfersupport@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

$$\varepsilon[s^2] = \frac{1}{n^2}\sum_{i-1}^{n}\sum_{j=1}^{n}\gamma(x_i-x_j, y_i-y_j)$$

This result states that the expected value of the sample variance, $\mathbf{S^2}$, is equal to the average value of the variogram between all $\mathbf{n^2}$ 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, *Geostaitsitcs 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

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

- The **File | Open** command opens the grid editor when a grid or image file is selected in the Open dialog.
- The **Grids | Editor | Grid Editor** command opens 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.

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.

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 normalized red, green, blue, and alpha.

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:

$$I = .30*1.0 + .59*0.0 + .11*0.0 = 0.3$$

Note the normalization process converted the color's (0 to 255) range to (0.0 to 1.0).

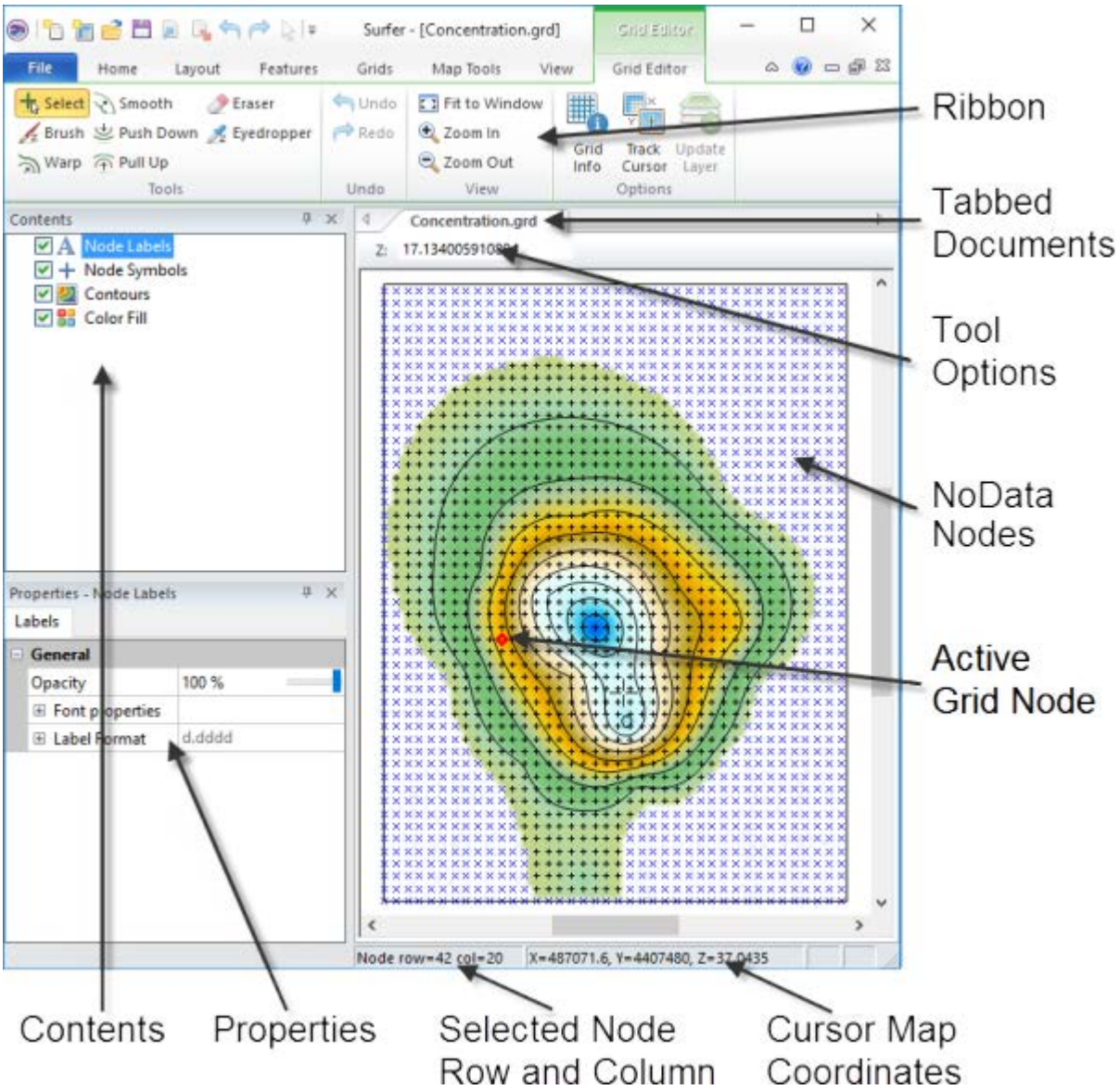
Grayscale Image

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, and Color Fill with the **Contents** window.
Properties	Edit Node Labels, Node Symbols, Contours, and Color Fill display 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
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 you wish to edit in the plot document **Contents** window. Only the grid for this map layer will be edited, even when multiple layers 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.

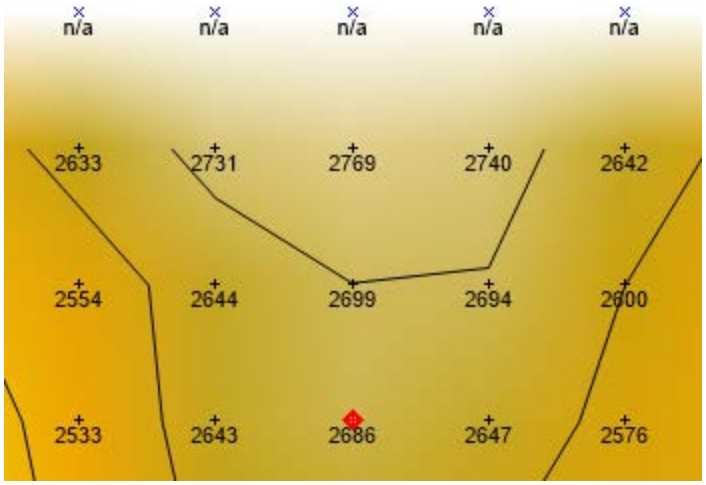
To edit a grid file

1. Click the **Grids | Editor | Grid Editor** command and select the grid file in the Open Grid dialog. Alternatively, click the **File | Open** command and select a grid file in the Open 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.

Grid Editor - Node Labels

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.

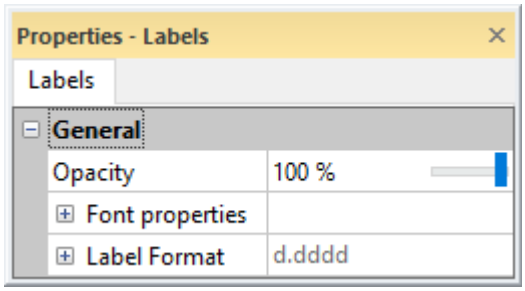
The visibility check box next to the *Node Labels* object is used to turn on or off the node label display in the grid editor window. When the check box is checked, a number appears below each grid node when zoomed in a sufficient amount. When the zoom or density is such that the labels would be too 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.

Grid Node Labels Properties

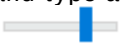
The **Labels** page is displayed in the Properties window when the *Node Labels* object is selected. Edit the label opacity, font properties, and format in the **Properties** window.



*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.

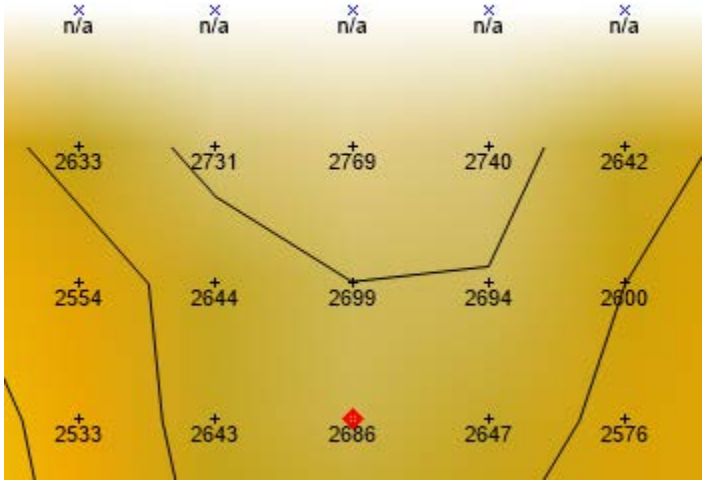
Label Format

Click the ⊞ next to *Label Format* to display the Label Format section. The numeric format is applied to all grid node labels.

Grid Editor - Node Symbols

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.

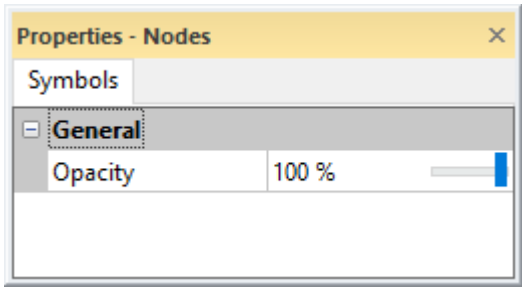
The visibility check box next to the *Node Symbols* object is used to turn on or off the symbol display in the grid editor window. When the check box is checked, a black "+" appears at the location of each grid node. NoData nodes appear with a blue "x" at each grid node location. When the command is 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.

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.*

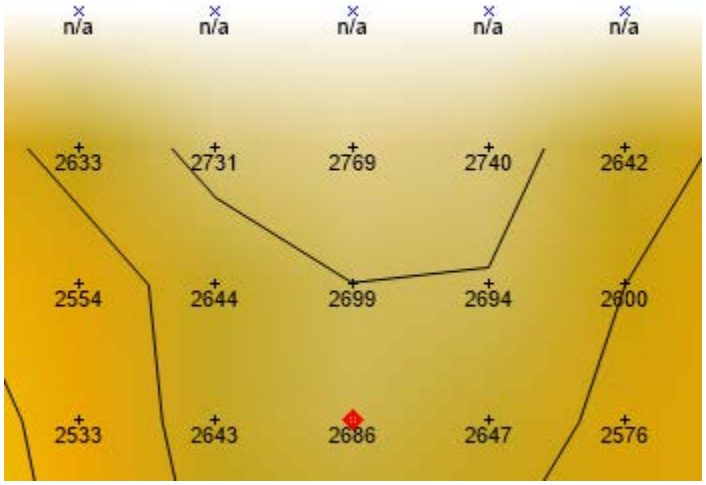
Opacity

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.

Grid Editor - Contours

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.

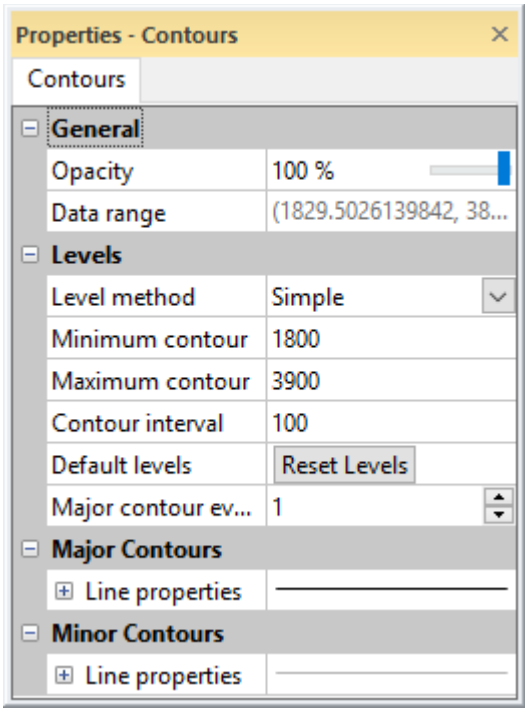
The visibility check box next to the *Contours* object is used to turn on or off the contour lines in the grid editor window. When the check box is checked, contour lines are displayed on the grid in the grid editor. When the command is not checked, the contour lines are not displayed.



This image displays grid node symbols, labels, contour lines, and a color fill for a grid file.

Grid Contour Properties

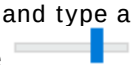
The **Contours** page is displayed in the Properties window when the *Contours* object is selected. Edit the contour opacity, level method and properties, and line properties in the **Properties** window.



*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

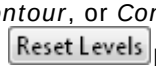
Set the *Maximum contour* to the value of the last contour line you want displayed in the grid editor. To set a new value, highlight the existing value and type the new value. Press ENTER on the keyboard or click another command in the **Properties** window for the value to be applied. Values are in 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 

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.

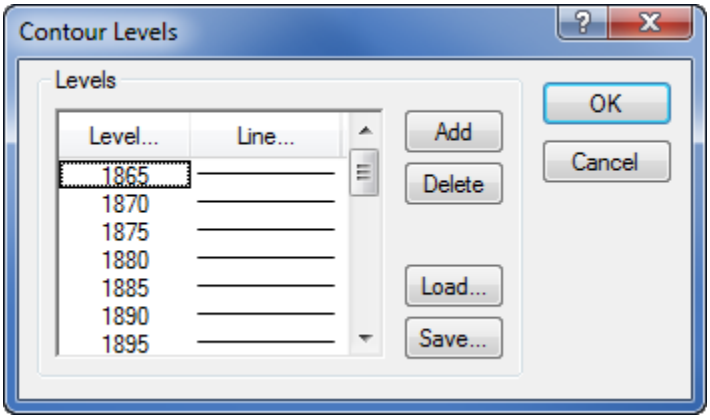
Minor Levels per Decade	Values of Contour Lines
8	1, 2, 3, 4, 5, 6, 7, 8, 9
7	2.125, 3.250, 4.375, 5.500, 6.625, 7.750, 8.875
5	2.5, 4.0, 5.5, 7.0, 8.5
4	2.8, 4.6, 6.4, 8.2
3	3.25, 5.5, 7.75
2	4, 7
1	5.5
0	no minor lines are displayed

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.

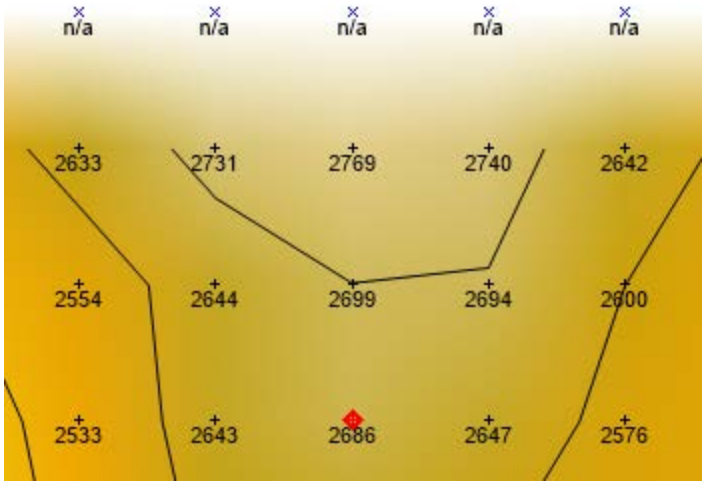
Save

The *Save* button allows you to save the current contour level and line property information to a level file .LVL.

Grid Editor - Color Fill

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.

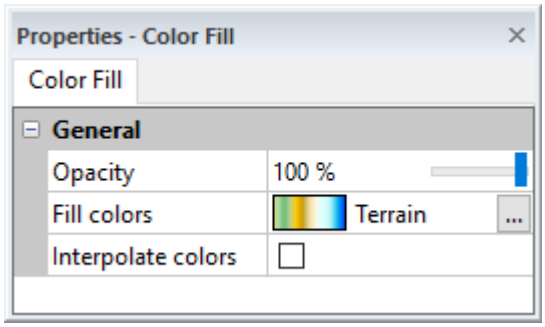
The visibility check box next to the *Color Fill* object is used to turn on or off the color fill display in the grid editor window. When the check box is checked, the grid editor displays a color relief map. When the command is not checked, the labels, nodes, and contours are displayed on a white background.



This image displays grid node symbols, labels, contour lines, and a color fill for a grid file.

Grid Node Color Fill Properties

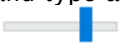
The **Color Fill** page is displayed in the Properties window when the *Color Fill* object is selected. Edit the fill opacity, color map, and interpolation option in the **Properties** window.



*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  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 dialog.

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.

Select

Click the **Grid Editor | Tools | Select** command 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 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

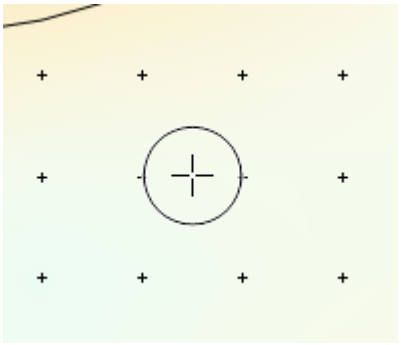
The *Brush size* and *Z value box* controls how the **Brush** tool changes the grid vales.

Brush Size

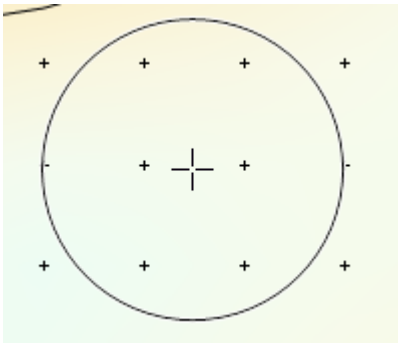
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- 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** Z value. 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.

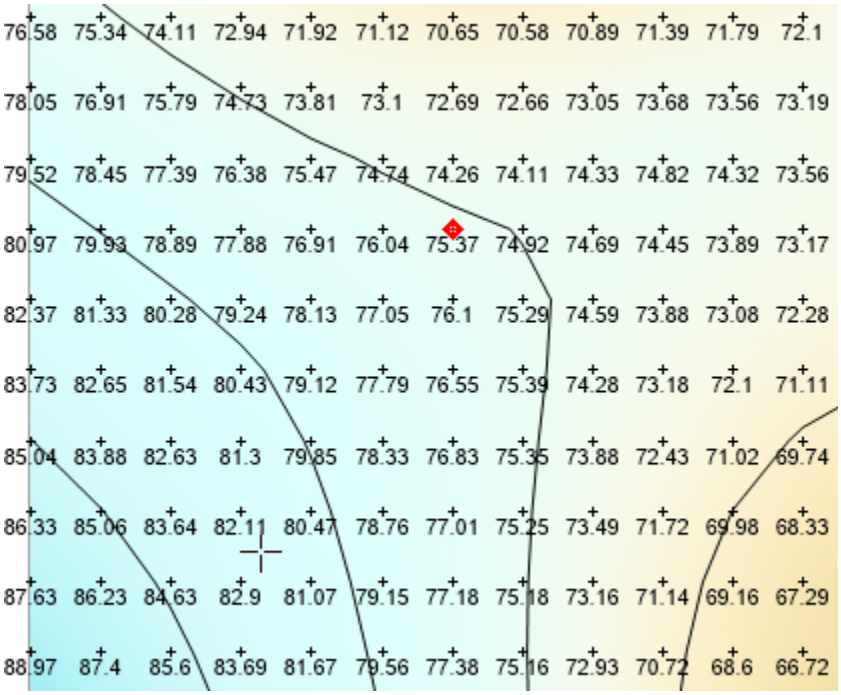
Undo

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

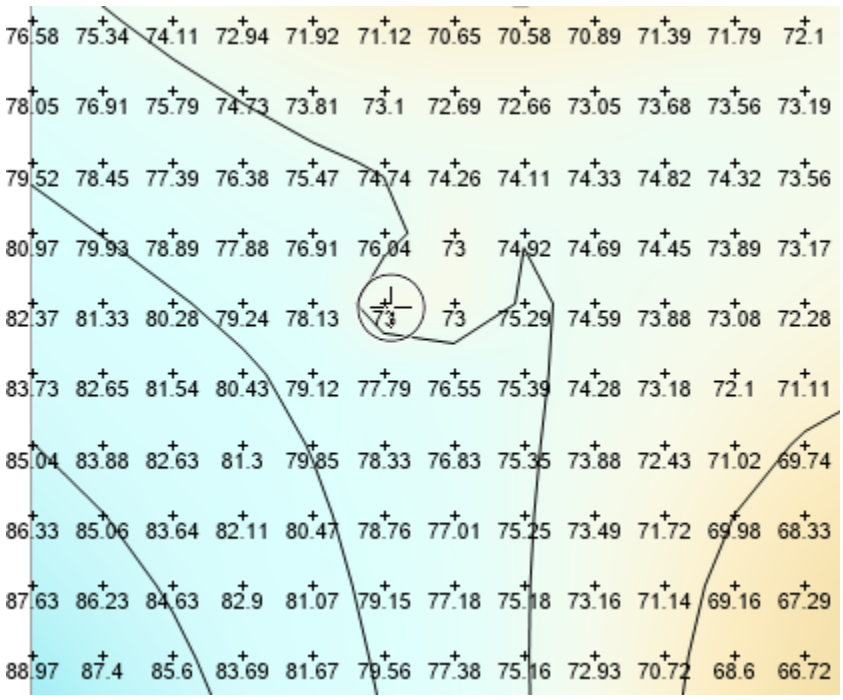
The following example will demonstrate editing the *Demogrid.grd* sample file with the **Brush** tool:

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.

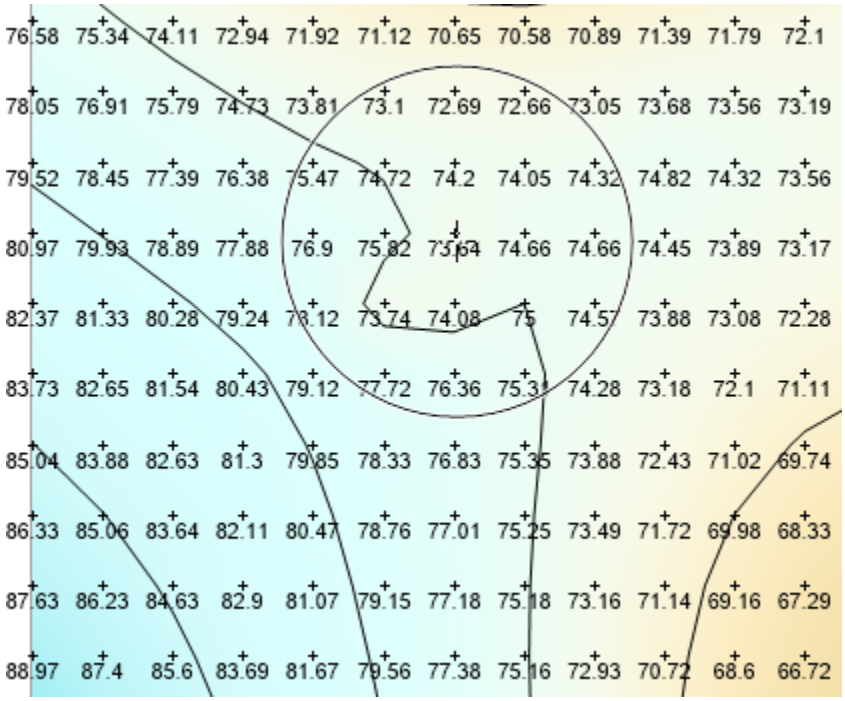
1. Click the **Grid Editor | Tools | Brush** command.
2. Set the *Brush size* to 1 by typing *1* in the *Brush size* field.
3. Set the Z value to 73 by typing *73* in the *Z value box*.
4. Click near the grid node at (29,6) and drag one row down and one column left.



*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 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

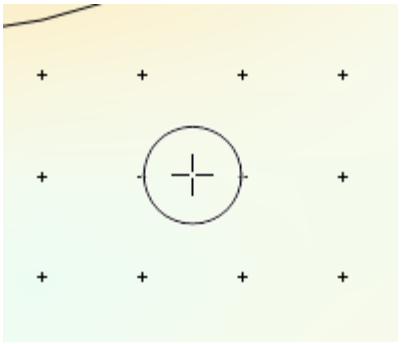
The *Brush size*, *Density*, and *Pressure* control how the **Warp** tool drags and changes the grid vales.

Brush Size

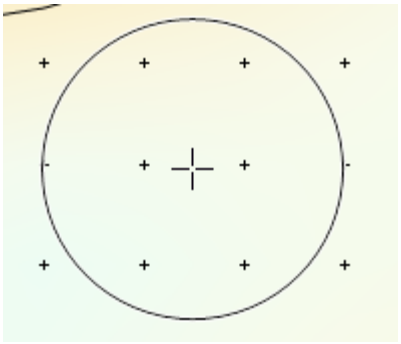
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- 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.



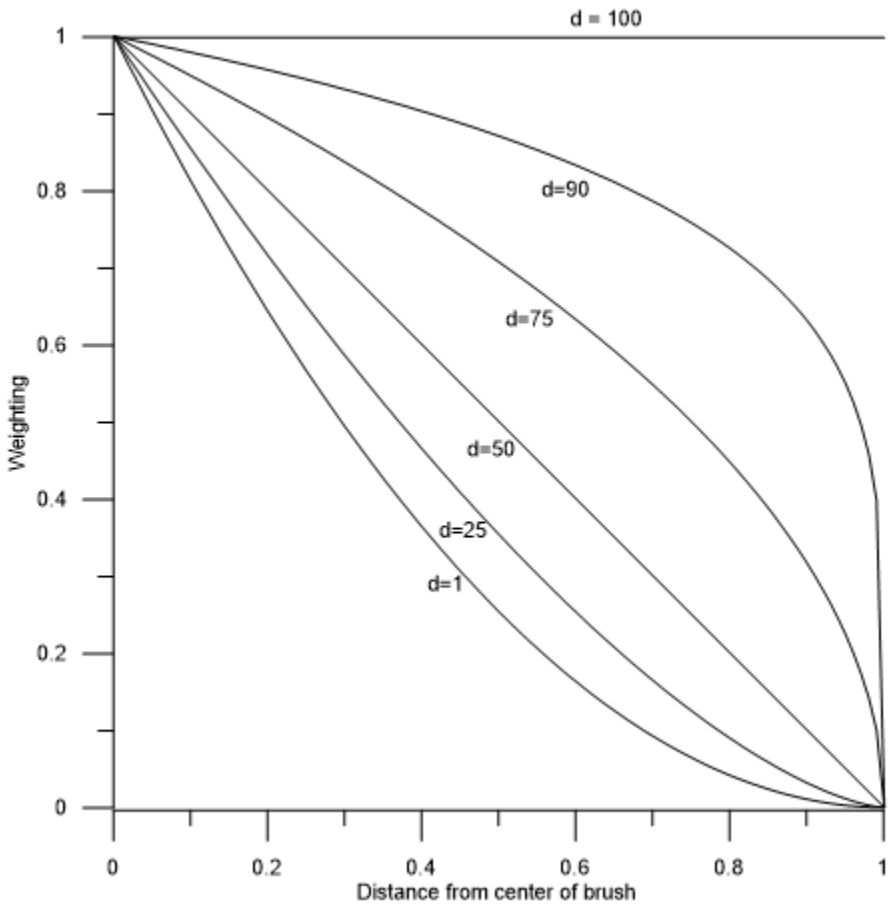
A Brush size of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.

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 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

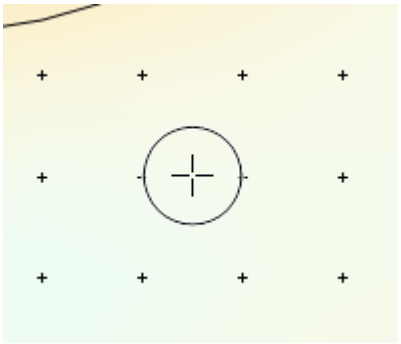
The *Brush size*, *Density*, and *Pressure* control how the **Smooth** tool smooths the grid vales.

Brush Size

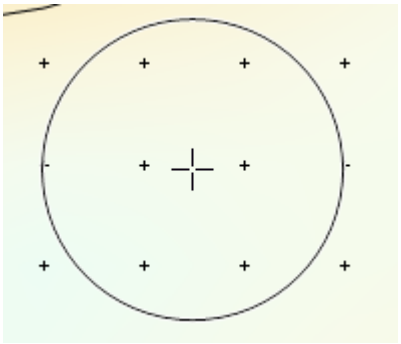
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- 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.

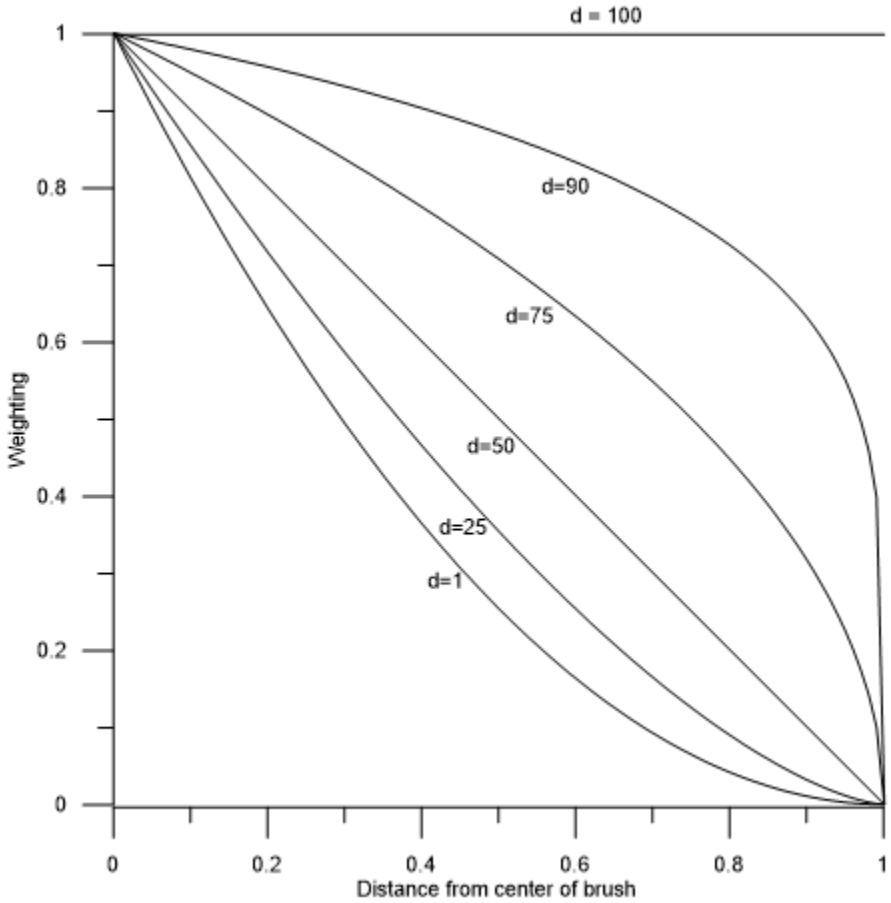
Density

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 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

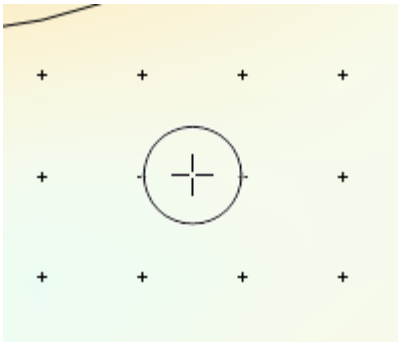
The *Brush size* controls the area around the cursor that is affected by the **Push Down** tool. The *Density* controls how the **Push Down** tool decreases the grid vales in the brush area. The *Pressure* controls how much the grid values are affected.

Brush Size

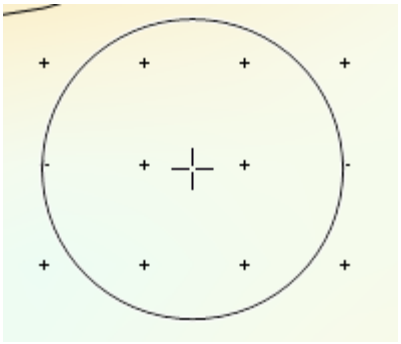
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- 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.



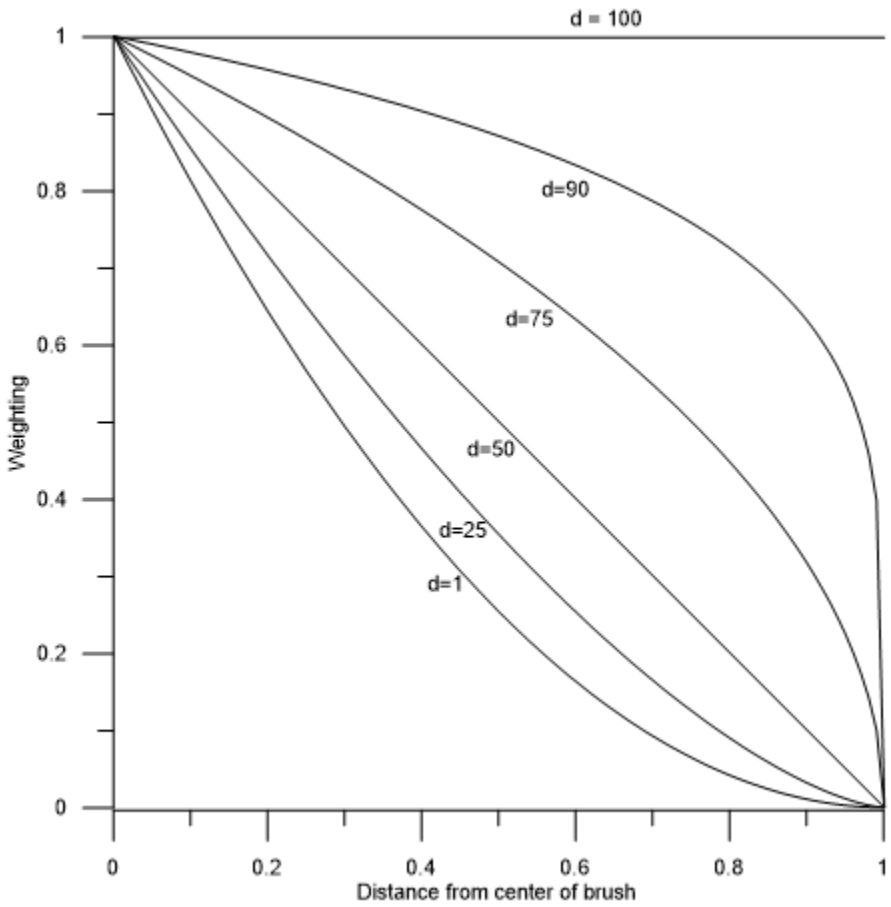
A Brush size of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.

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 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

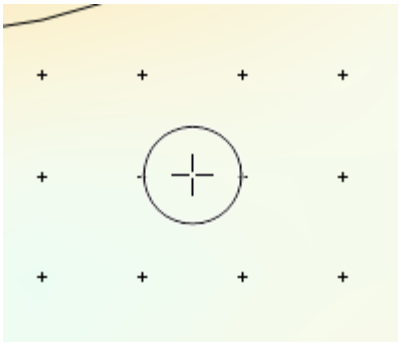
The *Brush size* controls the area around the cursor that is affected by the **Pull Up** tool. The *Density* controls how the **Pull Up** tool increases the grid vales in the brush area. The *Pressure* controls how much the grid values are affected.

Brush Size

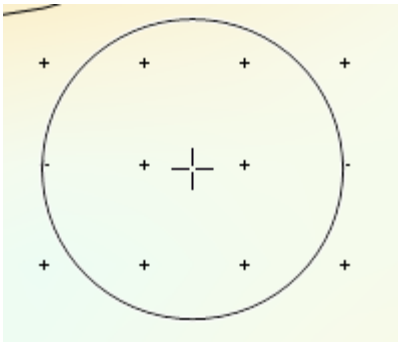
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- 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.



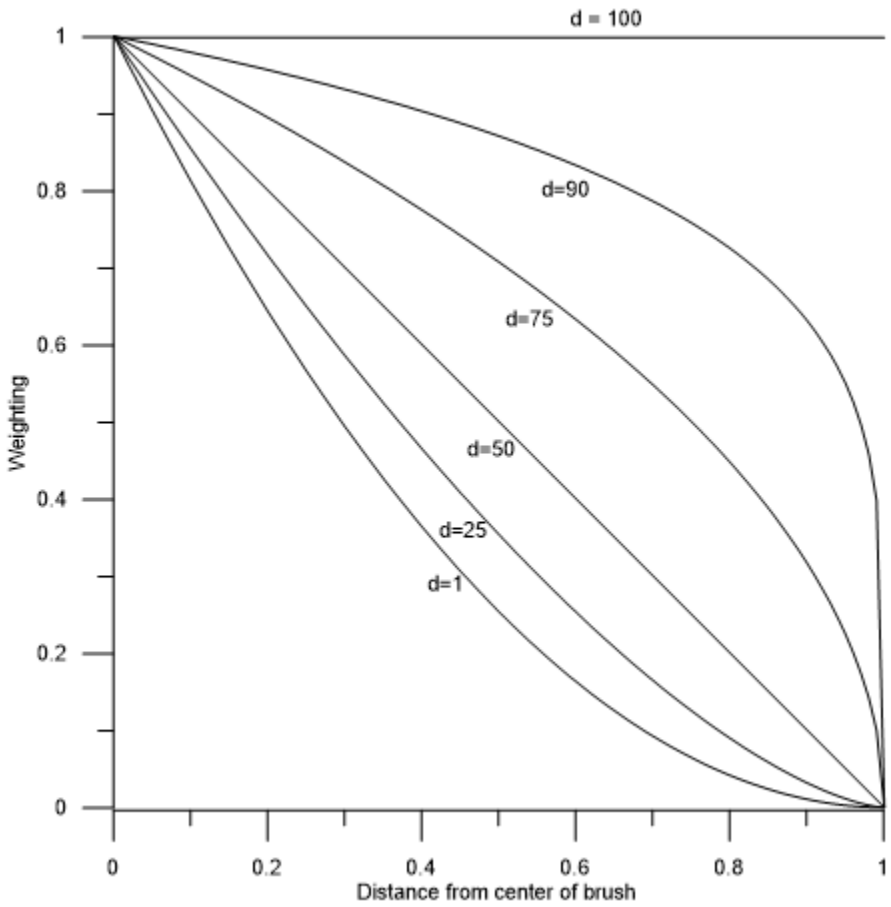
A Brush size of 3 makes the diameter of the brush equal to the horizontal spacing of three grid cells.

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 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.

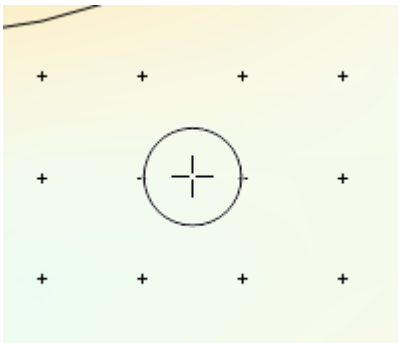
Press ESC or click another **Grid Editor | Tools** command to end **Eraser** mode.

Brush Size

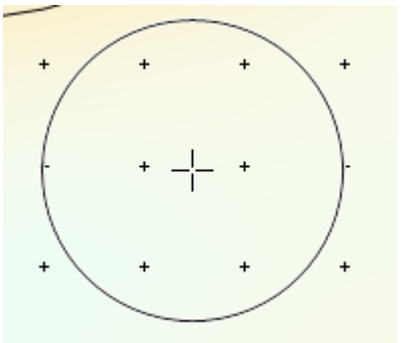
The *Brush size* tool option controls the diameter of the tool. The *Brush size* specifies the diameter in number of grid cells in the X direction. The *Brush size* setting is located in the tools options bar at the top of the grid editor window. There are three methods for changing the *Brush size*:

- 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



A Brush size of 3 makes the diameter of the brush equal to

the horizontal spacing of one grid cell.

the horizontal spacing of three grid cells.

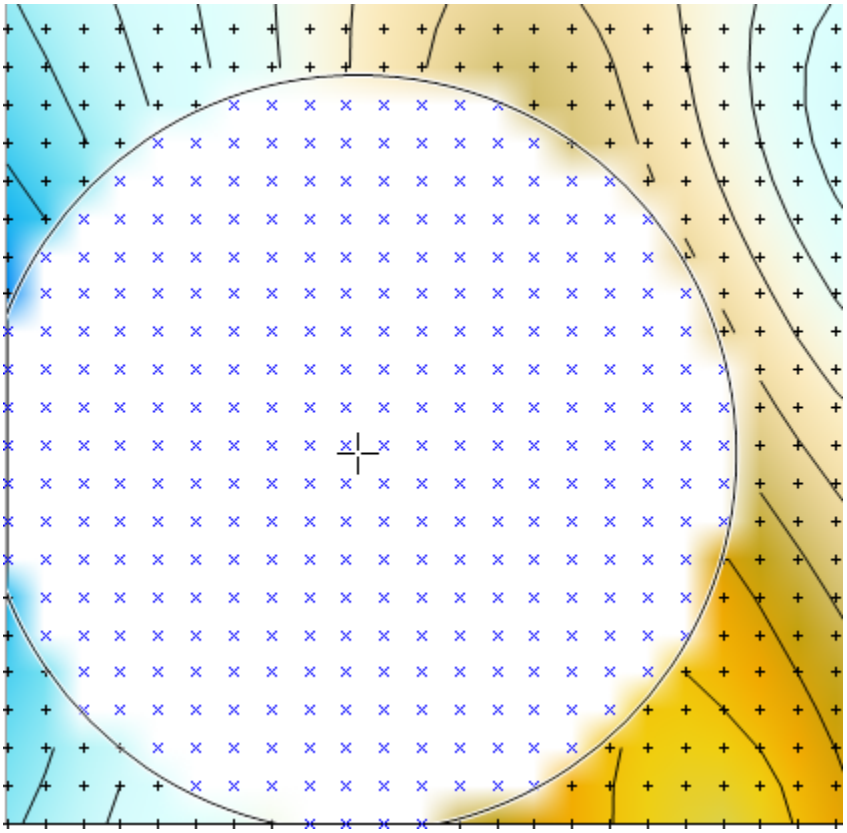
Undo

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

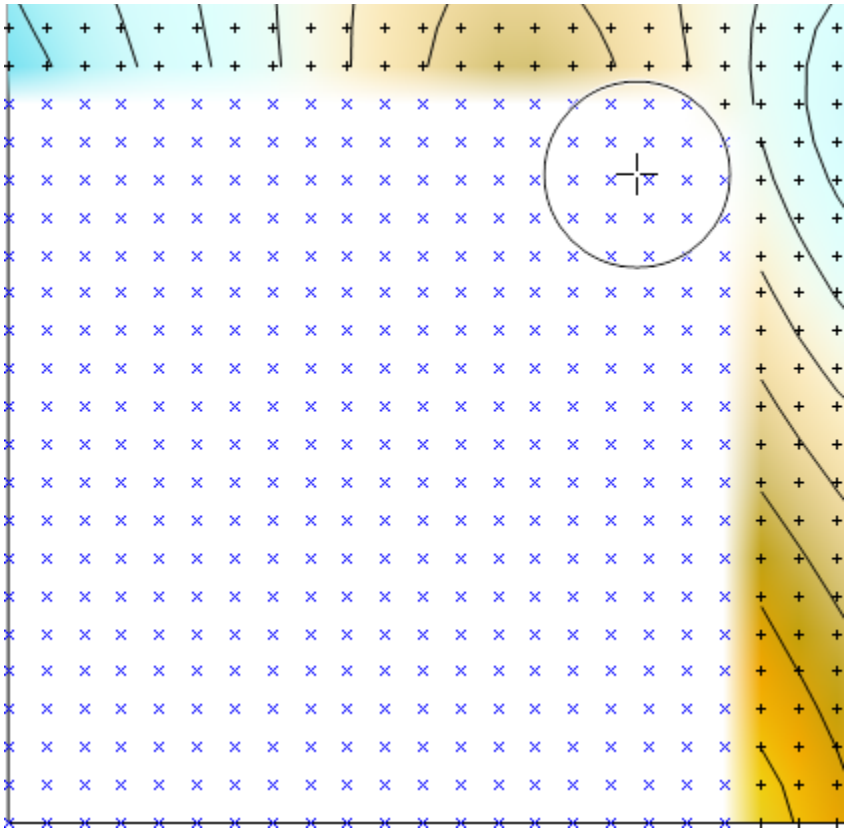
The following example will demonstrate using the **Erase** command to assign the NoData value to a 20x20 node section of the *Demogrid.grd* sample file grid with the **Erase** command,

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.



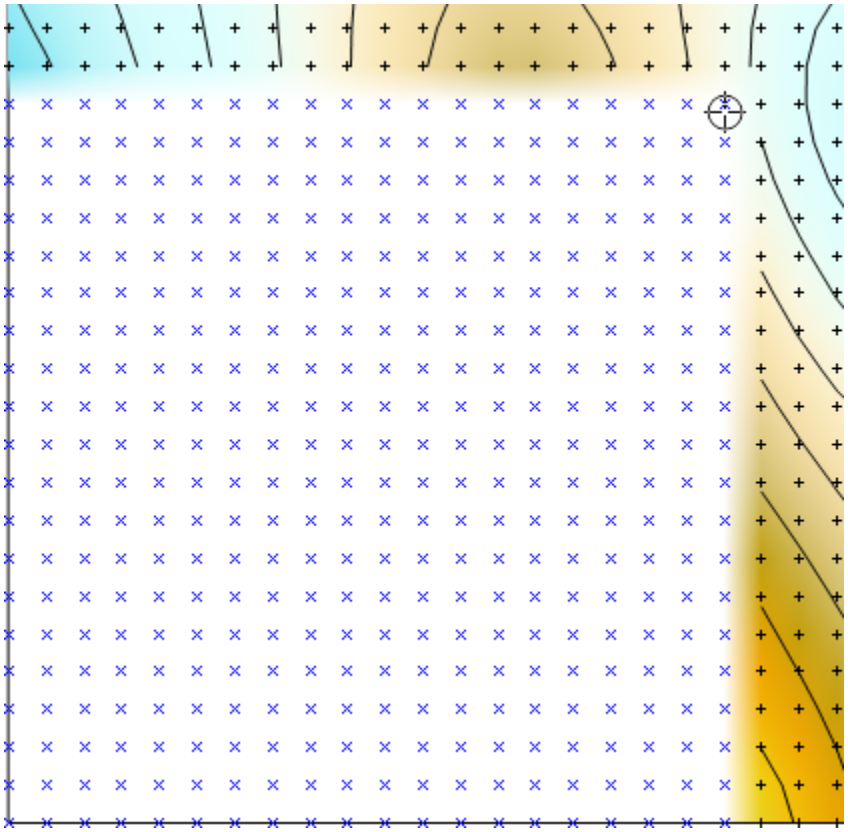
Many nodes can be erased in a single click when the Brush size is large.

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 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 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 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 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 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,

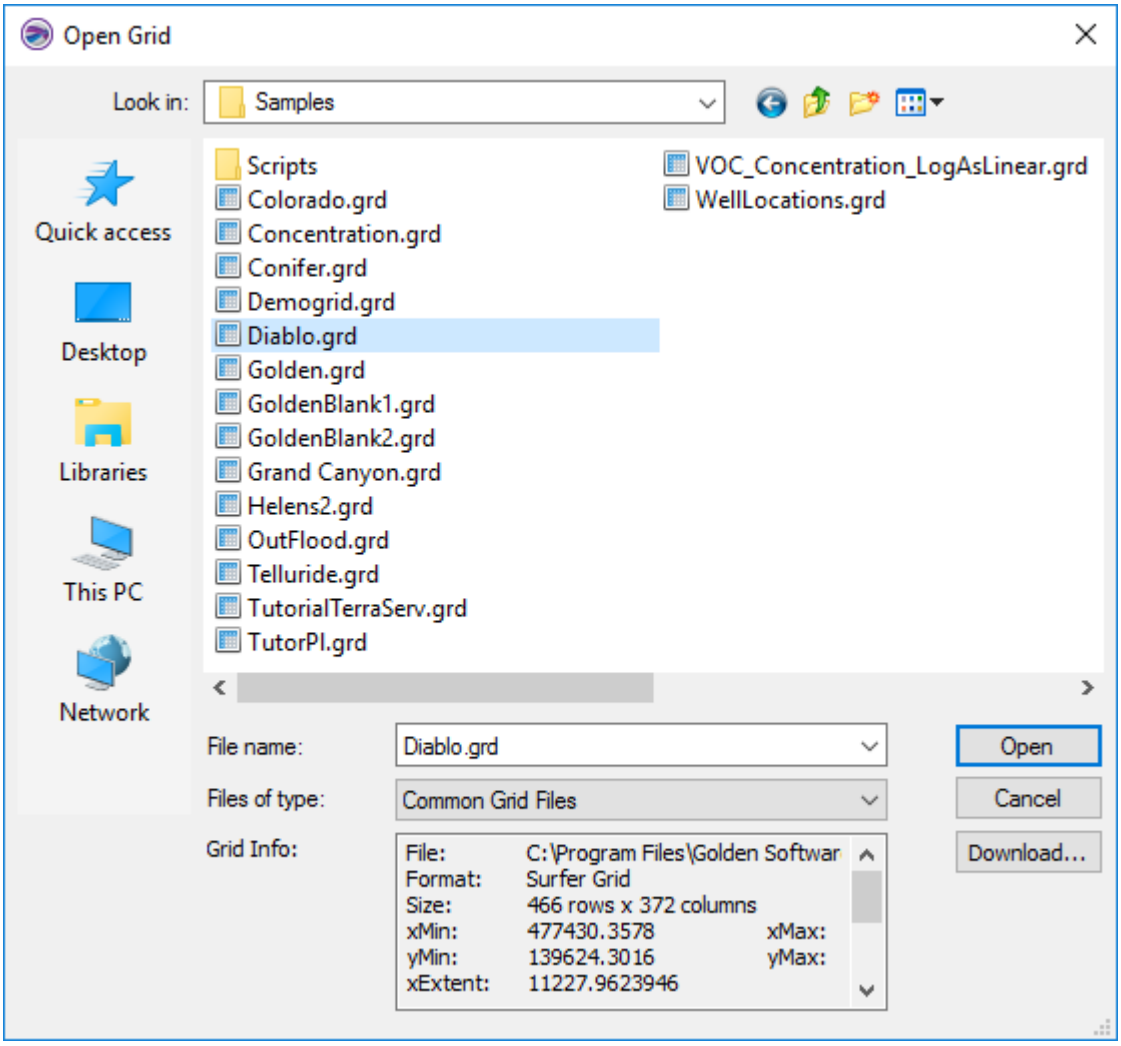
1. Return to the plot document by clicking the document tab for the plot window.
2. Click one of the map layers that is still using the original grid file in the Contents window.
3. Click the **General** tab in the Properties window.
4. Click the button in the *Grid file* property field.
5. Locate and 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, shaded 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 a grid file when creating a new grid-based map or layer or when using a grid command (i.e. **Grids | Edit | Convert**).



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.

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.

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

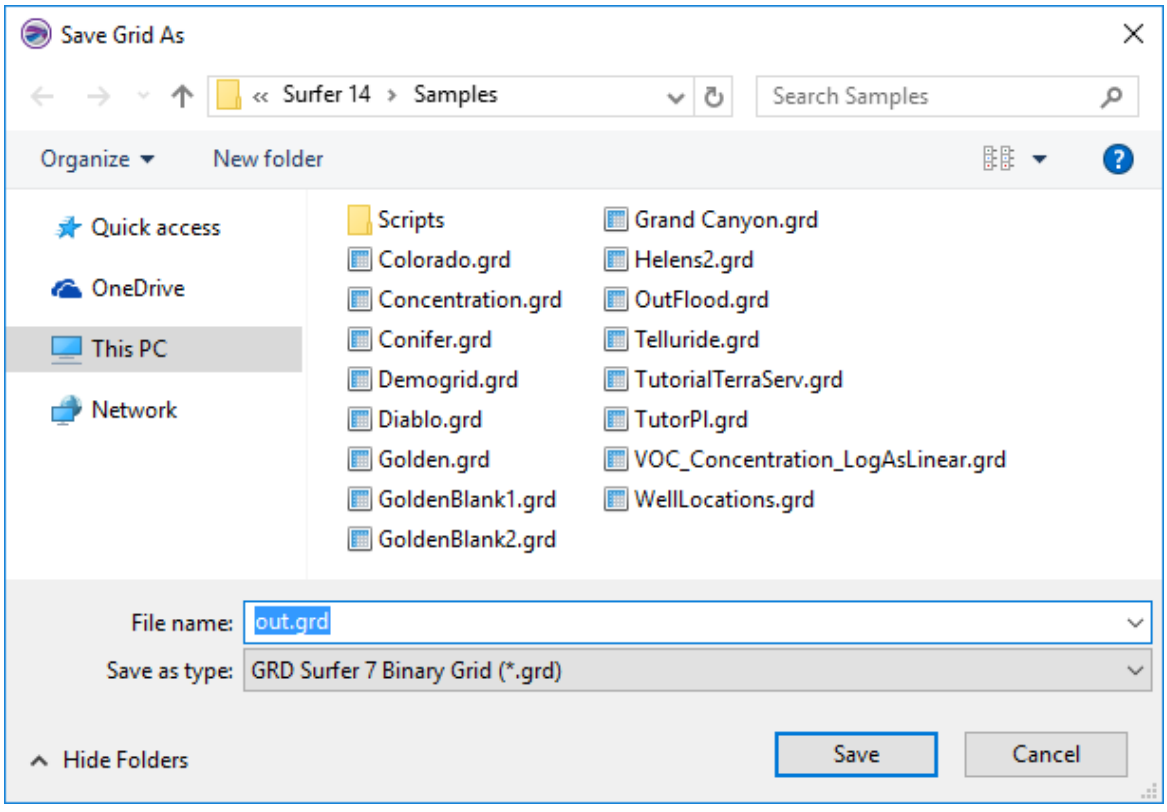
PNM/PPM/PGM/PBM Image
RAW Raw Binary Grid
RGB SGI-RGB Image
SID LizardTech MrSID Image
STK Metamorph
SUN Sun Raster Image
TAR SDTS DEM
TAR.GZ SDTS DEM
TGA Targa (TrueVision)
TGZ SDTS DEM
TIF Tagged Image
VTK Visualization Toolkit
X AVS X-Image
ZIP SDTS DEM
ZMap Plus Grid Format

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)
- 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

There are many ways to manipulate grid files in **Surfer**. The **Grids** tab 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 under the **Grids** tab.

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
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.

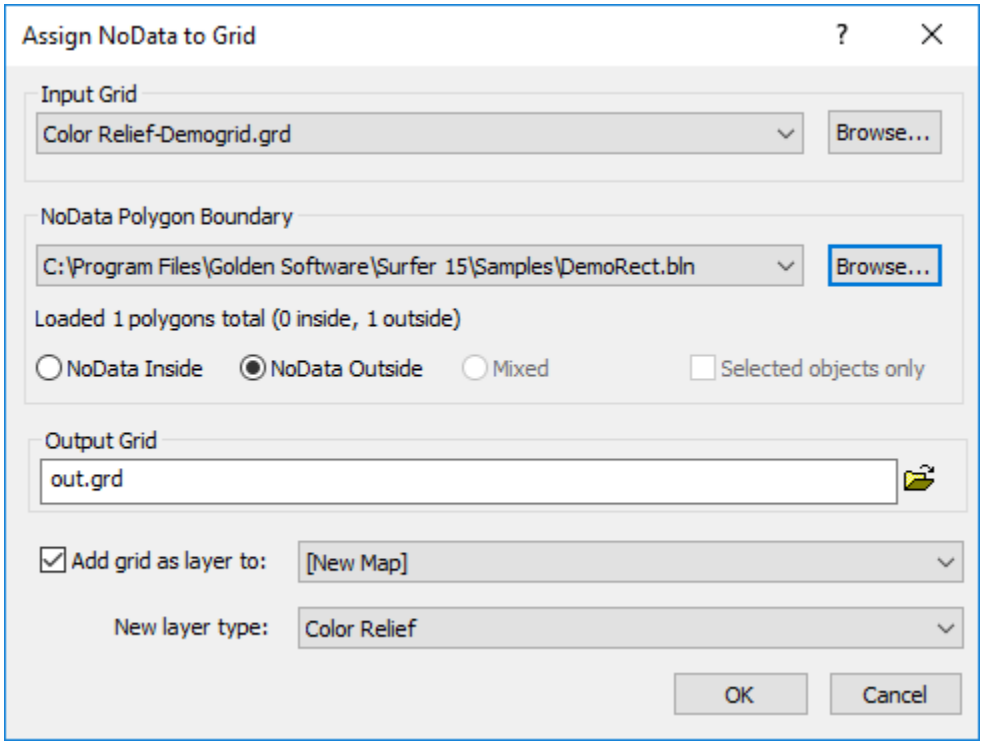
NoData polygon boundaries are defined in a vector file or in a base layer. The blanking file format can be used and is a special ASCII format file containing the X, Y coordinates defining the NoData polygon boundary. Blanking *.BLN files can be made in the **Surfer** worksheet. Blanking *.BLN files can also be created with the Digitize command. The NoData values can be assigned to areas inside or outside the polygon 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 boundaries is the 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.

Assign NoData to Grid Dialog

The **Assign NoData to Grid** dialog is displayed by clicking the **Grids | Edit | Assign NoData** command. 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.

NoData 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 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 *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 can be used for NoData polygon boundaries. The polylines in the base layer or vector file will be treated as polygons while assigning NoData values. The **Assign NoData** command is not recommended with open polylines, unexpected results my occur. Before clicking **Grids | Edit | Assign NoData**, consider converting polylines to polygons with the Polyline to Polygon command, 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.

NoData Inside, Outside, or Mixed

Select *NoData Inside* to assign the NoData value to the region inside the NoData polygon boundary or boundaries. Select *NoData Outside* to assign the NoData value to the region outside the NoData 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. A single polygon may be selected without entering editing mode. However, to select multiple polygons in a base layer, you must first begin editing mode. The polygon or polygons must be selected before clicking the **Grids | Edit | Assign NoData** command.

To select multiple polygons in a base layer, you must first begin edit group mode.

1. Select the base layer in the **Contents** window.
2. Click Features | Group | Start Editing.
3. Select the desired polygons in the plot window or **Contents** window.
4. Click **Grids | Edit | Assign NoData**.

Once you have assigned NoData values to the grid, stop editing the group by clicking **Features | Group | Stop Editing**.

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

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

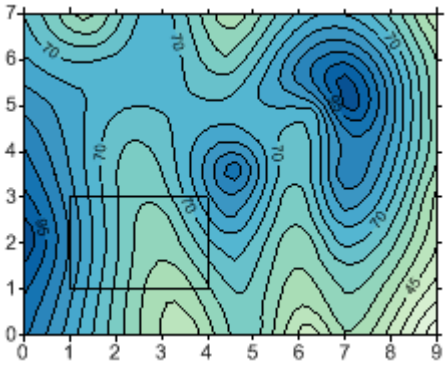
In this example, some of the nodes in the *Demogrid.grd* sample file will be assigned the NoData value with the *Demorect.bln* sample file.

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.

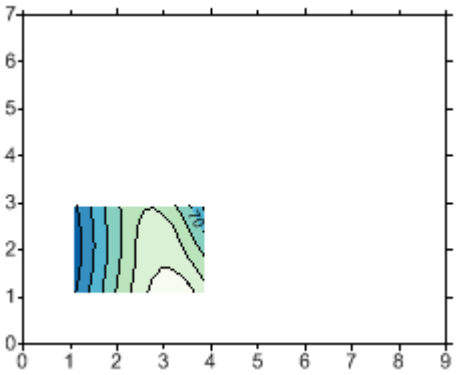
A new grid is created and saved to the file name specified in step 10. A new contour map is created and displayed in the plot window.

Before Assign NoData:



*The Blanking *.BLN file is displayed as a base map layer on the contour map.*

After Assign NoData:

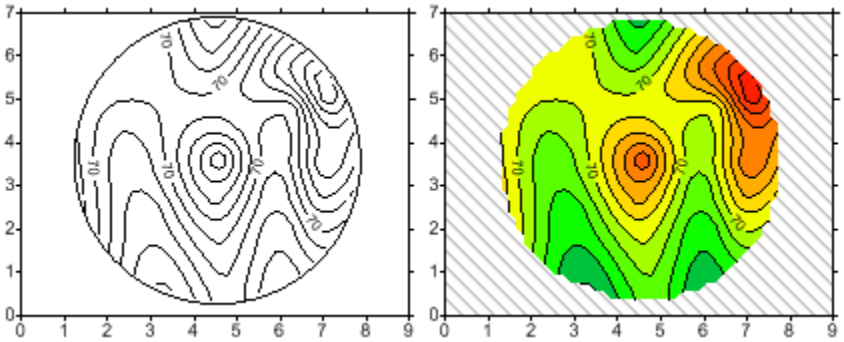


This is a contour map created from the new grid file.

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:

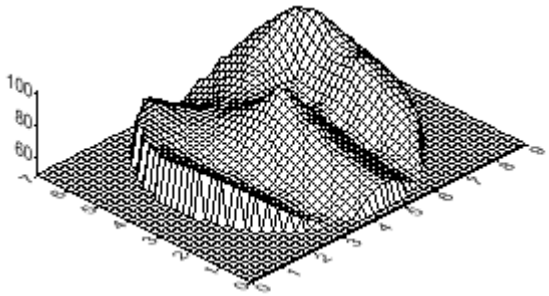
- Contours are not drawn.
- A separate fill color can be assigned to NoData regions/missing data in contour maps, color relief maps, and shaded relief maps.
- 3D wireframe maps display a flat surface at the minimum Z value in the grid file.
- Volumes are not calculated and areas are calculated separately.
- 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.

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* × *T*. Furthermore, the nodes in the neighborhood can be enumerated as:

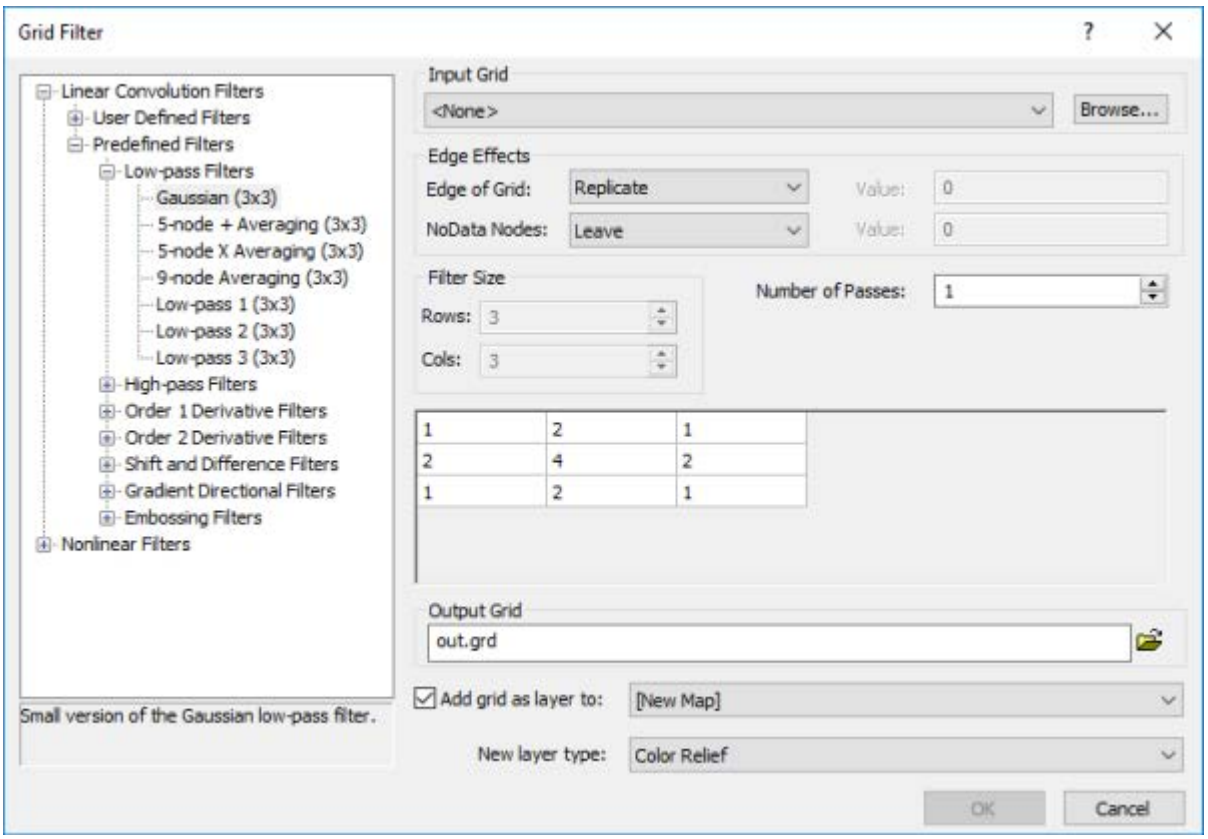
$$\left\{ Z(r+i, c+j) \text{ where } i = -\left\lfloor \frac{S}{2} \right\rfloor, \ldots, \left\lfloor \frac{S}{2} \right\rfloor \text{ and } j = -\left\lfloor \frac{T}{2} \right\rfloor, \ldots, \left\lfloor \frac{T}{2} \right\rfloor \right\}$$

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 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 c using a linear convolution filter with a neighborhood height *S* and width *T*.

$$\text{Numerator} = \sum_{i=-\left\lfloor \frac{S}{2} \right\rfloor}^{\frac{S}{2}} \sum_{j=-\left\lfloor \frac{T}{2} \right\rfloor}^{\frac{T}{2}} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ W(i,j)Z(r+i,c+j) & \text{otherwise} \end{cases}$$

$$\text{Denominator} = \sum_{i=-\left\lfloor \frac{S}{2} \right\rfloor}^{\left\lfloor \frac{S}{2} \right\rfloor} \sum_{j=-\left\lfloor \frac{T}{2} \right\rfloor}^{\left\lfloor \frac{T}{2} \right\rfloor} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ W(i,j) & \text{otherwise} \end{cases}$$

where *W(i, j)* are the weights defined for the specified filter. The output grid node value is then

$$Z_{out}(r,c) = \begin{cases} \dfrac{\text{Numerator}}{\text{Denominator}} & \text{if } |\text{Denominator}| > 0 \\ \text{Numerator} & \text{otherwise} \end{cases}$$

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 Weighting (mxn)

With the *Distance Weighting (mxn)* filter the weights fall-off with increased distance. The distance weighting function is

$$W(i,j) = \left(1 - \max\left(\frac{2|i|}{S+1}, \frac{2|j|}{T+1}\right)\right)^p \qquad \forall_i = -\left\lfloor \frac{S}{2} \right\rfloor, \ldots, \left\lfloor \frac{S}{2} \right\rfloor \forall_j = -\left\lfloor \frac{T}{2} \right\rfloor, \ldots, \left\lfloor \frac{T}{2} \right\rfloor$$

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

$$W(i,j) = \begin{cases} W_c & \text{if } i = 0 \text{ and } j = 0 \\ \left(\dfrac{1}{i^2 + j^2}\right)^{p/2} & \text{otherwise} \end{cases} \qquad \forall_i = -\left\lfloor \frac{S}{2} \right\rfloor, \ldots, \left\lfloor \frac{S}{2} \right\rfloor \forall_j = -\left\lfloor \frac{T}{2} \right\rfloor, \ldots, \left\lfloor \frac{T}{2} \right\rfloor$$

where W_C is the specified *Central Weight* and *p* is the *Power*. The higher the power *p*, the more rapidly the weights fall-off with distance. The resulting iso-weight contour lines are concentric circles. The higher the value W_C is, the more weight is given to the center point.

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* parameter 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. Conversely, 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 $$\text{Variance} = \frac{1}{N} \sum_{i=-\lfloor S/2 \rfloor}^{\lfloor S/2 \rfloor} \sum_{j=-\lfloor T/2 \rfloor}^{\lfloor T/2 \rfloor} \begin{cases} 0 & \text{if } Z(r+i, c+j) \text{ is blank} \\ \left(Z(r+i, c+j) - \overline{Z}\right)^2 & \text{otherwise} \end{cases}$$
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 N columns, the rules are

$$Z(r,c) = Z(a,b)$$

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 N columns, the rules are

$$Z(r,c) = Z(a,b)$$

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 N columns, the rules are

$$Z(r,c) = Z(a,b)$$

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.

$$Z(r,c) = \begin{cases} Z(r,r) & \text{if } 0 \le r < M \text{ and } 0 \le c < N \\ A & \text{otherwise} \end{cases}$$

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.
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.
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:

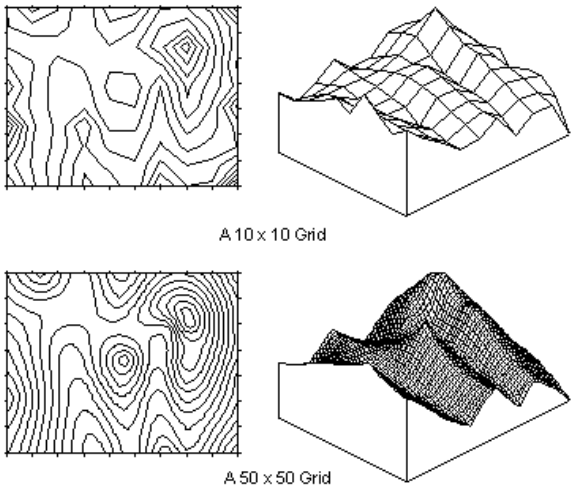
1. Click the **Grids | Edit | Spline Smooth** command.
2. In the **Spline Smooth** dialog, specify the input grid from a map layer or grid file in the *Input Grid* section.
3. Click the *Recalc Grid* option in the *Method* section.
4. In the *Final Grid Size* section, specify the number of rows and columns 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

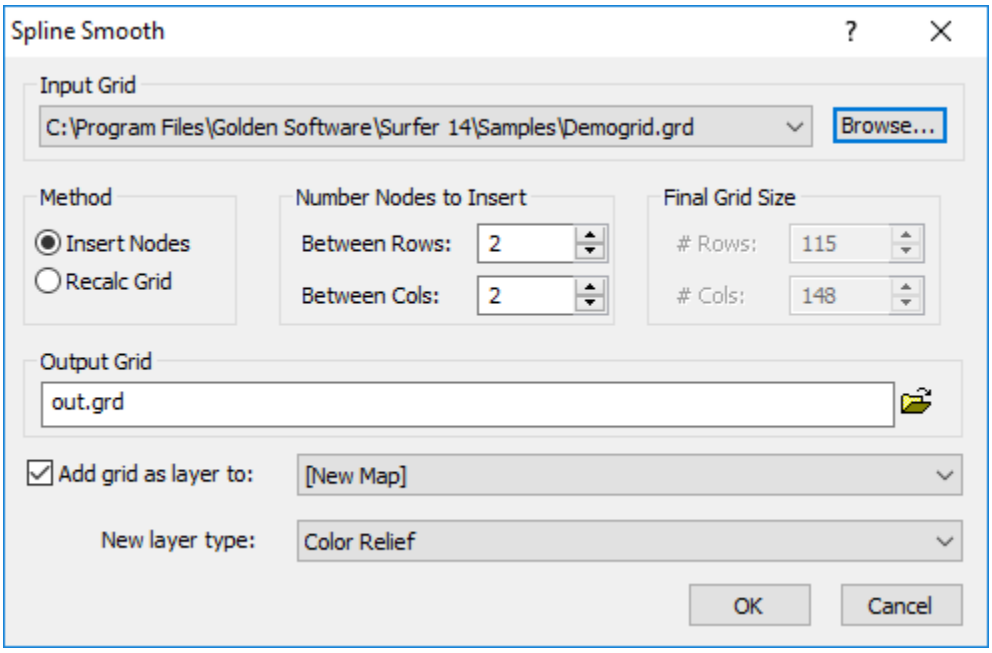
One of the purposes of spline smoothing is to fill in a sparse grid. A map produced from a sparse grid may have an angular appearance. For example, spline smoothing can be used to increase a 10 x 10 grid (a sparse grid) to a 50 x 50 grid. Denser grids produce smoother maps.



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 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.

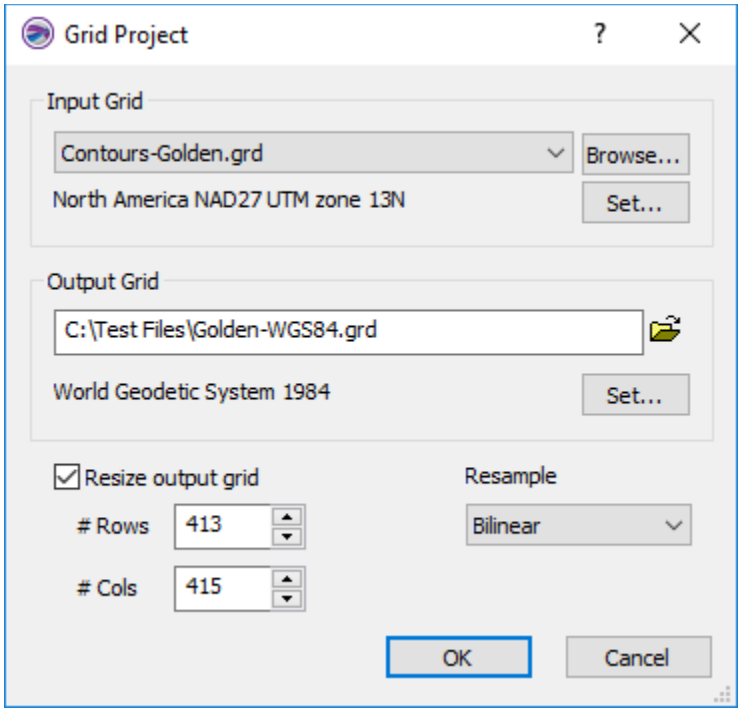
When any grid operation is performed on the referenced grid file, the **Export Options** dialog appears with the option to create a new .GSR2 file for the new output grid. For instance, if you use the Grids | Edit | Filter command and smooth the grid, the output smoothed grid will contain a .GSR2 file 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

The **Grids | Edit | Project** command creates 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.

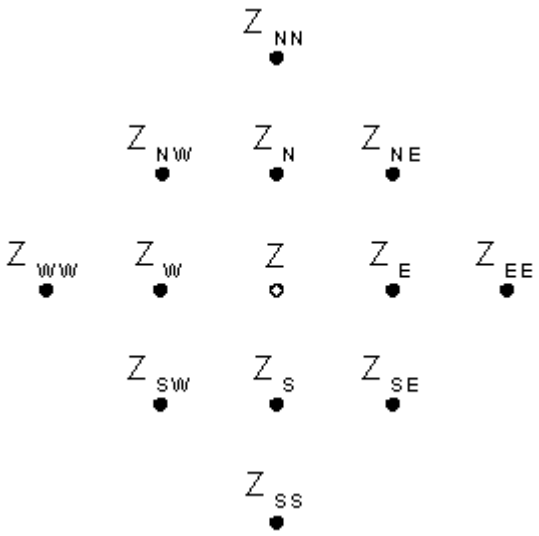
Grid Calculus

The **Grids | Calculate | Calculus** command provides you with 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:

$$\frac{dz}{dx} \approx \frac{Z_E - Z_W}{2\Delta x}$$

$$\frac{dz}{dy} \approx \frac{Z_N - Z_S}{2\Delta y}$$

$$\frac{d^2z}{dx^2} \approx \frac{Z_E - 2Z + Z_W}{\Delta x^2}$$

$$\frac{d^2z}{dy^2} \approx \frac{Z_N - 2Z + Z_S}{\Delta y^2}$$

$$\frac{d^2z}{dx\,dy} \approx \frac{Z_{NE} - Z_{NW} - Z_{SE} + Z_{SW}}{4\Delta x\,\Delta y}$$

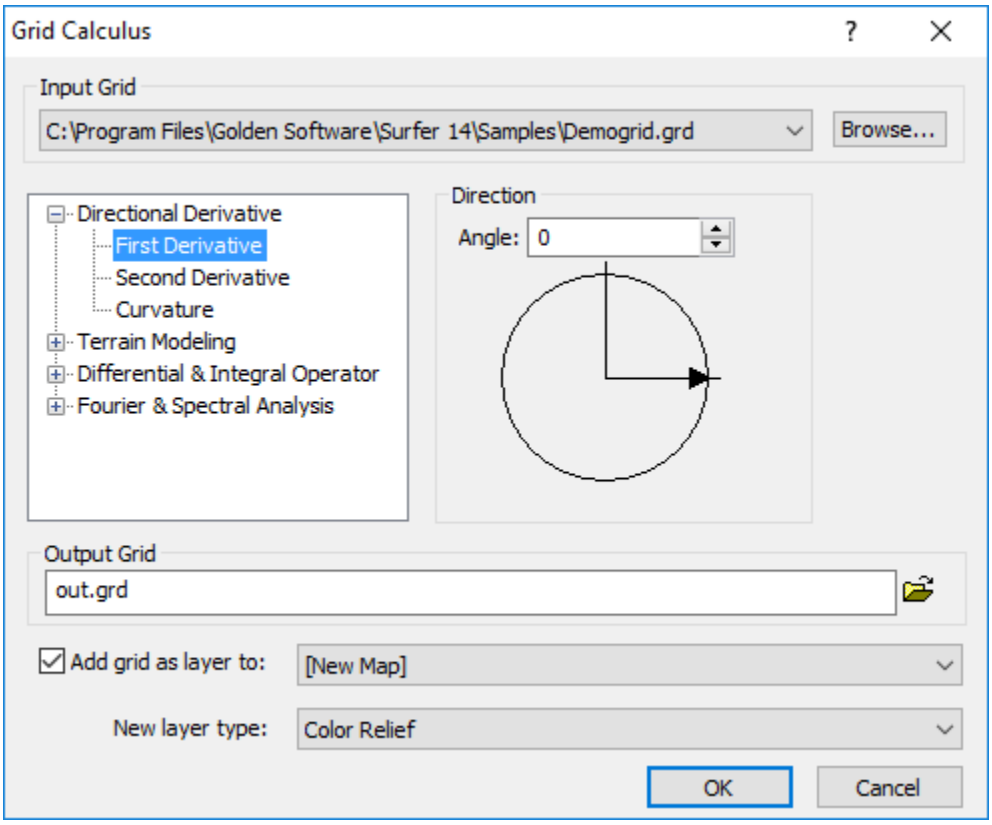
$$\frac{d^4z}{dx^4} \approx \frac{Z_{WW} - 4Z_W + 6Z - 4Z_E + Z_{EE}}{\Delta x^4}$$

$$\frac{d^4z}{dy^4} \approx \frac{Z_{NN} - 4Z_N + 6Z - 4Z_S + Z_{SS}}{\Delta y^4}$$

$$\frac{d^4z}{dx^2\,dy^2} \approx \frac{Z_{NW} - 2Z_N + Z_{NE} - 2Z_W + 4Z - 2Z_E + Z_{SW} - 2Z_S + Z_{SE}}{4\,\Delta x\,\Delta y}$$

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, y) be defined for domain D. Let P be a point of D, let L be a line through P, let Q be a point of L at a distance Δs from P,

$$\gamma(h) = C_0 + S \cdot h,$$

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 L, that is from P to Q.

There are three directional derivative options available: *First Derivative*, *Second Derivative*, and *Curvature*. You can specify the angular direction for the operation.

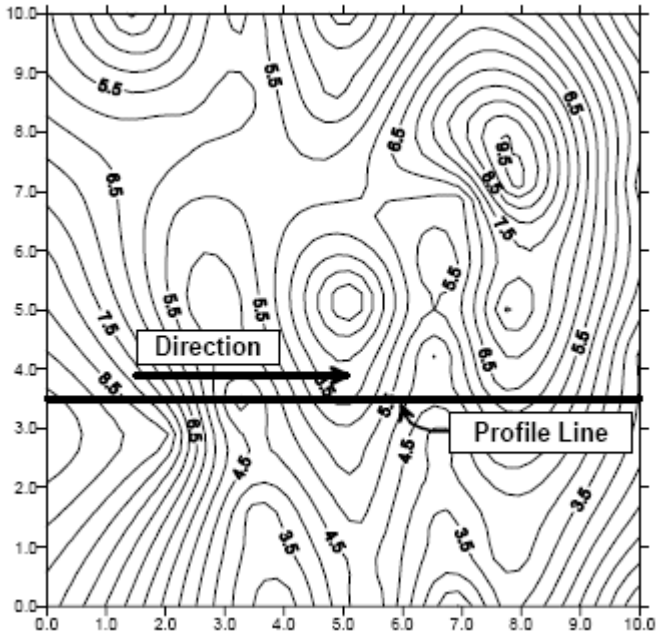
First Derivative

Second Derivative

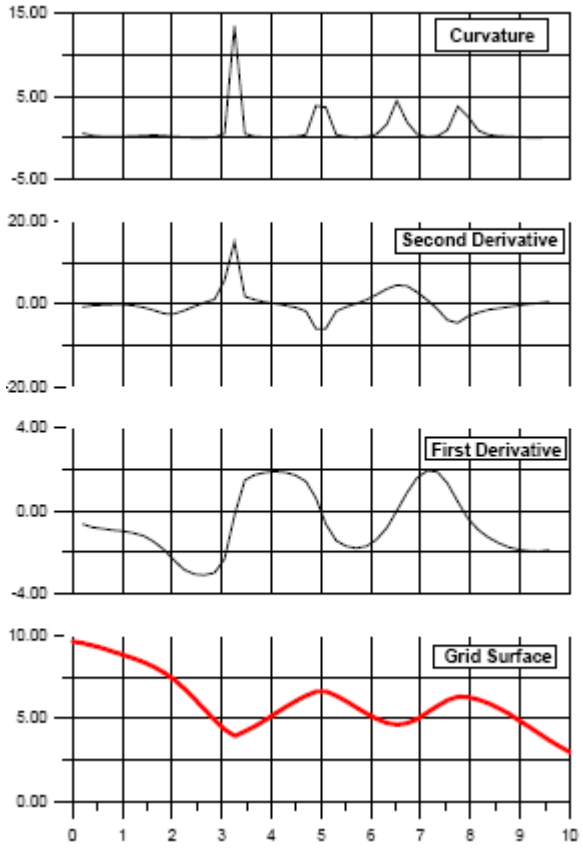
Curvature

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.

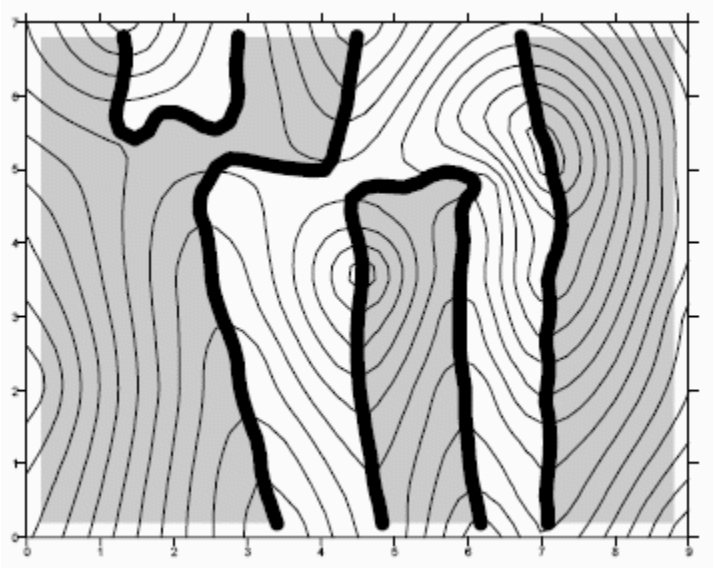


These are cross sectional views of Curvature, Second, and First Derivatives, and the original grid surface.

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:

$$\frac{df}{ds} = \vec{g} \cdot \begin{bmatrix} \cos(\alpha) \\ \sin(\alpha) \end{bmatrix}$$

$$= \left[\frac{df}{dx}, \frac{df}{dy}\right] \cdot \begin{bmatrix} \cos(\alpha) \\ \sin(\alpha) \end{bmatrix}$$

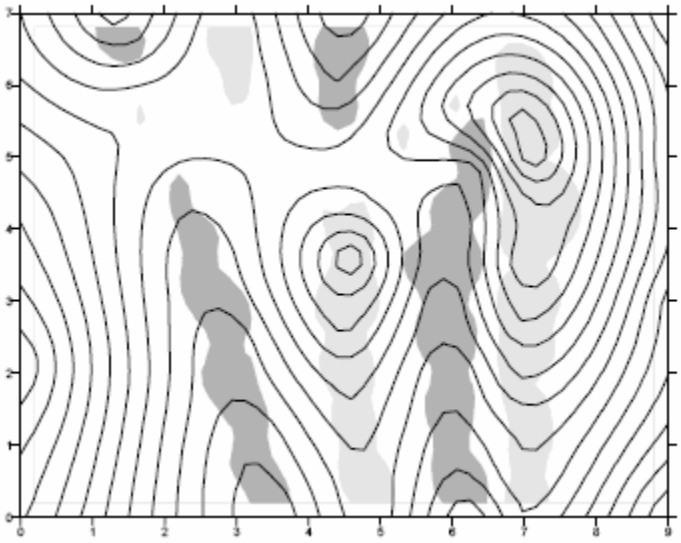
$$= \frac{df}{dx} \cdot \cos(\alpha) + \frac{df}{dy} \cdot \sin(\alpha)$$

where α 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:

$$\frac{d^2f}{ds^2} = \frac{d\left[\frac{df}{ds}\right]}{ds}$$

$$\frac{d^2f}{ds^2} = \frac{d\left[\frac{df}{dx}\cdot\cos(\alpha) + \frac{df}{dy}\cdot\sin(\alpha)\right]}{ds}$$

$$= \frac{d^2f}{dx^2}\cdot\cos^2(\alpha) + 2\frac{d^2f}{dxdy}\cdot\cos(\alpha)\cdot\sin(\alpha) + \frac{d^2f}{dy^2}\cdot\sin(\alpha)$$

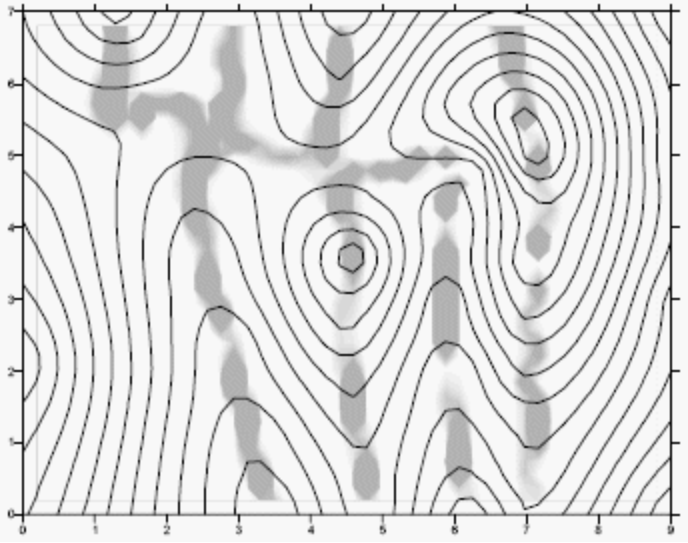
The compass-based grid notation difference equation used in **Surfer** takes the form:

$$\frac{d^2Z}{ds^2} \approx \frac{Z_E - 2Z + Z_W}{\Delta x^2}\cdot\cos^2(\alpha) + \frac{Z_{NE} - Z_{NW} - Z_{SE} + Z_{SW}}{4\Delta x\Delta y}\cdot\cos(\alpha)\cdot\sin(\alpha) + \frac{Z_N - 2Z + Z_S}{\Delta y^2}\cdot\sin^2(\alpha)$$

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, y) in a direction defined by s is:

$$K_s = \frac{\left|\frac{d^2f}{ds^2}\right|}{\left[1 + \left(\frac{df}{ds}\right)^2\right]^{3/2}}$$

(see Schwartz, 1974, p. 592).

$$K_\alpha = \frac{\left|\frac{d^2f}{dx^2}\cdot\cos^2(\alpha) + 2\frac{d^2f}{dxdy}\cdot\cos(\alpha)\cdot\sin(\alpha) + \frac{d^2f}{dy^2}\cdot\sin(\alpha)\right|}{\left[1 + \left(\frac{df}{dx}\cdot\cos(\alpha) + \frac{df}{dy}\cdot\sin(\alpha)\right)^2\right]^{3/2}}$$

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:

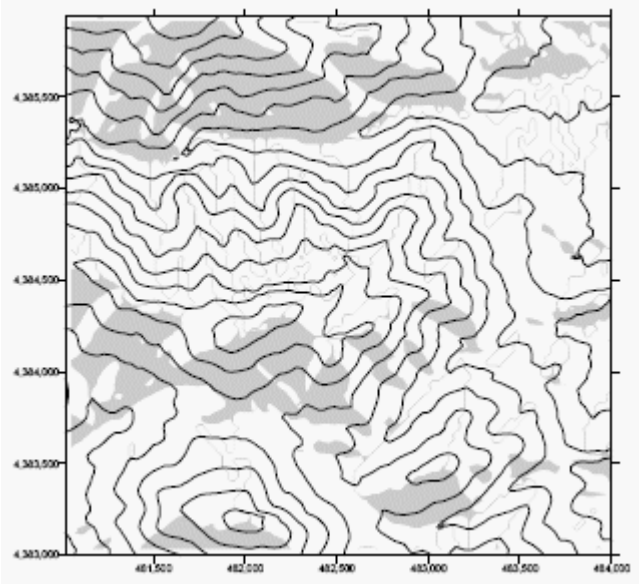
- Terrain Slope
- Terrain Aspect
- Profile Curvature
- Plan Curvature
- Tangential Curvature

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:

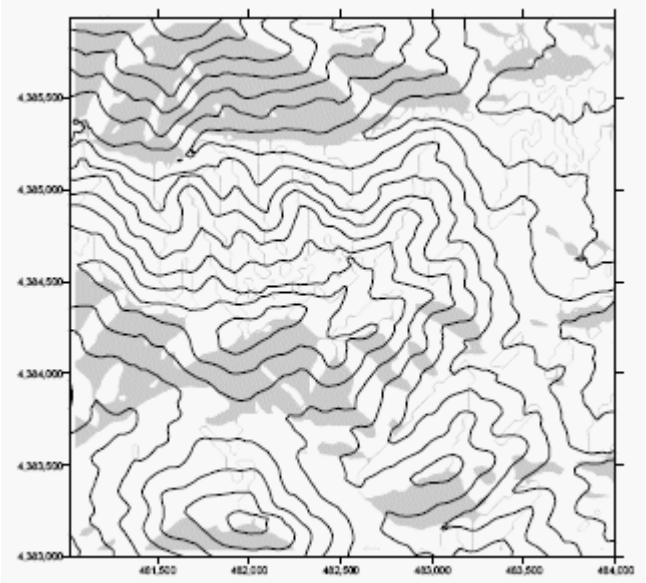
$$S \approx \sqrt{\left(\frac{Z_E - Z_W}{2\Delta x}\right)^2 + \left(\frac{Z_N - Z_S}{2\Delta y}\right)^2}$$

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):

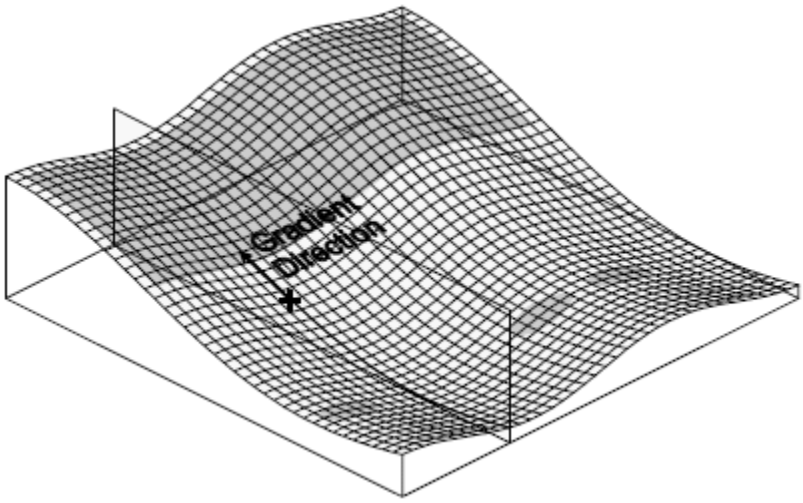
$$A_T = 270 - \frac{360}{2\pi} \cdot \text{atan2}\left[\frac{\partial z}{\partial y}, \frac{\partial z}{\partial x}\right]$$

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 K_P is given by:

$$K_p = \frac{\left(\frac{\partial^2 z}{\partial x^2}\right)\left(\frac{\partial z}{\partial x}\right)^2 + 2\left(\frac{\partial^2 z}{\partial x \partial y}\right)\left(\frac{\partial z}{\partial x}\right)\left(\frac{\partial z}{\partial y}\right) + \left(\frac{\partial^2 z}{\partial y^2}\right)\left(\frac{\partial z}{\partial y}\right)^2}{pq^{3/2}}$$

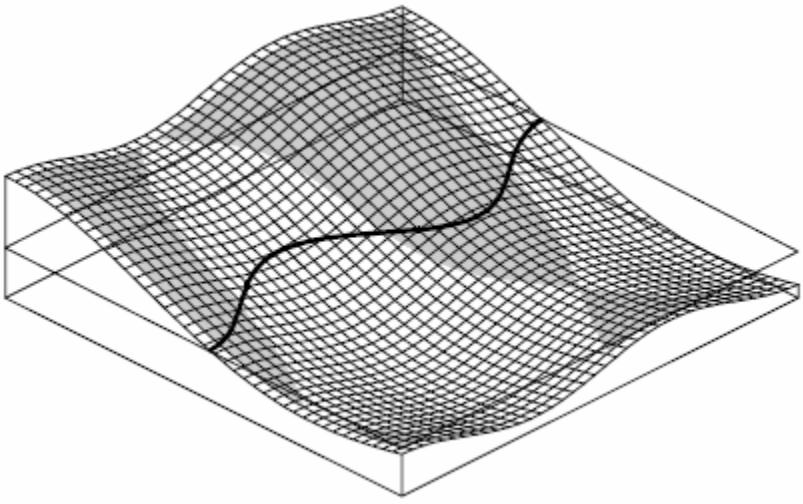
where

$$p = \left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2$$

$$q = 1 + p$$

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 K_H is given by:

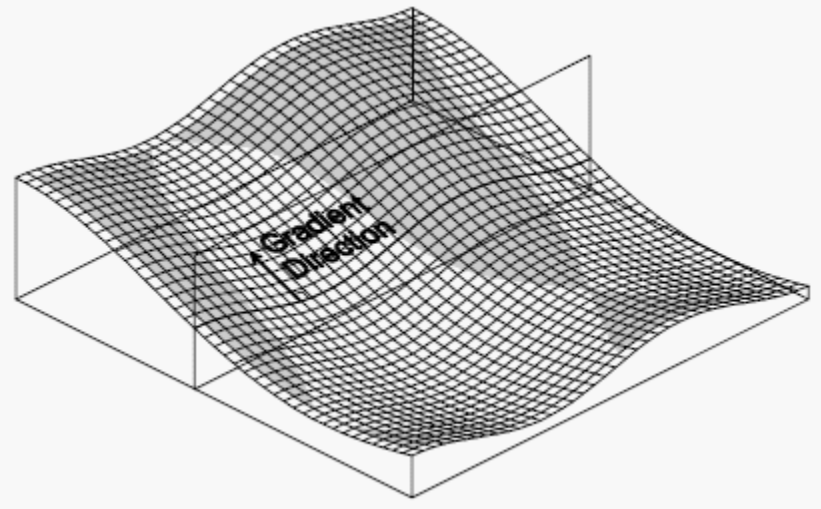
$$K_H = \frac{\left(\frac{\partial^2 z}{\partial x^2}\right)\left(\frac{\partial z}{\partial y}\right)^2 - 2\left(\frac{\partial^2 z}{\partial x \partial y}\right)\left(\frac{\partial z}{\partial x}\right)\left(\frac{\partial z}{\partial y}\right) + \left(\frac{\partial^2 z}{\partial y^2}\right)\left(\frac{\partial z}{\partial x}\right)^2}{p^{3/2}}$$

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* K_H by the sine of the slope γ: $K_H = K_T / \sin \gamma$.



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 K_T is given by the equation:

$$K_T = \frac{\left(\frac{\partial^2 z}{\partial x^2}\right)\left(\frac{\partial z}{\partial y}\right)^2 - 2\left(\frac{\partial^2 z}{\partial x \partial y}\right)\left(\frac{\partial z}{\partial x}\right)\left(\frac{\partial z}{\partial y}\right) + \left(\frac{\partial^2 z}{\partial y^2}\right)\left(\frac{\partial z}{\partial x}\right)^2}{pq^{1/2}}$$

where

$$p = \left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2$$

$$q = 1 + p$$

Terrain Modeling References

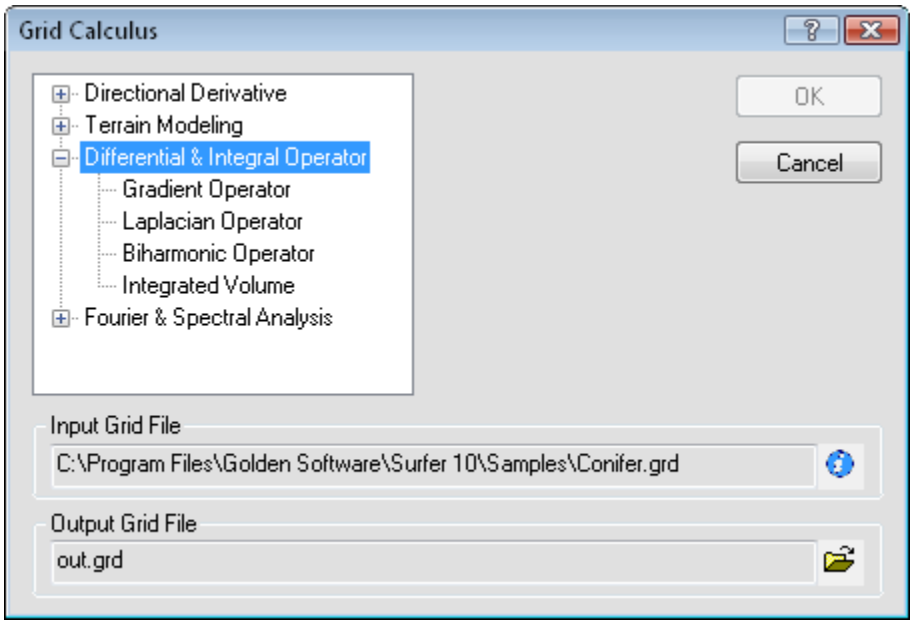
The mathematical definitions used in this implementation were taken from Mitasova and Hofierka (1993). For a general review of the methods and applications of topographic analysis, see Moore et al. (1991).

Mitasova, Helena and Jarosalav 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., 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.

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:

$$\|\nabla z\| = \sqrt{\left(\frac{\partial z}{\partial x}\right)^2 + \left(\frac{\partial z}{\partial y}\right)^2}$$

Using the compass based grid notation yields the equation used in **Surfer**:

$$\|\nabla z\| \approx \sqrt{\left(\frac{Z_E - Z_W}{2\Delta x}\right)^2 + \left(\frac{Z_N - Z_S}{2\Delta y}\right)^2}$$

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 subgrids of any shape. As an example, consider a **Surfer**-generated contour map showing the thickness of a coal seam in a surface coal mine. The *Integrated Volume* map could be used to compute the volume of coal mined in each month. Another example is rainfall intensity for a thunderstorm. If you computed the *Integrated Volume* and then overlaid a watershed boundary map, you could directly compute the quantity of water flowing into each stream.

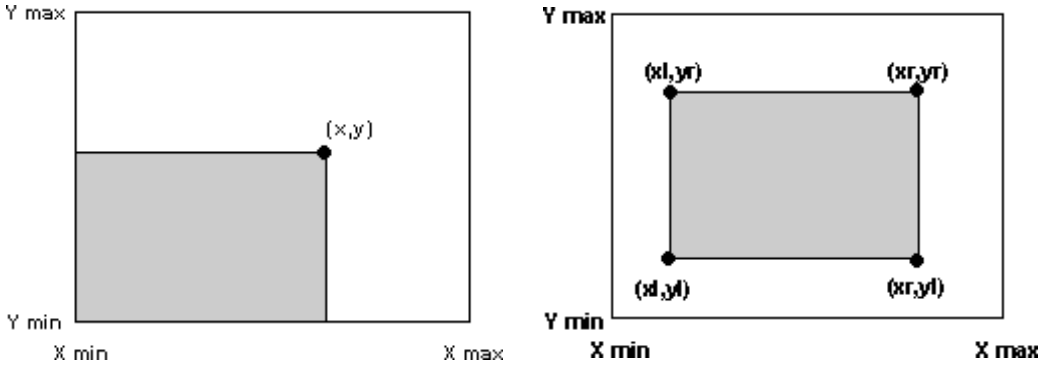
Mathematically, the integrated volume is defined by

$$V(x,y) = \int_{\alpha=xmin}^{x} \int_{\beta=ymin}^{y} z(\alpha,\beta)\, d\alpha\, d\beta$$

Consider a surface defined over a rectangular region extending from X min to X max and Y min to Y max:

V(x, y) is then the volume between the plane Z=0 and the surface under the shaded area. Thus, V(xMin,yMin) = 0, and V(xMax,yMax) equals the volume under the entire rectangular region, which is comparable to the value returned by the Grids | Calculate | Volume command in **Surfer**. If the grid node at (xMin,yMin) is assigned the NoData value, then the *Integrated Volume* cannot be calculated.

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, yr) - V(xl, yr) - V(xr, yl) + V(xl, yl). When applied to a surface specified by a **Surfer** grid, the *Integrated Volume* option generates a dimensionally equivalent grid with the values at grid nodes (x_i, y_i) equal to V(x_i, y_i). You can use the Grid Editor to obtain the volume values at different XY locations. The integration is carried out assuming that the surface follows a bilinear functional form over each grid cell.



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.

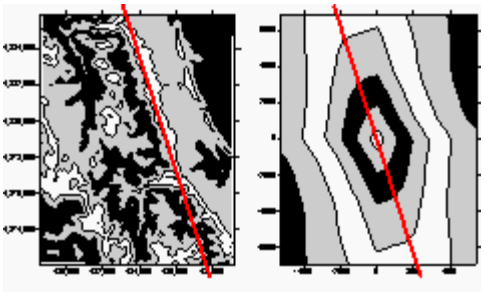
Example

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, y) = Z(-x,- y). Grid correlograms have the same dimensions as the parent grids, but the axes represent separation distances.



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) is defined as

$$C(r,s) = \frac{1}{NX \cdot NY} \sum_u \sum_v \left[z(u,v) - \bar{z} \right] \left[z(u+r, v+s) - \bar{z} \right]$$

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:

$$\text{corr}(r,s) = \frac{C(r,s)}{C(0,0)}$$

C(0,0) is the variance of the data in the grid.

The functions C(r,s) and corr(r,s) are symmetric in the sense that

$$C(-r,-s) = C(r,s)$$

but,

$$C(-r,s) \neq C(r,s)$$

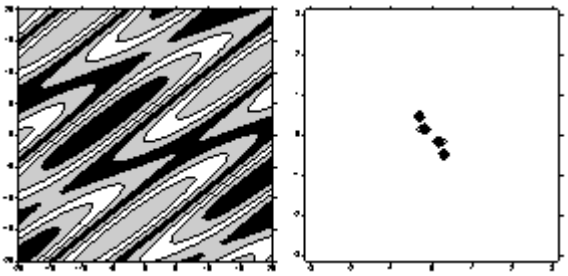
in general. As such, the grids generated by the grid correlogram option of the **Grids | Calculate | Calculus** sub-system show this symmetry.

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.

Surfer computes 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, y) = Z(-x,- y).

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, $I(\lambda, \mu)$ is formally defined as

$$I(\lambda,\mu) = \frac{N}{4\pi^2}\left|\frac{1}{N}\sum_{u}\sum_{v}\left[z(u,v)-\bar{z}\right]e^{-iu\lambda}e^{-iv\mu}\right|^2$$

where $i = \sqrt{-1}$.

The value of I(λ, μ) is the weight or strength of the sinusoid with a two dimensional frequency of (λ, μ).

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

$$-\pi < \lambda \leq +\pi$$
$$-\pi < \mu \leq +\pi$$

Surfer does not compute the grid periodogram, I(λ, μ), for all possible frequencies (λ,μ) in this range. Rather, it computes the grid periodogram for the two-dimensional *Fourier 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 (λ,μ). **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: I(j, k). The wave number is defined as:

wave number = 2π / wavelength (e.g. radians/meter)

The maximum wave number is the Nyquist frequency = $\pi/\Delta X$ (or Y), which defines the grid limits. Since **Surfer** maintains the grid dimensions (number of columns and rows) from the source grid to the destination grid, there may not be exactly the same number of rows and columns on both sides of the zero lines.

Grid Volume

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. The upper and lower surfaces are defined by a grid file or a plane of constant Z level. When two grid files are specified for the upper and lower surfaces, the grid files must have the same number of rows and columns and the same X and Y limits. 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.

Volume results are provided in cubic units based on the units of the input grid file.

Introduction to Volumes and Areas

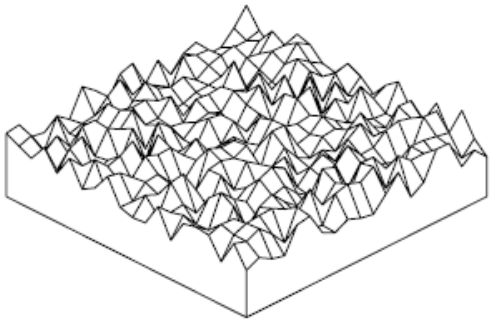
The procedures described here are used to provide more information about the data contained in grid files. The topics discussed include volumes and areas.

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:

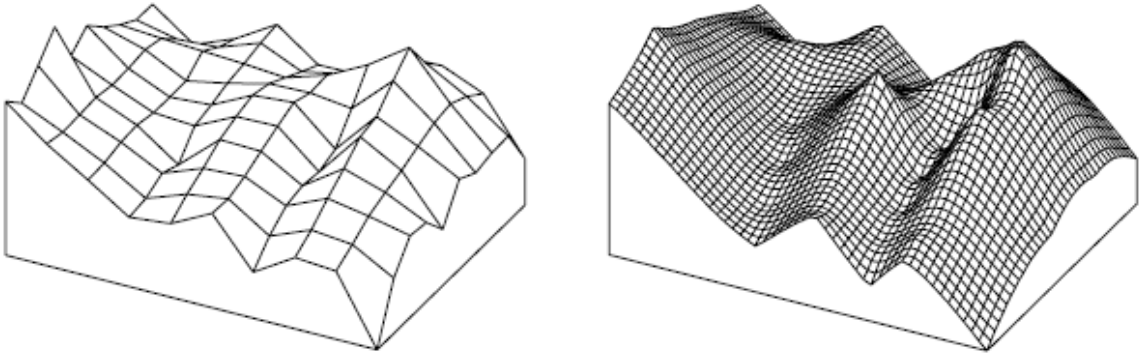
- Net volume between two grid files
- Net volume between a grid file and a horizontal plane
- Cut and fill volumes
- Surface area of a grid file above and below a specified level
- Planar area of a grid above and below a specified level

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.

Calculating Volumes and Areas

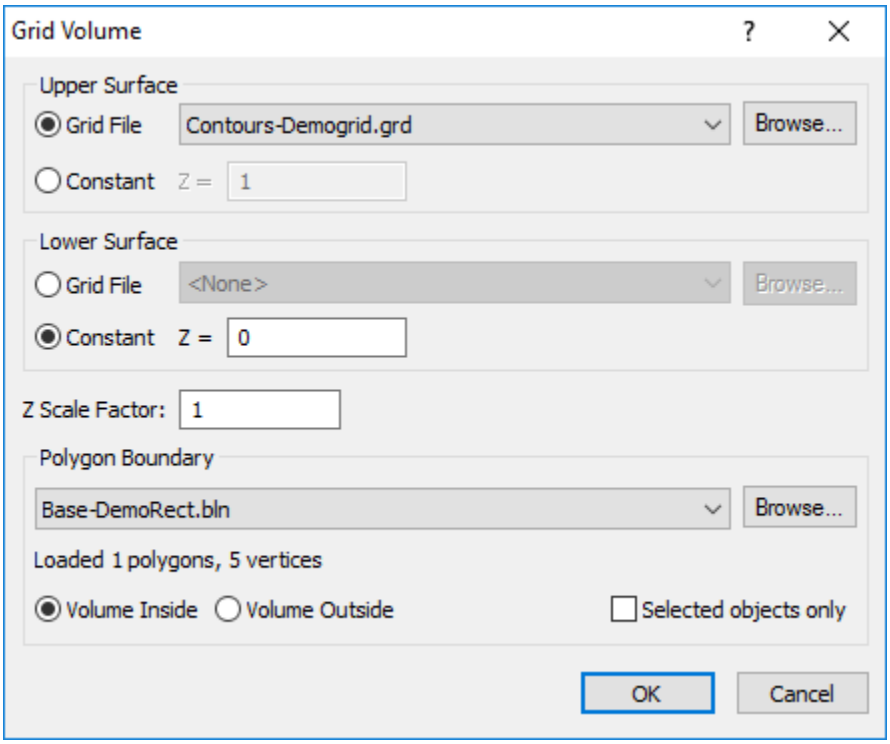
Use the **Grids | Calculate | Volume** command to calculate net volumes, cut and fill volumes, planar areas, and surface areas.

1. Click the **Grids | Calculate | Volume** command.
2. Specify the *Upper Surface* and *Lower Surface* parameters.
   - 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. 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. *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.
4. Click *OK* in the **Grid Volume** dialog and the results are displayed in a report.
5. 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.

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, the volume is calculated inside or outside all of the 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
- BNA Atlas Boundary
- DDF SDTS TVP
- DLG USGS Digital Line Graph
- DXF AutoCAD Drawing
- E00 Esri ArcInfo Export Format
- EMF Windows Enhanced Metafile
- GSB Golden Software Boundary
- GSI Golden Software Interchange
- KML Google Earth
- MIF MapInfo Interchange Format
- SHP Esri Shapefile
- WMF Windows Metafile

Volume Inside or Volume Outside

Click *Volume Inside* to calculate the volume inside of the polygon or polygons in the base layer or vector file. Click *Volume Outside* to calculate the volume outside of the polygon or polygons 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. A single polygon may be selected without editing the group. However, to select multiple polygons in a base layer, you must first begin editing the group. The polygon or polygons must be selected before clicking the **Grids | Calculate | Volume** command.

1. Select the base layer in the **Contents** window.
2. Click Features | Group | Start Editing.
3. Select the desired polygons.
4. Click **Grids | Calculate | Volume**.

Once you have calculated the volume, exit edit group mode by clicking **Features | Group | Stop Editing**.

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.

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 $(\text{meter})^3$. 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 $(\text{feet})^3$.

Net Volume = (feet * feet * feet)

Example 2

A grid with X, Y, and Z units in meters, the units for the results are $(\text{meter})^3$.

Net Volume = (meter * meter * meter)

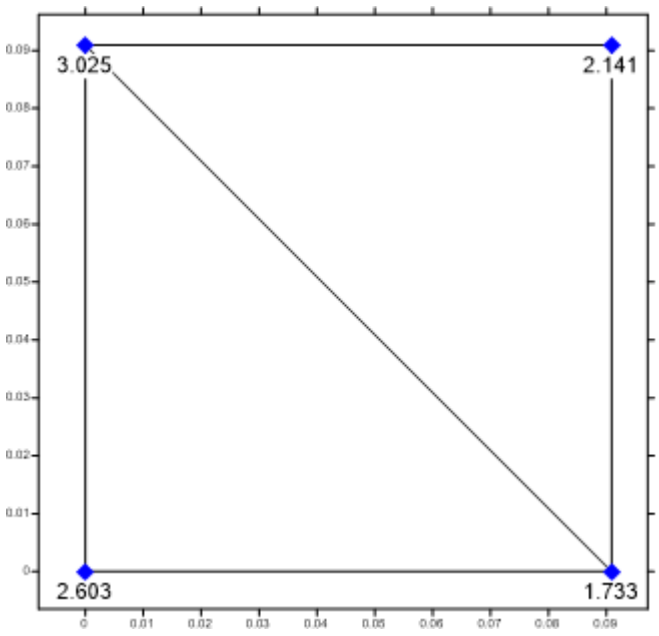
Cut and Fill Volumes

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

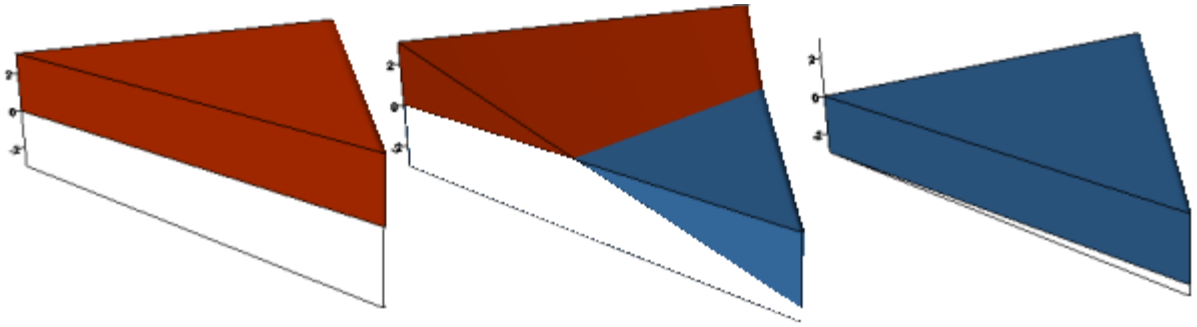
The *Cut & Fill Volumes* section uses a different calculation technique than the three methods used to define the volume, Extended Trapezoidal Rule, Extended Simpson's Rule, and Extended Simpson's 3/8 Rule. For determining cut and fill volumes, first the lower surface grid values are subtracted from 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.



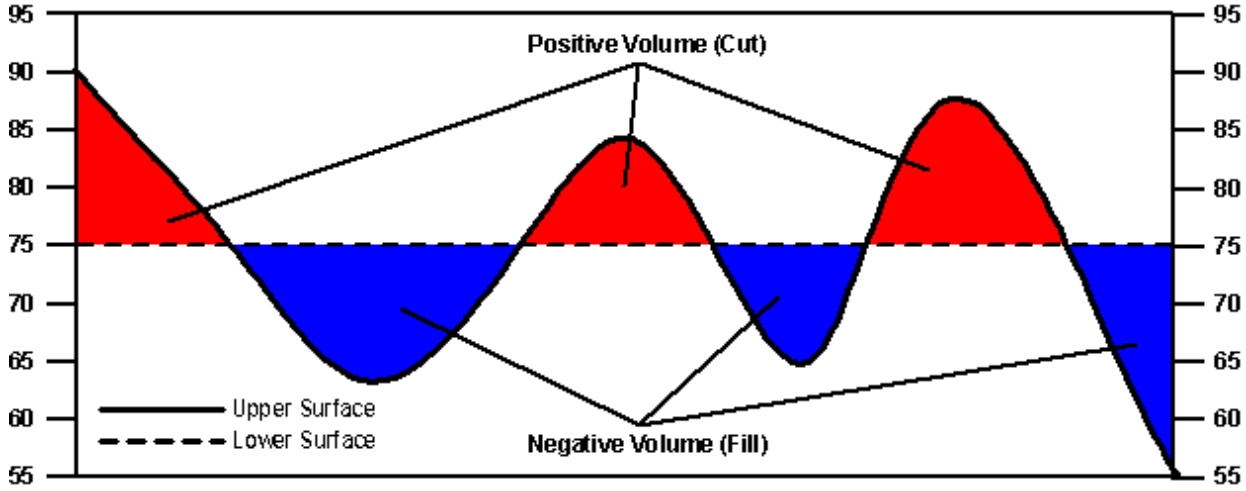
Prisms are created from the net grid values and the horizontal planar surface at Z=0. The red prisms above are positive (cut) volumes. The blue prisms are negative (fill) volumes.

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

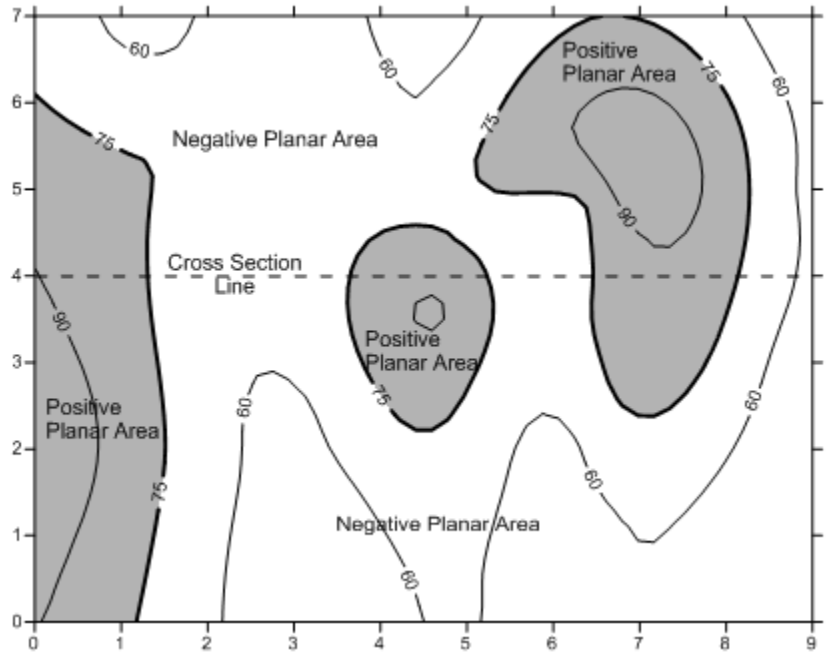
Planar area is computed by projecting the cut and fill portions of the surface onto a plane and calculating the area of the projection.

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.



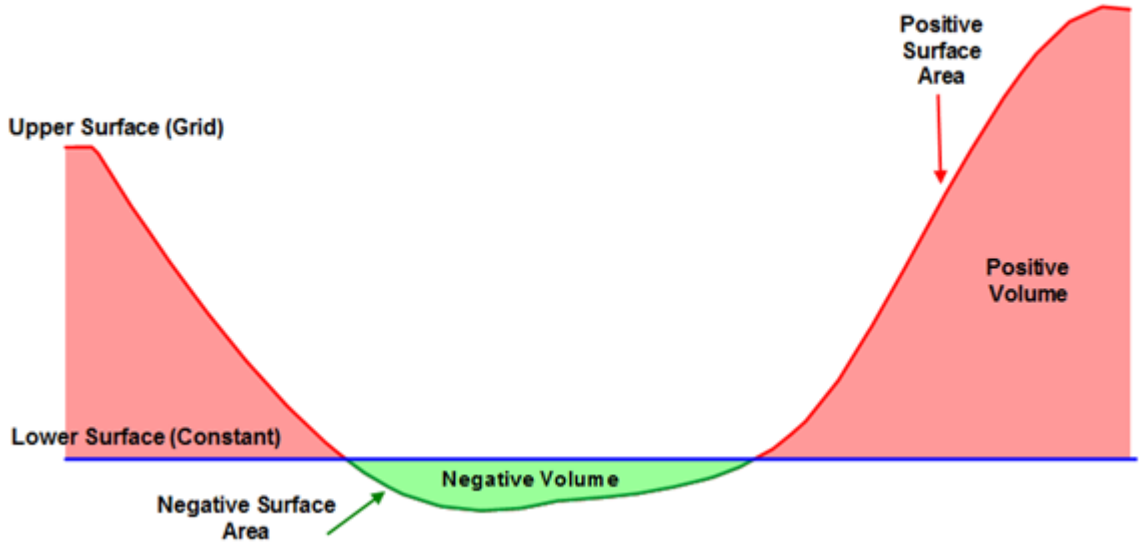
This is a contour map showing the relation between positive and negative planar area on a map. The lower surface is defined by a plane where Z = 75.

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.

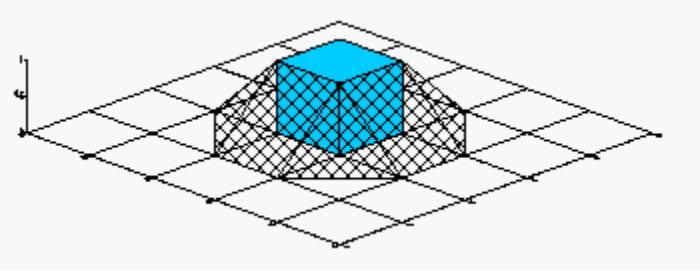
To find the volume between two surfaces:

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.

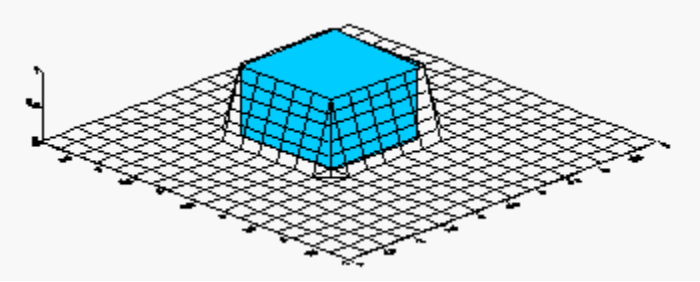
Calculating the Volume of a Grid with NoData Values

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.

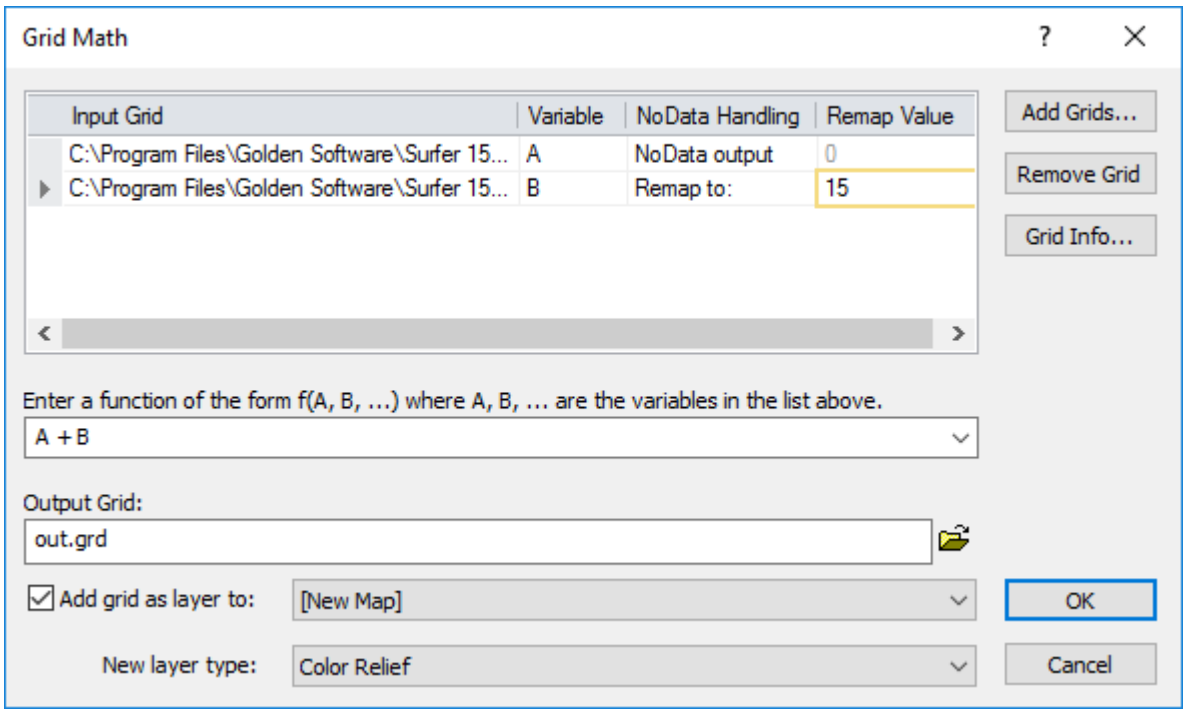
Grid Math

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. The defined function is performed on corresponding nodes with the same row and column coordinates from the input grid files, and 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.

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.

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.

All selected grids must contain the same number of grid rows and columns, and should extend over the same X and Y ranges.

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.

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 ten 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.
2. In the **Grid Math** dialog, click the *Add Grids* button. Select any number of grid files and click *Open*. All grids must contain the same number of grid rows and columns, and the same X and Y ranges.
3. In the *Output Grid* section, click the button to specify a different path or file name for the grid file to be created.
4. 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.
5. 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 Generate an Isopach Map

The **Grids | Calculate | Math** command can be used to generate an isopach map.

For example, let's say you have drill hole data that provides the elevation of the top and bottom of a coal seam, and want to generate a contour map representing the thickness of the seam. The elevation of the top of the coal seam can define the upper surface, and the elevation of the base of the coal seam could define the lower surface. Using grid math to subtract the lower surface from the upper surface would generate a grid file that represents the thickness of the coal seam.

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 XY ranges, and same number of rows and columns.
2. Select the Grids | Calculate | Math command.
3. In the **Grid Math** dialog, click the *Add Grids* button. Select the grid file for the upper and lower surface and click *Open*. The grid files are listed in the *Input Grid* column.
4. Click the button next to *Output Grid File*, enter the name for the thickness grid file in the **Save Grid As** dialog, and click *Save*.
5. In the *Enter a function of the form f(A,B,...) where A, B, ... are the variables in the list above.* box, enter the formula A - B. This formula assumes that the top of the surface is the *Variable* A and the bottom of the surface is the *Variable* B.
6. Click *OK* and a grid file is created which can be used to generate the isopach map of the thickness between the two input grid files.
7. To view a contour map of the thickness grid file, choose the Home | New Map | Contour command and choose the grid file you created with the **Grid Math** command.

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:

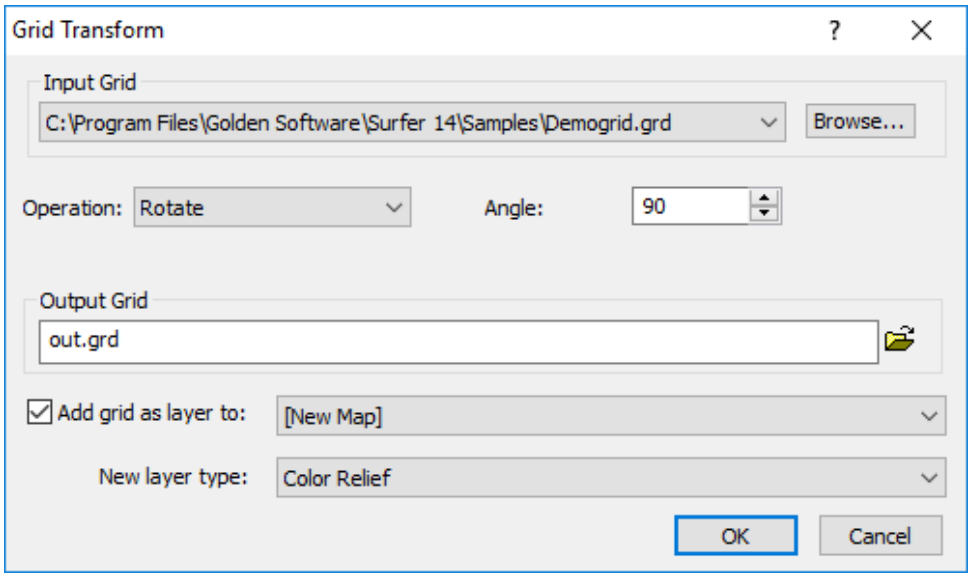
1. Click the **Grid | Math** command.
2. In the **Grid Math** dialog, click the *Add Grids* button. Select the grid file containing the unwanted negative values and click *Open*. This grid file appears in *Input Grid* column.
3. Select an output grid file name in the *Output Grid File* section by clicking 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.

Grid Transform Dialog

Click the **Grids | Calculate | Transform** command 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

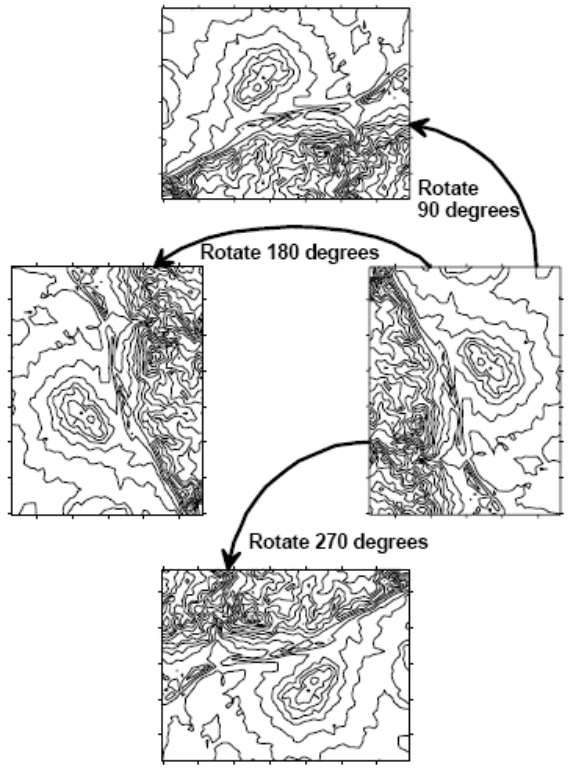
The *Offset* option adds or subtracts a number from the X and Y dimensions. Specify the *Offset* option from the *Operation* list, and specify the amount of the offset in the *X Offset* and *Y Offset* boxes in positive or negative units. For example, if the input grid ranges are X: 1 to 9 and Y: 2 to 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).

Rotate

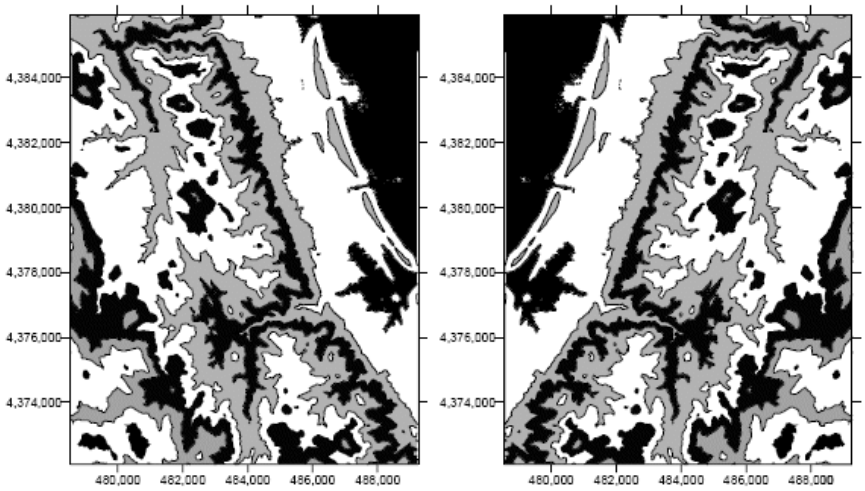
The *Rotate* option rotates a grid in multiples of 90 degrees (0, 90, 180, 270, …). Select the *Rotate* option in the *Operation* list, and specify the rotation angle in the *Angle* box. The origin of the map remains in the lower left, and the axes continue to increase in the same direction, with the 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

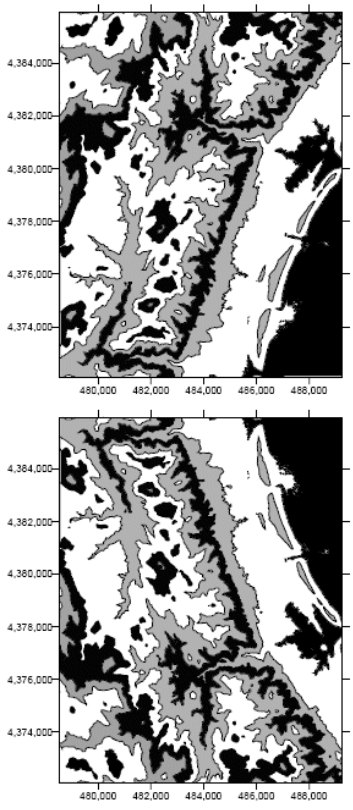
The *Mirror X* option creates a new grid file with the order of the grid nodes reversed in the X direction. The output grid file uses the same minimum and maximum X and Y coordinates as the input grid file. The origin of the grid remains in the lower left corner, with X increasing to the right and Y 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.
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

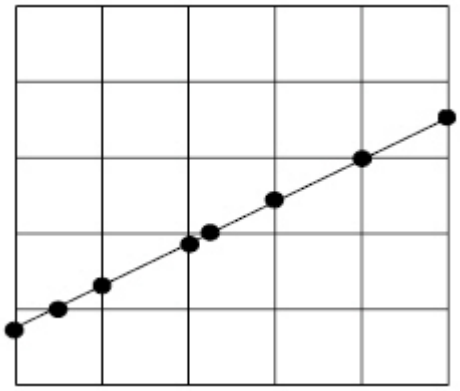
The **Grids | Calculate | Slice** command produces cross section data. The cross section is generated by taking a vertical slice through a gridded surface along a boundary line. Surface traces on a cross section are sometimes referred to as profile lines.

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
BNA Atlas Boundary
GSB Golden Software Boundary
GSI Golden Software Interchange

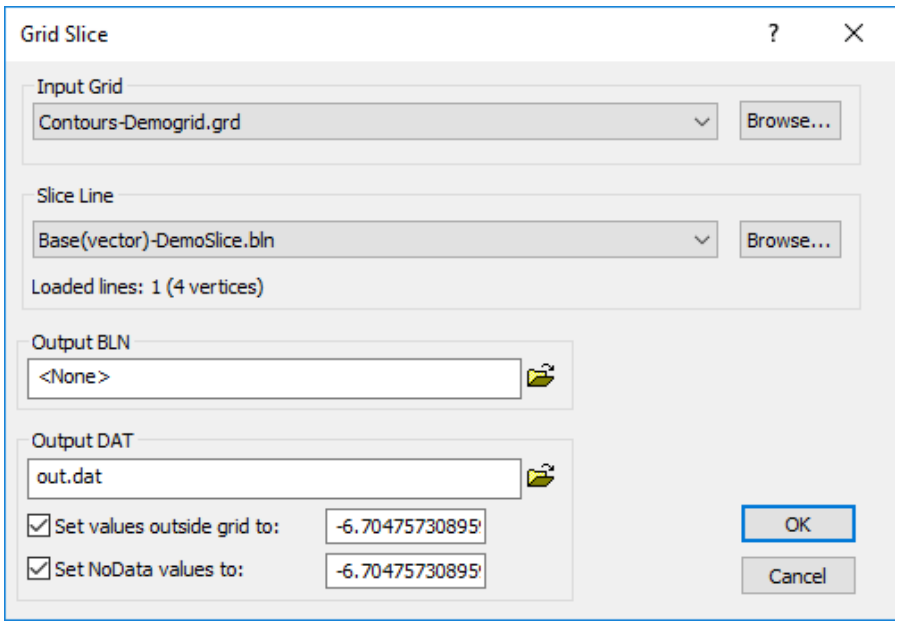
DDF SDTS TVP
DLG USGS Digital Line Graph
DXF AutoCAD Drawing
E00 Esri AcrInfo Export
EMF Windows Enhanced Metafile
GML Geo. Markup Language
GPX GPS Exchange Format
KML Google Earth
MIF MapInfo Interchange Format
PLT Golden Software Plotcall
PLY Stanford Polygon
SHP Esri Shapefile
STL Stereolithograph
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. 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. If desired, 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 are written to an ASCII data file .DAT during the **Slice** operation. 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

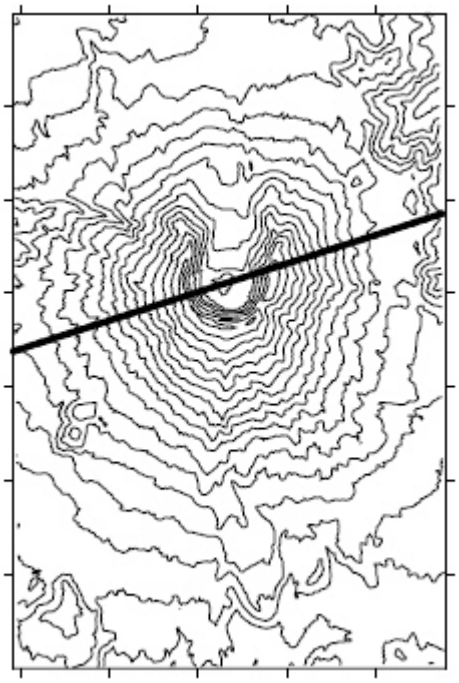
The *Set NoData values to* option assigns a special Z value to points that lie inside NoData regions when the check box is selected. If the check box is cleared, points inside the NoData areas are not included in the output data file.

OK or Cancel

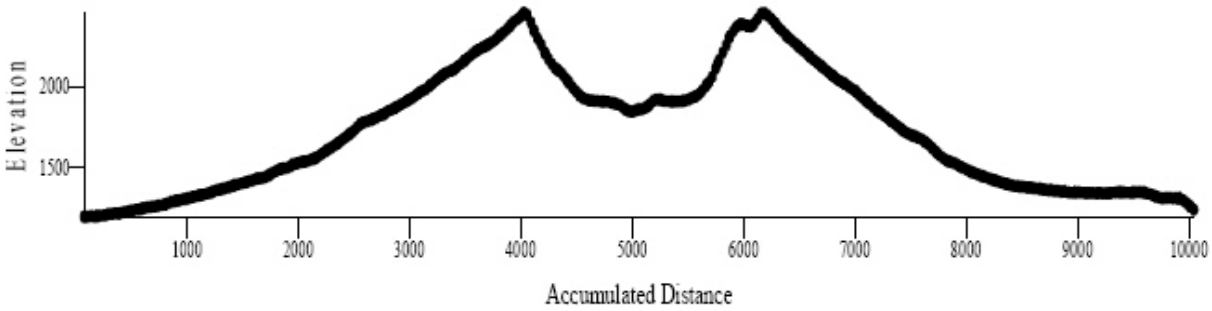
Click *OK* and a data file (and a blanking file, if specified) is created. Click *Cancel* to exit the dialog without creating the data.

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.

Grid 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:

$$Z_{res} = Z_{dat} - Z_{grd}$$

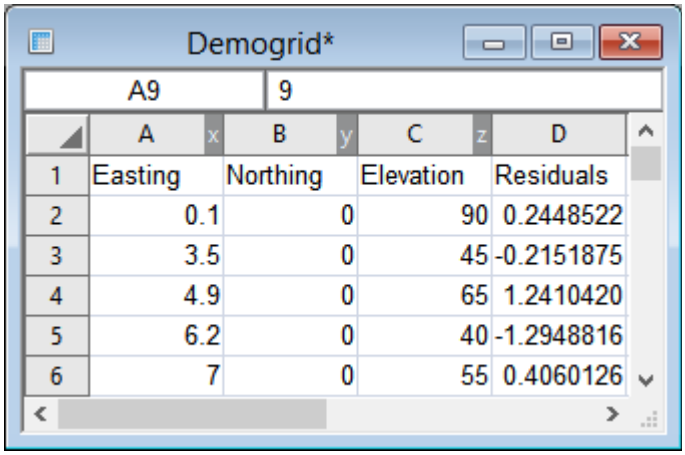
where:

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

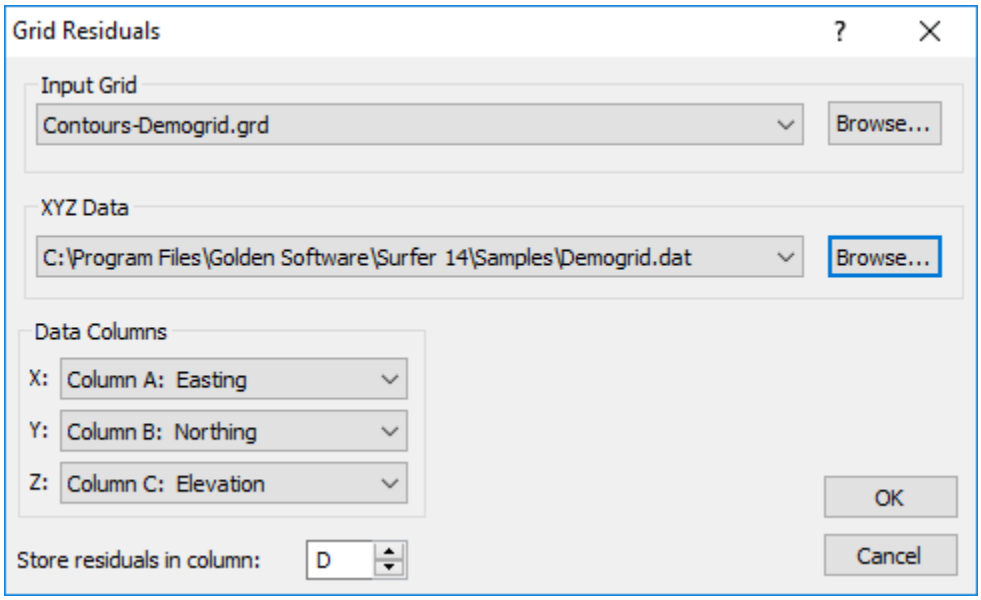
A worksheet window is automatically opened after the **Residuals** command is used. The worksheet includes the data file used for the calculation and the newly generated residual information. Use the worksheet's Data | Data | Statistics command to obtain statistical information on the calculated residuals.



*After clicking OK in the **Grid Residuals** dialog, the worksheet window opens with the original data and the new residuals column.*

Grid Residuals Dialog

Click the **Grids | Calcluate | Residuals** command 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.
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

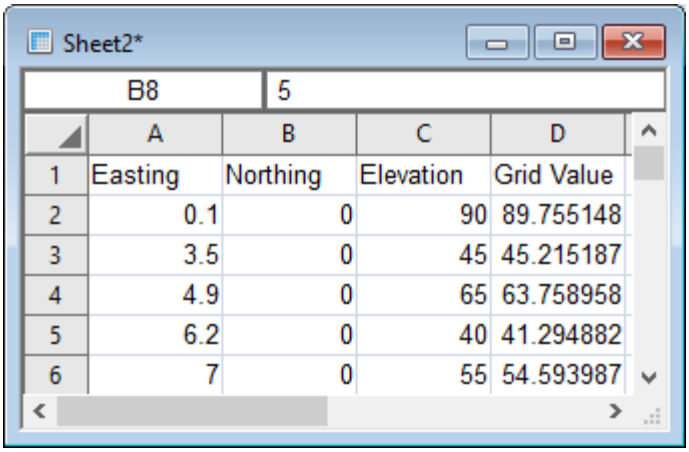
After performing the Grids | Calculate | Residuals command:

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.

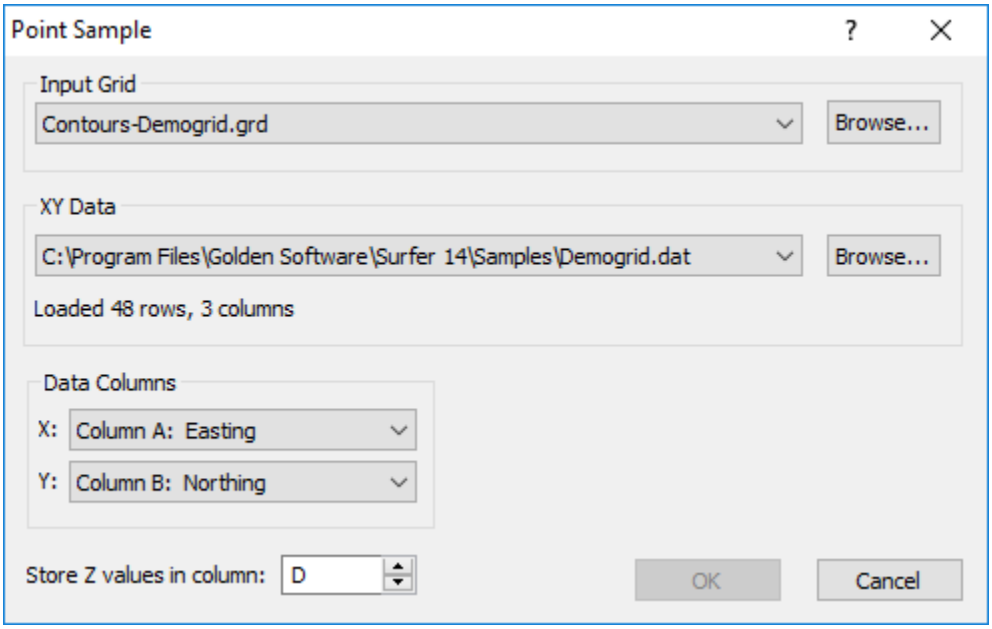


	A	B	C	D
1	Easting	Northing	Elevation	Grid Value
2	0.1	0	90	89.755148
3	3.5	0	45	45.215187
4	4.9	0	65	63.758958
5	6.2	0	40	41.294882
6	7	0	55	54.593987

*After clicking OK in the **Point Sample** dialog, the worksheet window opens with the original data and the new Grid Value column.*

Point Sample Dialog

Click the **Grids | Calculate | Point Sample** command 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.

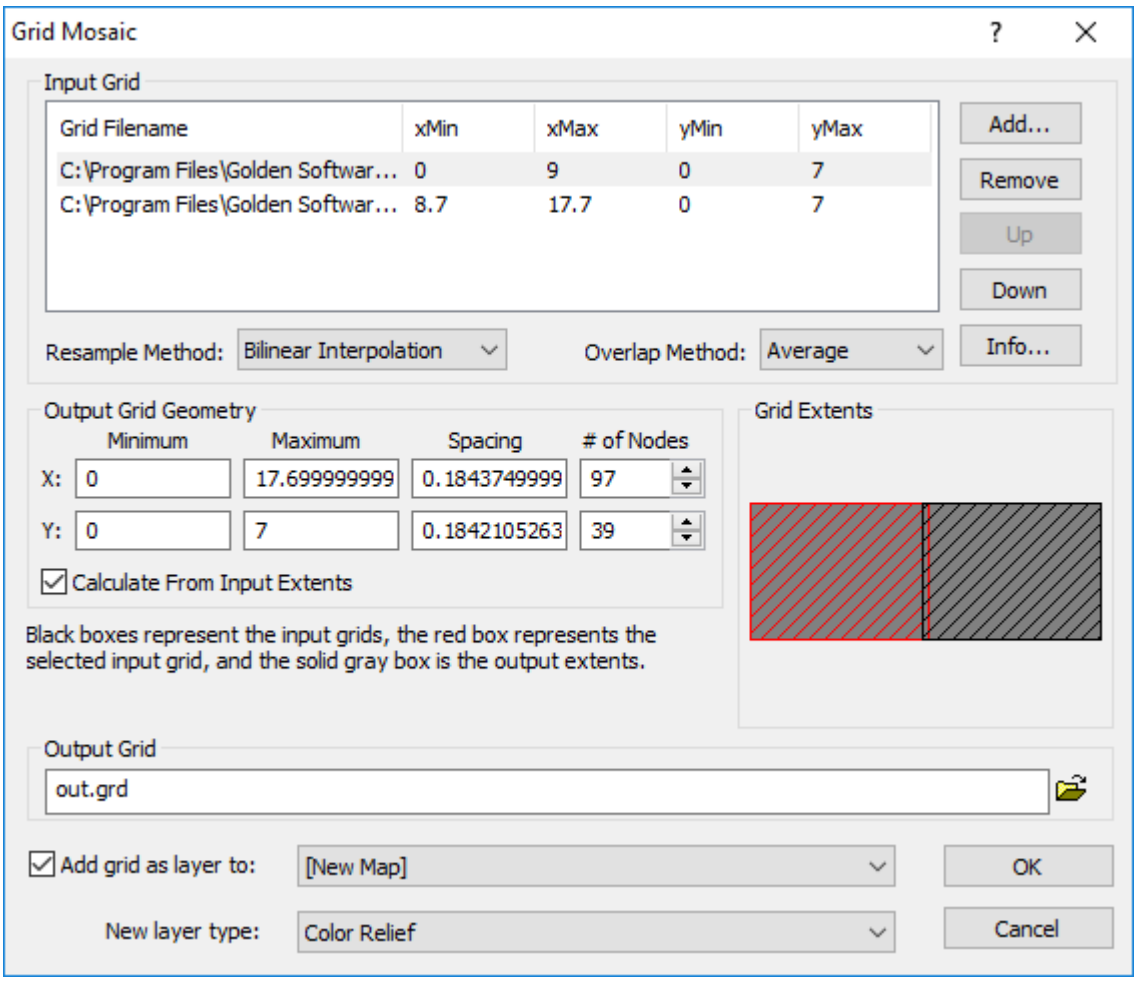
1. Open a new worksheet window with **File | New | Worksheet**.
2. In column A, type the X coordinates for the points you wish to find the Z value, e.g. type 1 in Row 1 and 4 in Row 2. In column B, type in the Y coordinate, e.g. type 3 in Row 1 and 2 in Row 2. Add as many points as you want into the 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.

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, 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.

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.
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

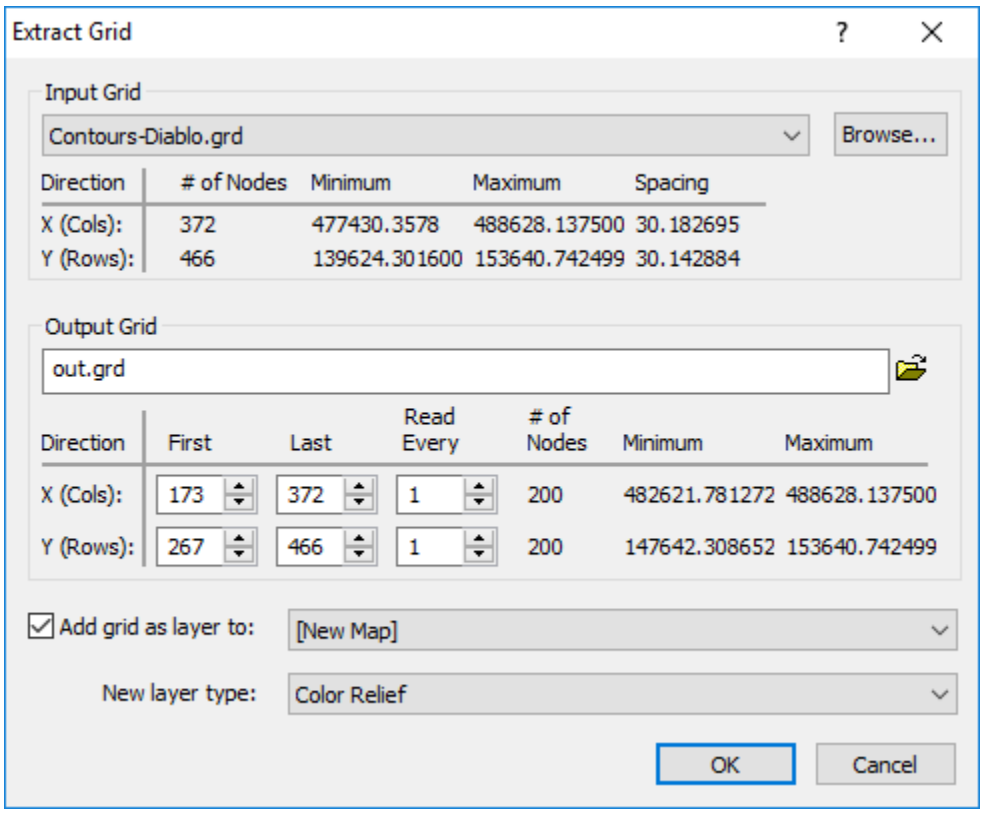
The **Grids | Resize | Extract** command creates a subset of an existing grid file. Grid files consist of rows and columns (lines of constant Y and X value) containing Z values. When you extract information from a grid file, you can specify the starting and ending rows and columns to extract. For example, 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 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.
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 in the plot window allows 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.*

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 Edtior** 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.

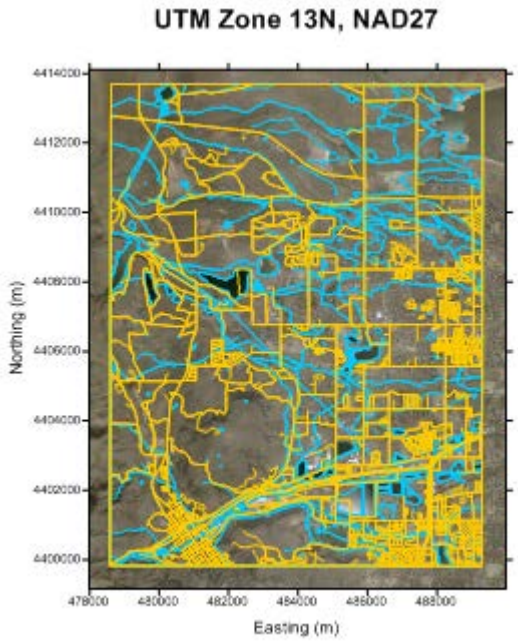
Ripley, B. D., (1981), *Spatial Statistics*, John Wiley and Sons, New York, 252 pp.

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 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.

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 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. There are five 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.

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. Edit the group to edit multiple sub-objects at once, add new features to the base(vector) layer, or reshape the existing features. 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.

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.

Base (raster) layers cannot be edited with Start Editing. 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 creates 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 creates 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 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 creates an empty base map with the intention of creating new geometry within the empty base map limits by using the Start Editing command.

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 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 contains the following pages:

View

Scale

Limits

Frame

Coordinate System

Info

Base Map from Data

Click the **Home | New Map | Base from Data** command 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. Click the base layer and then click the Start Editing command to perform geoprocessing operations on the points in the base layer.

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*.

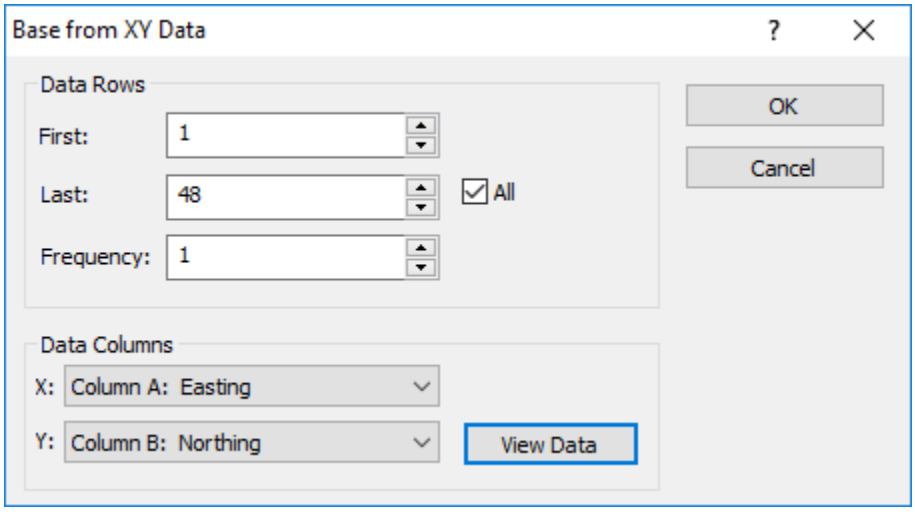
Base from Data Layer vs Post Layer

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

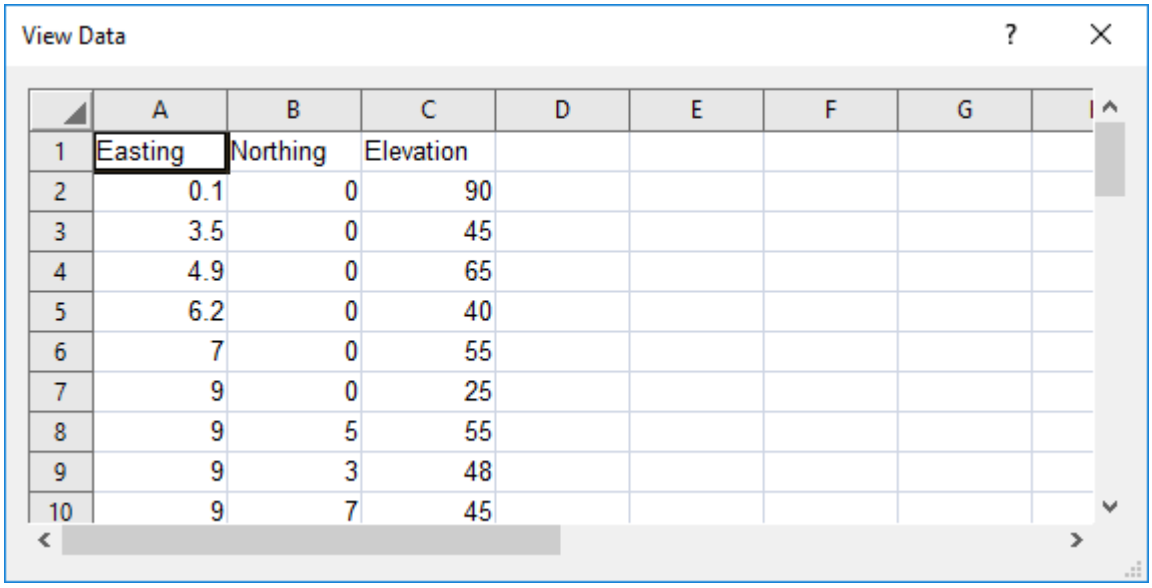
Specify the column containing X coordinates in the *X* field. Click the current selection and click the desired column in the list to change the value. Only columns containing data will be included in the *X* list. Specify the column containing Y coordinates in the *Y* field. Click the current selection 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** dialog to 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, **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 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.

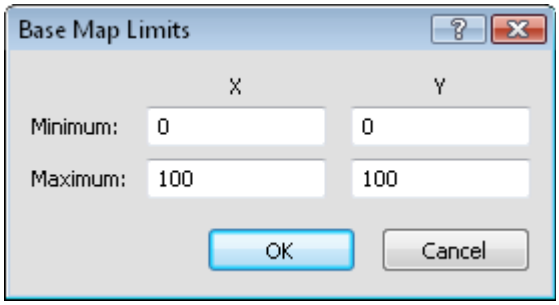
- When selecting a WMS or OSM server, an image is downloaded to be used as a base (raster) layer.
- When selecting a WCS server, data values are downloaded to be used as a grid file. WCS servers cannot be used with the **Base from Server** command. Use the Grid from Server command or the *Download* button 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 creates an empty base map with the intention of creating new geometry within the empty base map limits by using the **Start Editing** command. The **Home | Add to Map | Layer | Empty Base** command adds an empty base map layer to an existing map.

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. Right-click the new *Empty Base* layer in the Contents window and click **Start Editing**. The base layer object is now being edited.
3. Click the **Features** tab and select the item you want to draw (text, polylines, polygons, symbols, etc).
4. Draw the objects over the map. Create as many objects as you wish.
5. When you are finished, right-click the *Base* layer in the **Contents** window and click *Stop Editing*. The base map, and all the drawn objects, will now move, scale and resize with the original map.
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 (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. Right-click the *Empty Base* layer in the **Contents** window and click **Start Editing**.
4. Click **Home | Clipboard | Paste**. The objects are pasted in the base layer.
5. 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.
6. When they are positioned correctly, right-click the empty base layer in the **Contents** window and click **Stop Editing**.
7. You can delete the original objects if you wish, or hide them by clearing their visibility check boxes in the **Contents** window.
8. 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.

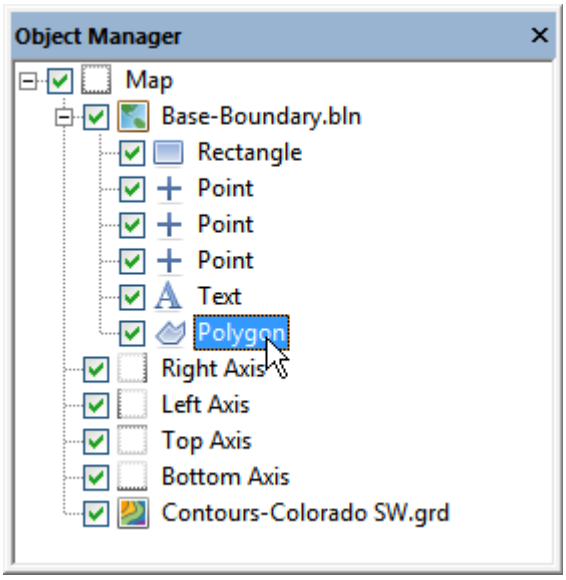
The base map is automatically redrawn. The specified properties are applied to all the polyline, polygon, text, or symbols in the base map.

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.

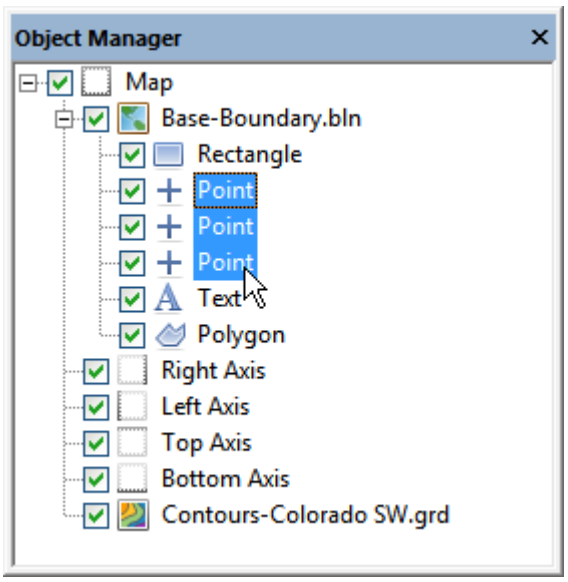


Select the sub-object to change individual properties.

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. Click **Features | Group | Start Editing**. Alternatively, right-click on the *Base* map layer and select **Start Editing**.
3. Use the SHIFT or CTRL key to select multiple sub-objects of the base map layer.
4. The shared properties are displayed in the **Properties** window. Make changes as needed. The objects will automatically update to reflect the new properties.
5. Click **Features | Group | Stop Editing** to exit the base map group. Alternatively, right-click on the *Base* map layer and select **Stop Editing**.

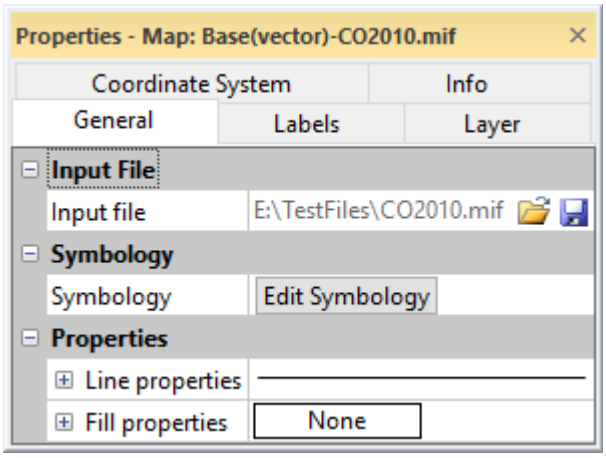


Start editing and select multiple sub-objects to change properties.

General Page - Base (vector) Layer

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 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.

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. There are five 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 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.

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.

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 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 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. There are five 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.

Adding Symbology to a Base Layer

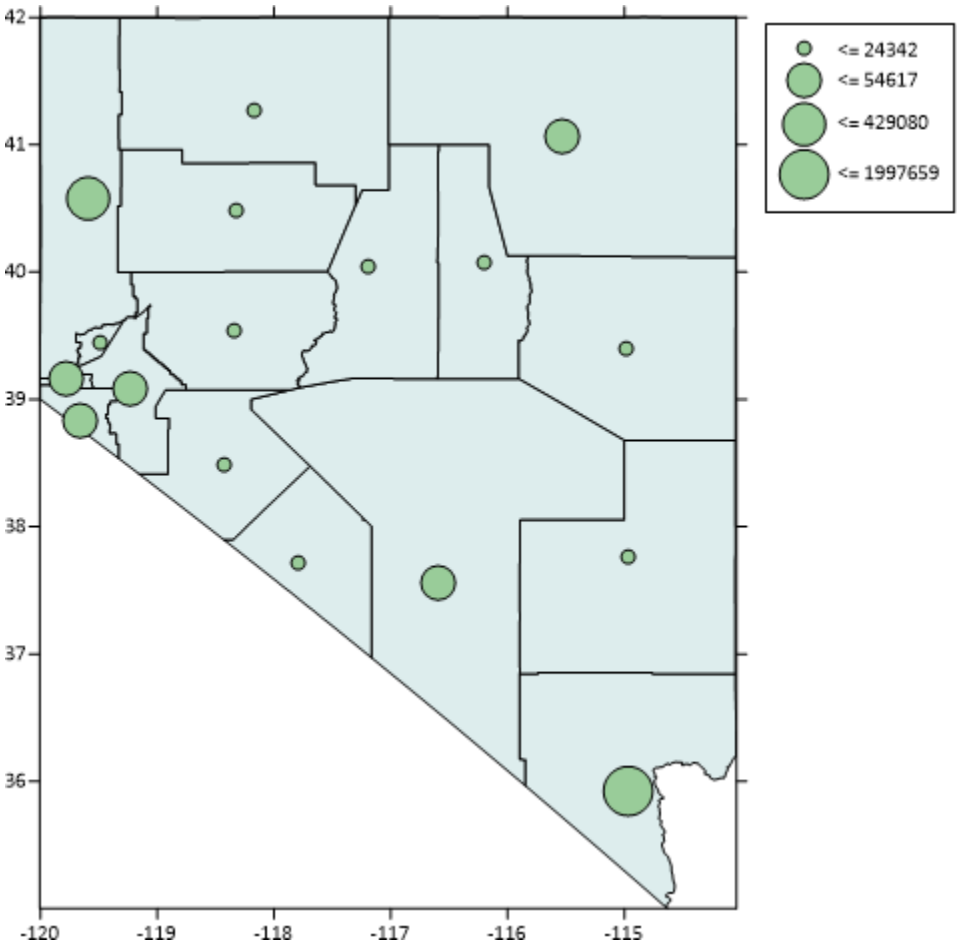
A symbology can be added to a base (vector) layer by clicking *Edit Symbology* in the **Properties** window General page. Select the symbology type, specify the attribute field for the symbology, and then specify the various line, fill, and/or symbol properties for the symbology in the **Symbology** 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. The *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 15\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.

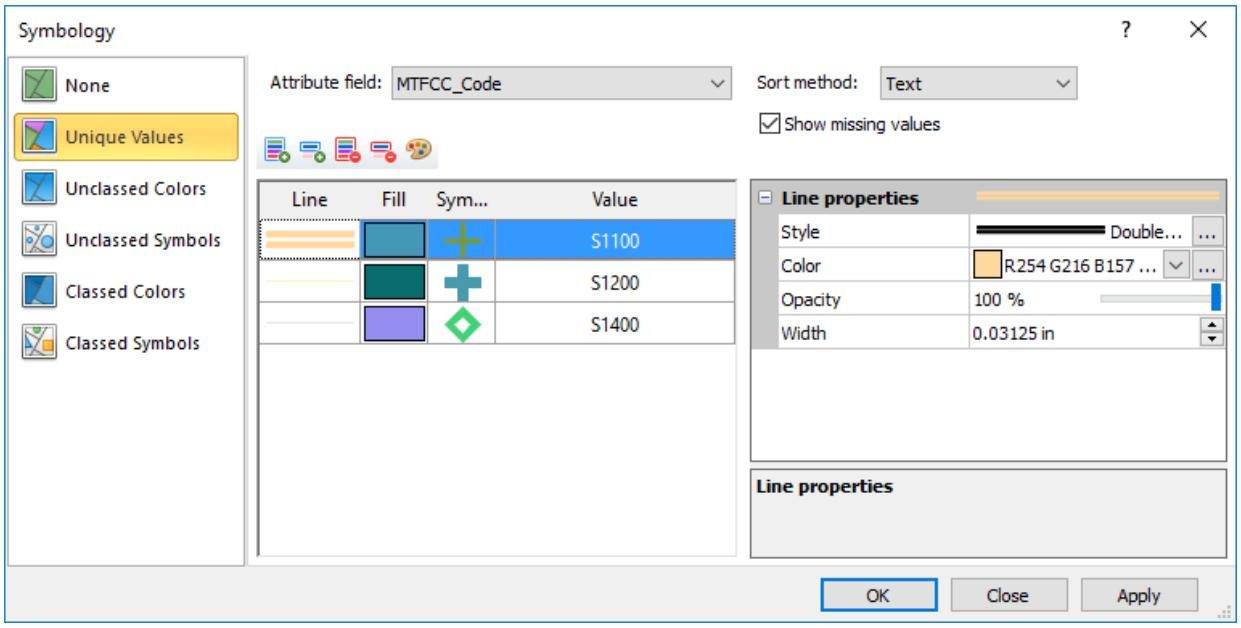


The symbology classifies the counties in Nevada by population and displays proportionally sized symbols to indicate the class. The legend indicates the maximum population for each class.

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

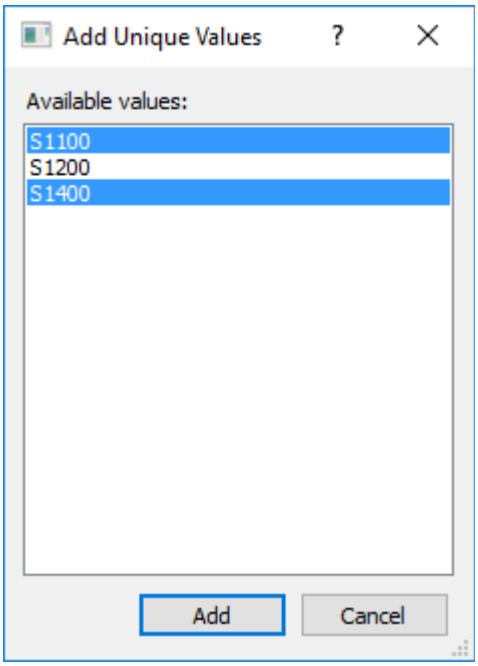
The values list displays the selected unique values and their associated properties. Values are added and removed by clicking the command buttons above the list. Click the *Line*, *Fill*, or *Symbol* property preview to display the properties in the properties pane. The properties preview is updated 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

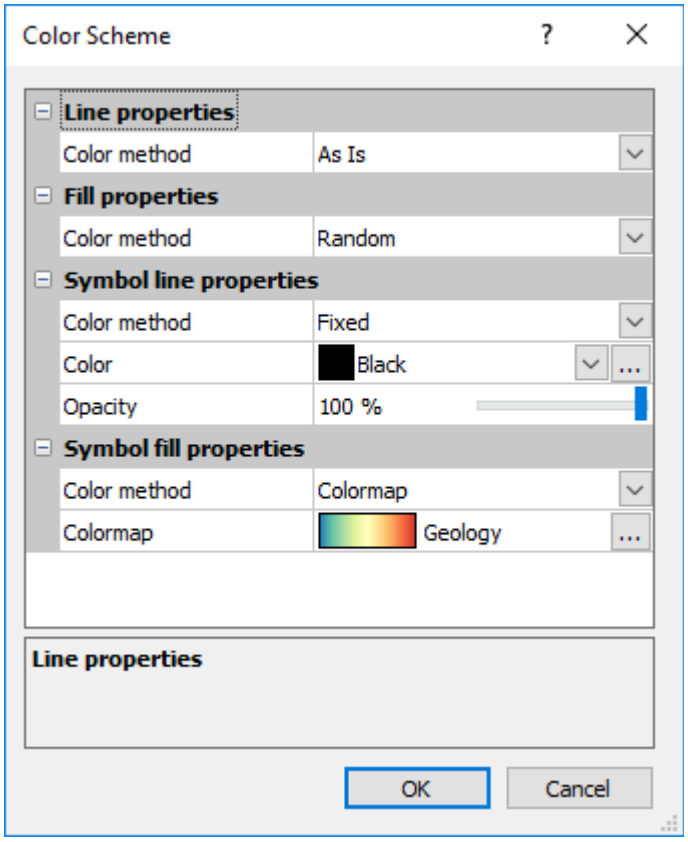
Click the **Remove All** command to remove all unique values from the values list.

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.

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*.

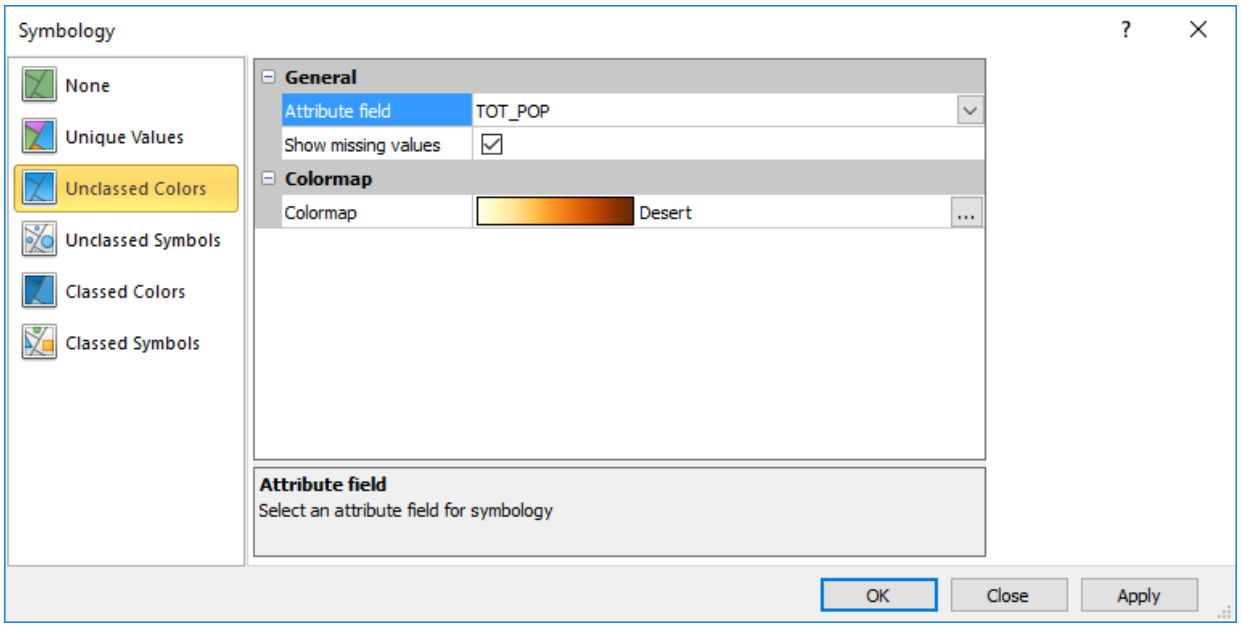
OK, Cancel, and Apply

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 dialog.

Beyond selecting the colors for the color spectrum, the **Colormap** dialog 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** dialog.

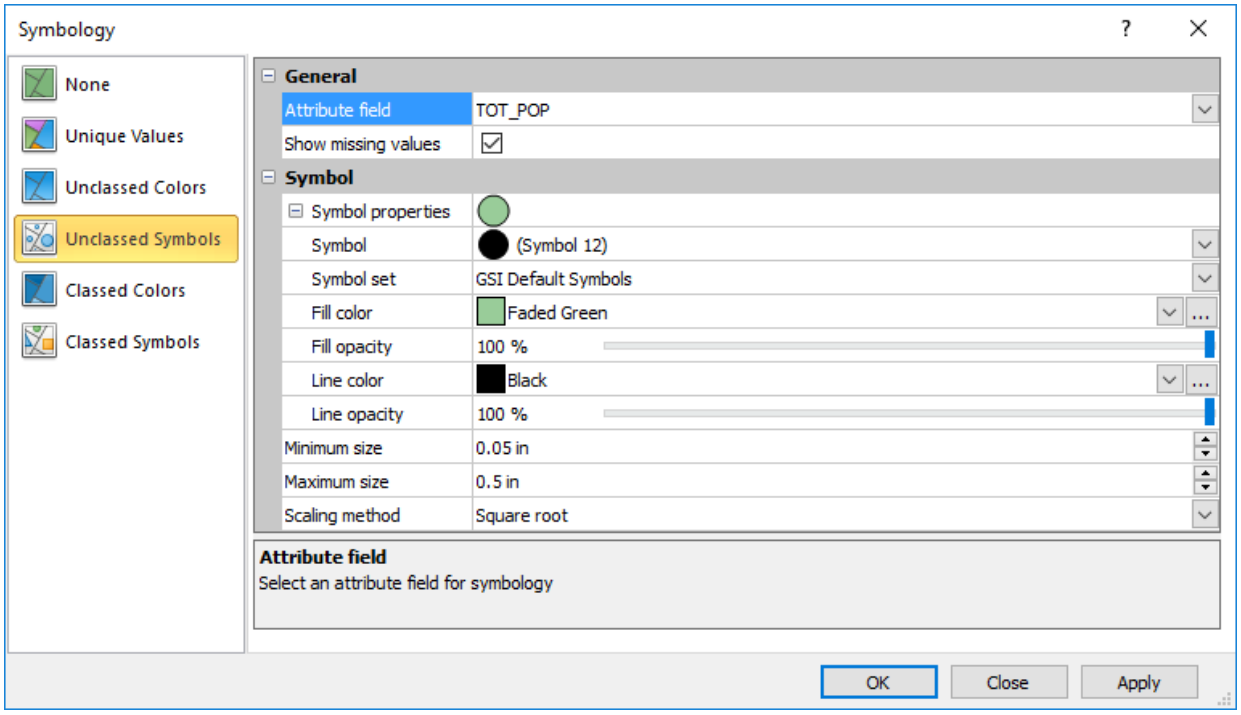
OK, Cancel, and Apply

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.

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 Properties

The properties in the *Symbol* section control the appearance of the symbols, including the symbol properties, symbol size, and scaling method.

Symbol 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:
$$H_n = [((Z_n - Z_1)/(Z_2 - Z_1)) * (H_2 - H_1)] + H_1$$

Square Root:
$$H_n = \left[\sqrt{(Z_n - Z_1)/(Z_2 - Z_1)} * (H_2 - H_1)\right] + H_1$$

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.

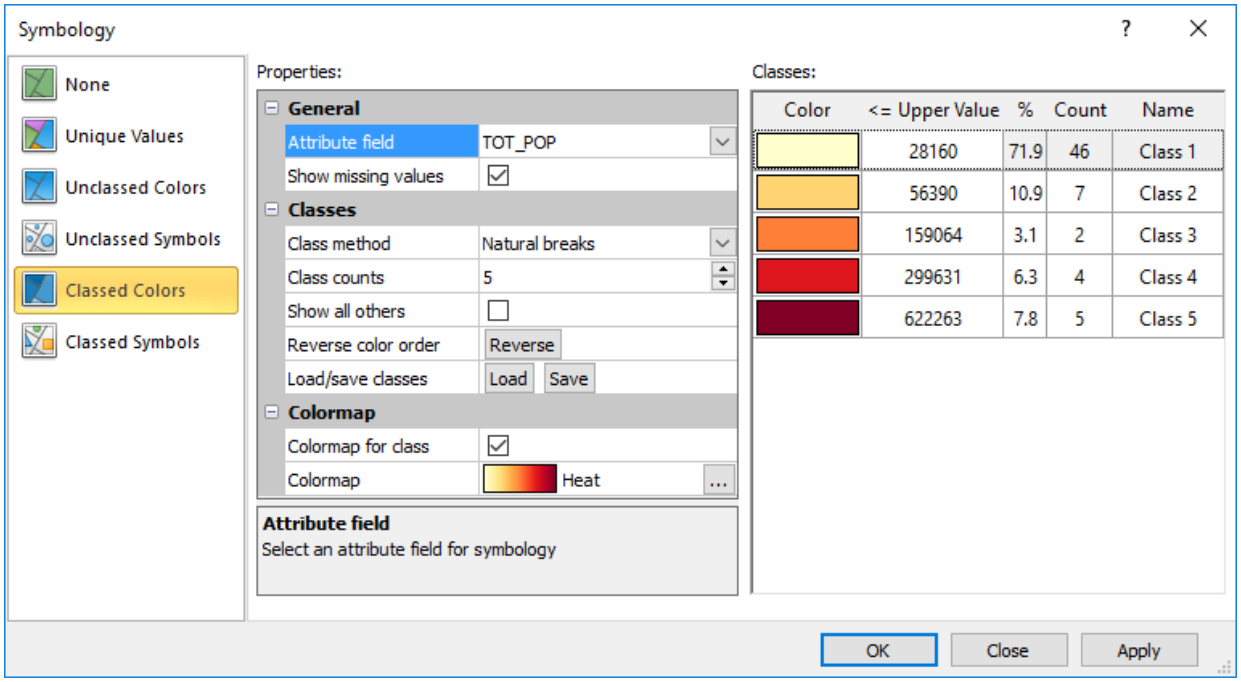
OK, Cancel, and Apply

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 dialog.

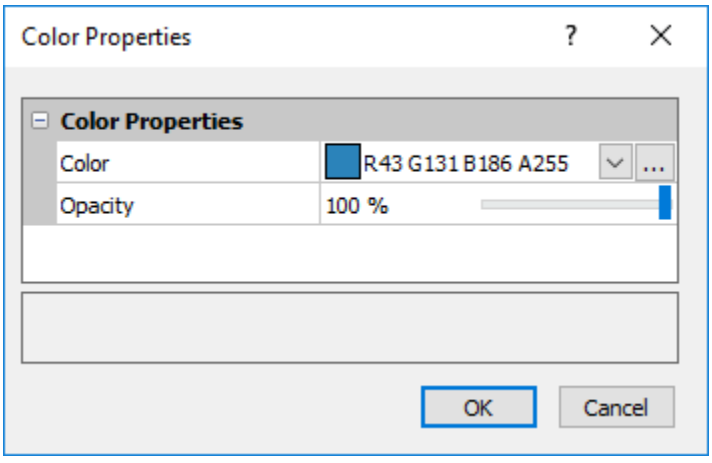
The *Colormap* property applies colors to the classes based on class number, not the data value. Therefore the *Data to Color Mapping* section of the **Colormap** dialog does not have any effect on the class colors by default. We recommend you do not change the *Data to Color Mapping* properties in the **Colormap** dialog. 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 to Color Mapping* 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* list is 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.

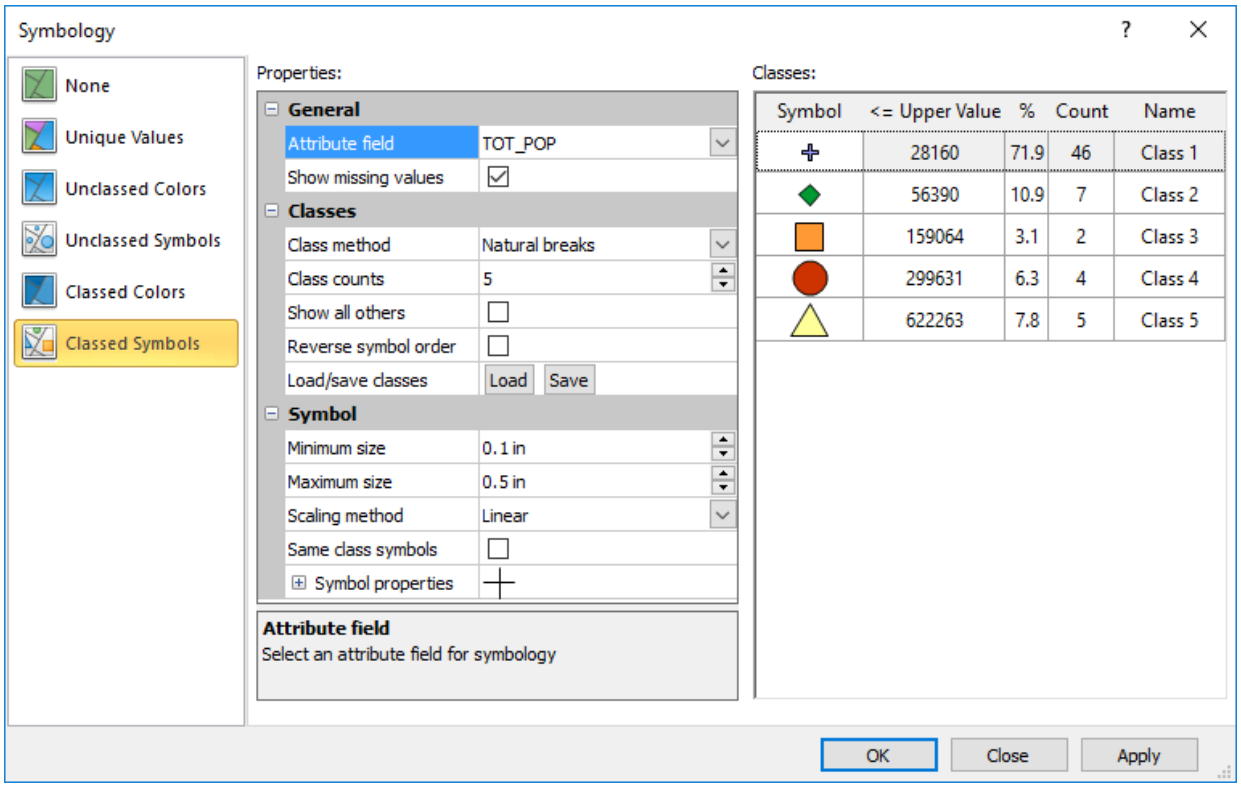
OK, Cancel, and Apply

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: $$H_n = \left[\left(\left(Z_n - Z_1\right)/\left(Z_2 - Z_1\right)\right) * \left(H_2 - H_1\right)\right] + H_1$$

Square Root: $$H_n = \left[\sqrt{\left(Z_n - Z_1\right)/\left(Z_2 - Z_1\right)} * \left(H_2 - H_1\right)\right] + H_1$$

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.

Symbol Properties

The *Symbol properties* 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.

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

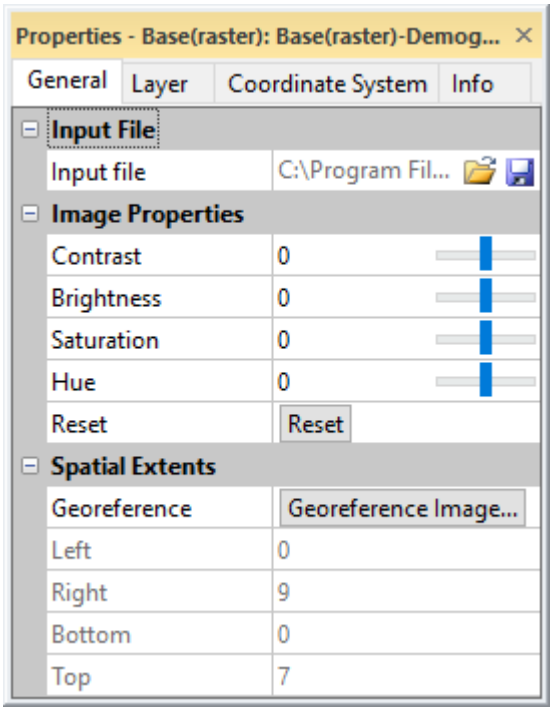
The *Name* column displays the class names. Specify a class name in the *Classes* list to display class names in a legend. By default, classes are named *Class 1*, *Class 2*, etc. Type a name in the *Name* column to change the class name.

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.

General Page - Base (raster) Layer

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 **File | Import** 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

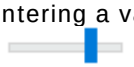
saving a base layer. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* option be checked to generate a .GSR2 file. Set any export options and click *OK* to save the file.

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.

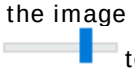
Image Properties

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.

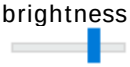
Opacity

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

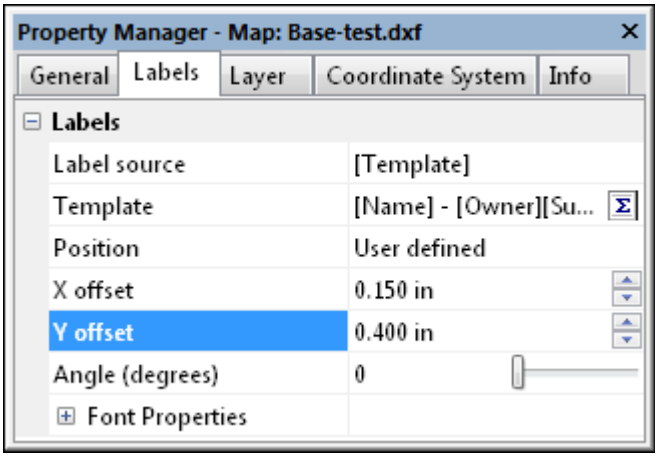
Click *Georeference Image* in the *Georeference* field to apply control points to the image and reference it to your map's coordinate system. Clicking *Georeference Image* opens the Georeference Image window. The **Georeference Image** command is only available when the image is a base (raster) layer. 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.

Labels Page - Base Map

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. It is recommended that a polygon be created instead of a rectangle, rounded rectangle, or ellipse if labels are desired. 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.

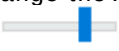
The square brackets are mandatory. Math text options (such as \up50) must appear outside of the square brackets. If the font or font properties are changed for the attribute, highlight the entire text including the square brackets before changing the properties in the Text Editor.

Click the 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.

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:

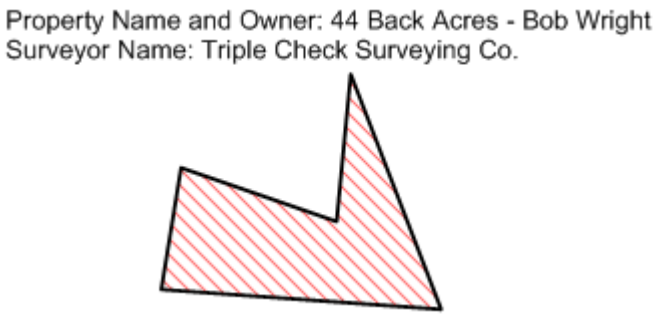
[Name] - [Owner]

5. To create multiple lines, click the Σ 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.

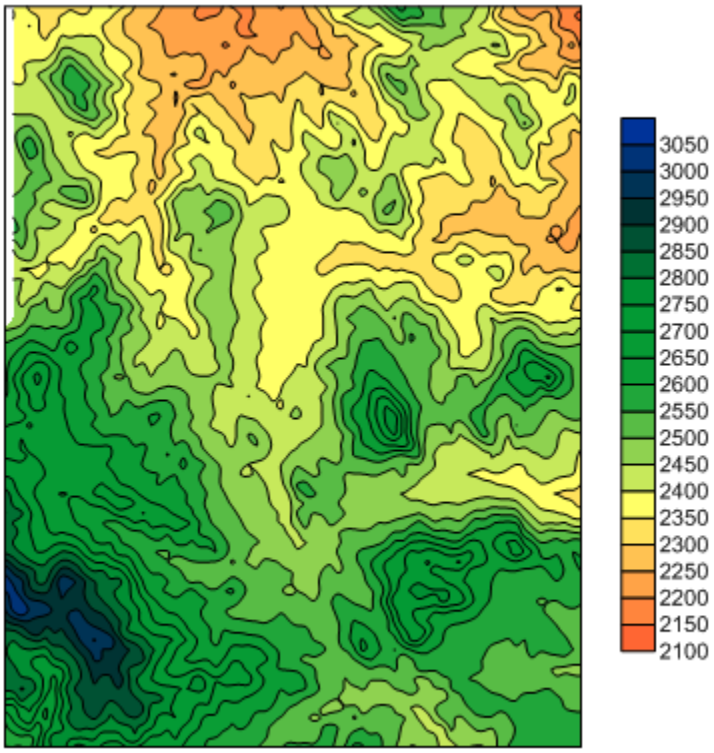


The label appears in the desired location, substituting the attribute value for the attribute names in square brackets.

Chapter 9 - Contour Maps

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** 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

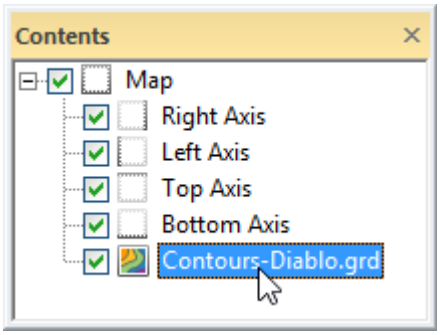
To create a new contour map:

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

Map Properties

The map properties contains the following pages:

View

Scale

Limits

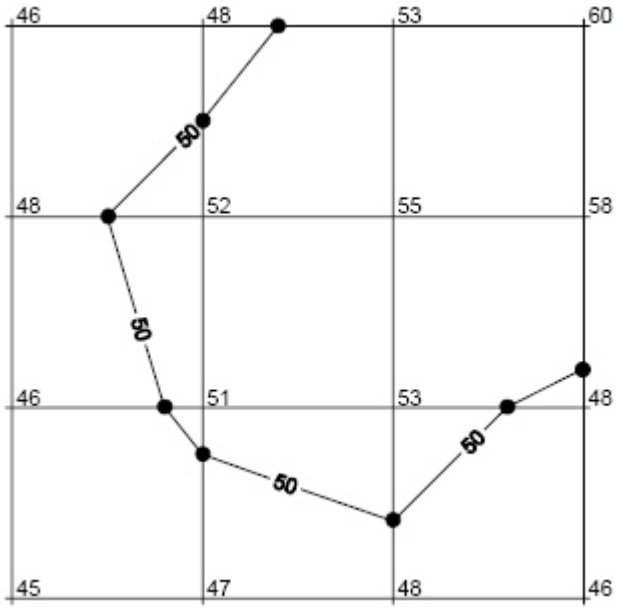
Frame

Coordinate System

Info

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.



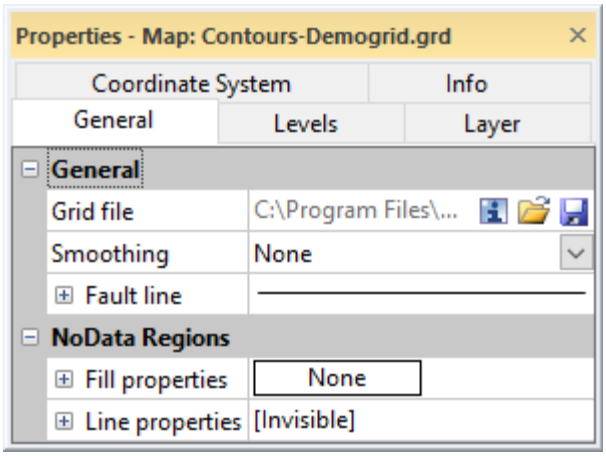
The path of a contour line (Z = 50) through a grid is defined by interpolating between grid nodes. This example shows the straight-line segments that define the contour line. The dots are added to show the ends of each line segment.

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 Page

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 click *OK*, the grid file is replaced, but no contour lines are drawn. You can update the contour levels being displayed on the Levels tab.

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 Page

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, linear, or advanced method. All methods are described below. The advanced option is useful when more control over individual contour line properties is desired.

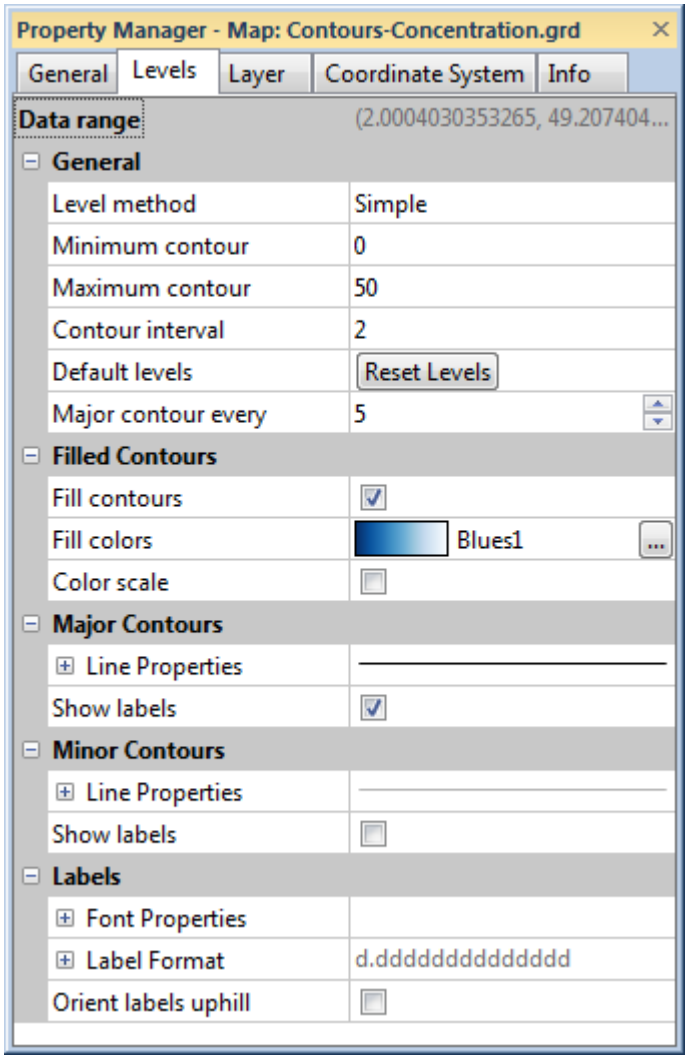
Display Simple or Logarithmic Options

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* from *Simple* or *Logarithmic* to *Advanced*, all *Simple* or *Logarithmic* options are initially shown in the **Levels for Map** dialog that contains the *Advanced* options. However, when switching the *Level method* from *Advanced* to *Simple* or *Logarithmic*, 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, and how many minor contours are in a major contour interval.

Level Method

The *Level method* determines which options are available for setting the contour map contour line 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 labels, hachures, fill, and line properties set individually. These 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*.

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 Levels for Map dialog, where the advanced level options are set.

Filled Contours

The *Filled Contours* section controls the display of color fill between the contour lines as well as the display of the color scale bar.

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 dialog 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.

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. See color scale bar for more information on the color scale properties.

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 dialog. 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 orientation 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.

Advanced Levels

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 or logarithmic 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

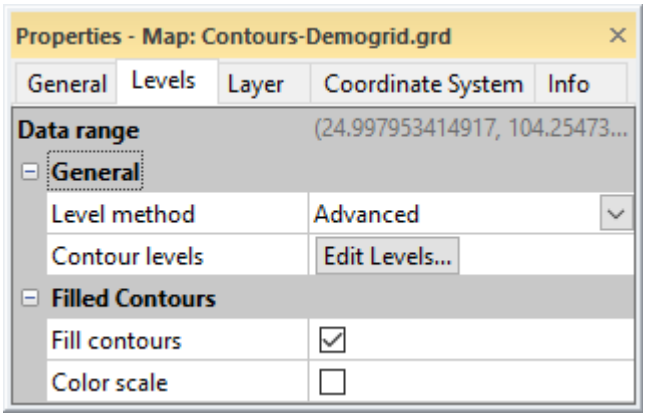
To use the advanced options, click on the contour map to select it. In the Properties window, click on the Levels tab. Change the *Level method* to *Advanced*. To open the advanced **Levels for Map** dialog, click the *Edit Levels* button next to *Contour levels.*

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* from *Simple* or *Logarithmic* to *Advanced*, all *Simple* or *Logarithmic* options are initially shown in the **Levels for Map** dialog. However, when switching the *Level method* from *Advanced* to *Simple* or *Logarithmic*, 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.

Filled Contours

The *Filled Contours* section in the **Properties** window controls the display of color fill between the contour lines as well as the display of the color scale bar.

Fill Contours

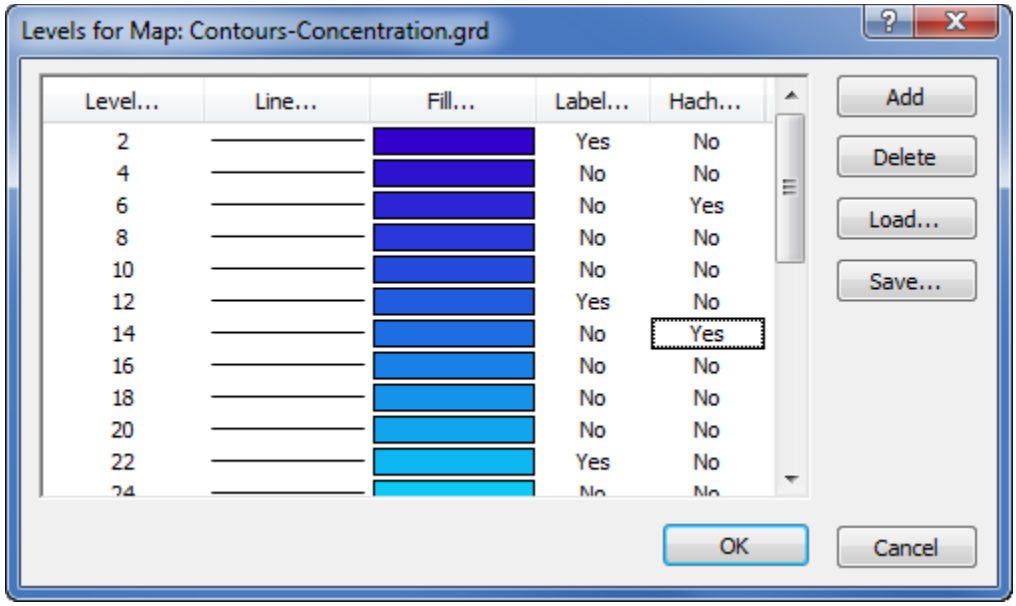
Check the box next to *Fill contours* to fill the space between each contour line with color. See Adding Color Fill between Contours for more information on filled contours. To set the *Fill colors* when the *Level method* is set to *Advanced,* click on the *Edit Levels* button and set the colors in the **Levels for Map** dialog.

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. See color scale bar for more information on the color scale properties.

Levels for Map Dialog

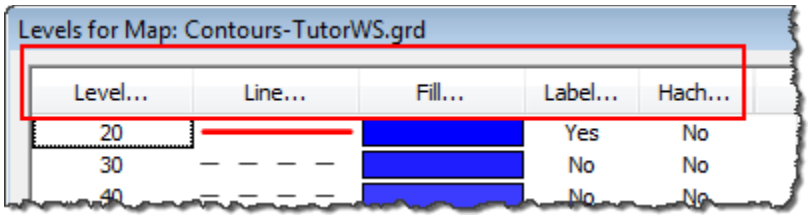
Control advanced settings for the *Level, Line, Fill, Label,* and *Hach* properties of the contour map in the **Levels for Map** dialog. Properties can be adjusted for all contours at once (by clicking on the column buttons), or for individual contours (by double-clicking on the specific contour level).



*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 dialog 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** tab.
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.

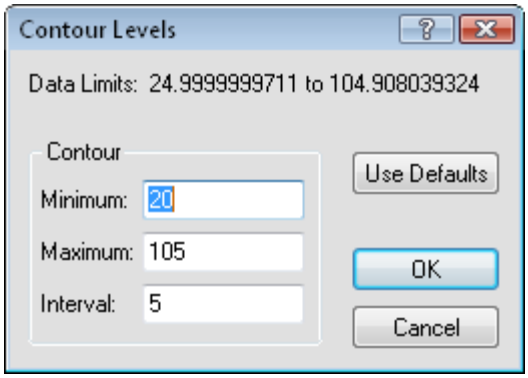
Specifying Contour Levels

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.

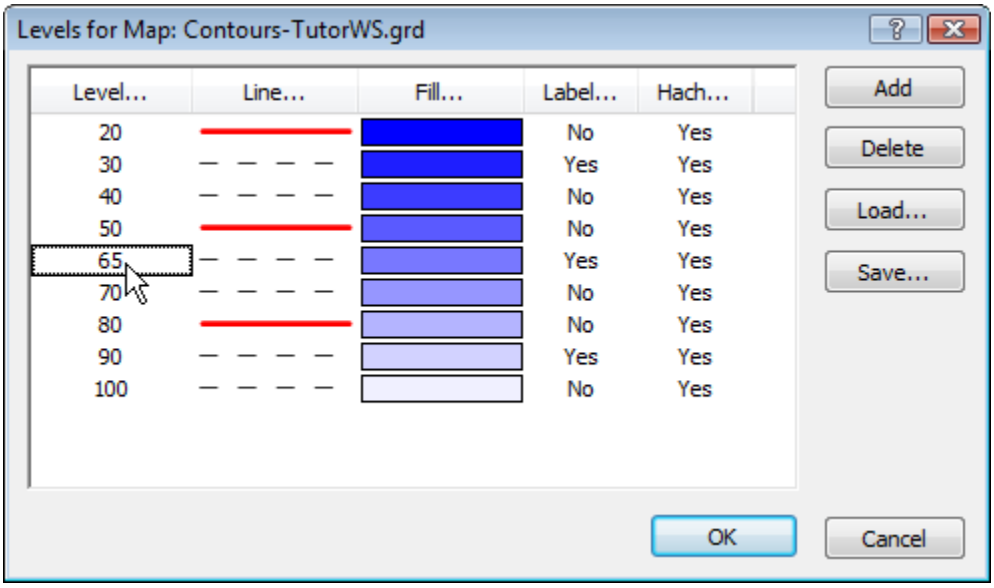
Using advanced options allows the possibility of controlling additional features for each contour line separately. Use these steps for advanced contour lines:

1. Click on a contour map to select it.

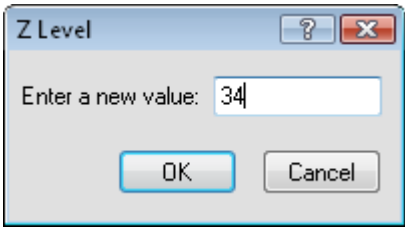
2. In the Properties window, click on the **Levels** tab.
3. Set the Level method to Advanced.
4. Click the *Edit Levels* button next to *Contour levels* to open the advanced options **Levels for Map** dialog.
5. To assign evenly spaced contour levels, click the *Level* button and the **Contour Levels** 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.*



*Change individual contour lines with the **Z Level** dialog.*

New Value

Enter a new value into the *Enter a new value* box, and click *OK* to add the new value to the 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.

1. Enter level values into column A.

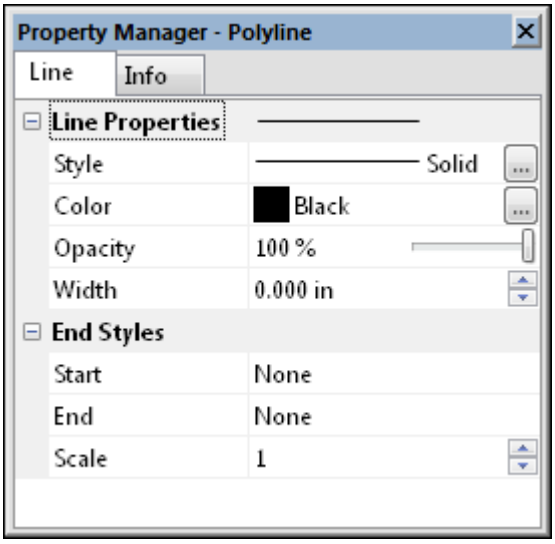
2. Add any other information you wish to include for the levels (i.e. line color) according to the level file format.
3. After entering the level information, click **File | Save As**.
4. In the *File name* field, enter the level file name and .LVL extension enclosed in quotes ("colors.lvl").
5. Choose *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

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*.

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.

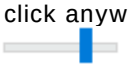
Complex line styles can be selected for most object types. Wireframe map layers and surface mesh lines do not support complex line styles. When a complex line style is selected, the line *Width* automatically increased to 0.03125 inches (0.079 cm). Most complex line styles require lines with the *Width* 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.

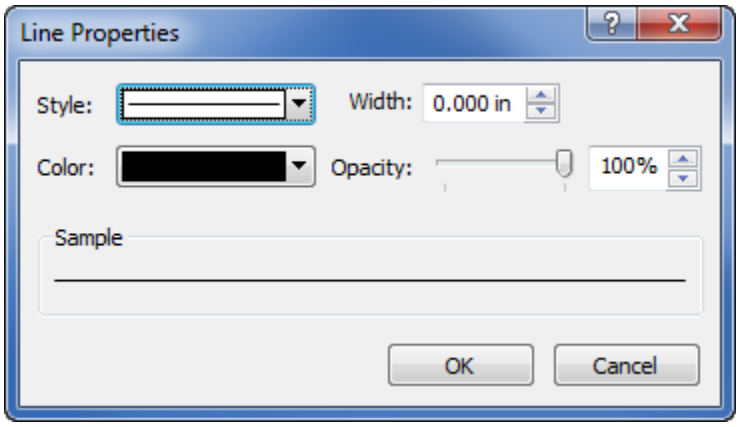
The *Scale* determines the scale factor of the arrowhead. To change the size, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the values. A value of 1 makes the arrow the default size.

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.

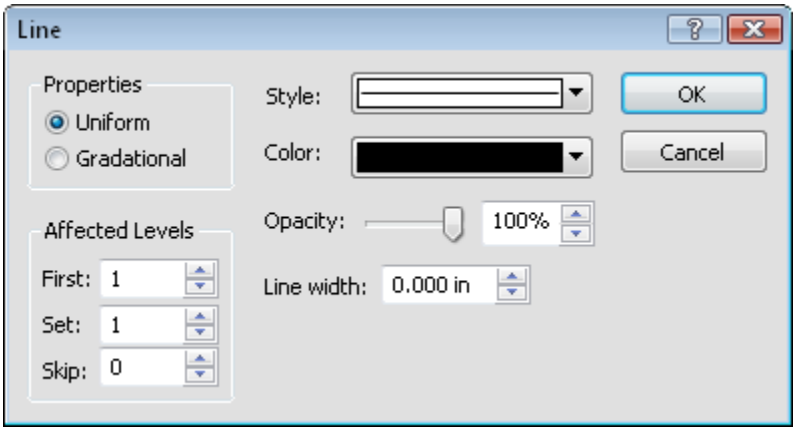
Sample

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.



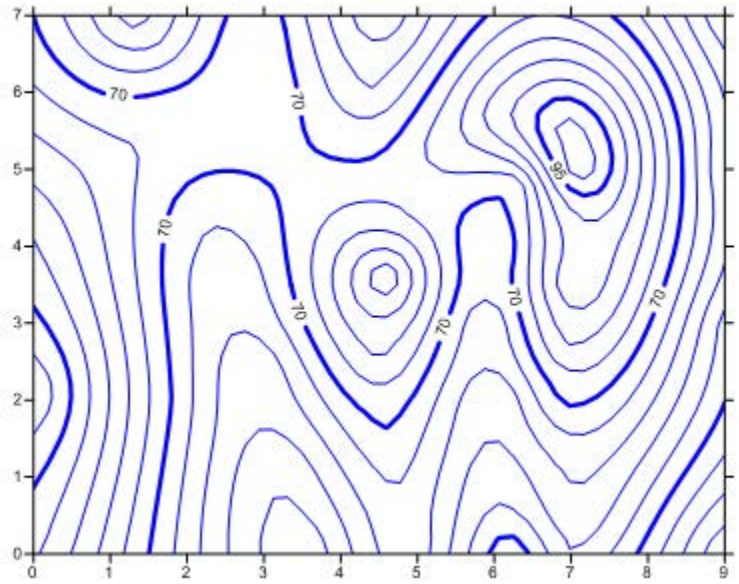
*Specify line properties for multiple contour lines in the **Line** dialog.*

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 dialog. Set the properties in the **Colormap** dialog and click *OK* to return to the **Line** dialog.

Opacity

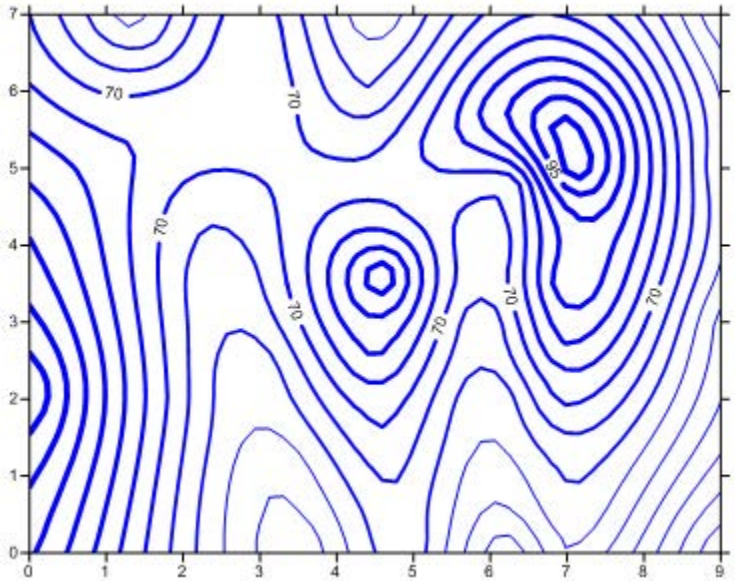
The *Opacity* is the amount of transparency of the lines. This is a value from 0% (completely transparent) to 100% (completely opaque). To change the value, highlight the existing value and type a new value, use the buttons to the right of the box to increase or decrease the value, or click and drag the to change the opacity percentage.

Line Width

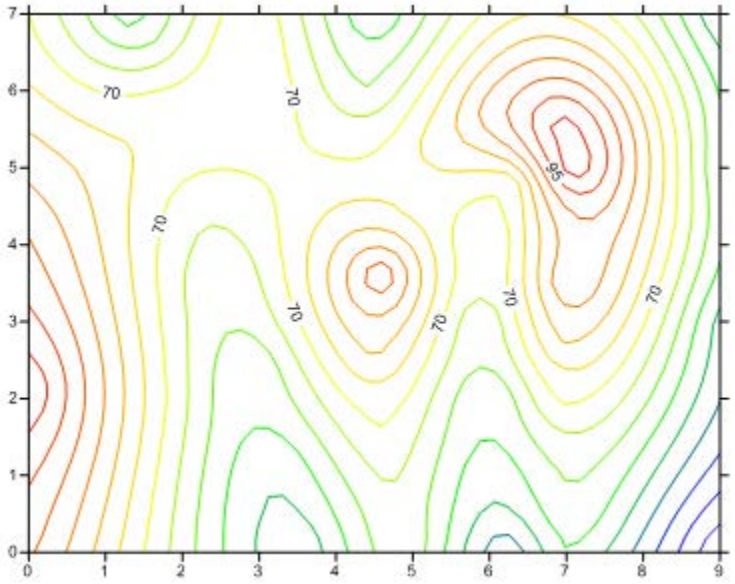
If you have *Uniform* selected, change the *Line width* by typing a new number into the box or by using the buttons to the right of the box to increase or decrease the value. The line width can be zero to 0.5 inches (1.27 cm) wide. A width of zero is one pixel wide.

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:

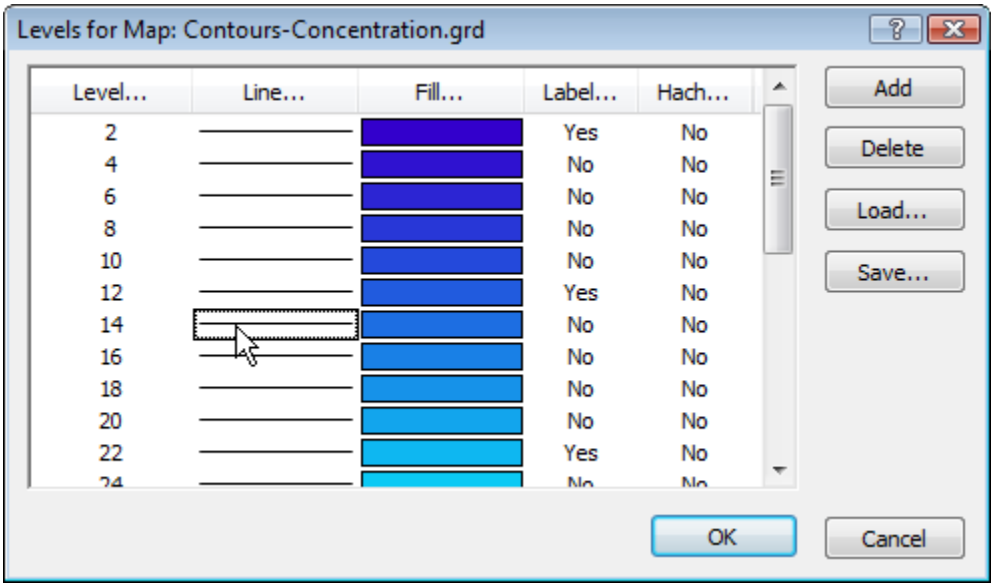
1. In the **Line** dialog, enter *First*=one, *Set*=one, *Skip*=four and click the *OK* button. This applies the properties to the first contour level, skip four levels, set the next level, and so on.
2. In the **Line** dialog, enter *First*=two, *Set*=four, *Skip*=one and then click the *OK* button. 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.

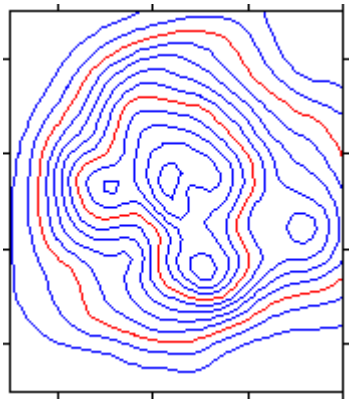


In this example, the line sample for the 14 Level is selected.

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.

Color Filled Contour Maps

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 dialog. 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.

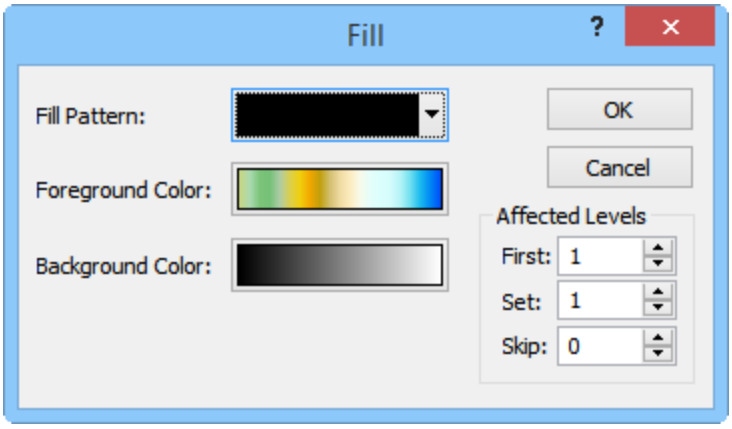
Use the *Fill colors* option in the Properties window to change the fill color properties for a simple level contour map.

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 dialog 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 dialog.

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 dialog. Foreground color *Opacity* can be controlled in the **Colormap** dialog.

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** dialog. Background color *Opacity* can be controlled in the **Colormap** dialog.

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 dialog 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 dialog 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 dialog. 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

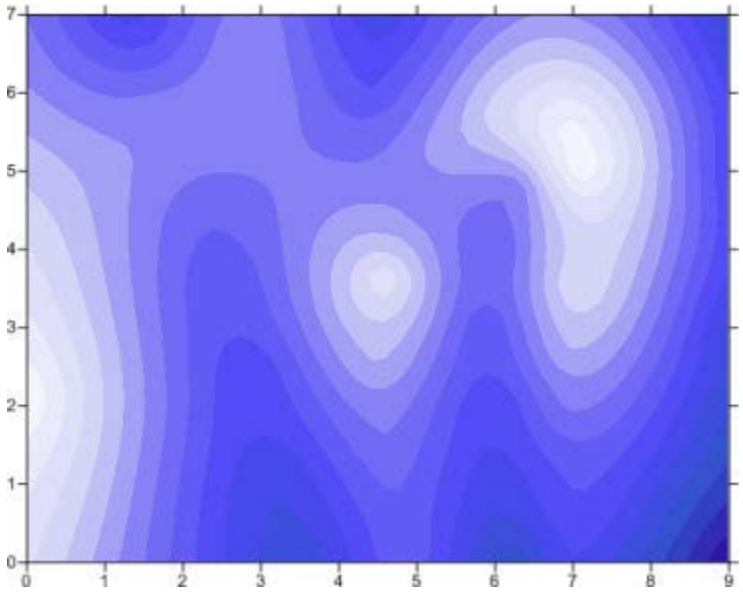
To create a filled contour map without contour lines using a simple or logarithmic level contour map,

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,
    1. Click the ⊞ next to *Line Properties.*
    2. Set the Style to Invisible.
    3. Uncheck the box next to *Show labels*.
6. In the *Minor Contours* section,
    1. Click the ⊞ next to *Line Properties*.
    2. Set the Style to Invisible.
    3. 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.

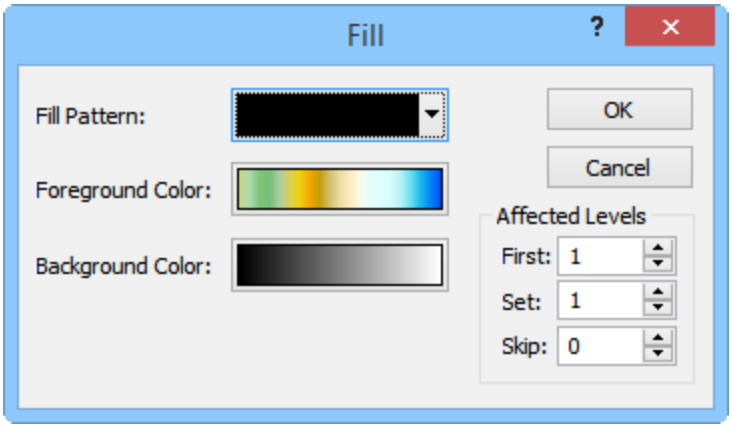
Setting Fill Properties on a Frequency Basis

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, Backgorund 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.

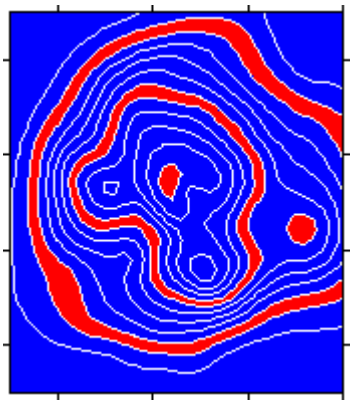
Skip

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, *Set*=one, *Skip*=four means apply the properties to the first contour level, skip four levels, set the next level, and so on. Then, you can enter *First*=two, *Set*=four, *Skip*=one. This applies the new properties starting at the second level. 4 consecutive levels are modified, 1 is skipped, 4 are modified, 1 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 fill 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



This is an example of a map using multiple fill styles.

Contour Labels

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,

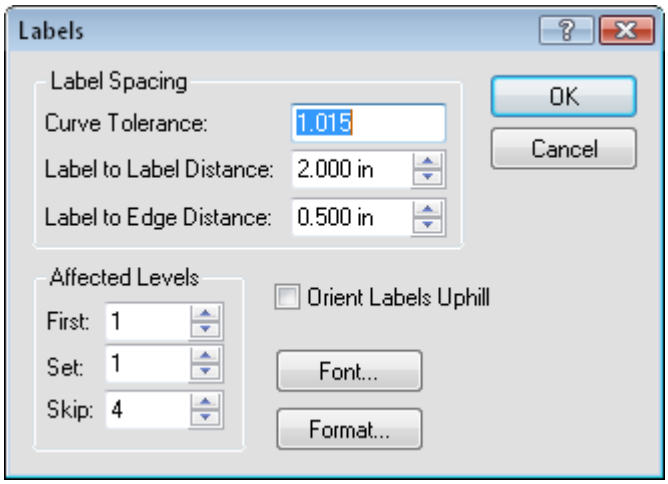
1. Click on the contour map to select it.
2. In the Properties window, open the Levels tab.
3. Set the Level method to Simple or Logarithmic.
4. Check the box next to *Show labels* in the *Major Contours* section to show labels along the major contour lines.
5. Check the box next to *Show labels* in the *Minor Contours* section to show labels along the minor contour lines.
6. Click the ⊞ next to *Font Properties* to set the text 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. 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. Hachures are only displayed when using the advanced level contour map settings.

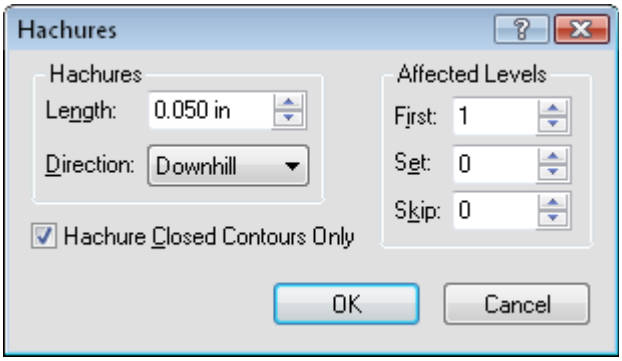
To display hachures on a contour map:

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* in the list.
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

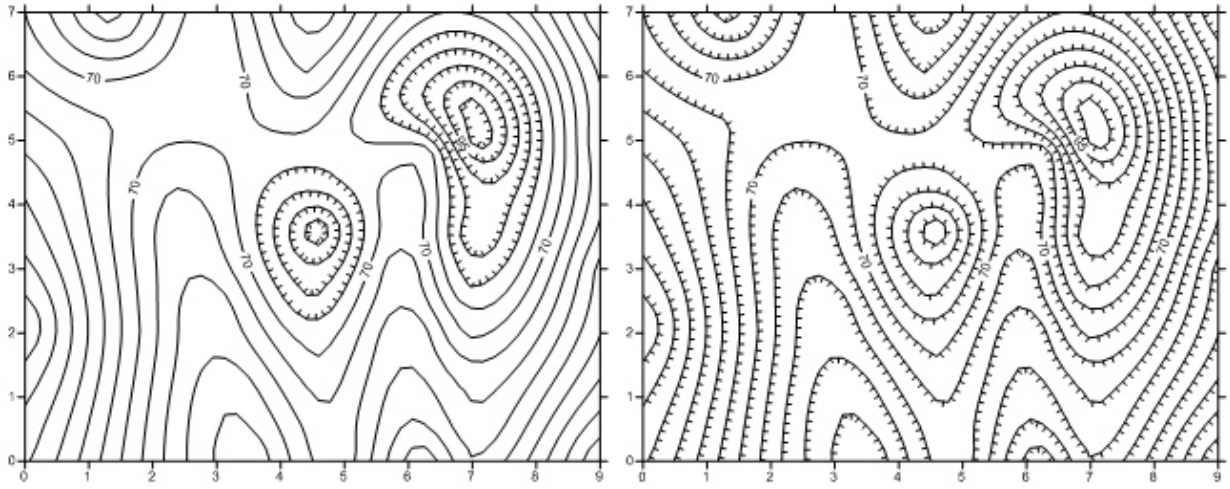
The *Direction* box controls whether hachures point in an upward or downward direction. For example, to point hachures towards contours of lower value, choose the *Downhill* option. To change the *Direction*, click on the existing option and select the desired option from the list.

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 map.
3. Select the base map in the **Contents** window and click the Features | Group | Start Editing command. Everything you draw will be added to this base map layer until you exit the group.
4. 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.
5. Click the Features | Group | Stop Editing command.
6. When the base map is selected, the base map properties are displayed in the **Properties** window. Click on the **General** tab.
7. 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.
    1. Select the .GRD file created in step 1 as the *Input Grid*.
    2. Select the .BLN file created in step 2 as the *NoData Polygon Boundary.*

3. 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:

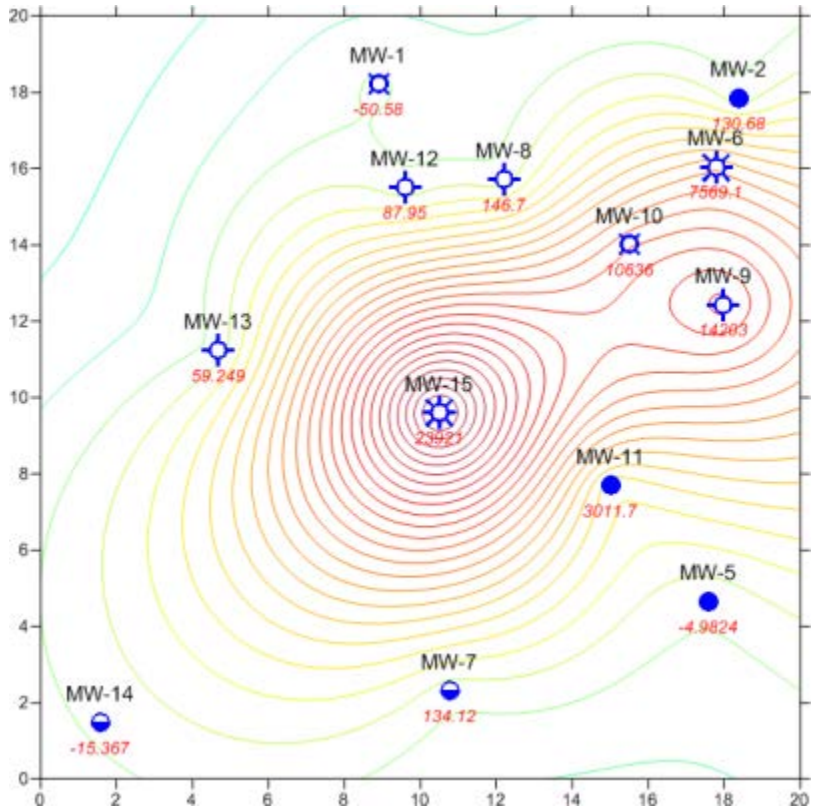
- By using Grids | New Grid | Grid Data to produce a denser grid file from your original data. As a rule, contour maps made from high-density grids are visibly smoother than contour maps made from low-density grid files. For example, a 10 X 10 grid file (10 rows and 10 columns) results in more angular 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.
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

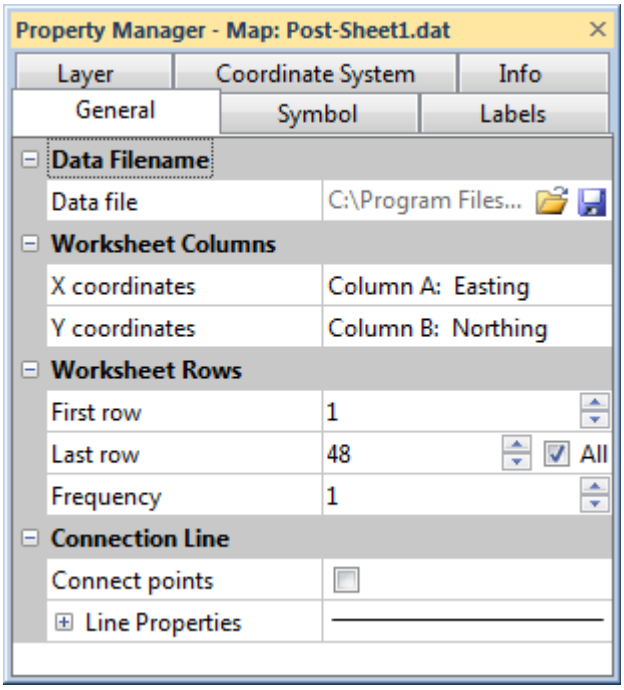
Map Properties

The map properties contains the following pages:

View
Scale
Limits
Frame
Coordinate System
Info

General Page - Post Map

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. Alteratively, 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.

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.

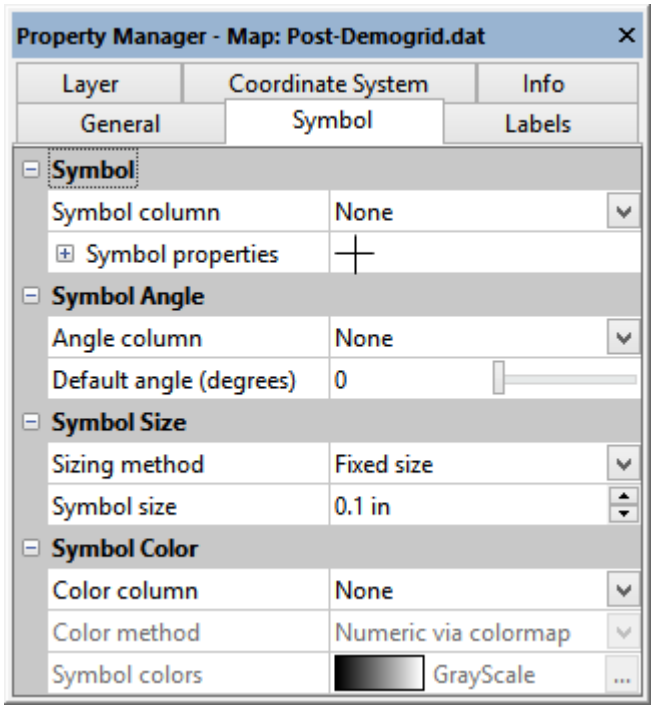
The line is drawn below the points. With open symbols, this can cause the line to appear inside the symbol. If the line should not be drawn through the symbol, use a closed symbol with the *Fill color* set to white instead of the open symbol.

Line Properties

Click on the ⊞ next to Line Properties to set the connecting line properties. The *Style, Color, Opacity*, and *Width* can be set.

Symbol Page - Post Map

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 *Symobl 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%.

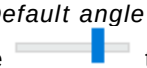
Symbol Angle

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  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 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.

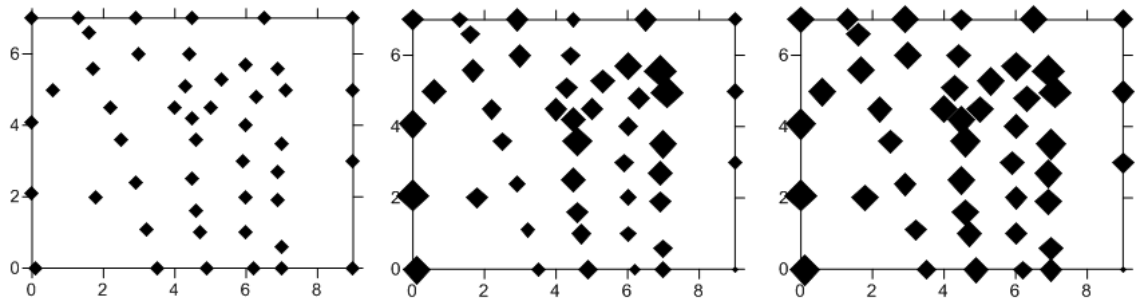
When the *Color method* is set to *Numeric via colormap*, the *Color column* should contain numeric values. The values are mapped to a colormap using the *Symbol colors* option.

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 dialog appears. Make any changes and click *OK* to see the change on the map.

Proportional Scaling

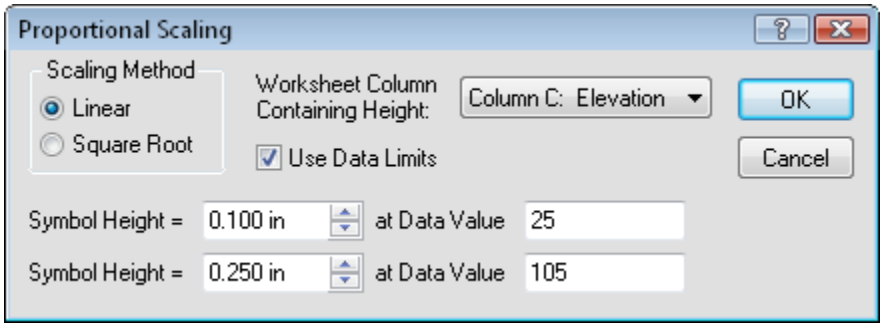
The *Symbol Size* group on the Symbol page of the post map properties allows you to specify the symbol *Sizing method* in two ways: fixed or proportional. *Fixed size* uses the same size symbols throughout the map. Set the *Sizing method* to *Fixed size* and set the *Symbol size* in page units. Change 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 H_n is the symbol height for a symbol with the proportional value Z_n.

Linear Scaling Method

If the *Scaling Method* is set to *Linear*, a standard linear interpolation is used:

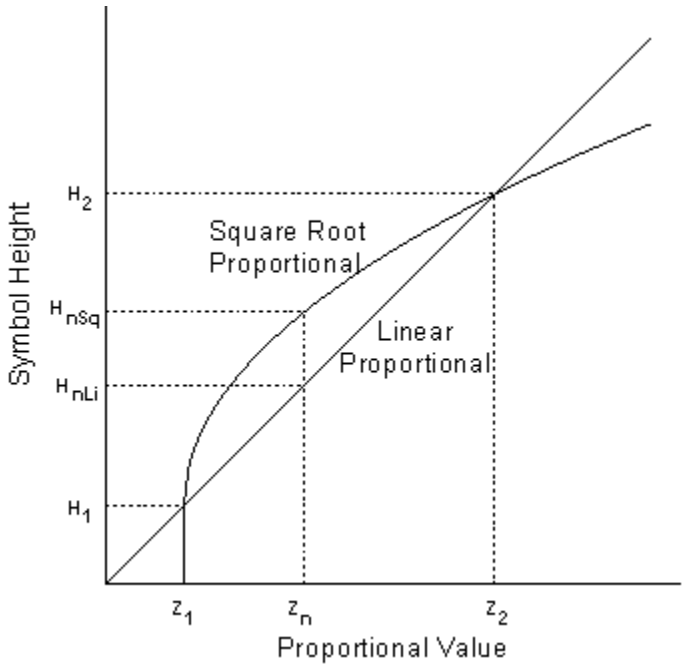
$$H_n = [((Z_n - Z_1)/(Z_2 - Z_1)) * (H_2 - H_1)] + H_1$$

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.

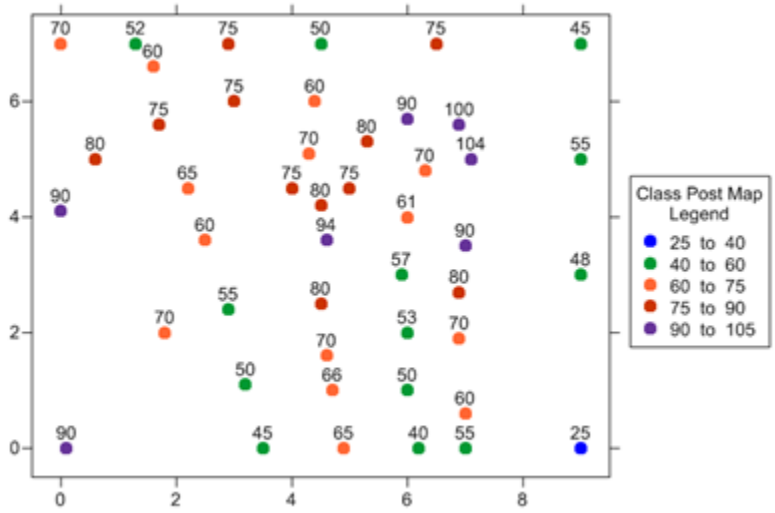
Use Data Limits

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.
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:

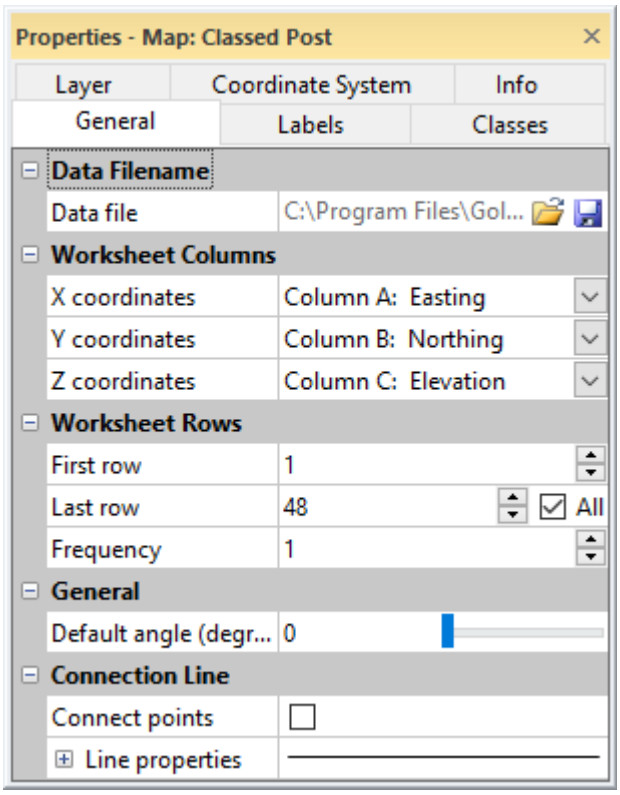
View

Scale

Limits
Frame
Coordinate System
Info

General Page - Classed Post Map

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*.

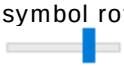
Frequency

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*.

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  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.

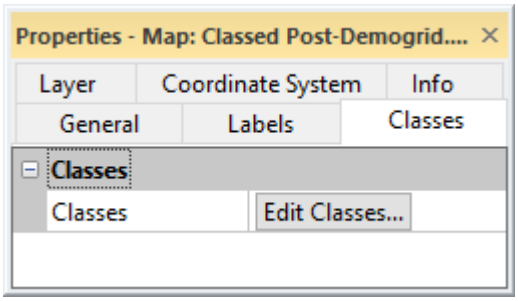
The line is drawn below the points. With open symbols, this can cause the line to appear inside the symbol. If the line should not be drawn through the symbol, use a closed symbol with the *Fill color* set to white instead of the open symbol. Symbols are set on the Classes tab.

Line Properties

Click the ⊞ next to Line Properties to set the connecting line properties. The *Style, Color, Opacity*, and *Width* can be set.

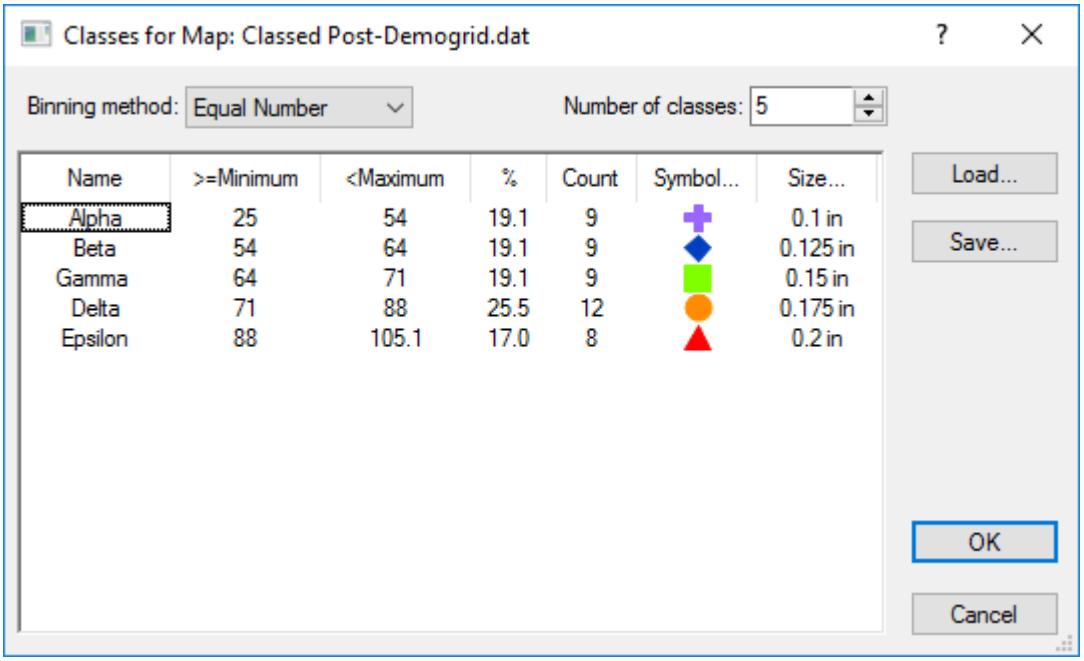
Classes Page

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. 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.

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 a 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

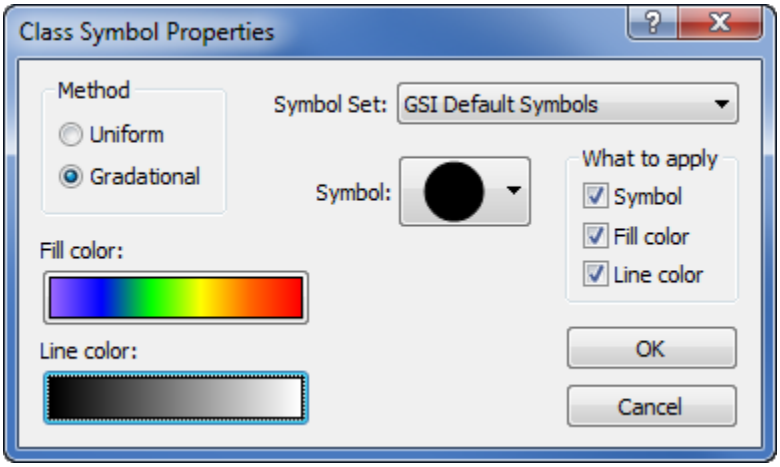
Click the *Save* button to save a classed post map .CLS class file. The range minimum, range maximum, symbol set, symbol index number, color, and symbol size are saved in the .CLS file. The color is in RGBA format.

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** dialog, 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** dialog, 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

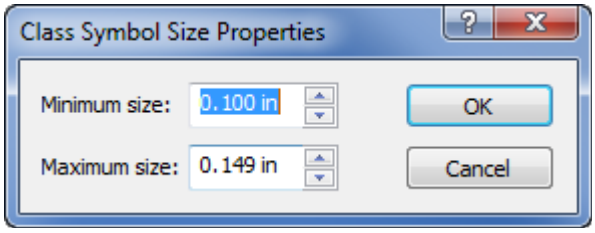
Click *OK* to make the changes to the class symbol properties. Click *Cancel* to close the dialog without making changes.

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*.

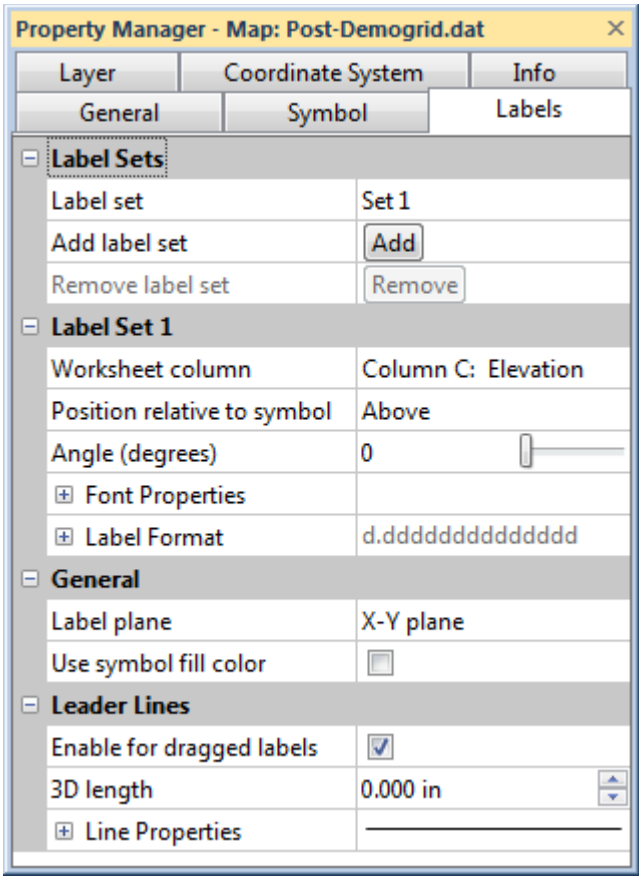
OK or Cancel

Click *OK* to make the changes to the symbol sizes for all classes. Click *Cancel* to close the dialog without making changes.

Labels Page

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.

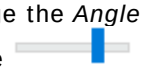
X Offset and Y Offset

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

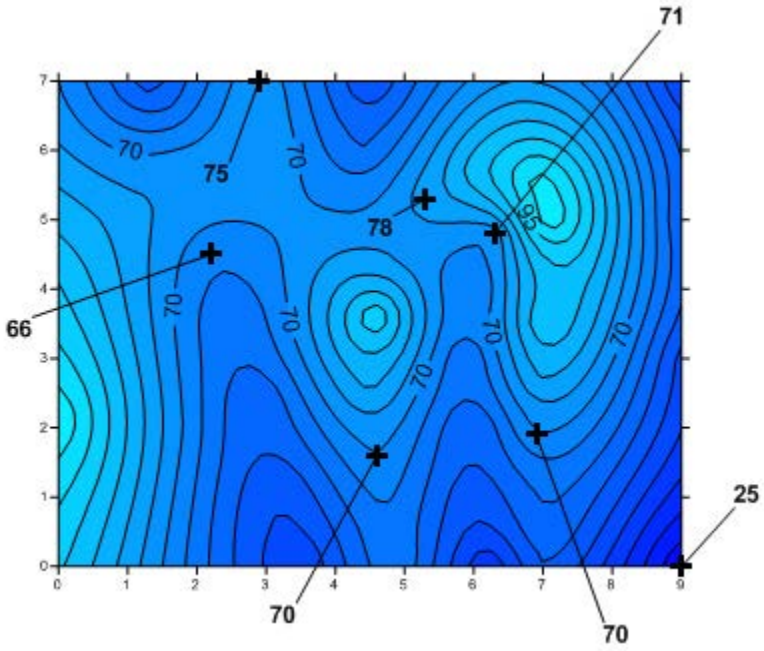
The *Use symbol fill color* option controls whether the color of the symbol is used for all label sets. When the *Use symbol fill color* option is checked, all label sets for a point use the symbol color for the text color instead of the *Font Properties* section *Foreground color* option. When the symbol 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

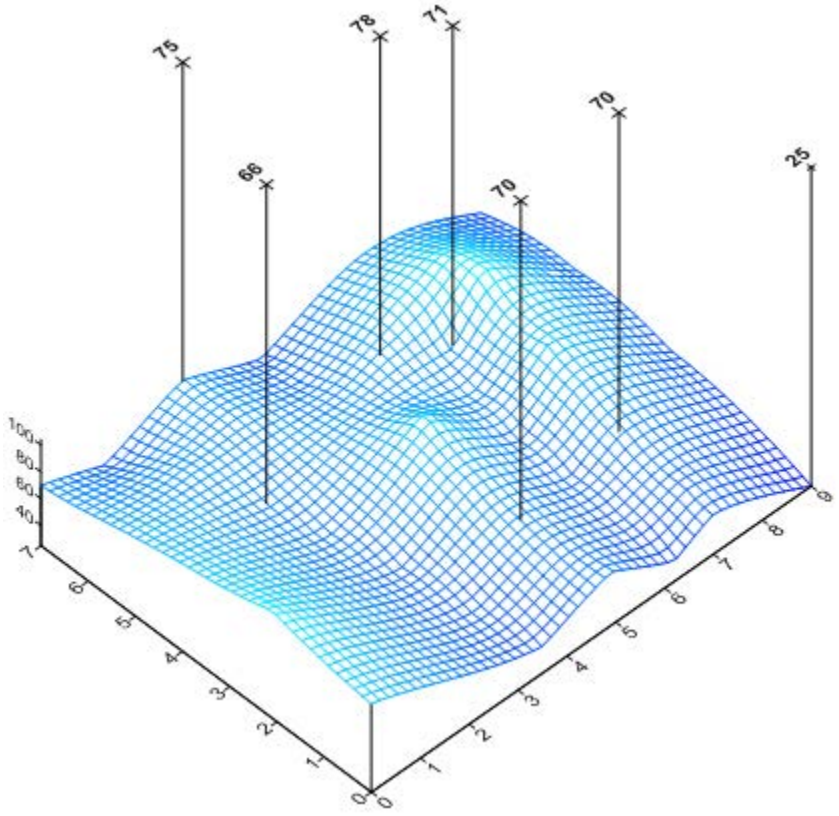
Check the *Enable for dragged labels* option to display leader lines when labels are manually dragged to a new location with the Edit Post Labels command. The leader line will display a line from the original label location to the dragged location.



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*,
    1. In the *Label Set 1* section, set the *Worksheet column* for the first label.
    2. Change the *Position relative to symbol* to the desired position.
    3. 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*,
    1. In the *Label Set 2* section, set the *Worksheet column* for the second label.
    2. Change the *Position relative to symbol* to the desired position.
    3. 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 | Post 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 | Post Labels** command. Alternatively, right-click on the selected map and click **Edit Post Labels**. 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 | Post Labels** command again.

Move Around the Plot Window in Edit Mode

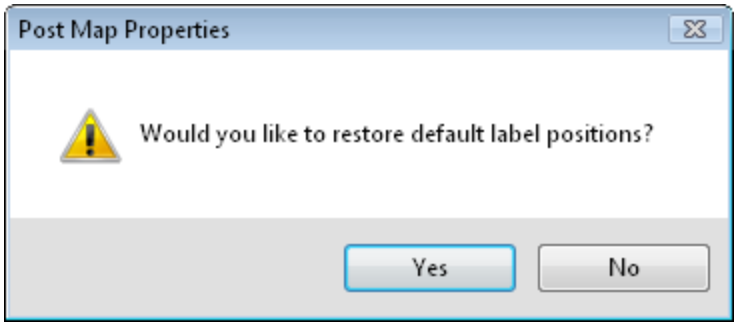
If you are zoomed in on the post map, use the scroll bars to move to locations that appear off the screen. Alternatively, click and hold the scroll button of a mouse wheel to pan the plot window.

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.

Plot1* | Sheet1.dat

B10 | 7

	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 **Surfer 13** 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 **Surfer 15** 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 **Surfer 15** 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

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. Use the **Home | New Map | 3D Surface** 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 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.

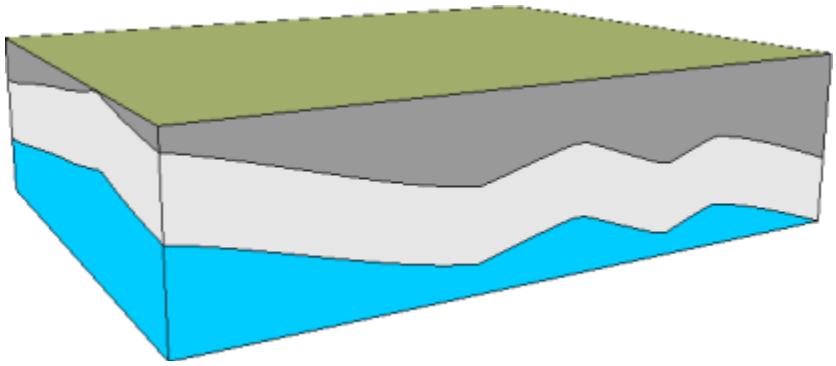
Another alternative, is to create both maps using the **Home | New Map** commands. Then, select one map and drag the map layer to the other map object. This is equivalent to using the **Overlay Maps** command to overlay maps. For example, create a contour map with the **Home | New Map | Contour** command. 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 Properties

The surface properties contains the following pages:

General

Mesh

Lighting

Overlays

Info

Map Properties

The map properties contains the following pages:

View

Scale

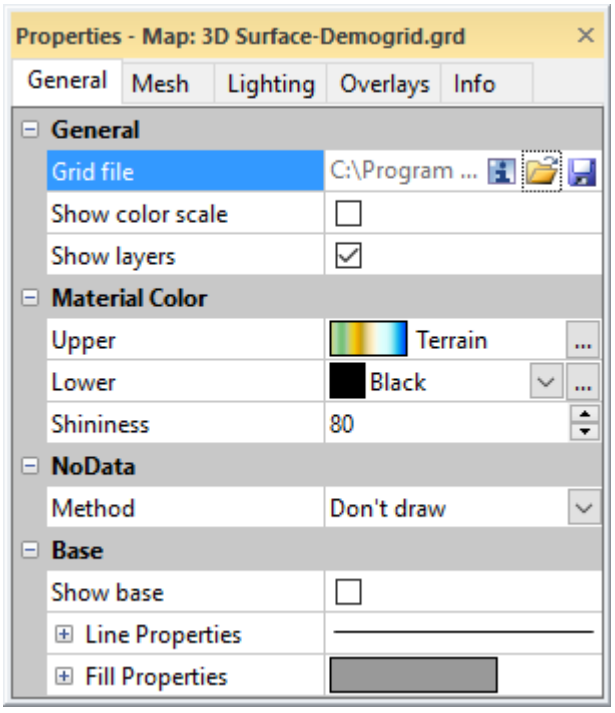
Limits

Frame

Info

General Page - 3D Surface

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 properties **General** page contains the following options:



*Change 3D surface 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 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 dialog, 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 dialog 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 dialog, 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** dialog.

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 from the edge of the surface to the axis lines. This creates 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 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. Alternatively you can adjust the line with by clicking the buttons.

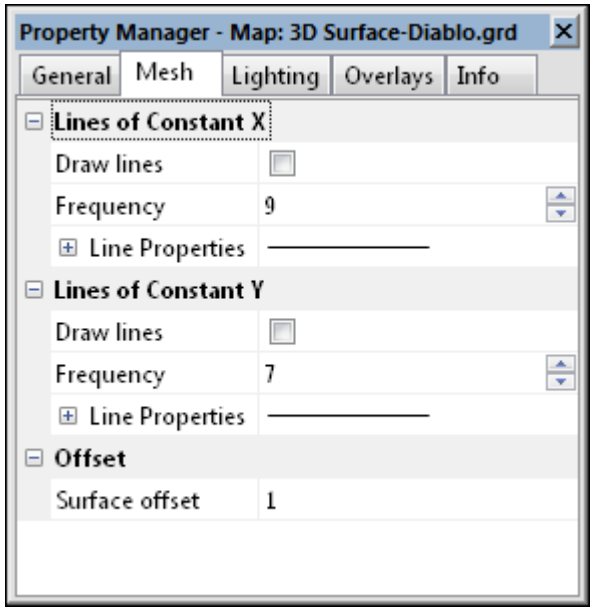
Base Fill Properties

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.

Mesh Page - 3D Surface

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 Page

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. 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 and any overlays that may have been combined with the surface. The light source is fixed relative to the surface, so if the surface is rotated, the light rotates with it.

Hill Shading and Shaded Relief in the 3D View

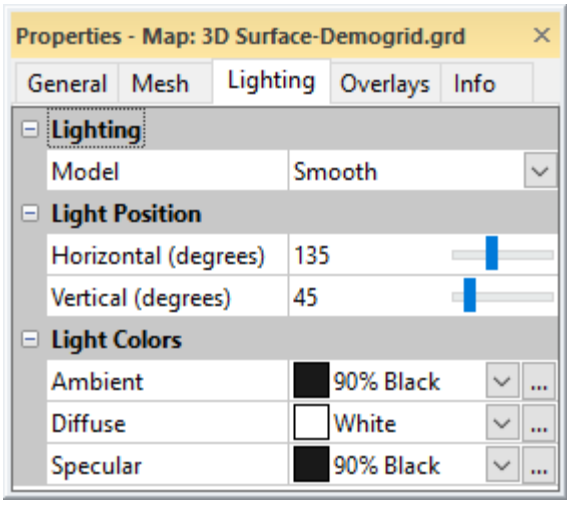
The 3D view lighting is turned off when the map in the 3D view contains a hill-shaded color relief layer or a shaded relief layer. This is because the map is already shaded by the color relief layer hill shading or shaded relief layer. The cumulative effect of both lighting systems would result in very dark regions of the map in the 3D view.

To use the 3D view **Lighting** properties instead of the shaded relief layer, the shaded relief layer must be removed from the map in the plot window. The shaded relief layer can be deleted to remove it permanently, or the shaded relief layer can be broke apart from the map to remove it temporarily.

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.

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.

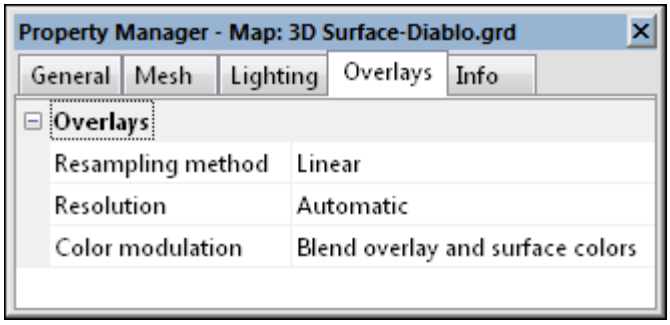
Surfer uses a pure white light source. The light "strikes" the surface and some of the light is absorbed based on the color of the surface material at the point the light ray struck it. Some light is reflected to the viewer according to the type of light (*Ambient*, *Diffuse*, and *Specular*), and the 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.

Overlays Page - 3D Surface

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 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

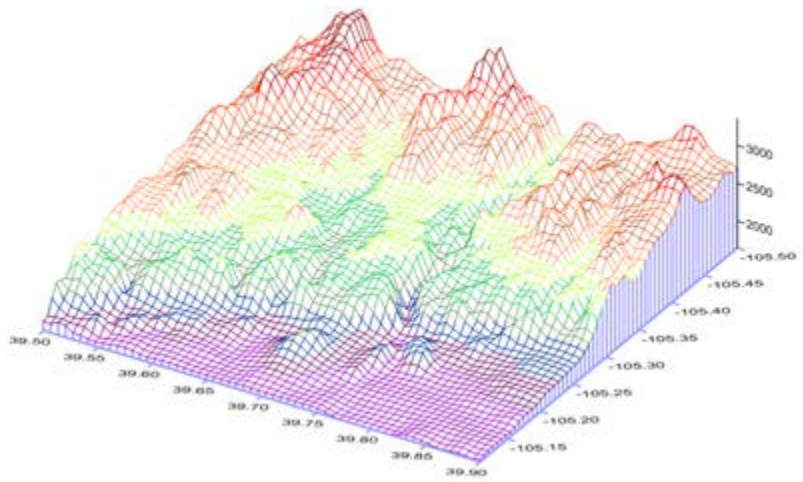
Color modulation refers to the method used to combine the texture map and surface material colors. You can *Use surface color only*, *Use overlay color only*, or you can *Blend overlay and surface colors*. When set to *Use surface color only*, any other overlays will not be shown in the map. The surface 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

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 Grids | Edit | 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 the **Home | New Map | 3D Surface | 3D Wireframe** 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 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, shaded relief maps, and base (raster) layers.. Wireframes can be overlaid with contour maps and other map types.

Wireframe Properties

The wireframe properties contains the following pages:

General
Z Levels
Color Zones
Layer
Info

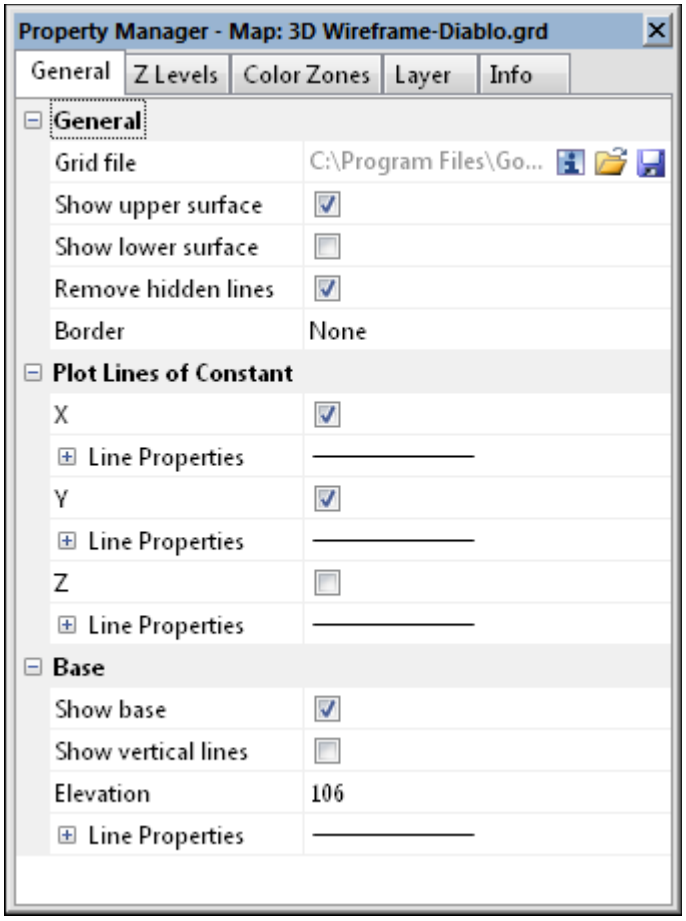
Map Properties

The map properties dialog contains the following pages:

View
Scale
Frame
Info

General Page - Wireframe

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

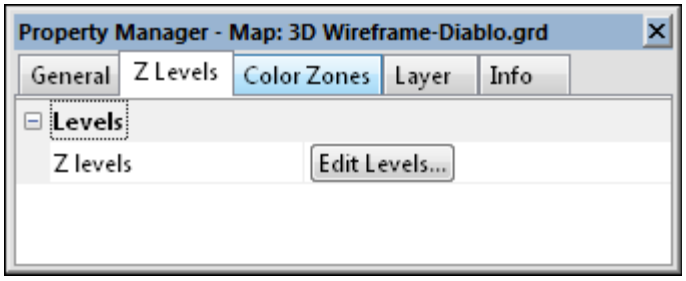
You can adjust the elevation of the base in the *Elevation* box. To change the elevation, highlight the existing value and type a new one.

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.

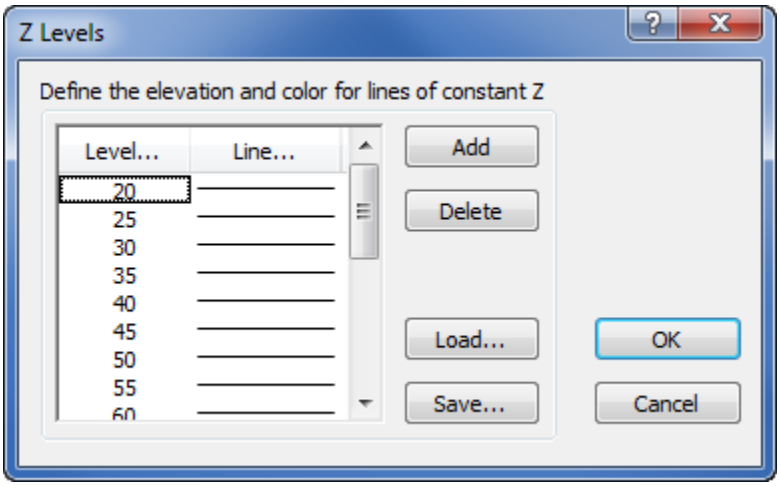
Z Levels

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 the Z Levels options 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

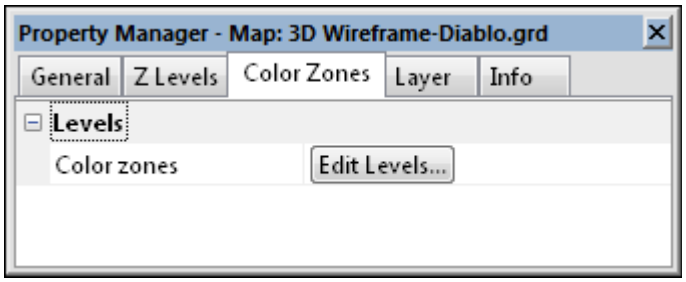
Click the *Save* button to save a level file.

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.

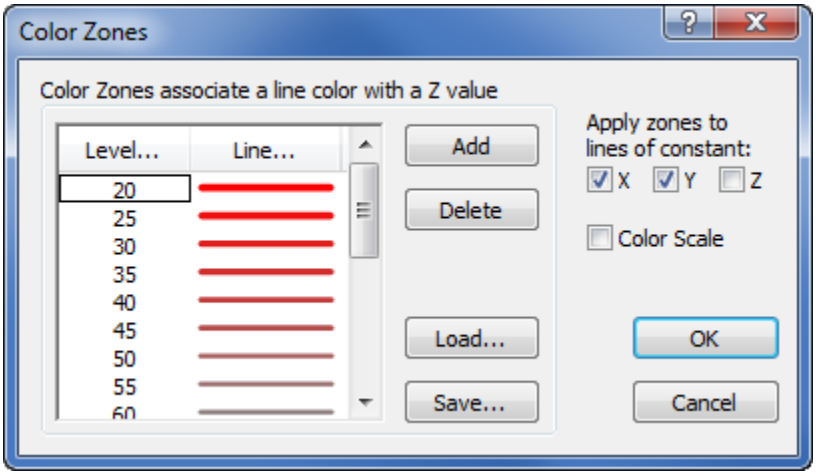
Color Zones

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

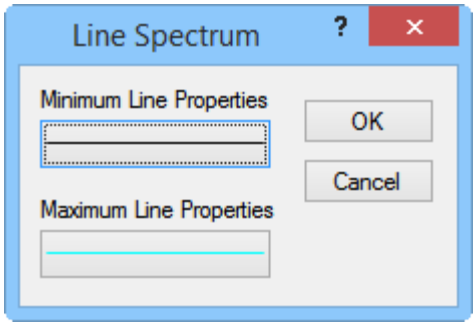
A color scale bar can be displayed on the map by checking the box next to the *Color Scale* option on this page.

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.

Level Files - Wireframe

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.
1. 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.
2. 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**.
   3. Set any other desired options on the General page.
   4. 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.

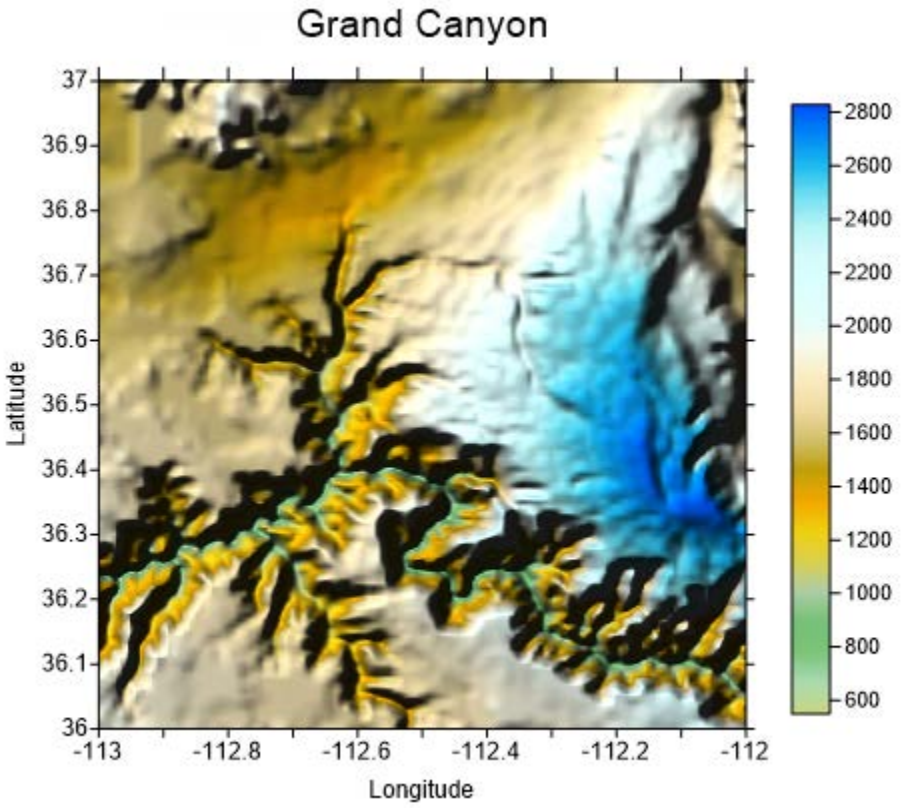
Wireframe NoData Regions

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 Reilef Maps

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 can be enabled to enhance contrast and visualization.



This color relief map shows the Grand Canyon. Colors show elevation change 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.

Creating a Color Relief Map

To create a color relief map:

1. Click the **Home | New Map | Color Relief** command.
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 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

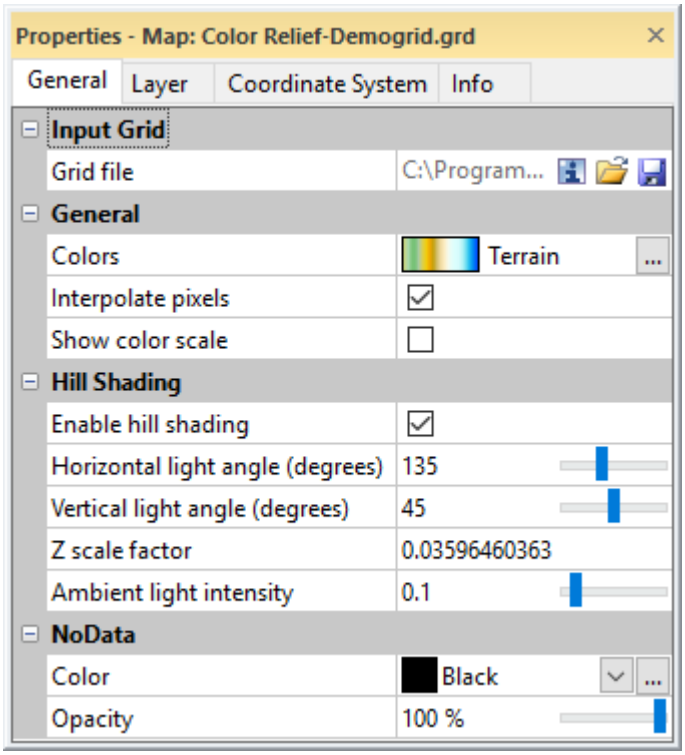
The map properties contains the following pages:

View

Scale
Limits
Frame
Coordinate System
Info

General Page - Color Relief

To edit an color relief map 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 layer 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.

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 dialog 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.

Enable Hill Shading

Hill shading is a method of representing relief by depicting shadows cast by a point light source. Click the *Enable hill shading* check box 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 remaining hill shading options are enabled when the *Enable hill shading* box is checked.

Hill shading replaces the 3D view Lighting properties. To use the 3D view lighting properties when viewing the map in the 3D view, clear the *Enable hill shading* check box.

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

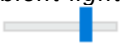
The *Vertical light angle (degrees)* box rotates the light source in the vertical plane. A vertical angle of zero degrees places the light source at the horizon and shining horizontally. An angle of 90 degrees places the light source directly overhead and shining down onto the map. The default

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.

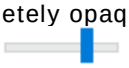
NoData

The *NoData* section sets the color for grid nodes that have been assigned the NoData value.

NoData Color

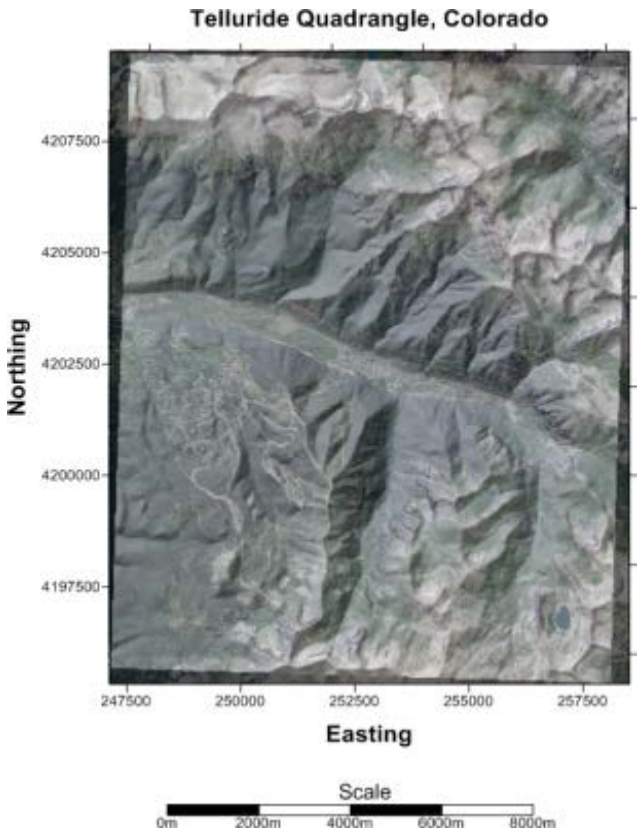
To select a different NoData region color, click on the existing color to the right *Color* and click on the desired color from the palette. Click the button to the right of the selected color to define new colors in the Colors dialog.

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.

Chapter 14 - Shaded Relief Maps

Shaded relief maps are raster maps based on grid files. These maps use 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.



This shaded relief map shows Telluride, Colorado with the default shaded relief shading and gradient settings.

The colors on a shaded 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.

Because colors are assigned to entire grid cells, sparse grids (grids with relatively few rows and columns) are poor candidates for shaded relief maps. Shaded relief maps based on grids with too few cells look blocky or fuzzy. The Spline Smooth or Mosaic commands can be used to make a denser grid from an existing grid.

You can assign colors to represent the various reflectance values. Colors between the assigned values are automatically blended to make a smooth gradation. NoData regions on shaded relief maps are shown as a separate color.

Any color scheme selected for a shaded relief map may be used with any other shaded relief map, even if the associated grid files result in significantly different orientations of the surface relative to the light source. Color schemes can be saved in color spectrum .CLR files.

Creating a Shaded Relief Map

To create a new shaded relief map:

1. Click the **Home | New Map | Speciality | Shaded Relief** command.
2. Select a grid file in the **Open Grid** dialog and click *Open*.

The map is created with the default settings.

Editing an Existing Shaded Relief Map

To change the features of the shaded relief map, click once on the shaded 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 shaded relief maps are created, they are independent of other maps in the plot window. For example, creating a shaded 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 shaded relief layer on the contour layer, 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 shaded relief map directly to the existing contour map by creating the shaded relief map using the **Home | Add to Map | Layer | Shaded Relief** or **Map Tools | Add to Map | Layer | Shaded Relief** command. This automatically adds the shaded relief 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 shaded relief map using the **Home | New Map | Speciality | Shaded Relief** command. This creates two separate maps. Click on the shaded relief map, hold down the left mouse button, and drag the shaded relief map into the contour map. A single map with two map layers, using one set of axes and scaling parameters is created.

Shaded relief maps cannot be added as a map layer to 3D wireframe maps, but a shaded relief map can be added to any other map layer, including 3D surface maps.

Shaded Relief Layer in the 3D View

When the map in the 3D view contains a shaded relief layer, the 3D view Lighting is turned off. This is because the map is already shaded by the shaded relief layer. 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 shaded relief layer, the shaded relief layer must be removed from the map in the plot window. The shaded relief layer can be deleted to remove it permanently, or the shaded relief layer can be broke apart from the map to remove it temporarily.

Shaded Relief Layer Properties

The shaded relief map layer properties contains the following pages:

General

Layer

Coordinate System

Info

Map Properties

The map properties contains the following pages:

View

Scale

Limits

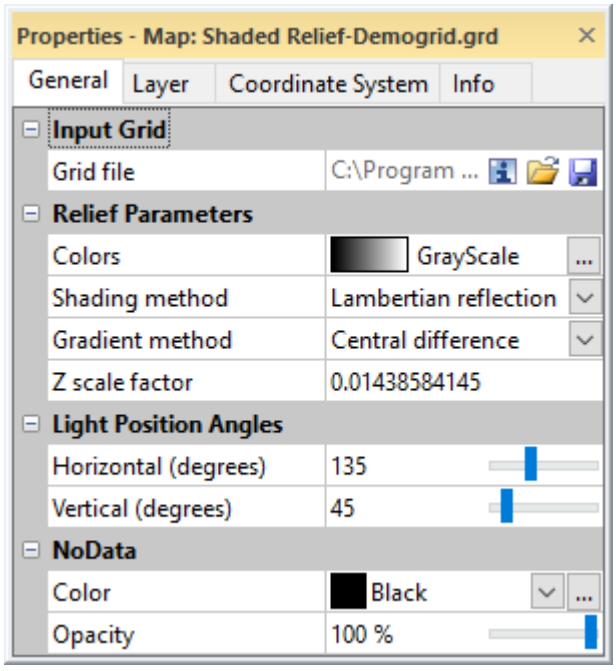
Frame

Coordinate System

Info

General Page - Shaded Relief Map

To edit a shaded relief map layer, click once on the shaded relief layer to select it. The properties for the shaded relief are displayed in the Properties window. The shaded relief **General** page contains the following options:



*Change shaded relief 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 layer. 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 shaded relief layer. 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 shaded relief layer 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.

Relief Parameters

Specify the Colors, Gradient method, Shading method, and Z scale factor in the Relief Parameters section.

Colors

The *Colors* option defines the colormap used to fill the shaded relief map. 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 color. The Colormap dialog appears. Make any changes and click *OK* to see the change on the map.

Shading Method

The Shading Method specifies the algorithm used to compute the reflected light at each grid cell. The *Shading Method* determines how the colors are distributed in relation to the slopes and slope directions (aspect) over the extent of the map.

Gradient Method

The *Gradient method* controls the smoothness of the shaded relief map.

- The *Central difference* method averages the slope and orientation of the surface across three adjacent grid nodes. This results in a smoother shaded relief surface. Because a node on either side is required, the NoData color and opacity are applied to edge cells.
- The *Midpoint difference* method computes the gradient at the center of each grid cell. This method provides less smoothing, but does not assign the NoData color and opacity to the grid cells at the edge of the map.

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}.

Light Position Angles

The *Light Position Angles* section specify 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.

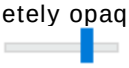
NoData

The *NoData* section alters the fill properties for areas where grid nodes have been assigned the NoData value.

NoData Color

The *Color* shows the color that is used to draw NoData regions of the grid. To select a different color, click the existing color and choose a new color from the palette. You may click the button to the right of the selected color to open the Colors dialog, where you can define new colors.

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.

References

Many of the parameters for the shaded relief map are technical in origin, and a detailed development is beyond the scope of this document. For a complete presentation of the underlying theory and mathematics, see Horn (1982).

Horn, B.K. (1982), *Hill Shading and the Reflectance Map*, Geo-Processing, v. 2, p. 65-146.

Shading Methods

There are four shading methods available for shaded relief layers. To set the *Shading method*, click on the shaded relief layer to select it. In the Properties window, click on the General tab. Change the *Shading 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 *Shading method.*

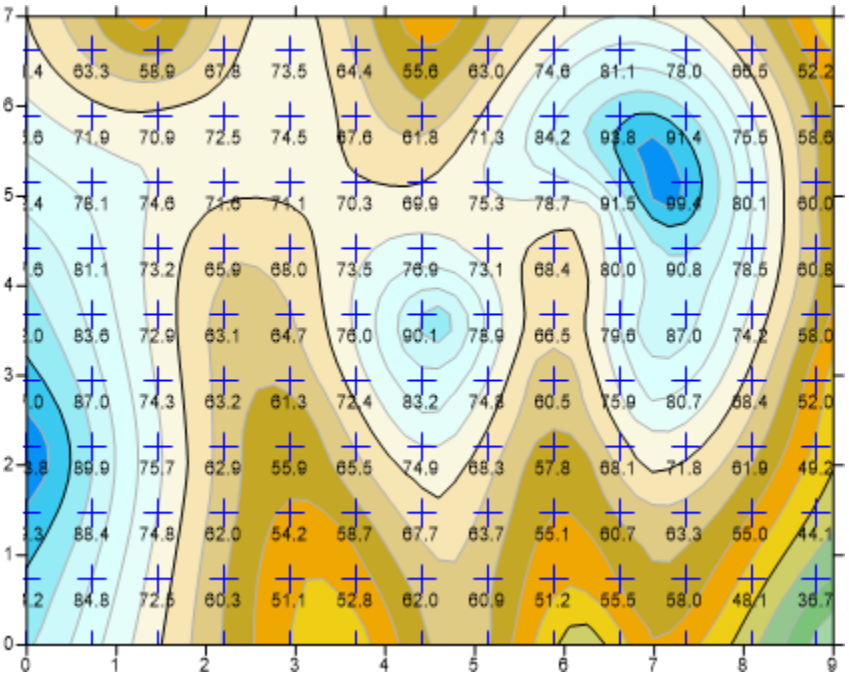
Lommel-Seeliger Law

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 15 - Grid Values Maps

The **Home | New Map | Speciality | 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

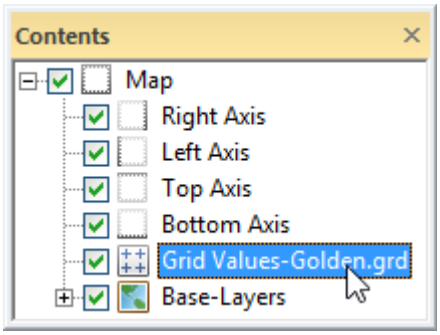
To create a new grid values map:

1. Click **Home | New Map | Speciality | 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

Map Properties

The map properties contains the following pages:

View

Scale

Limits

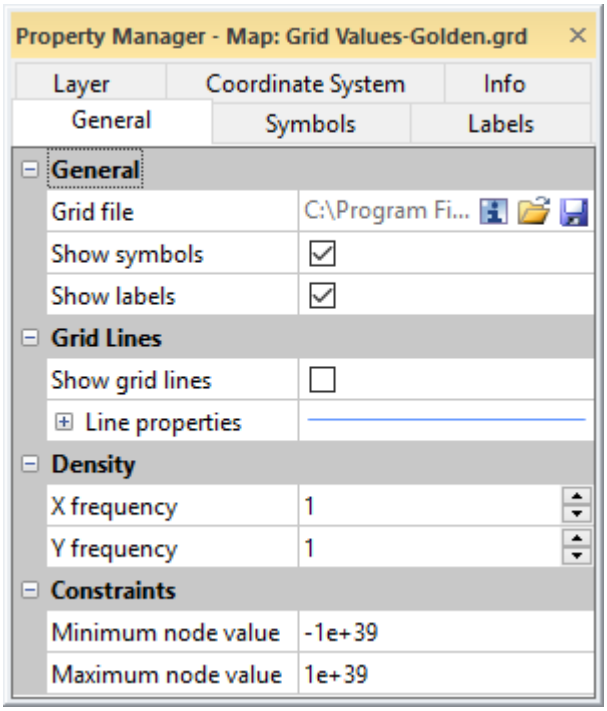
Frame

Coordinate System

Info

General Page - Grid Values Layer

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

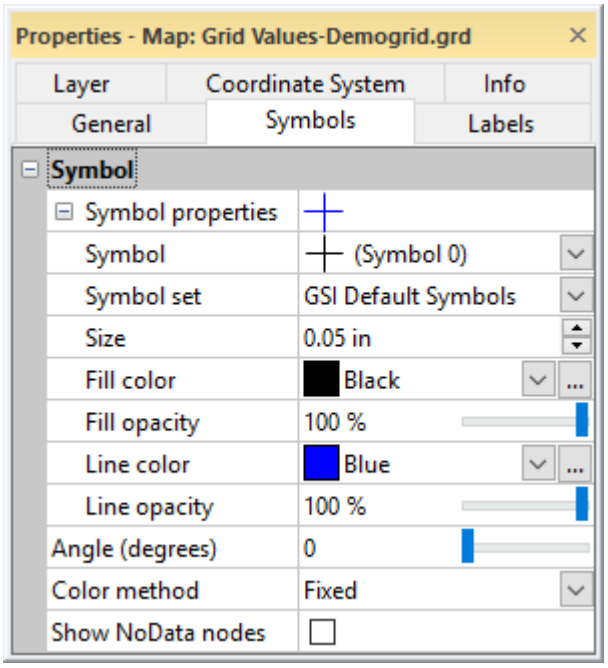
The *Minimum node value* sets the smallest grid node value that will have a symbol and/or label displayed on the grid values map. By default the *Minimum node value* is -1e+39, i.e. -1×10^{39}. Only grid nodes with a value greater than or equal to the *Minimum node value* will be displayed.

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. 1×10^{39}. Only grid nodes with a value less than or equal to the *Maximum node value* will be displayed.

Symbols Page - Grid Values Layer

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

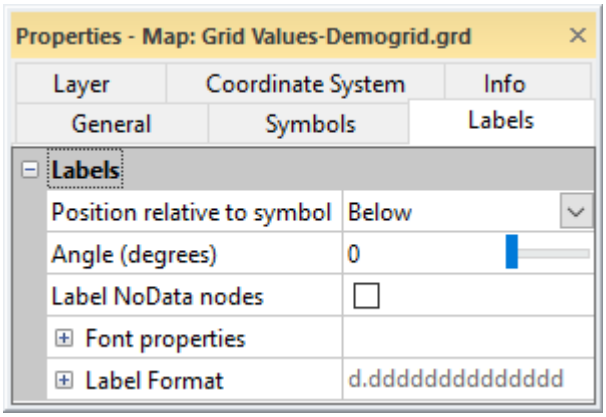
The *Colors* property is enabled when the *Color method* is set to *Colormap*. Click the current colormap and select a predefined colormap from the list or create a custom colormap in the Colormap dialog by clicking the ... button in the *Colors* field.

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.

Labels Page - Grid Values Layer

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.

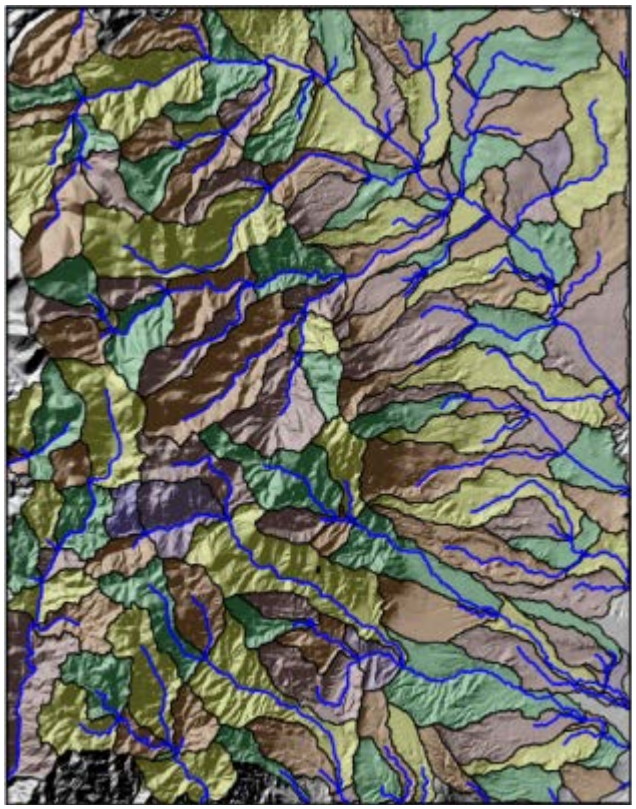
Label Format

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 16 - Watershed Maps

The **Home | New Map | Speciality | Watershed** command 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 shaded relief layer is overlaid.

Creating a Watershed

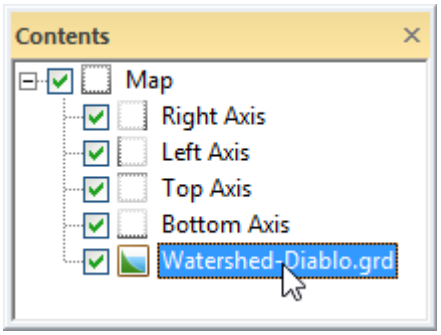
To create a watershed:

1. Click the **Home | New Map | Speciality | 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 | Speciality | Watershed Map** 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 watershed maps can be combined with any other map layers.

Watershed Properties

The watershed 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

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.

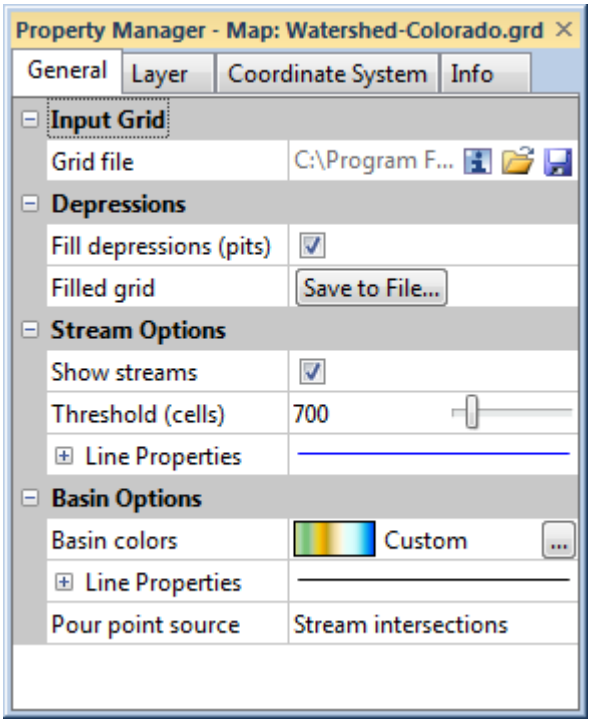
Welsh, D. J. A.; Powell, M. B. (1967), "An upper bound for the chromatic number of a graph and its application to timetabling problems." *The Computer Journal*, Vol. 10 (1): 85–86.

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.

General Page - Watershed

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.

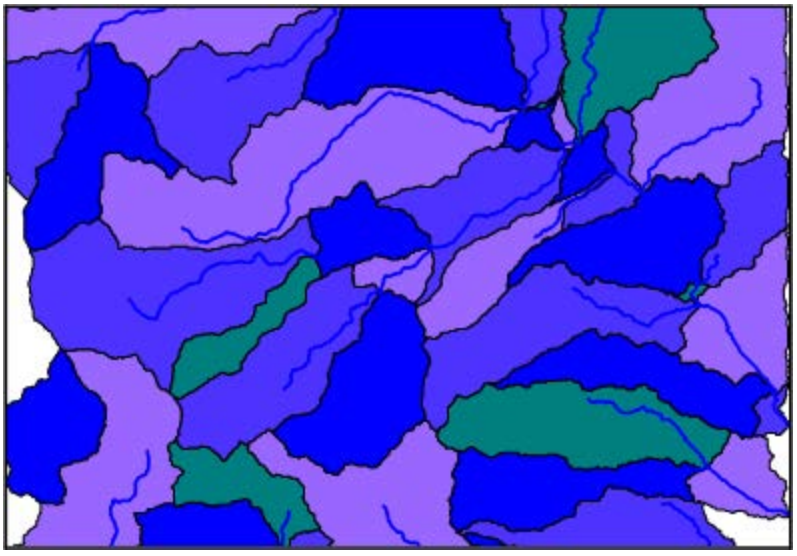
Basin Colors

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 dialog.

Surfer applies 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.

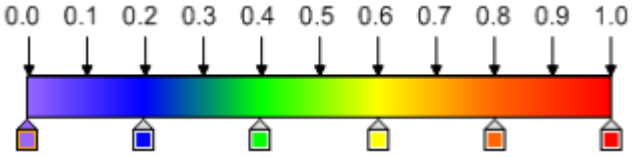
Example

Consider this watershed map and the *Rainbow* colormap.



The watershed is filled with four colors, ranging from a purple to green.

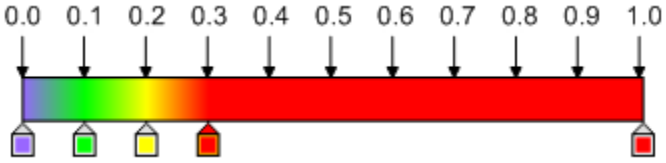
Because the watershed requires four colors, the colors at 1/10th increments (at values 0, 0.1, 0.2, and 0.3) are selected to fill the basins. These correspond to the light purple, blue-purple, blue, and blue-green colors.



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** dialog. 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.

The maximum number of pour points is 65535. If you exceed the maximum number of pour points 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

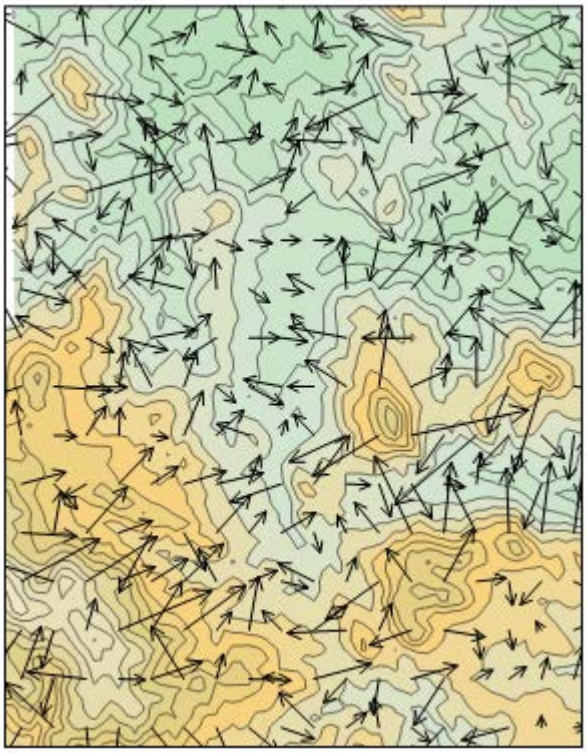
The *Snap pour points* option is used when the *Pour point source* is set to *File*. A common problem with using digitized points to define the pour points is locating the point slightly off the desired location, such as on the side of a hill. Checking this option moves the points in the *Pour point 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 17 - 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 | Speciality | 1-Grid Vector** command.
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 | Speciality | 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

Map Properties

The map properties contains the following pages:

View

Scale

Limits

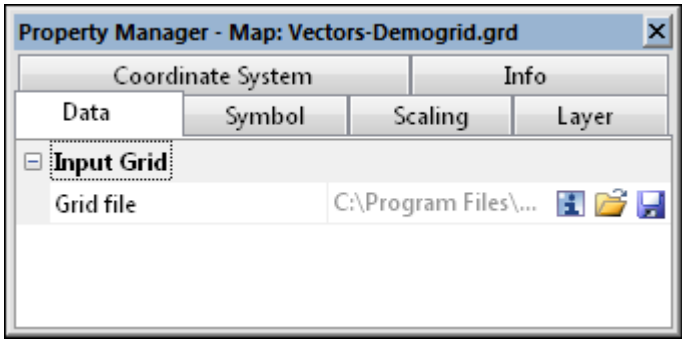
Frame

Coordinate System

Info

Data Page - 1-Grid Vector Map

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

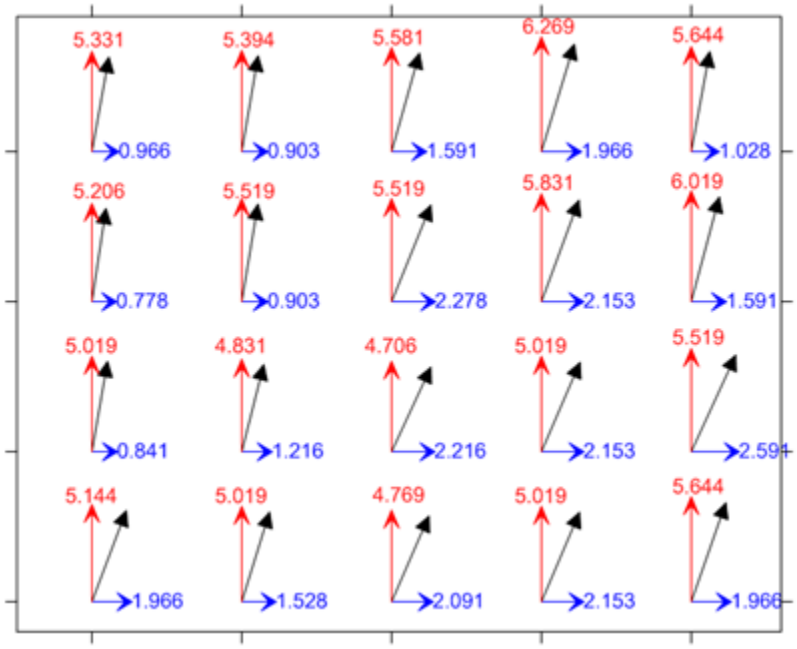
Regardless of the file selected with either the **File | Reload Map Data** command or by clicking the button in the **Properties** window, the coordinate system for the original map layer is used after updating the file. If this is not the correct coordinate system, click on the map layer to select it. 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 | Speciality | 2-Grid Vector** command.
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 | Speciality | 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

Map Properties

The map properties contains the following pages:

View

Scale

Limits

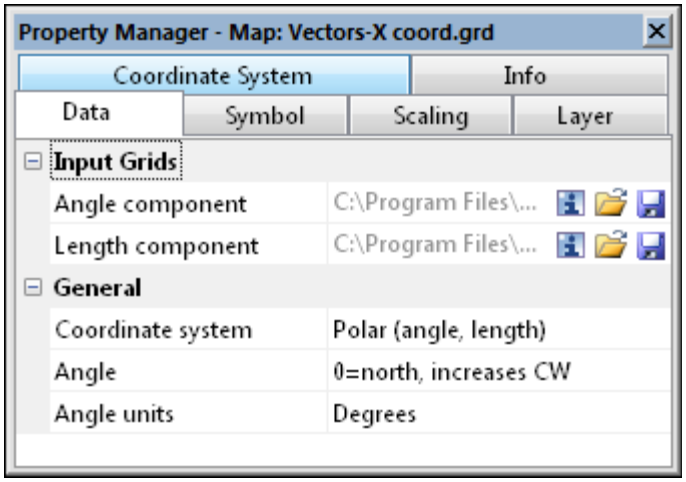
Frame

Coordinate System

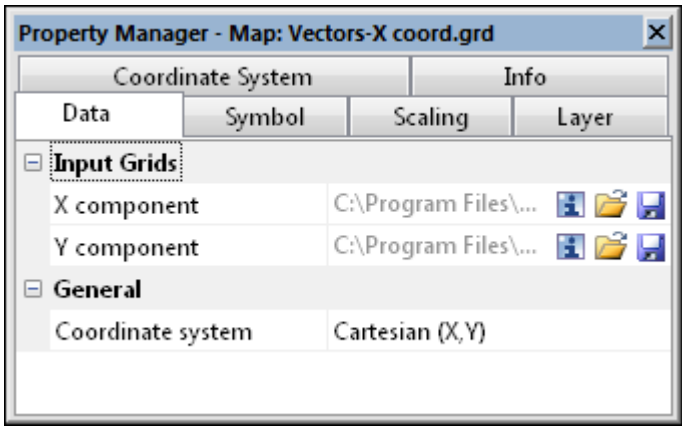
Info

Data Page - 2-Grid Vector Map

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

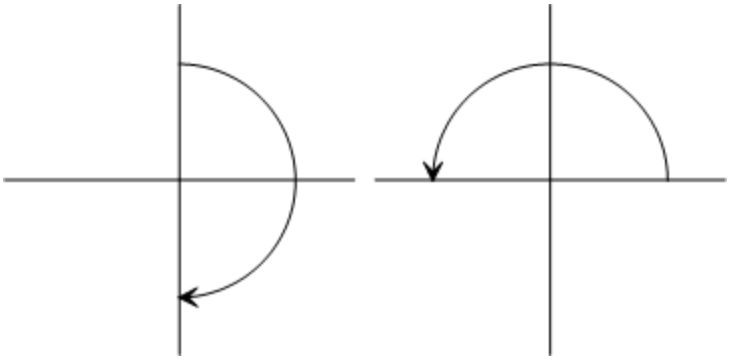
Click the button to display the **Save Grid As** dialog. This allows the grid file used for the 2-grid vector map to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file. If a coordinate system has been defined on the Coordinate System tab in the **Properties** 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=easth, 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

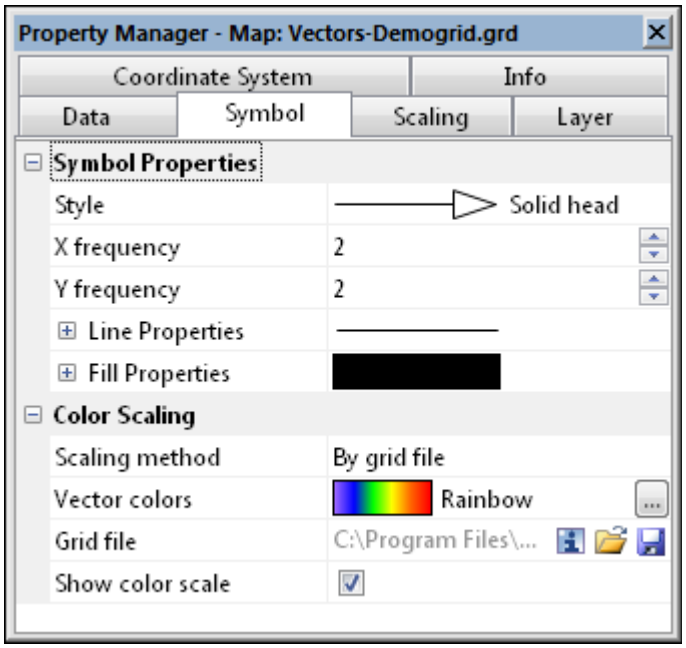
Magnitude values cannot be negative. If you have chosen a gridding technique that displays trends in the data, such as Kriging, you may end up with negative values in your grid even if the original data are all positive.

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 Grids | Calculate | 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).

Symbol Page - Vector Map

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 dialog 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** dialog. 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

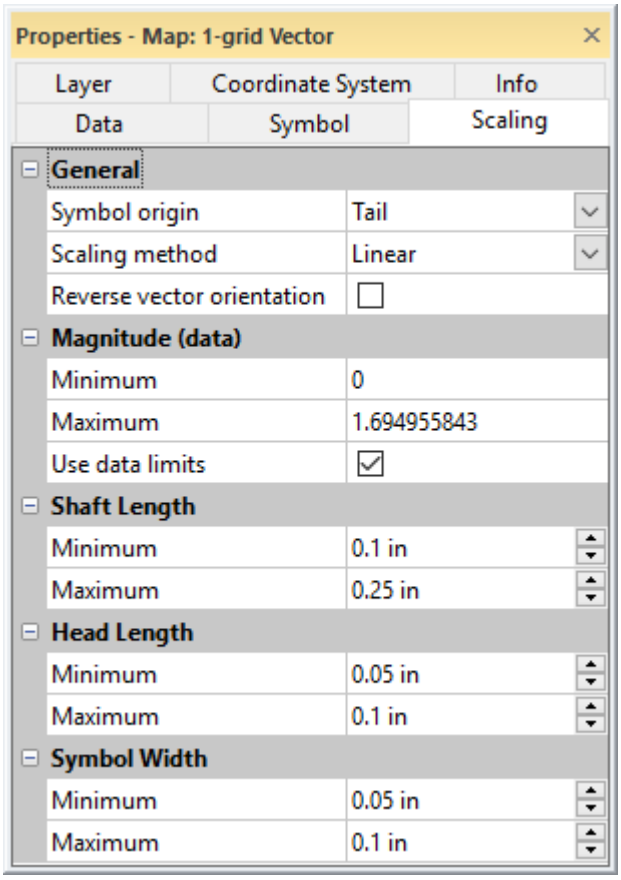
Click the button to display the **Save Grid As** dialog. This allows the grid file used to be saved to a new name. Set the *File name*, *Save as type,* and click *Save* to save the file.

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.

Scaling Page - Vector Map

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).

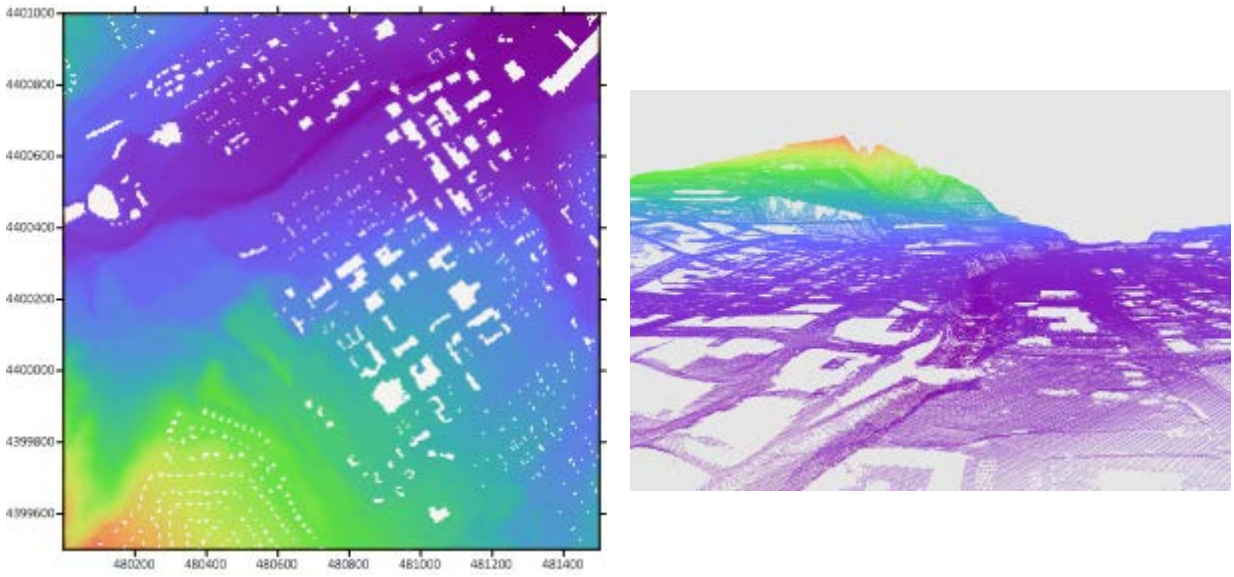
Clipping Symbols on Vector Maps

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 | Speciality | 1-Grid Vector Map** or **Home | New Map | Speciality | 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 18 - Point Cloud Maps

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 | Speciality | Point Cloud** command.
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 | Speciality | 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

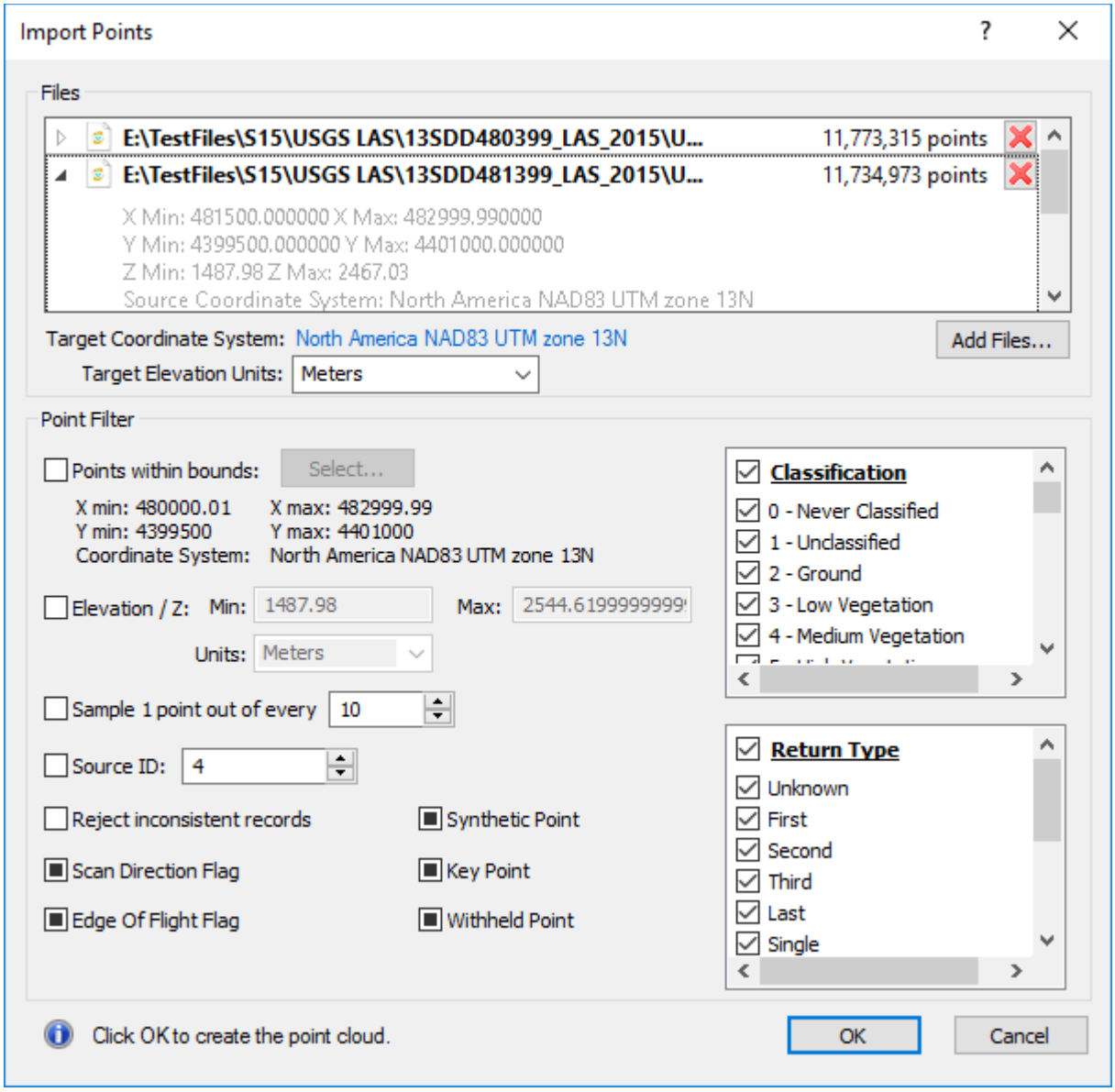
Coordinate System

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 n^{th} 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

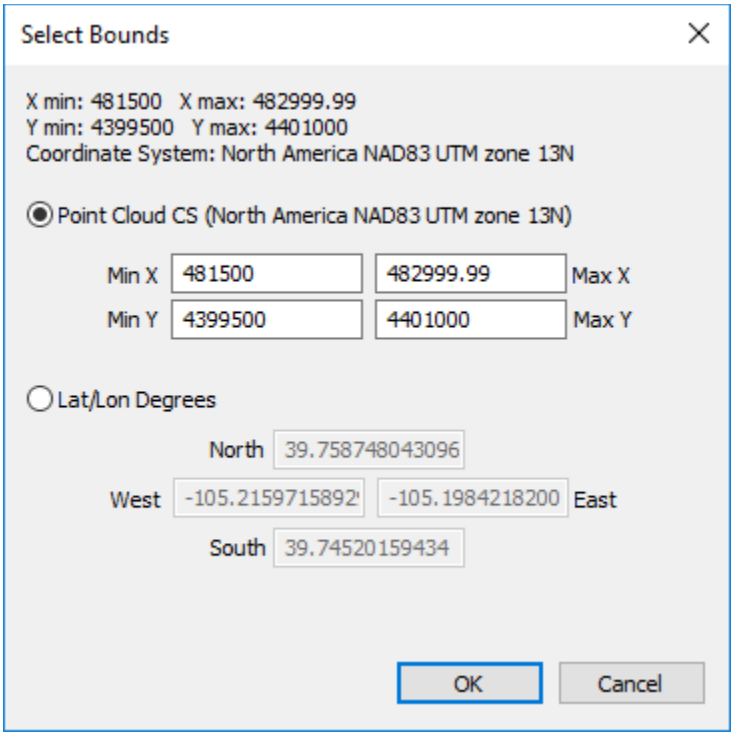
classes. Select individual classes for import by selecting the check boxes next to the desired classes.

Return Type Filter

The *Return type* filter limits the imported points to only the specified return types. 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

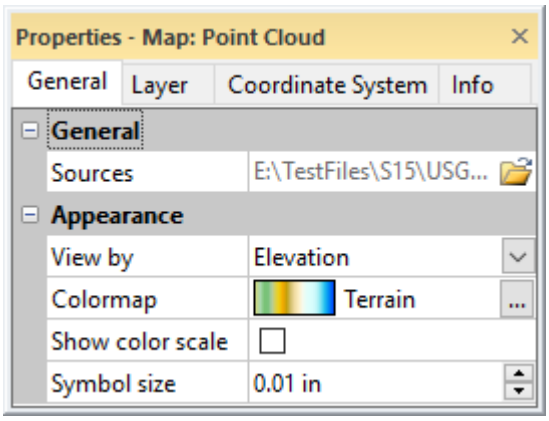
Select *Point Cloud CS* to limit the imported points to, or select the points within, bounds defined in the same coordinate system as the point cloud layer. Type the desired extents in the *Min X*, *Max X*, *Min Y*, and *Max Y* fields.

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.

General Page - Point Cloud Layer

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 dialog.

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 Commands

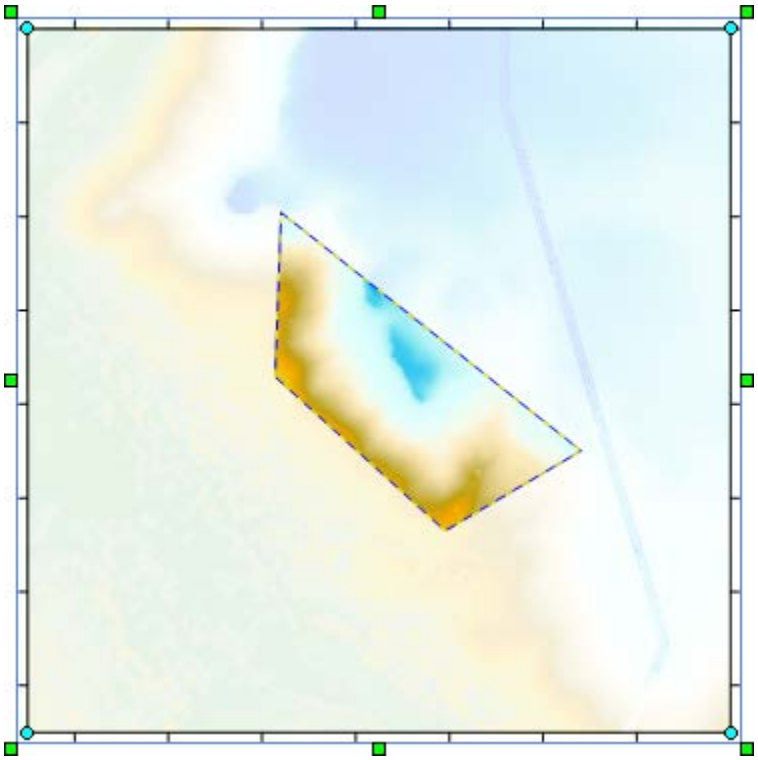
When a point cloud layer is selected, the contextual **Point Cloud** ribbon tab is available and automatically selected. The **Point Cloud** tab contains commands for selecting, classifying, removing, and exporting points in the point cloud.

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

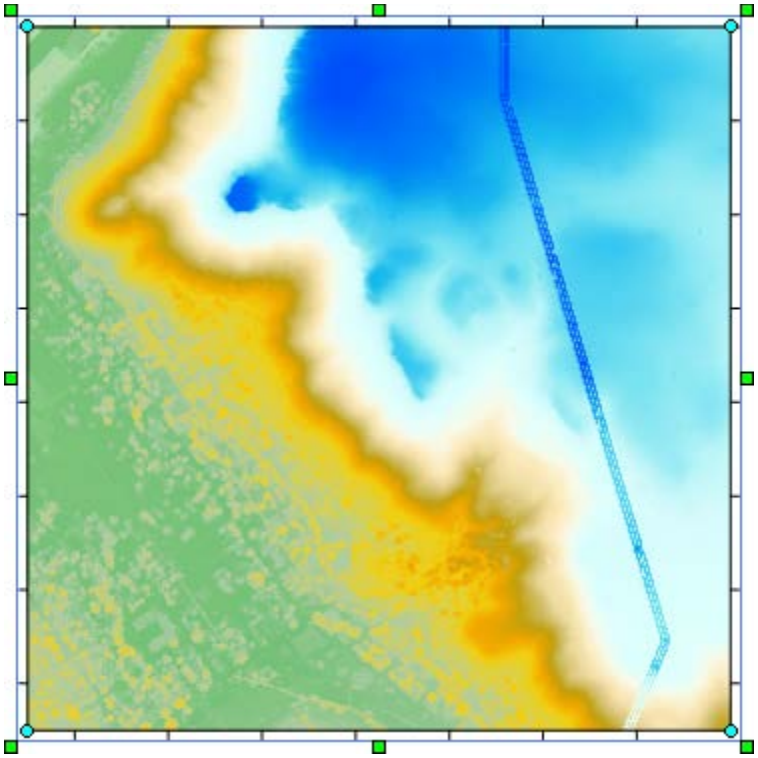
Points in the point cloud layer can be reclassified, removed, and/or exported. First the points must be selected. The Point Cloud ribbon tab includes three commands for selecting points in the point cloud layer: Criteria, Rectangle, and Polygon. When one or more points is selected the selected point(s) 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

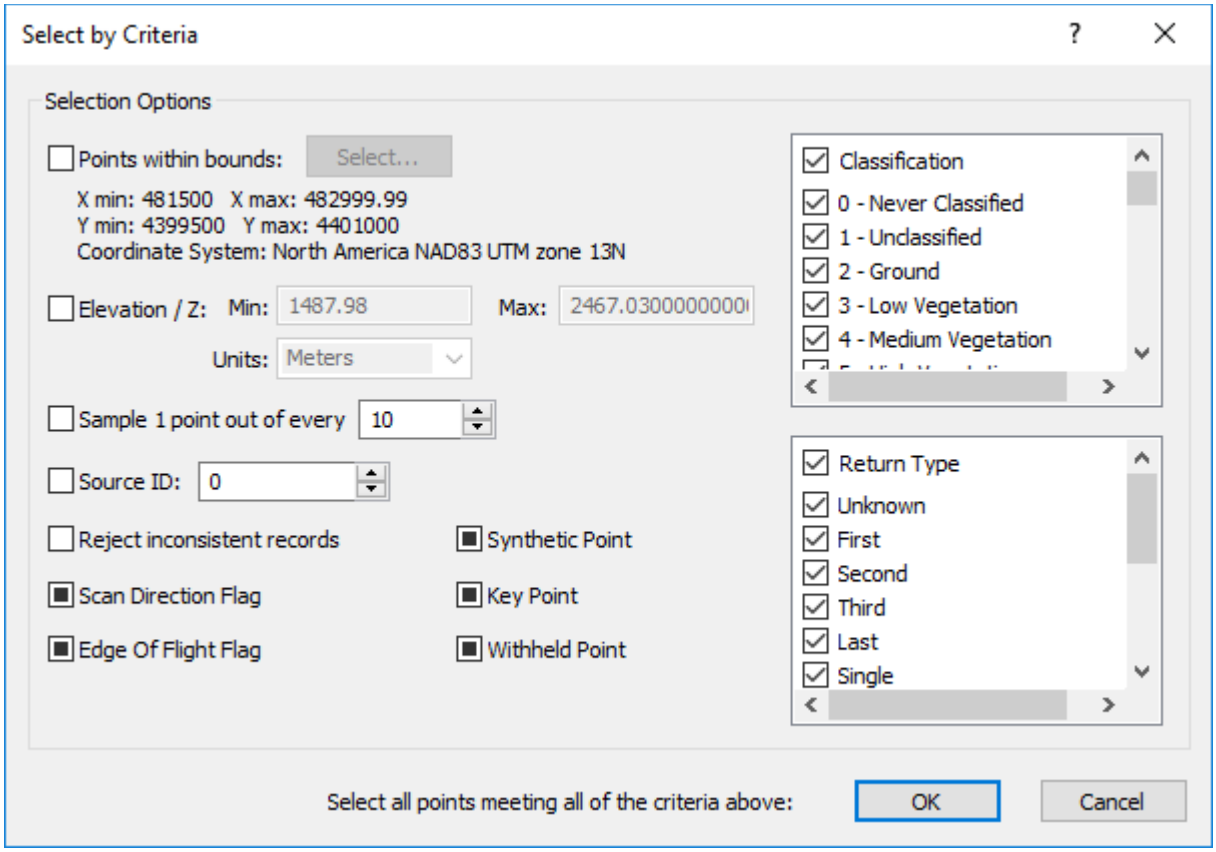
The **Point Cloud | Point Selection | Criteria** command selects points in the point cloud layer based on spatial, random sampling, flag, classification, or return type criteria. Click the **Point Cloud | Point Selection | Criteria** command to select points based on specific criteria. Specify the desired 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 n^{th} 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.

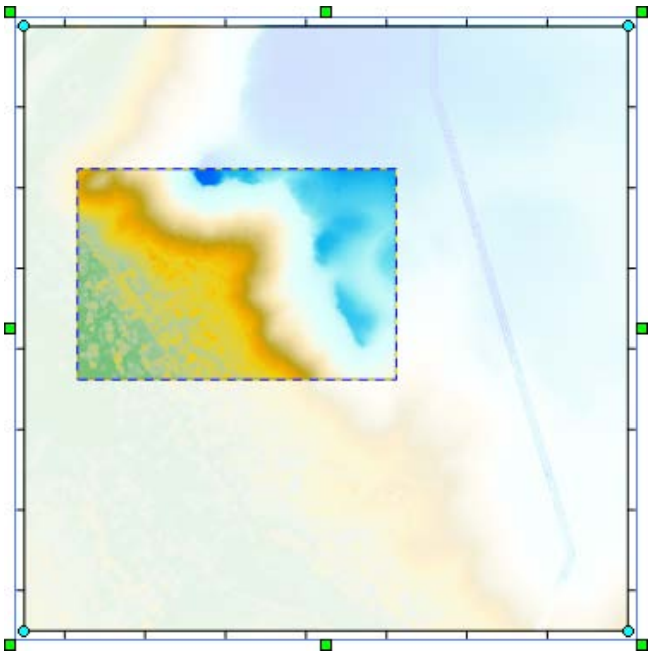
Return Type Filter

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 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.

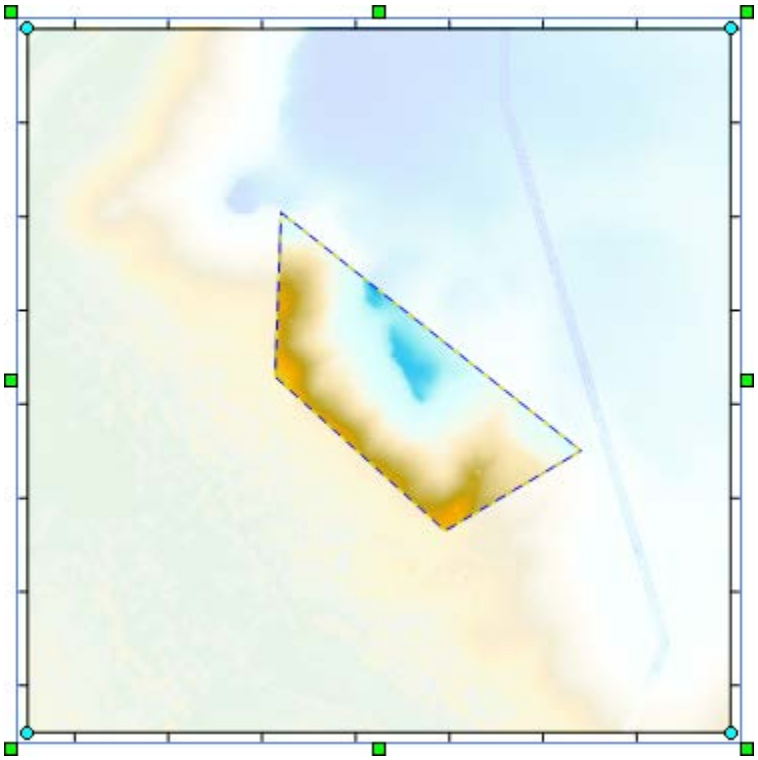
Drawing Tips

- 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 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
- Overlap
- Wire Guard
- Wire Conductor
- Transmission Tower
- Wire Structure
- High Point Noise
- Bathymetric Point
- Water Surface
- Derived Water Surface
- Submerged Object
- IHO S57 Object
- No Bottom Found Bathymetric Point

Remove Classifications

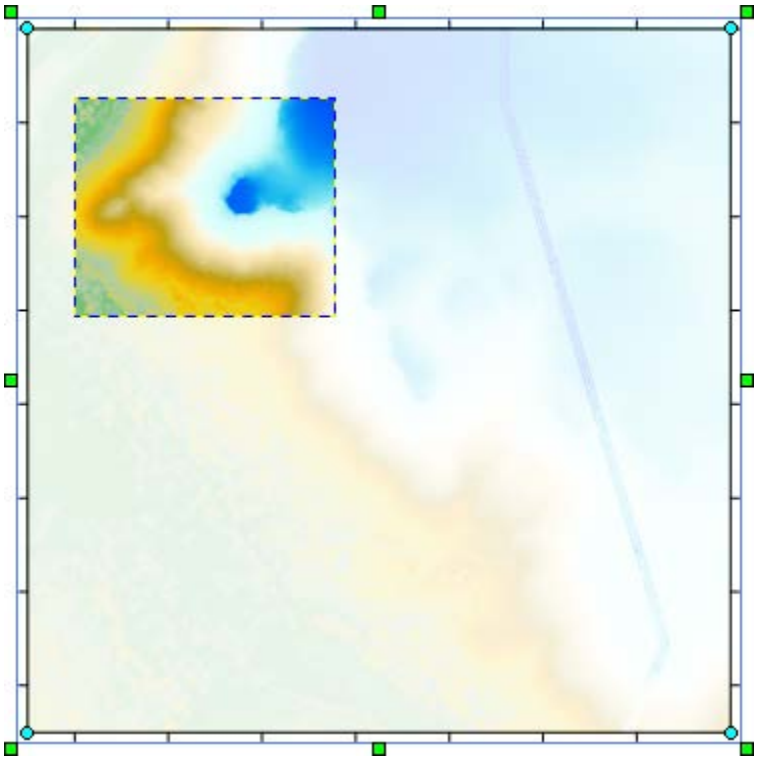
Click the **Point Cloud | Classify | Remove Classifications** command to remove the classifications from the selected points.

Crop to Selection

Click the **Point Cloud | Modify | Crop to Selection** command 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.

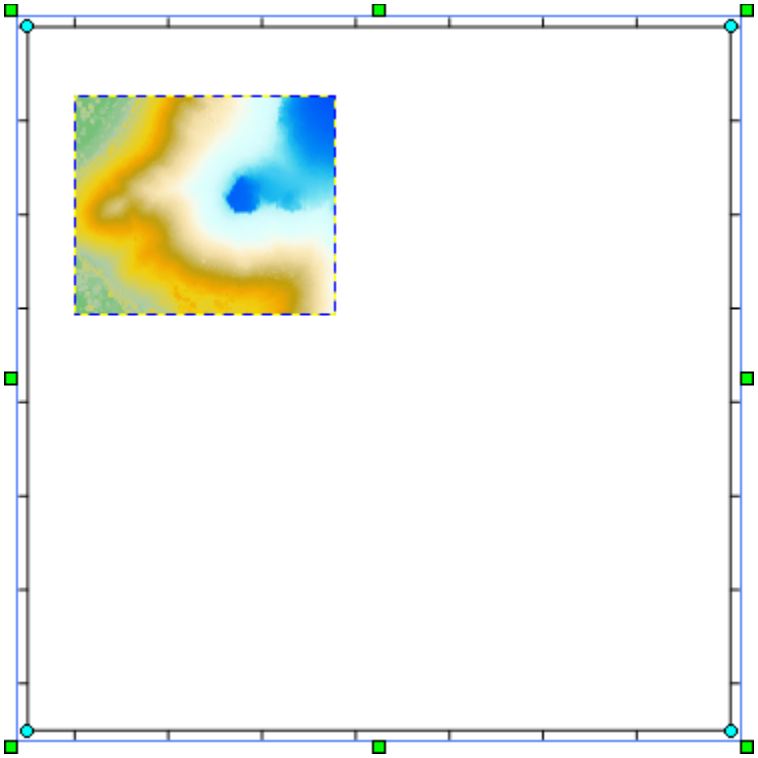
To crop the layer to the selected points,

1. Select points using the Criteria, Rectangle, or **Polygon** commands.



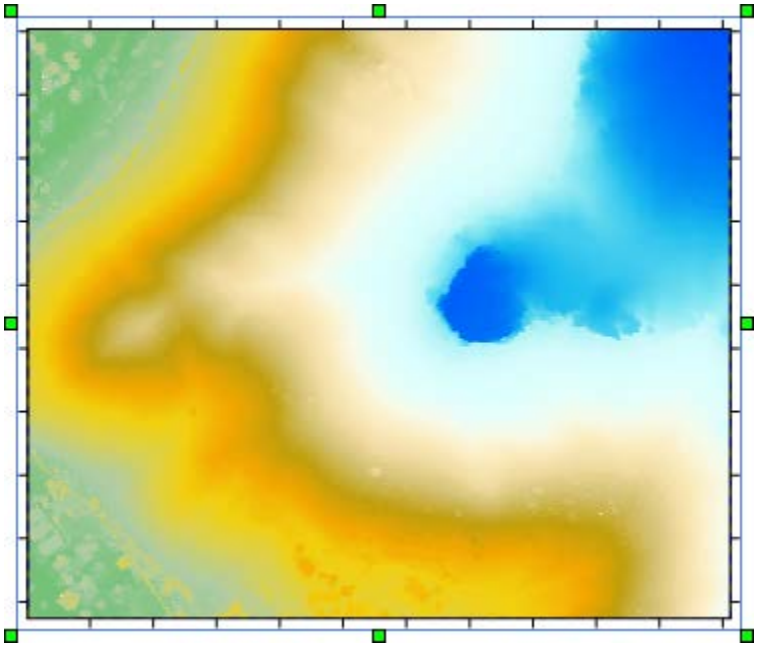
*Select the points you wish to keep when using the **Crop to Selection** command.*

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 to remove the selected points from the layer.

Grid Point Cloud

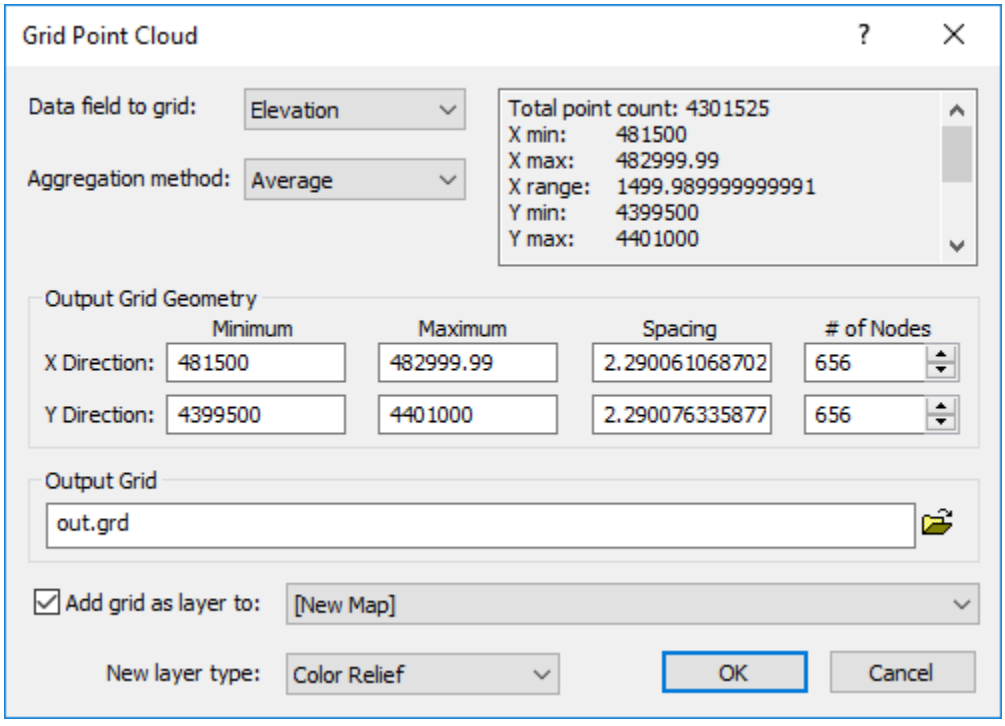
Click **Point Cloud | Features | Create Grid** to create a grid file from the point cloud layer. The **Grid Point Cloud** dialog is displayed when the **Create Grid** command is clicked.

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

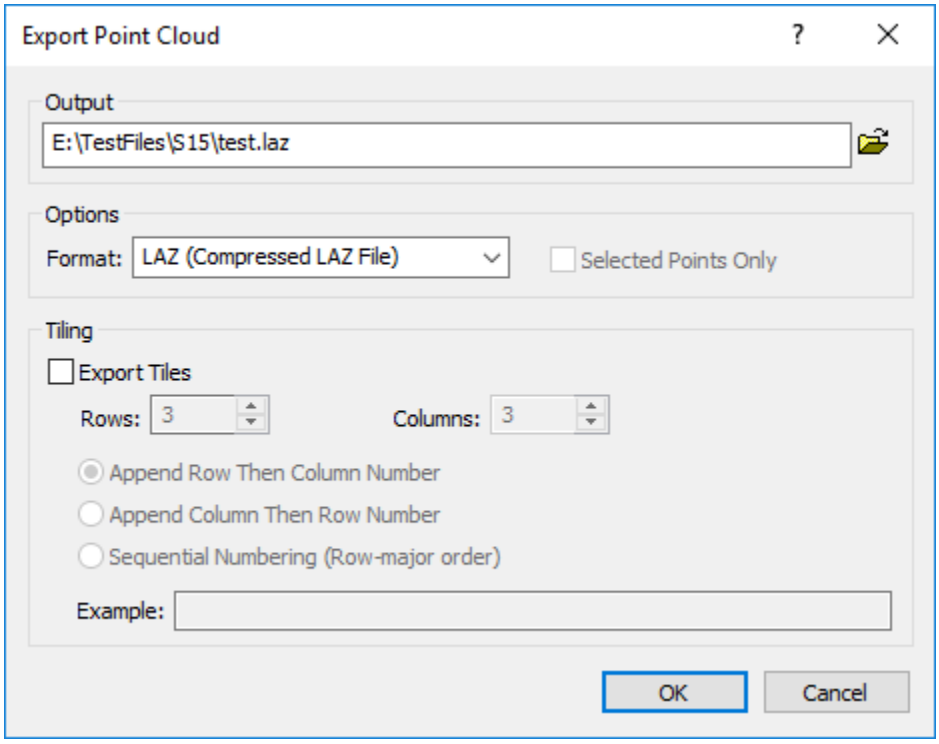
Check the *Add grid as layer to* check box to automatically add the created grid to a new or existing map. Select *[New Map]* in the *Add grid as layer to* field to create a new map. Click the current selection and select an existing map to add a new layer to the map. Select the layer type by clicking 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** 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

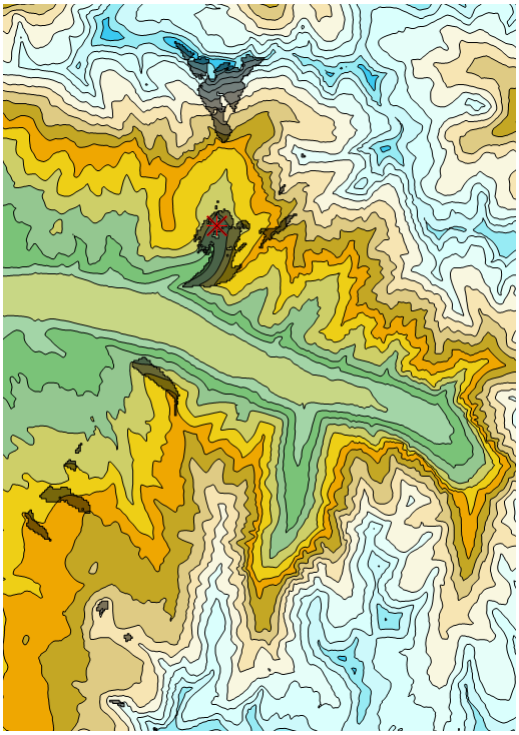
The table above demonstrates the indexing for the tile file names.

Example

The *Example* shows an example of the tile file names for the first three tiles in row 0.

Chapter 19 - Viewshed Layers

The **Home | Add to Map | Layer | Viewshed** 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, shaded 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

Map Properties

The map properties contains the following pages:

View

Scale

Limits

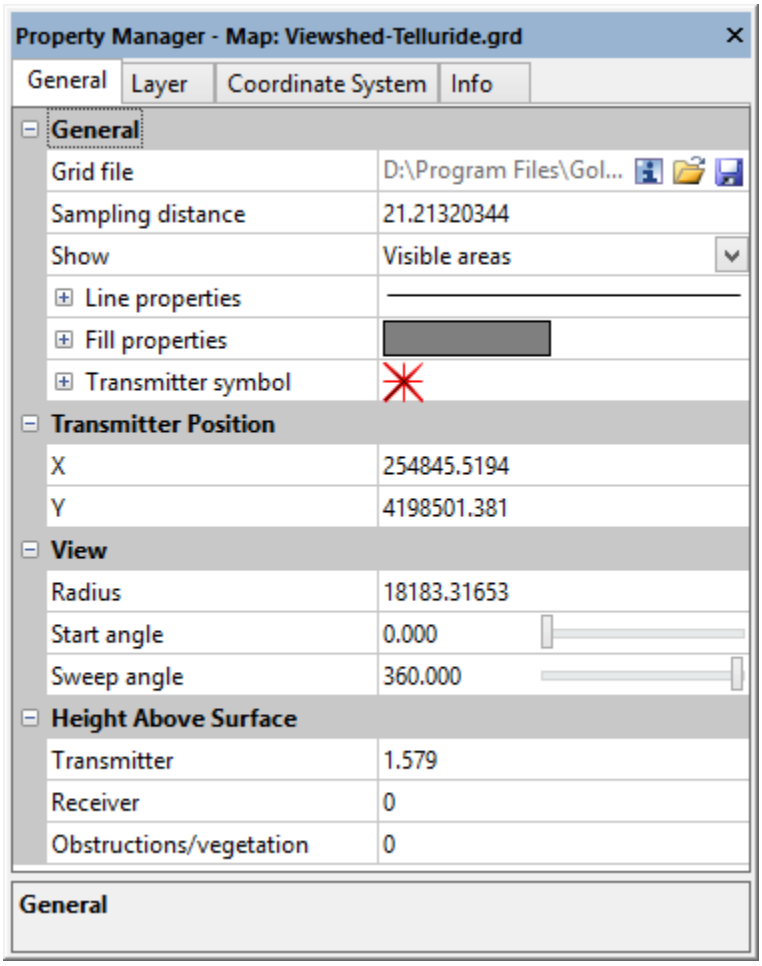
Frame

Coordinate System

Info

General Page - Viewshed

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.

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 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

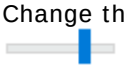
By default the *Radius* is equal to the diagonal of the grid extents, so the entire grid is included in the viewshed analysis. Change the viewshed *Radius* by clicking the current value in the *Radius* field and typing a new value. The viewshed radius is in the linear distance units specified by the 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 20 - Downloading Layers from a Server

Click the **Home | New Map | Base | Base from Server** command, **Home | Add to Map | Layer | Base from Server**, 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 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 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 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. **Surfer** comes with one WCS server by default, under the *Terrain Data* category, called *SRTM Global 90m (GMU)*.
- 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.

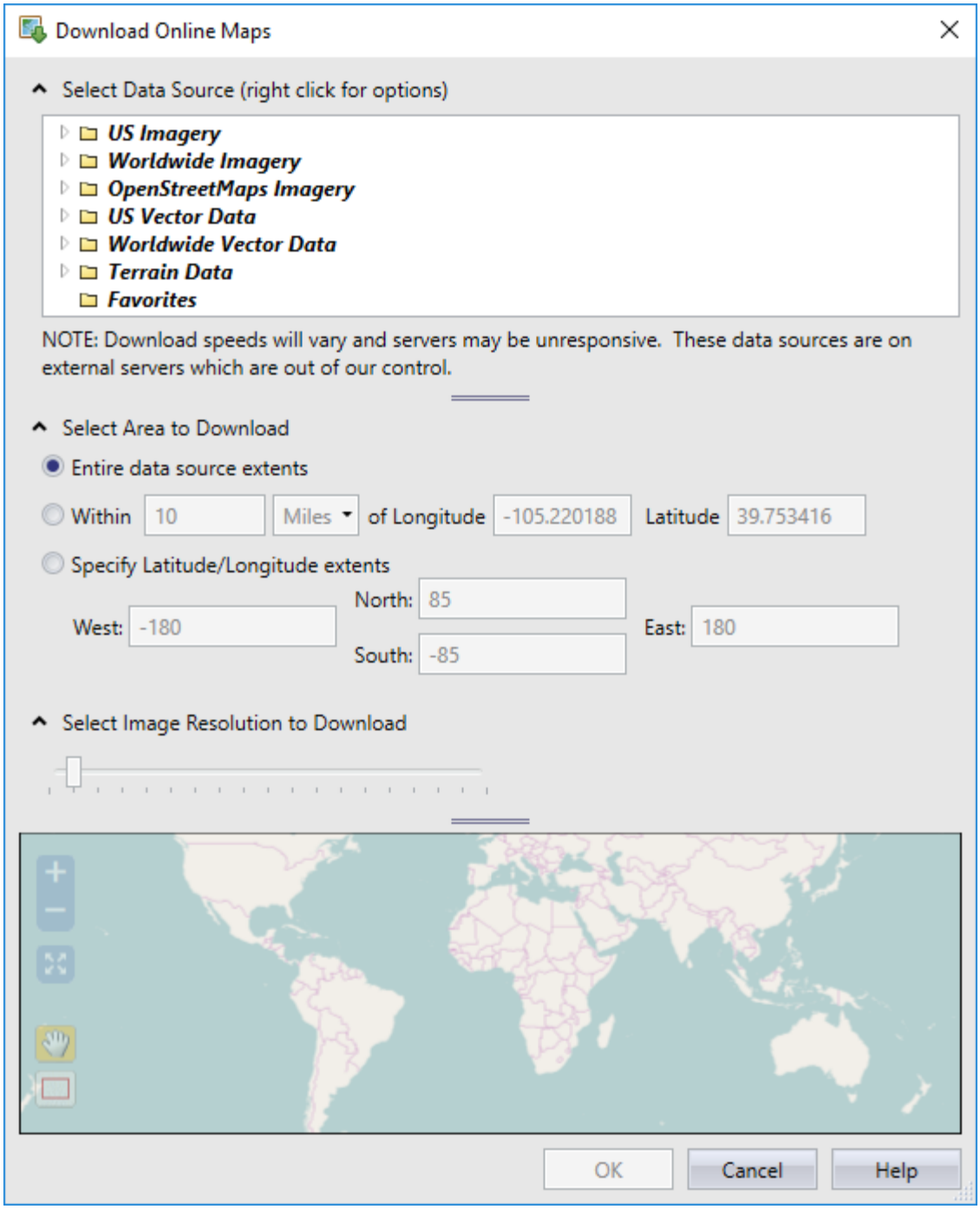
Click the **Grids | New Grid | Grid from Server** or the *Download* button in the Open Grid dialog to open the **Download Online Grids** dialog.

This dialog contains a server list and download options for downloading data from a web mapping server (WMS), Open Street Map server (OSM), web coverage server (WCS), or web features server (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 in the *Terrain Data* category. 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

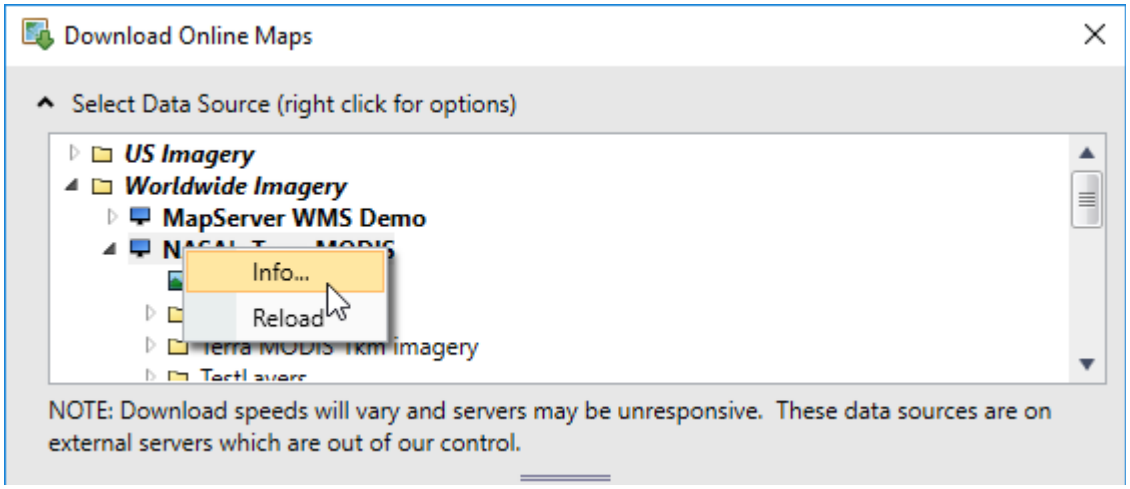
The *Select Data Source* list contains servers and layers, separated in categories, from which map data or images can be downloaded. The categories are *US Imagery, Worldwide Imagery*, *OpenStreetMaps Imagery*, *US Vector Data*, *Worldwide Vector Data*, *Terrain Data*, and *Favorites*. These categories 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 **Surfer** 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 **Surfer 15**, 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. For example, if the Geode.xml file from **Surfer 15** is shared with **Surfer 13**, only WMS servers will be accessible in **Surfer 13**.

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.

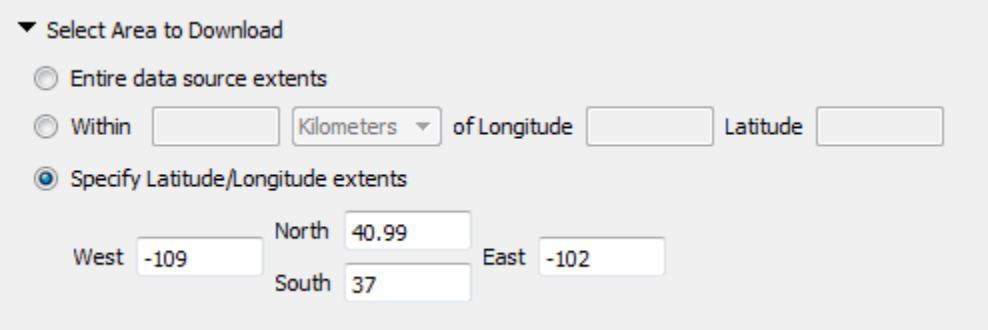
To view information about the server, right-click on the server name and click **Info**. The View Map Source dialog opens and displays the server and layer information. The *Name*, *Type*, and *URL* cannot be edited in the **View Map Source** dialog.

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.

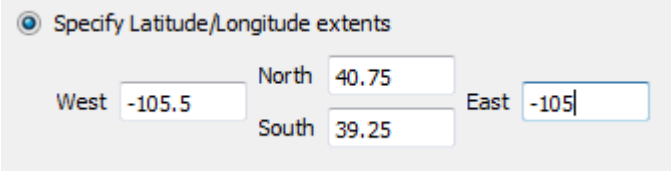
The *Within <distance> <units> of Longitude <Lon> Latitude <Lat>* option allows a single longitude and latitude value to be entered. A square area is downloaded based on the *distance* value and the *units* option. The square is centered on the value entered for the *Longitude* and *Latitude*. To use 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.

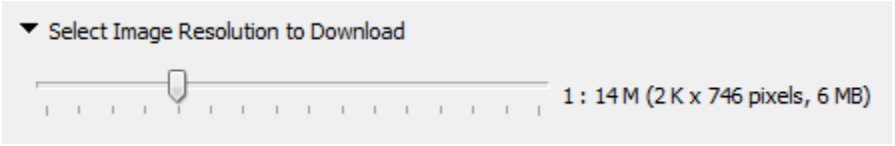
The selected WMS server may have an original data source resolution (for example, 1 meter/pixel) for its imagery. You may be able to select an image resolution higher or lower than this original data source resolution. Increasing the image resolution here cannot give you better image detail over the 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.

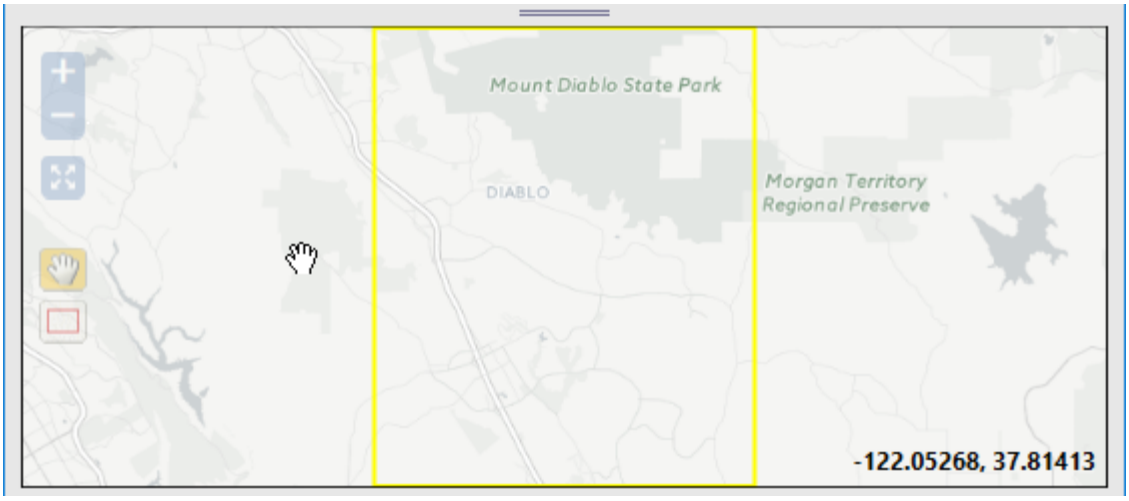


Set the image quality by dragging the slider to the right or left.

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

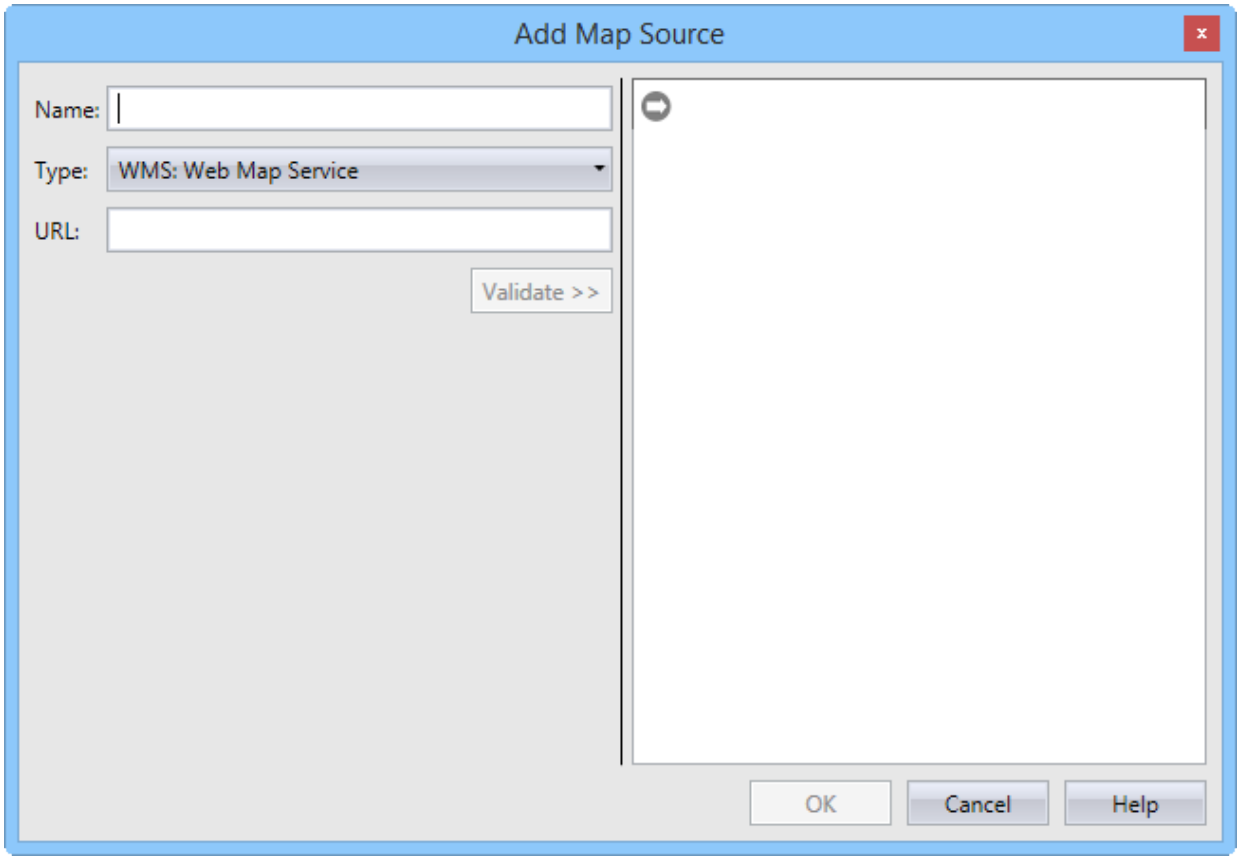
The **Download Online Maps** and **Download Online Grids** dialogs can be resized by clicking and dragging a side or corner, as with normal windows. The image preview and *Select Data Source* sections can also be enlarged or reduced by clicking and dragging the double horizontal bars between the sections 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

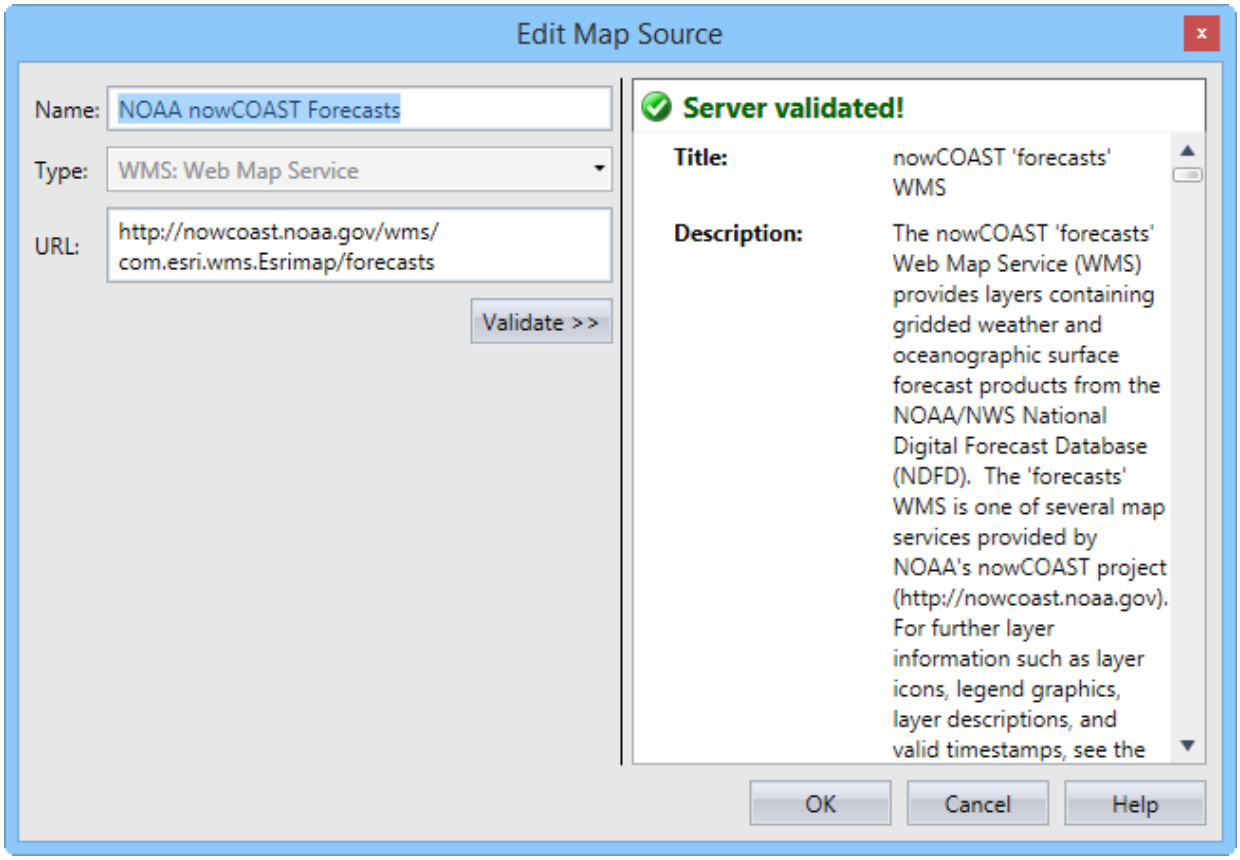
Right-click on an existing category in the Download Online Maps dialog, and then click *Add Map Source* to open the **Add Map Source** dialog. This dialog adds new web servers to the list in the **Download Online Maps** dialog.



*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.

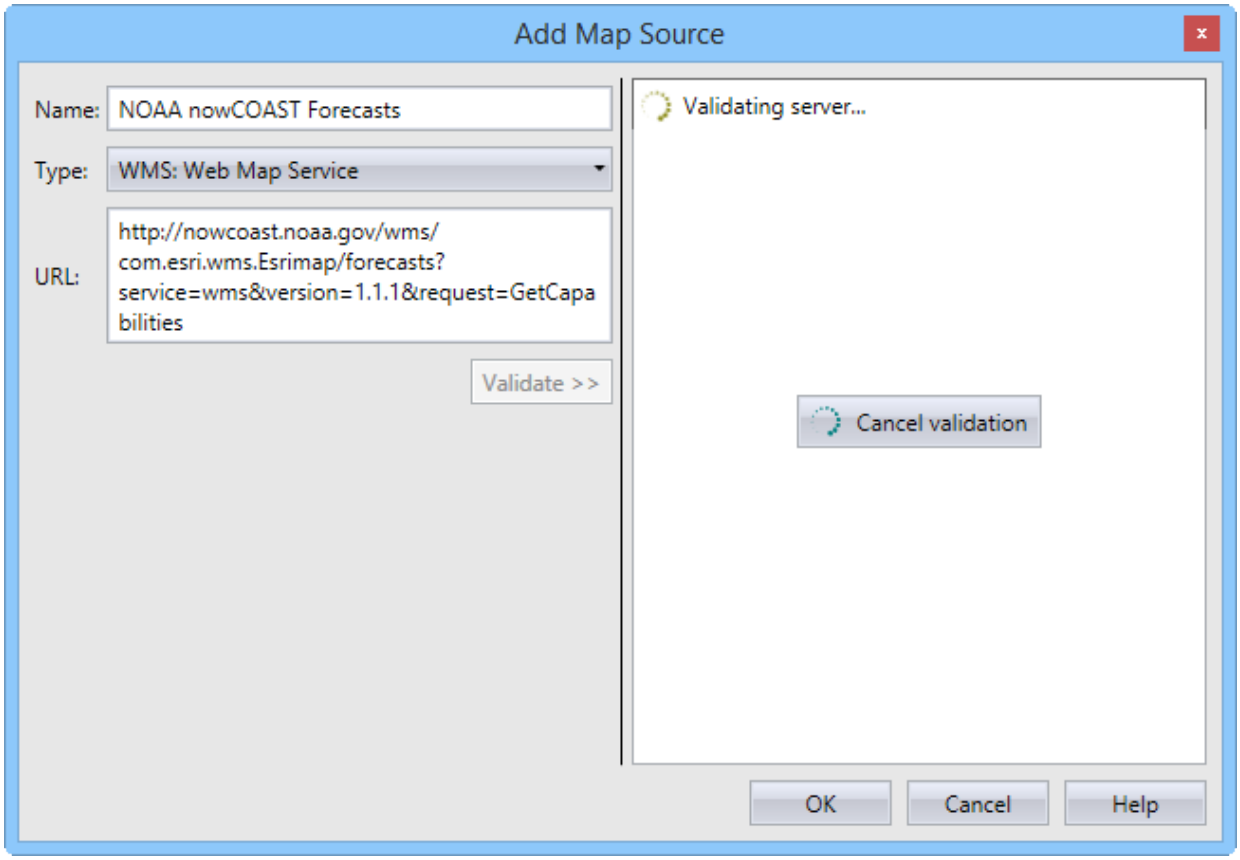
Surfer supports WCS protocols 2.0.1, 2.0.0, 1.1.3, 1.1.2, 1.1.1, 1.1.0, and 1.0.0. **Surfer** supports WFS protocols 1.0.0, 1.1.0, and 2.0.0.

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 21 - 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.

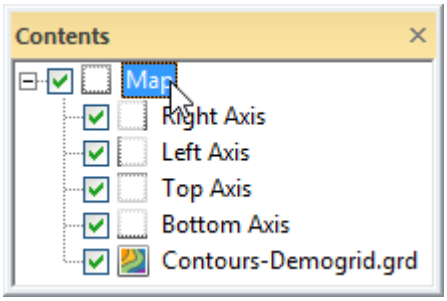
Info

The Info page provides additional information about the map. Comments can be added, deleted, and edited to clarify information contained in the map.

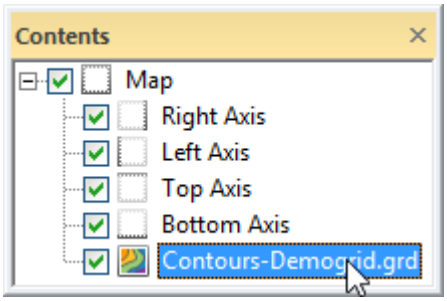
Map Properties

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.

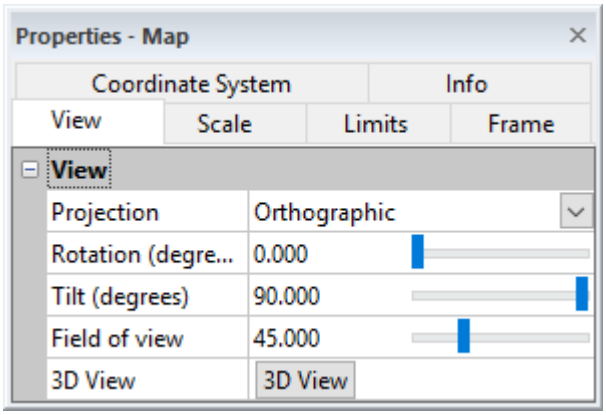


Click on the Map object to open the map properties.



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.

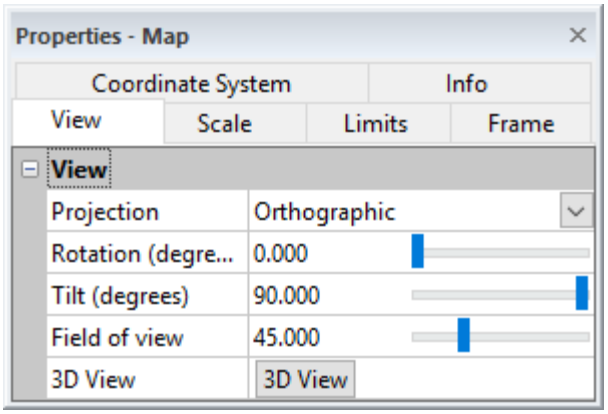


*Specify the View, Scale, Limits, Frame, Coordinate System, and Info in the **Properties** window when a map is selected.*

View

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.

The **View** page is located in the map properties. To open the map properties, click on the *Map* object in the Contents window**.** The map properties are displayed in the Properties 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

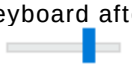
The *Rotation (degrees)* value controls the rotation of the map about the center. As the *Rotation (degrees)* value increases, the map rotates in a counterclockwise direction.

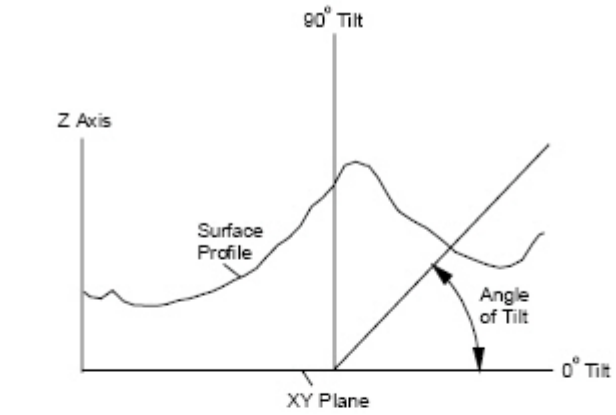
To change the *Rotation (degrees),* click in the box next to *Rotation (degrees)*. Highlight the existing value and type a new value. Values are always displayed with three digits after the decimal. Press ENTER on the keyboard after typing a new value to show the value on the map.

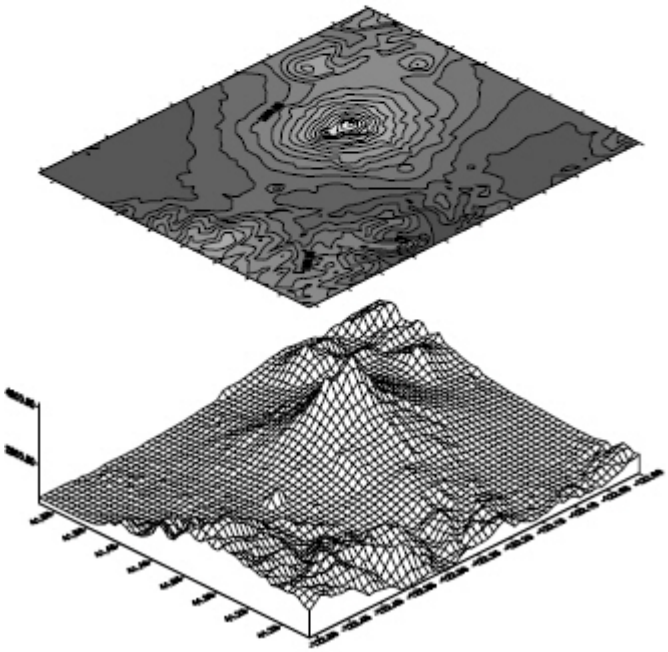
Alternatively, click and drag the to a new value. Values must range between zero and 360.000.

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.

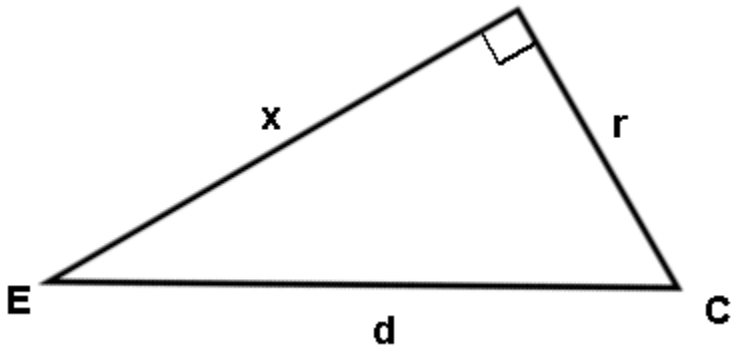




Maps can be displayed at any rotation angle and at any tilt angle from zero to 90 degrees. When more than one map is selected, the view parameters are applied to all selected maps.

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).

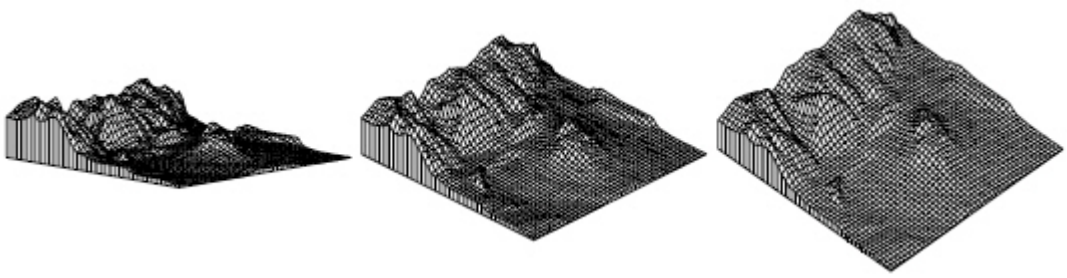
With an *Orthographic* projection the *Field of View* values have no effect. Note that the *Field of View* parameter does not influence the map scale.

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.

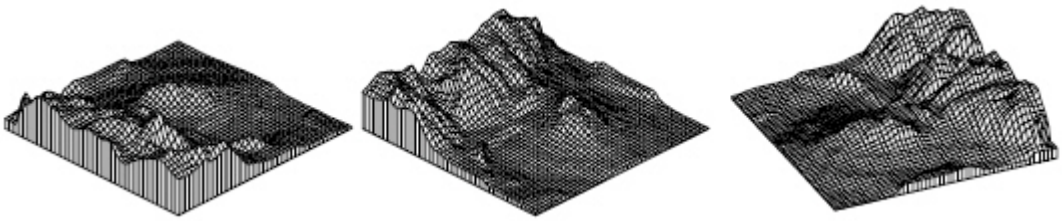
3D View

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.

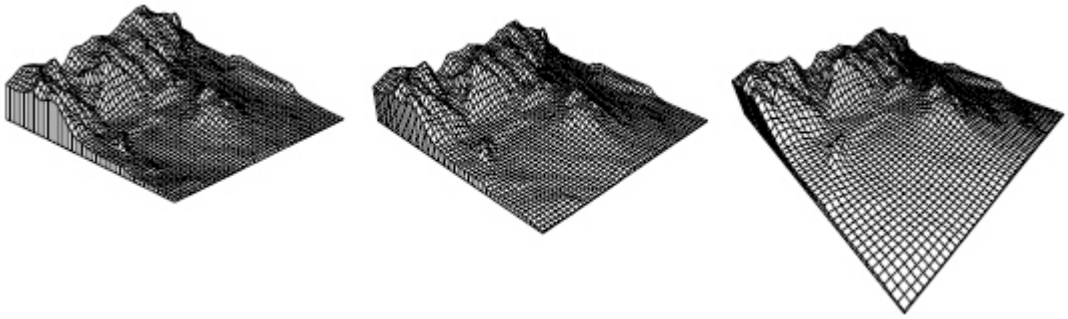
Example of Tilt, Rotation, and Projection Parameters



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.

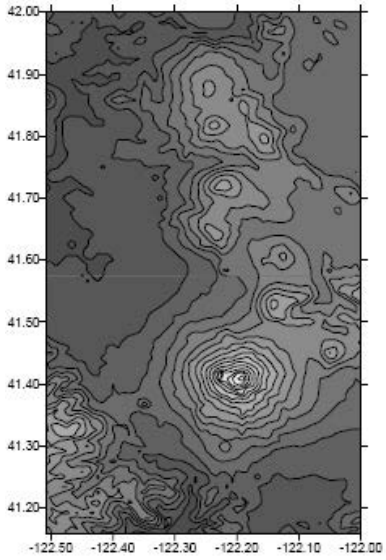
Scale

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 map properties. 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.

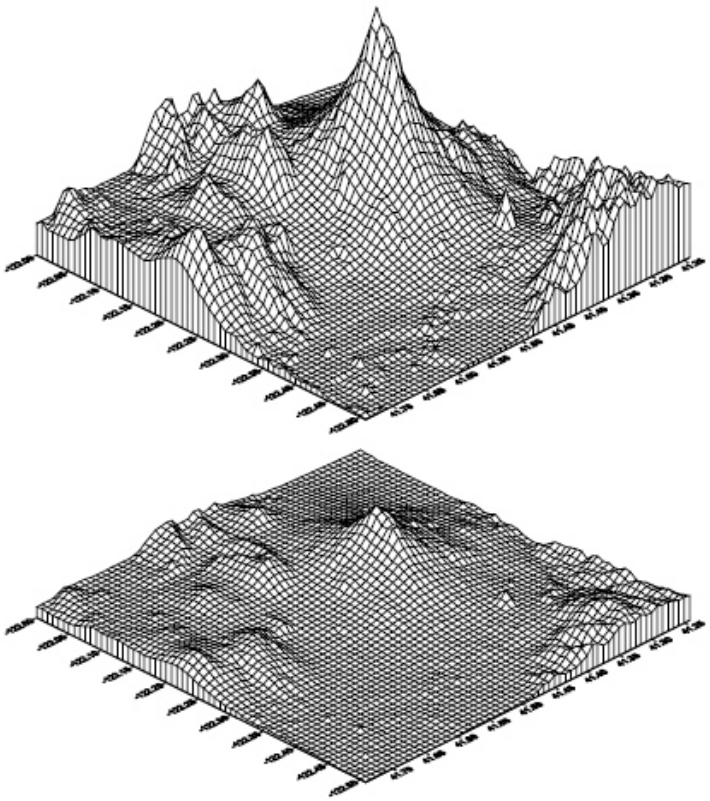
Axes are part of the map on which they are drawn. Therefore, when the scale is redefined, the axes are modified accordingly. Because **Surfer** calculates reasonable tick spacing based on the map lengths and map limits, redefining the scales can result in different tick spacing along the axis.

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.



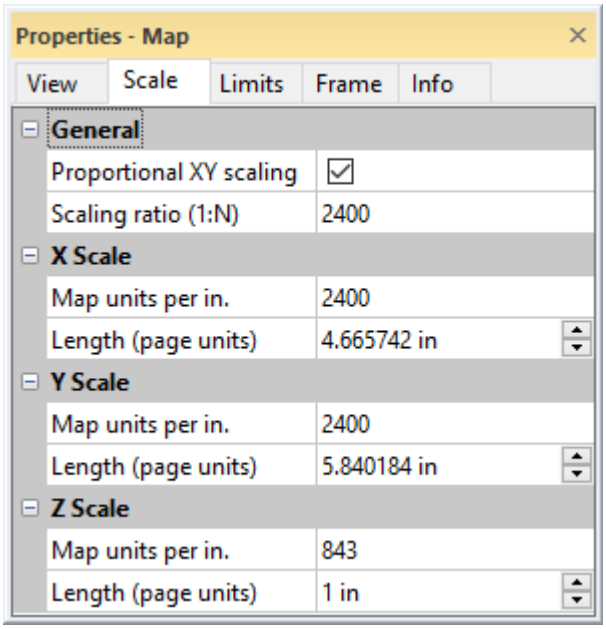
*These are two 3D wireframe maps of the same grid, each using a different Z scale. The Z scale is exaggerated in the top view. Use the **Scale** page to change the length of the Z axis.*

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 does not rescale the map internally, it just resizes the map on the screen. 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:

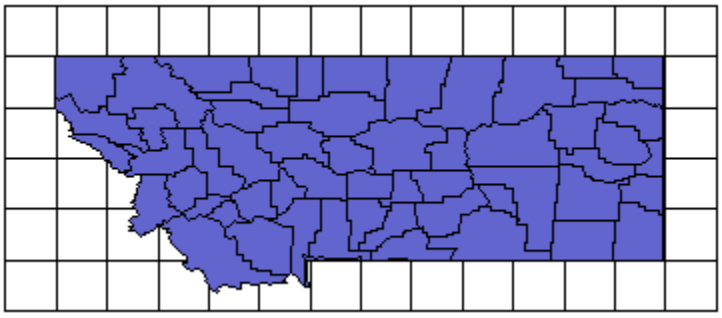
1. Determine the latitude for the parallel through the center of the map, and obtain the cosine for this latitude value. The center parallel in ca2010.gsb is approximately 37.27 degrees. The cosine of 37.27 degrees is 0.80.
2. Click on the *Map* in the Contents window to select it. The map properties 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.

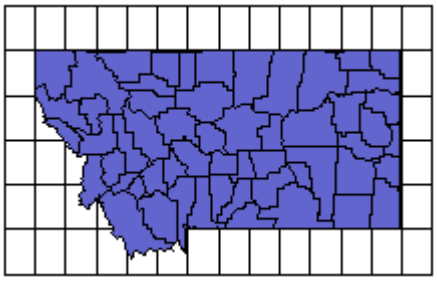
Example

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).

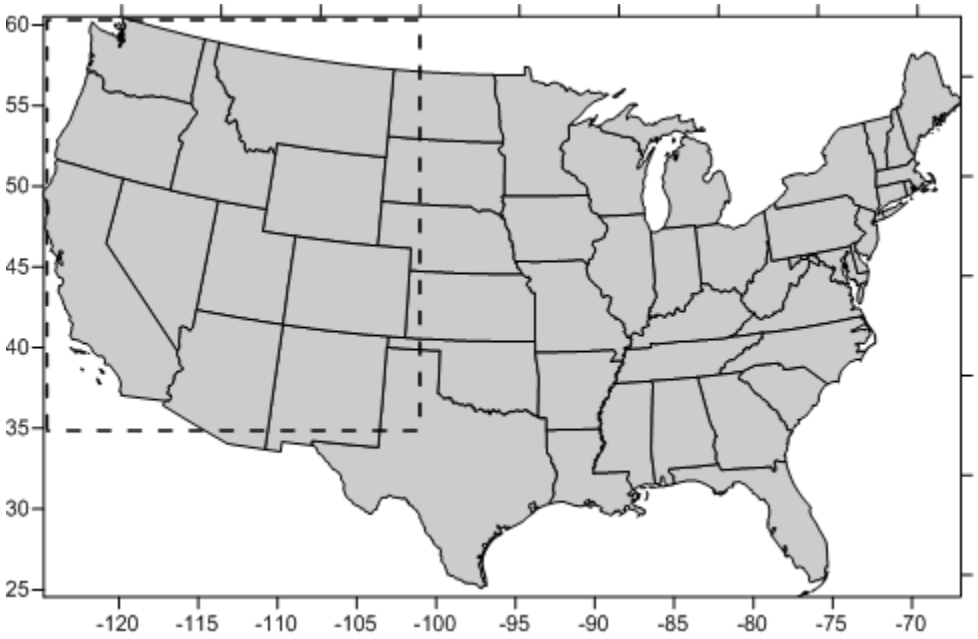


When you apply scaling factors, map distances are represented more accurately. The rectangular spacing of graticule lines indicate the difference in scaling.

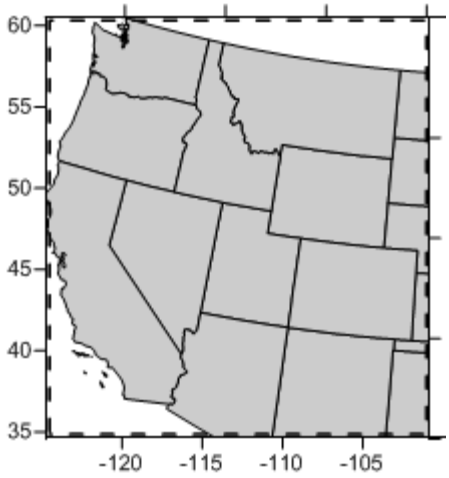
Limits Page

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.



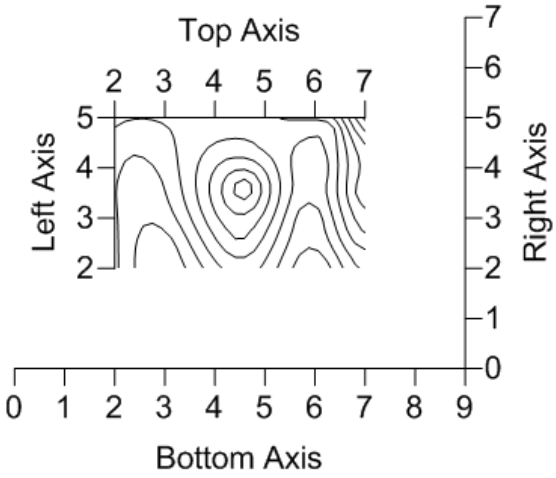
The map limits of the map have been adjusted to the limits of the area of interest.

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

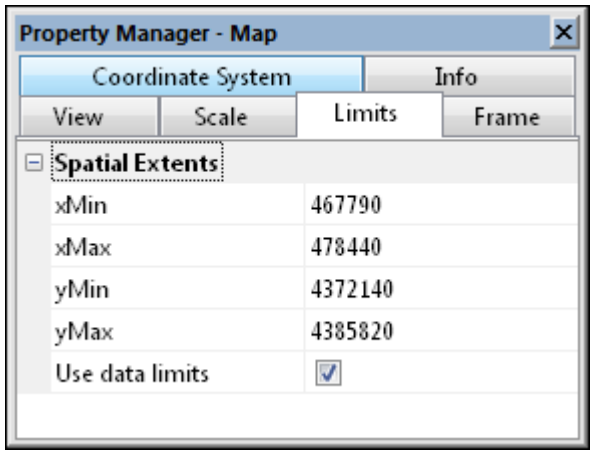
When an axis is reversed on the Scaling tab, the limits do not change. The xMin value must still be less than the xMax value. The reverse axis option only changes the display of the axes and map layers, not the actual limits.

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

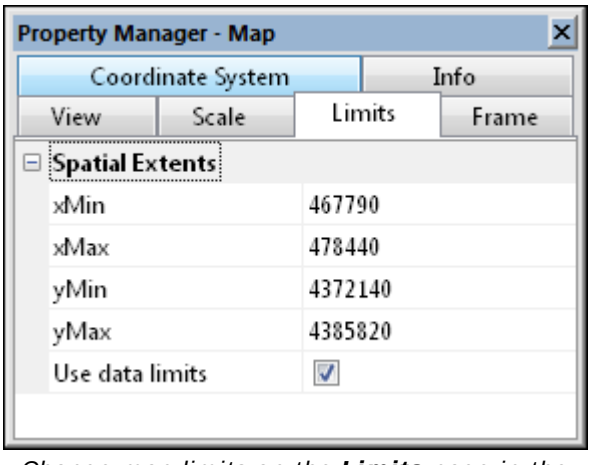
The *Use data limits* check box resets the map limits to the limits of the grid file, base map, or post map data file limits.

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.

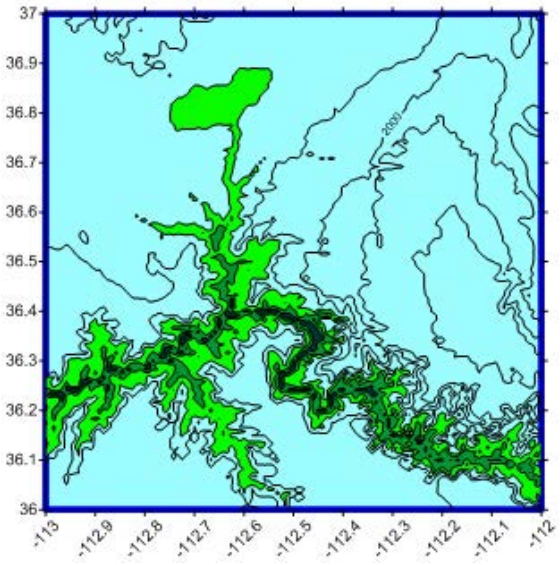


*Change map limits on the **Limits** page in the **Properties** window.*

Frame

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.

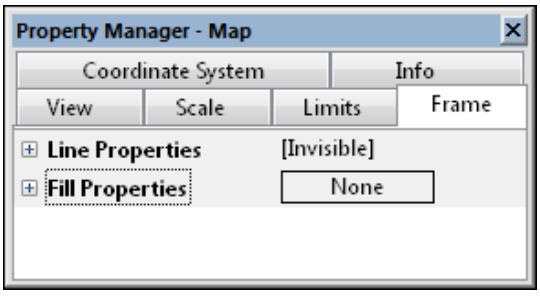
Map backgrounds lie underneath all other objects on the map. This means that the background can be obscured by other map features. For example, when you create color filled contour maps, the fill between contours obscures the map background wherever a solid or image pattern is used for the contour fill. 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 Page

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 Map Data

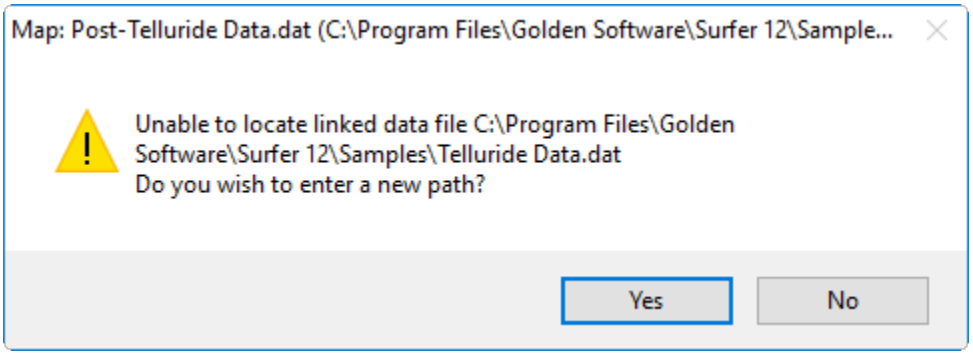
You can reload external map data for all maps in the currently loaded **Surfer** file with the **File | Reload Map Data** command.

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 also be reloaded for individual map layers directly from the Properties window by changing or reloading the file. Click the 📂 button to change or reload the file used to create the map. For example, to update the grid file used to create a contour map, click on the contour map to select it. In the **Properties** window, click on the General tab and change the file used in the *Grid file* box.

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 22 - Map Features

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, Shaded 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.

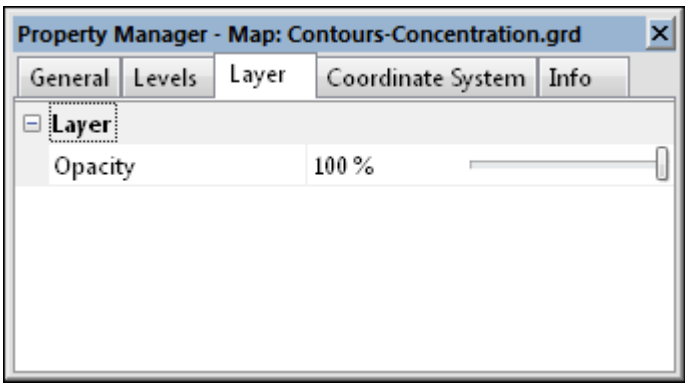
Map Limits

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 Page

The **Layer** page in the Properties window for a base, post, classed post, grid values, contour, color relief, shaded relief, vector, 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). To change the opacity, highlight the existing number and type a new number. Press ENTER on the keyboard or click on another item in the **Properties** window for the value to be accepted. 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

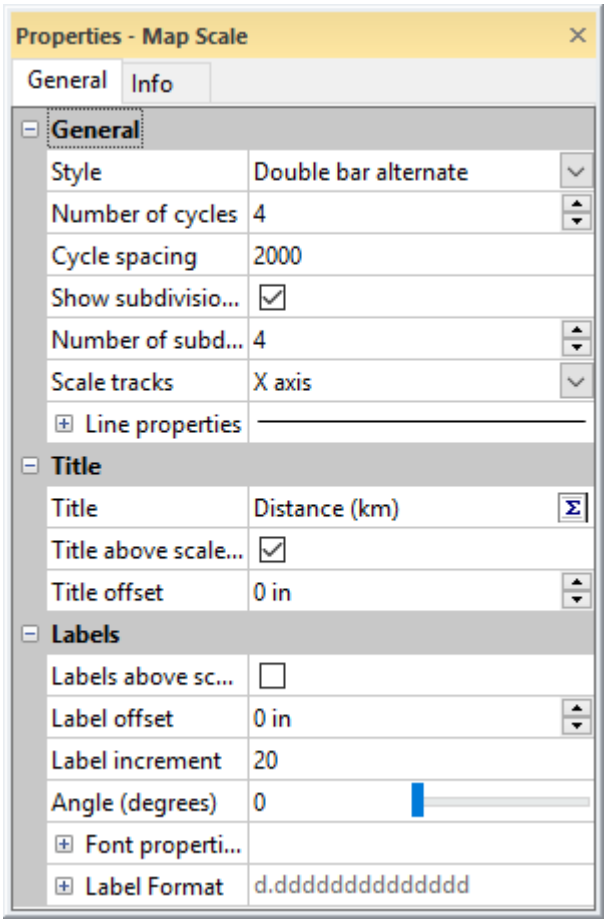
The **Map Tools | Add to Map | Scale** command places a scale bar on the map. Alternatively, right-click on an existing map and select **Add to Map | Scale Bar** to add a scale bar. Scale bars are divided into equally spaced cycles. Any number of cycles between one and 100 can be used. Each cycle is 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

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.



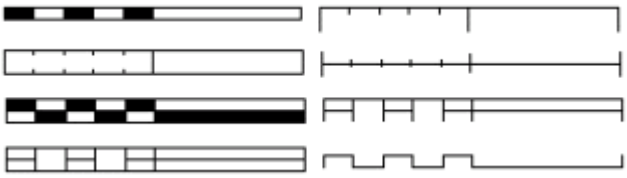
*The scale bar properties are set in the **Properties** window.*

General

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.

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.

Title

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*.

Labels

The *Labels* section contains properties for the scale par 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*.

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.

Legends

Click the **Map Tools | Add to Map | Legend** command 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

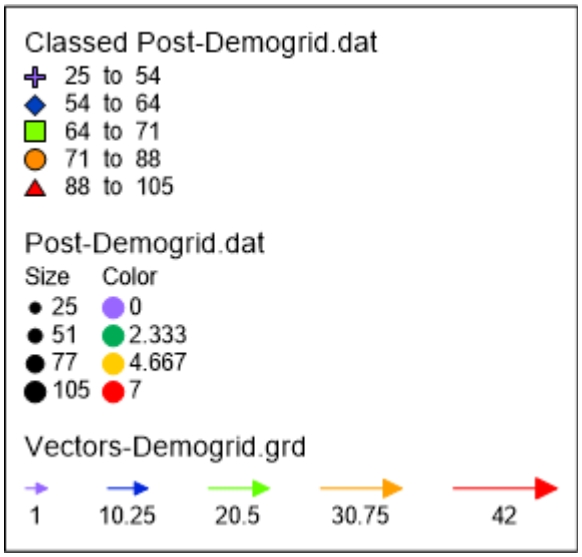
When a base layer includes a symbology, a legend can be displayed for the base layer. Unique values legends can include the value, fill, line, and symbol samples, and the legend order is determined by the order in the **Symbology** dialog. Unclassed colors legends can include fill value and fill samples. 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.

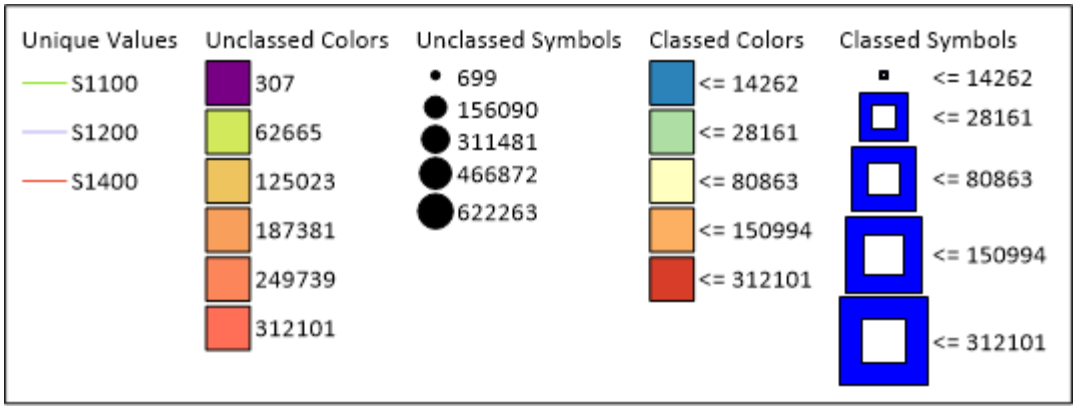
Post Layer Legends

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:

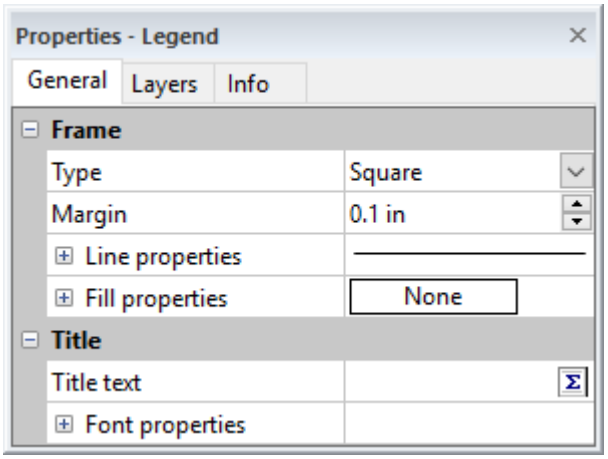
General

Layers

Info

General Page

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.

Type

The *Type* list specifies the type of border to use for the legend. You can select *None*, *Square*, or *Rounded* 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 legend at the *Margin* distance. Setting *Type* to *Rounded* creates a rounded rectangle at the edge of the legend 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. 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 the *Style*, *Color, Opacity*, and *Width* of the line to use for the legend 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 legend area.

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.

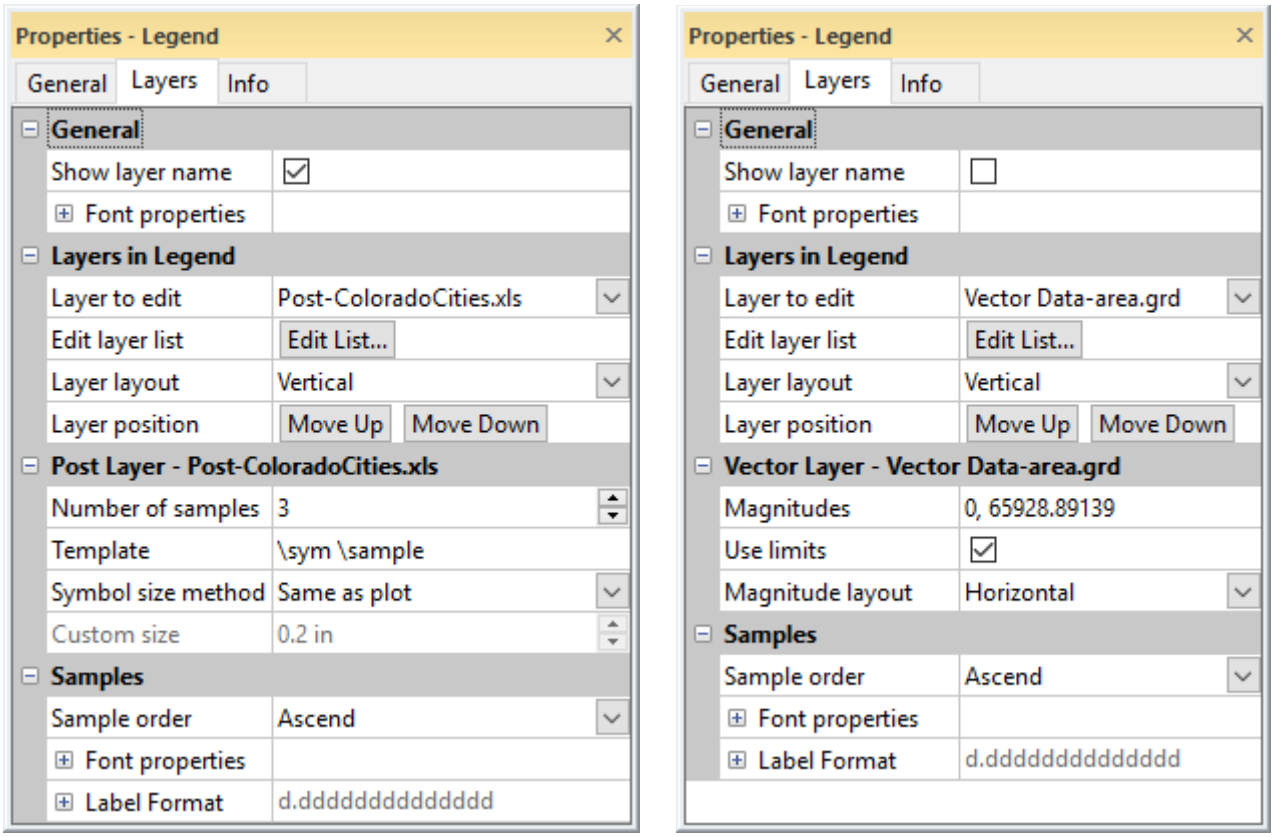
Titles can contain multiple lines. To create a multiple line title, click the button. The Text Editor appears. Type the text in the dialog. To add a new line of text, press the ENTER key on the keyboard. To change the properties of the text, highlight the text and change the desired property. When finished, 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 Page

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.*

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.

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-Demogrid.dat*. 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.

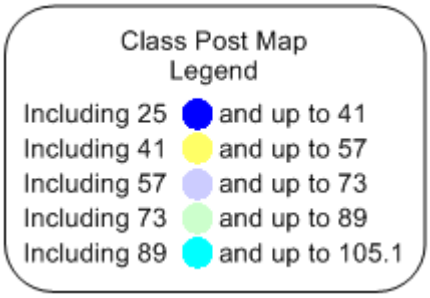
Template

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
\fill			fill sample	color sample		color sample	
\line			line sample				
\lower		lower class limit					
\name		class name				class name	class name
\symbol	symbol sample	symbol sample	symbol sample		symbol sample		symbol sample
\tab	0.25 inch tab character						
\upper		upper class limit					
\value	sample value		sample value	sample value	sample value	"<=" upper class value	"<=" upper class value

Any number of directives can be included in any order.

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.

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

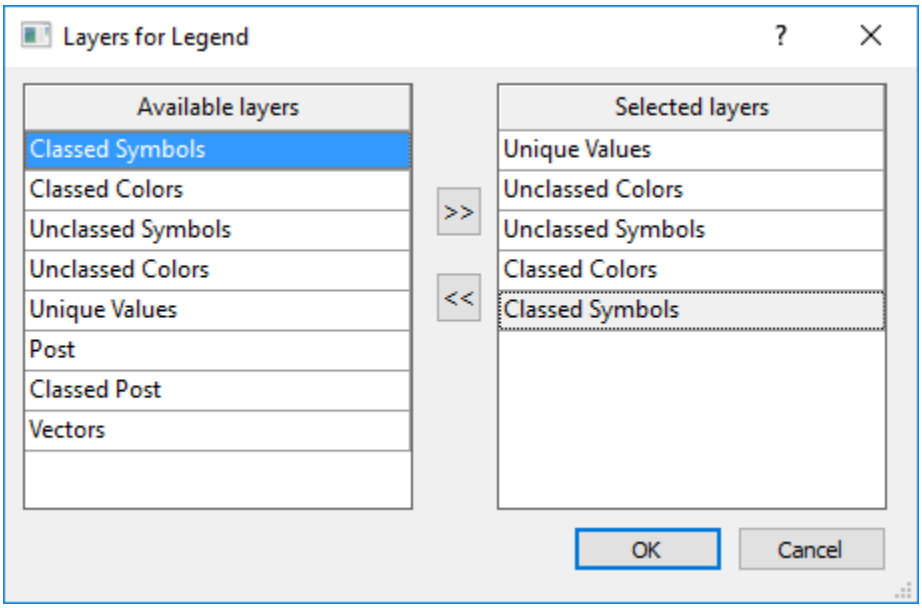
Control the sample font properties in the *Font properties* section. The font properties apply to the samples in all layers in the legend.

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.

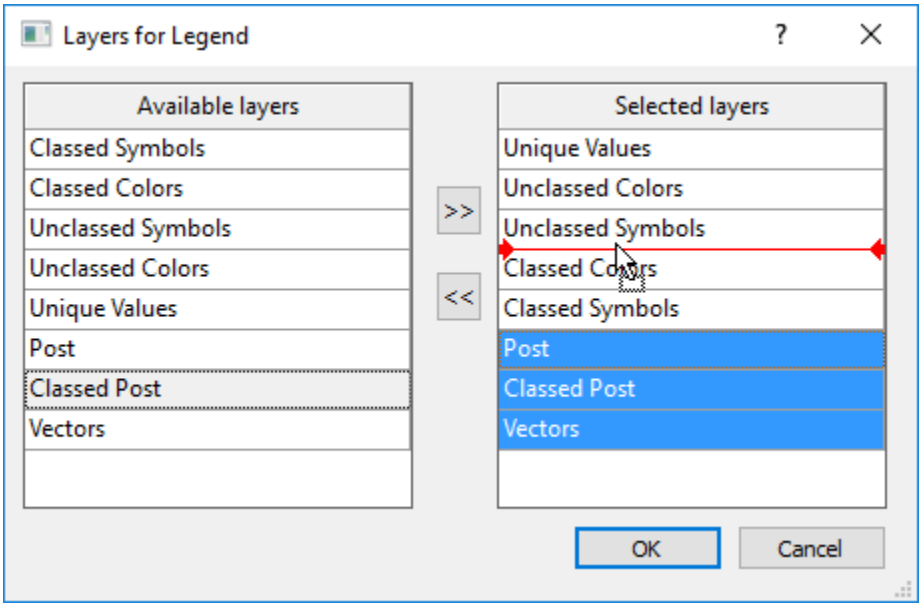
The first, last, and all layers between are selected. Deselect a layer by selecting another layer, or by holding CTRL and clicking the selected layer.

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. Click on the color scale bar to open the color scale properties in the Properties window. Color scale bars for contour and wireframe 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 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:

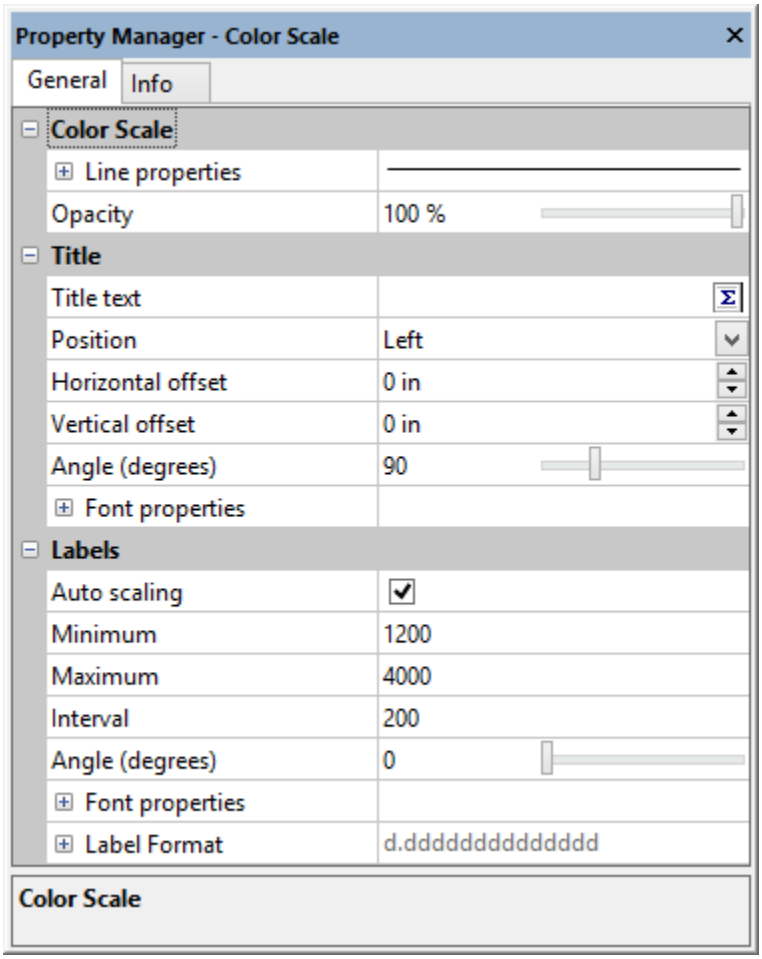
1. Click on the vector map layer to select it.
2. In the **Properties** window, click on the **Symbol** tab.
3. Click the ⊞ next to *Color Scaling* to open the *Color Scaling* section.
4. Set the Scaling method to By magnitude or By grid file.
5. Click the ... next to the *Vector colors* colormap.
6. 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.



Change color scale display properties for vector, color relief, 3D surface, and point cloud maps in the ***Properties*** *window.*

Line Properties

Click the ⊞ button next to Line Properties to specify the line properties for the color scale bar border. This line property is used for the outside of the color scale bar box and for the lines separating the colors inside the color scale bar. You can set *Style, Color, Opacity,* and *Width* of the line.

Opacity

Change the *Opacity* of the color scale by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the to change the opacity percentage. This is the opacity for the filled color portion of the color scale bar.

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 Σ 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 . 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.

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

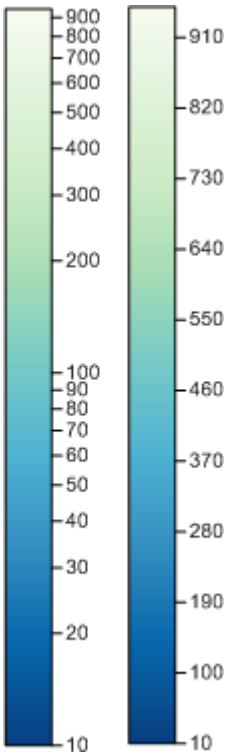
Set the last label value in the *Maximum* box. This is the last possible value that will be labeled. This may not be actually labeled, depending on the starting *Minimum* value and the *Interval* specified. This is a Z value in Z coordinates. To change the *Minimum* label, highlight the existing value and type a new value. When the color scale is displaying a logarithmic scale, the *Minimum* value must be greater than 0.

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.

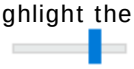
Labels per Decade

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.



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 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 - Contour and 3D Wireframe

A color scale bar is a legend that shows the fill color and pattern assigned to each contour level on a filled 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, image, 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 filled contour map:

1. Click on a contour map to select it.
2. In the Properties window, click on the Levels tab.
3. Check the *Fill contours* box to fill the contours with color.
4. Check the *Color scale* box to display the color scale bar.

To add a color scale bar to a 3D wireframe:

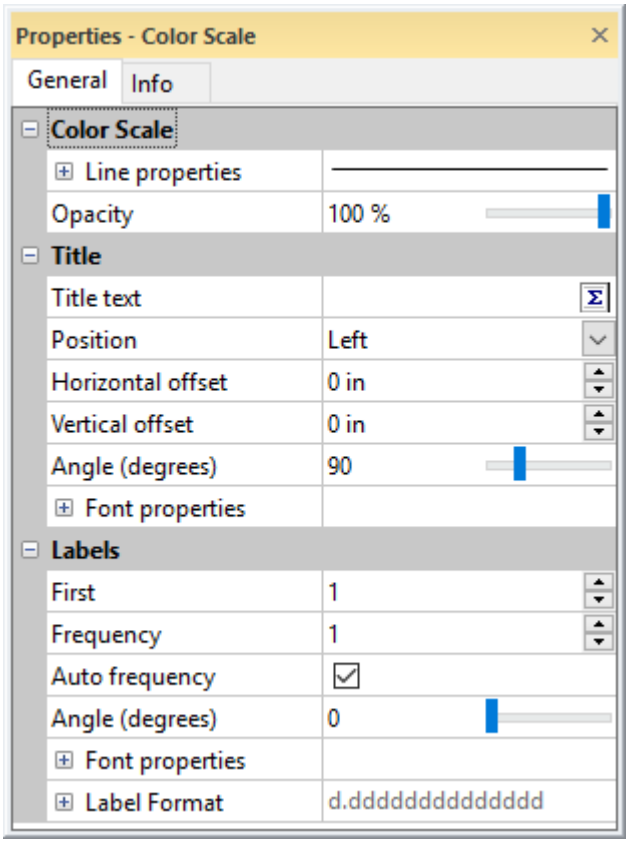
1. Click on a wireframe map to select it.
2. In the Properties window, click on the Color Zones tab.
3. Click the *Edit Levels* button next to *Color zones*.
4. In the **Properties** dialog, check one or more of the *Apply zones to lines of constant* boxes.
5. Check the *Color Scale* box.
6. Click *OK* 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 uncheck 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.



*Change color scale properties for contour and 3D wireframe maps in the **Properties** window when a Color Scale bar is selected.*

Line Properties

Click the ⊞ button next to Line Properties to specify the line properties for the color scale bar border. This line property is used for the outside of the color scale bar box and for the lines separating the colors inside the color scale bar. You can set *Style, Color, Opacity,* and *Width* of the line.

Opacity

Change the *Opacity* of the color scale by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the to change the opacity percentage. This is the opacity for the filled color portion of the color scale bar.

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 Σ 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 . 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.

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.

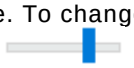
Label Frequency

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.

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

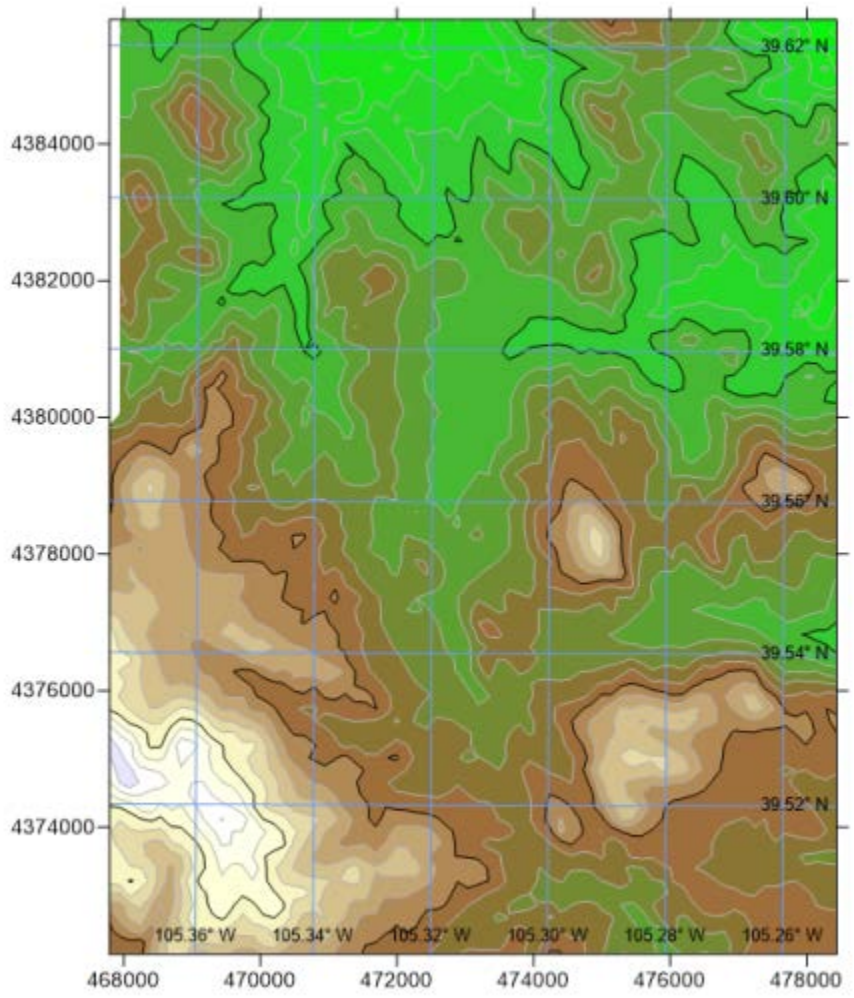
Click the ⊞ next to Label Format to choose the numeric label format.

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.

Graticules

Clicking the **Map Tools | Add to Map | Graticule** command 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:

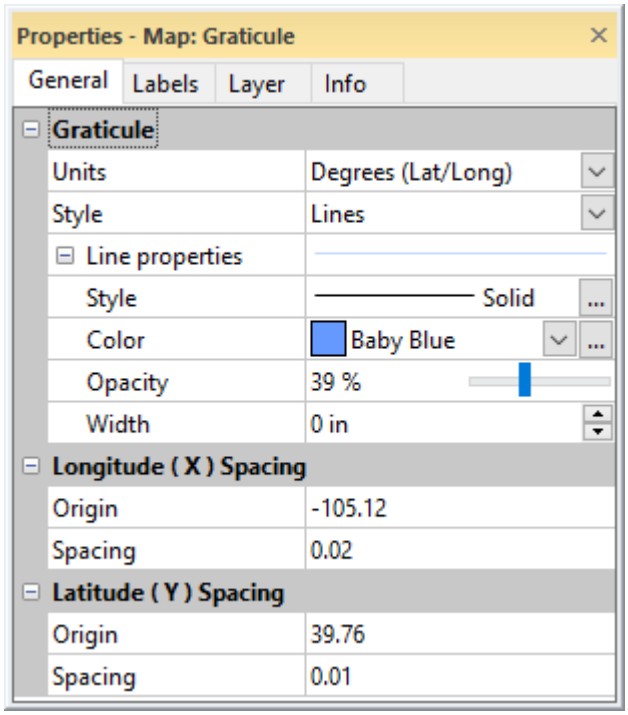
1. In the Contents window or plot window, click on the map to which the graticule should be added.
2. Click the **Map Tools | Add to Map | Graticule** command. Alternatively, right-click the map and select **Add | Graticule** in the context menu.
3. A graticule is added to the map. Select the graticule 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

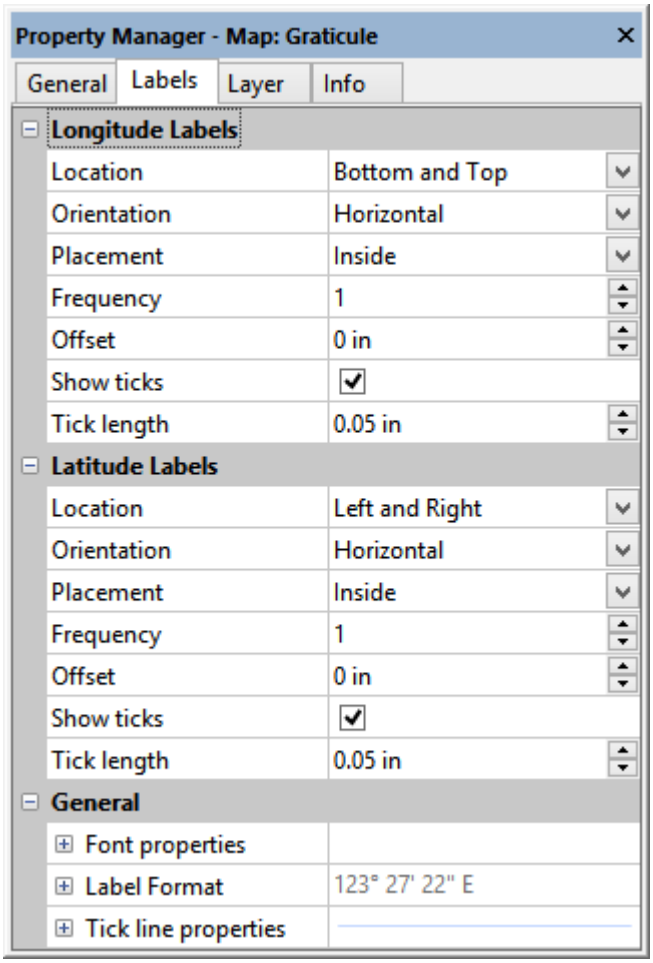
Graticule lines are drawn at a specified latitude and longitude and equally spaced from the origin. The *Longitude (X) Spacing* section controls the placement and spacing of longitude graticule lines. The *Latitude (Y) Spacing* section controls the placement and spacing of latitude graticule lines.

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** window into *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

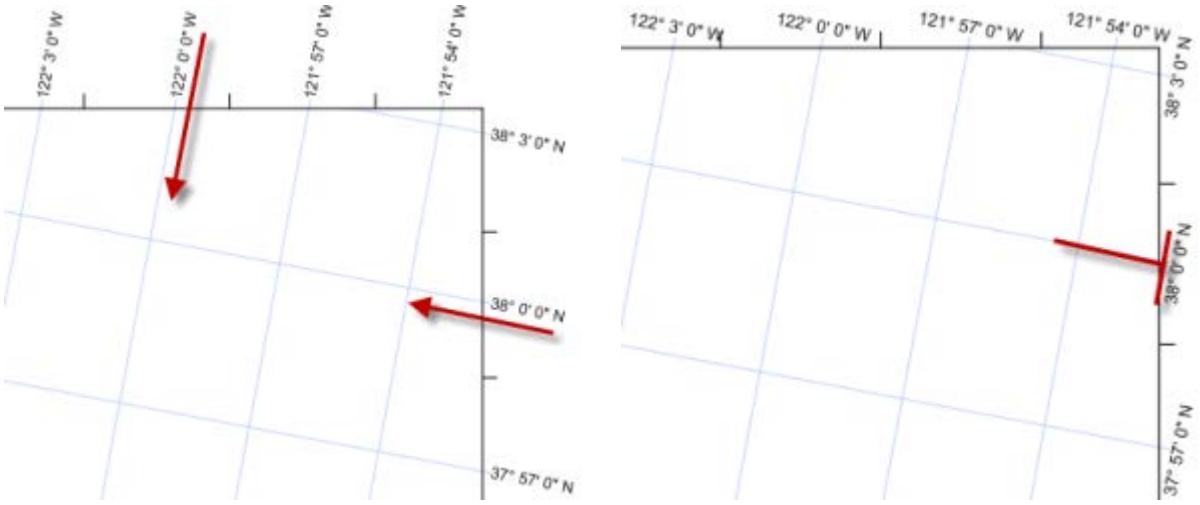
The *Location* property in the *Longitude Labels* section controls the label placement in reference to the graticule. Click the current selection in the *Location* field and select *None*, *Bottom*, *Top*, or *Bottom and Top* from the list. *None* disables longitude labels. *Bottom* displays the labels 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.

Profiles

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** 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 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.

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 on the *Base - Profile* layer in the map to select it. Click the Start Editing command. You can then click on the *Polyline* object and move it to the desired location. When the polyline is located in the new position, click the **Stop Editing Group** command. 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,

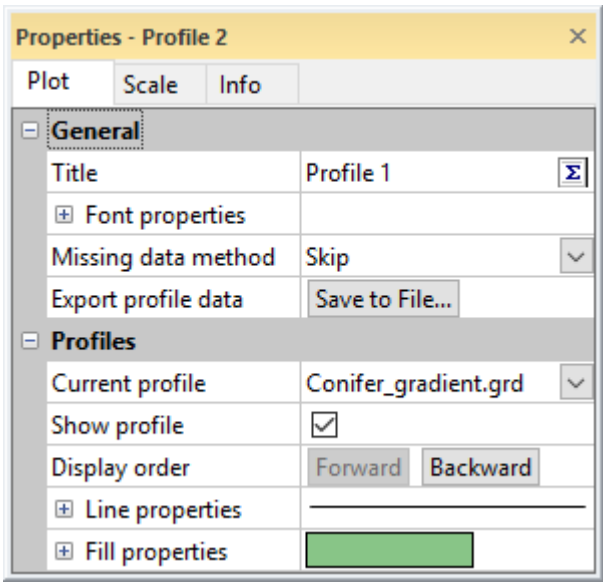
1. Click on the *Map* object in the **Contents** window.
2. In the **Properties** window, click on the **View** tab. Change the *Projection* to *Orthographic*.
3. Click and drag the *Tilt (degrees)* to 90.
4. Then, click the **Map Tools | Add to Map | Profile** command.

Once the profile is located on the map, the *Projection* and *Tilt (degrees)* properties can be changed back to their previous values.

Profile - Plot Page

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.

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 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.

Missing Data

The *Missing data method* property controls how NoData areas in the map are displayed in the profile. Available options are *Skip, Missing,* and *Custom Value*. When the *Missing data 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.

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.

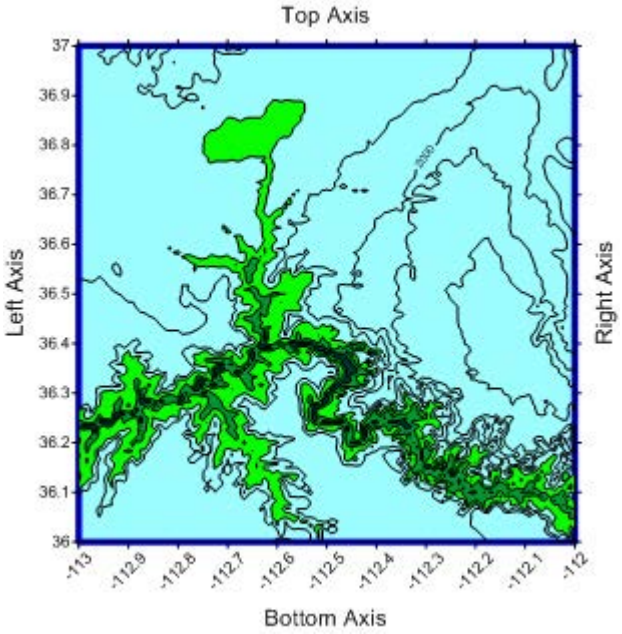
Fill Properties

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.

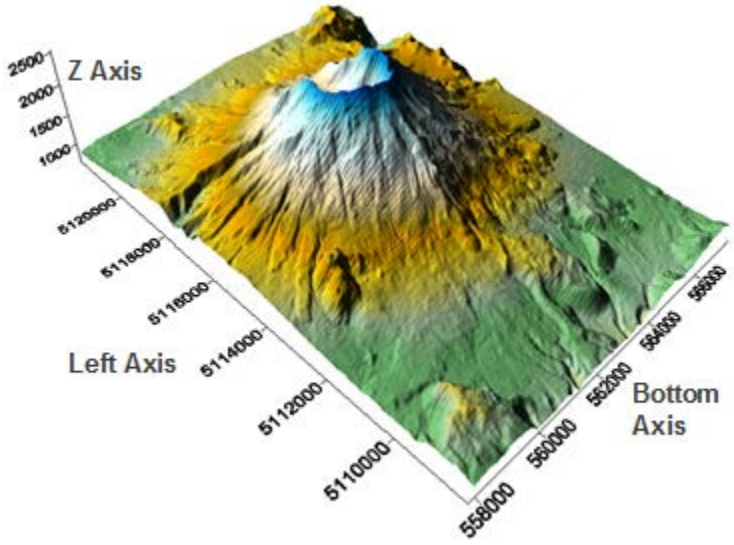
Axes

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

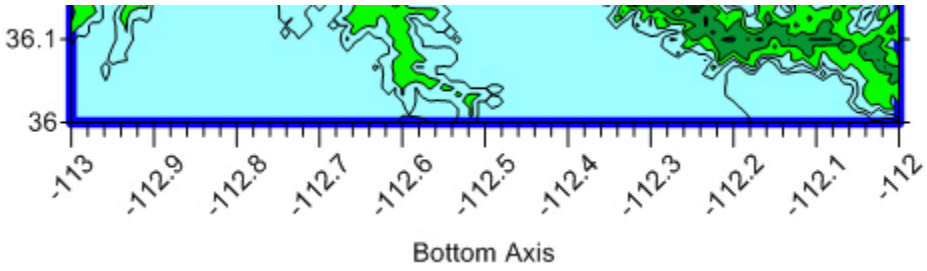
Contents window. Click the check box adjacent to the axis name to turn on or off the axis. On a 3D wireframe or 3D surface, only the Left Axis, Bottom Axis, and Z Axis are initially visible.

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

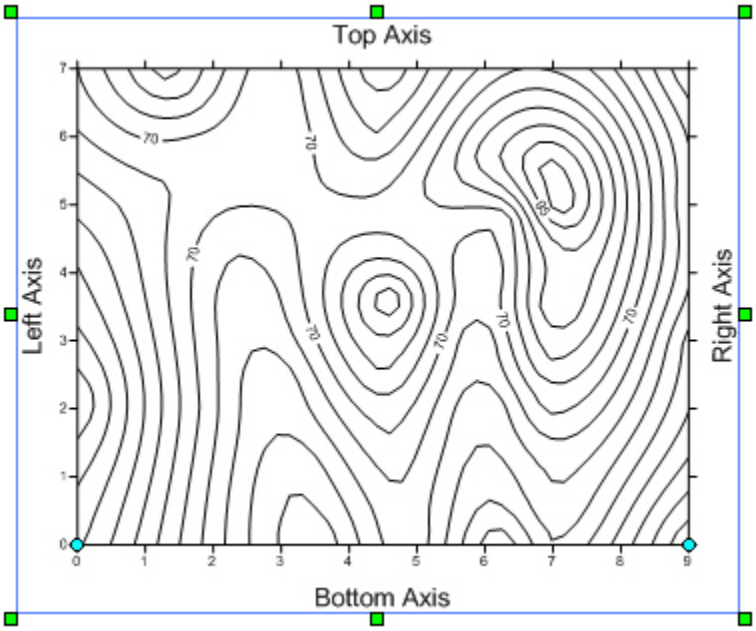
Map axes can display both major and minor ticks. Major ticks can display tick labels. Minor ticks are displayed at regular intervals between major ticks, and do not have associated tick labels. When tick labels are displayed, all major ticks along the axis are represented with a label. The display of 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.



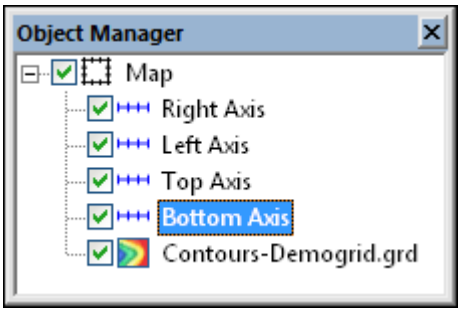
On this axis, four minor ticks are included with every major tick.

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

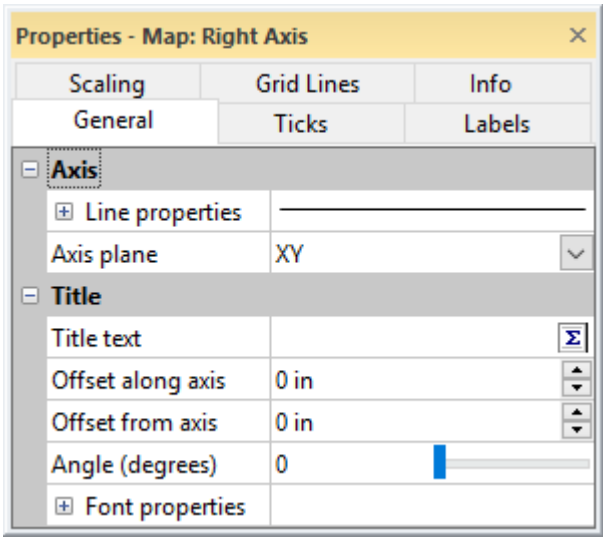
Click the one of the **Map Tools | Add to Map | Axis** commands, **Right Axis**, **Top Axis**, **Bottom Axis**, or **Z Axis**, to add any number of additional X, Y, or Z Axes to your current map. Alternatively, right-click on an existing map and click **Add to Map** to add a map layer, axis, or scale 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.

General Page - Axis 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.

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.

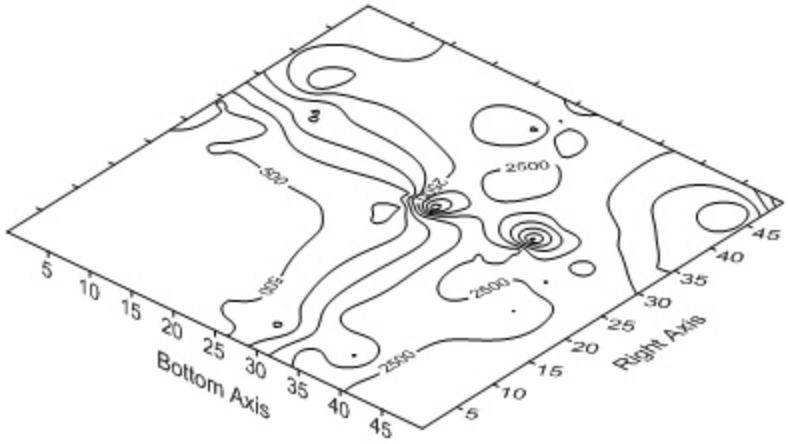
Line Properties

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

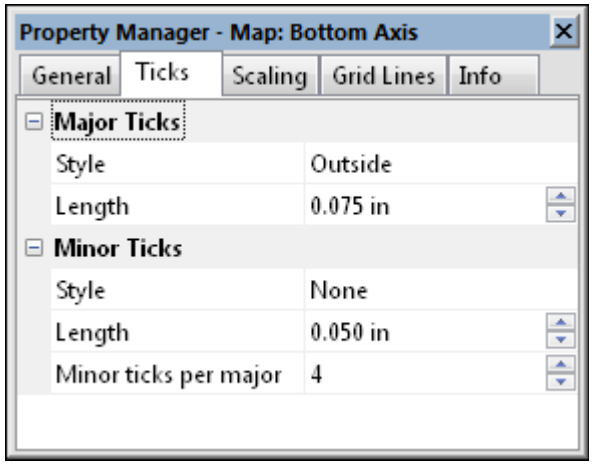
Click the ⊞ button next to *Font Properties* to specify the text properties for the axis title.

Ticks Page - Axis 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.

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.

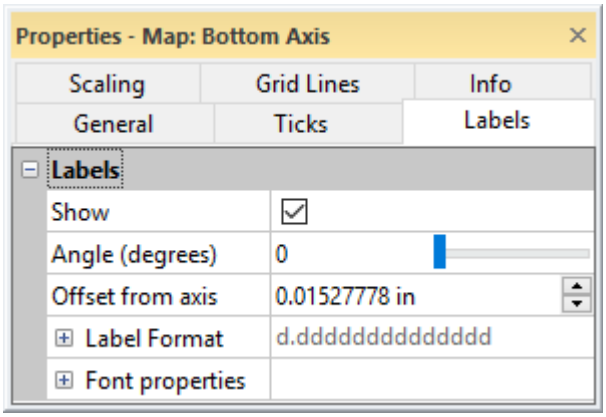
Axis Plane

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 - Axis 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.

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.

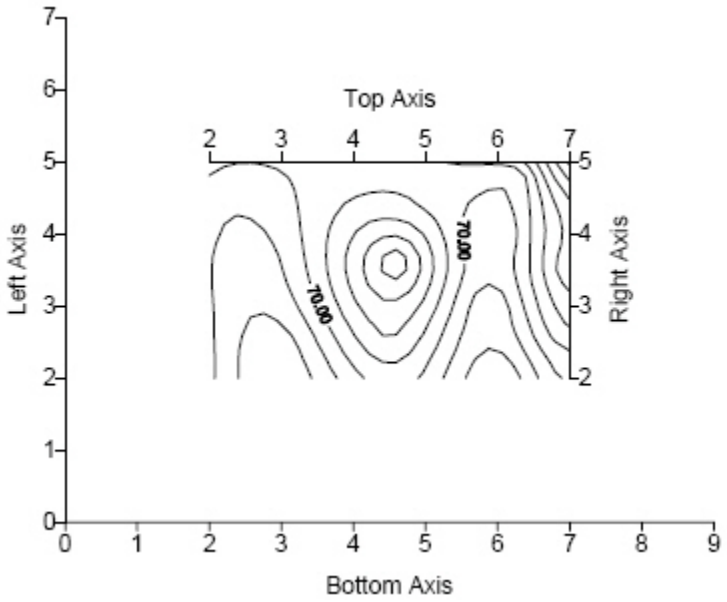
Axis Plane

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.

Scaling Page - Axis Properties

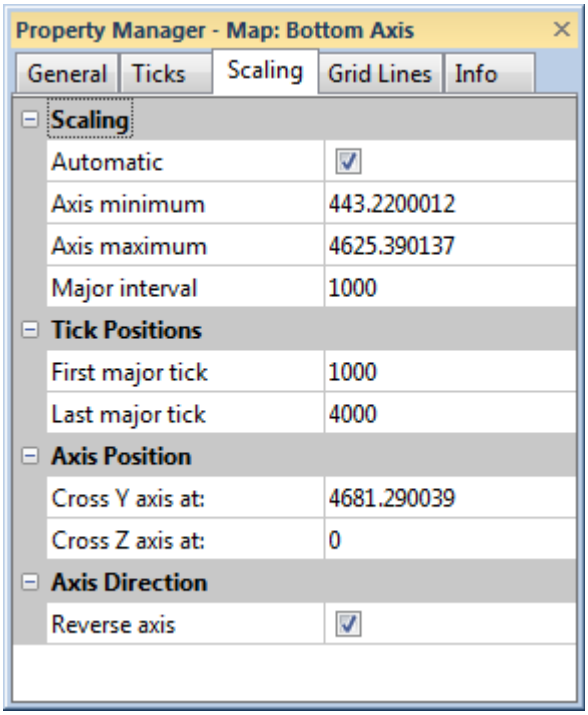
Axis scaling controls the axis limits, the major tick starting and ending values, and the position of the axis relative to the other axes in the map. To set axis scaling, click on the axis to select it. Click on the **Scaling** page in the Properties window.

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.

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.

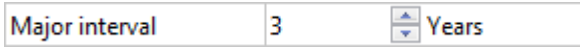
Major Interval

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* $\leq$ *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

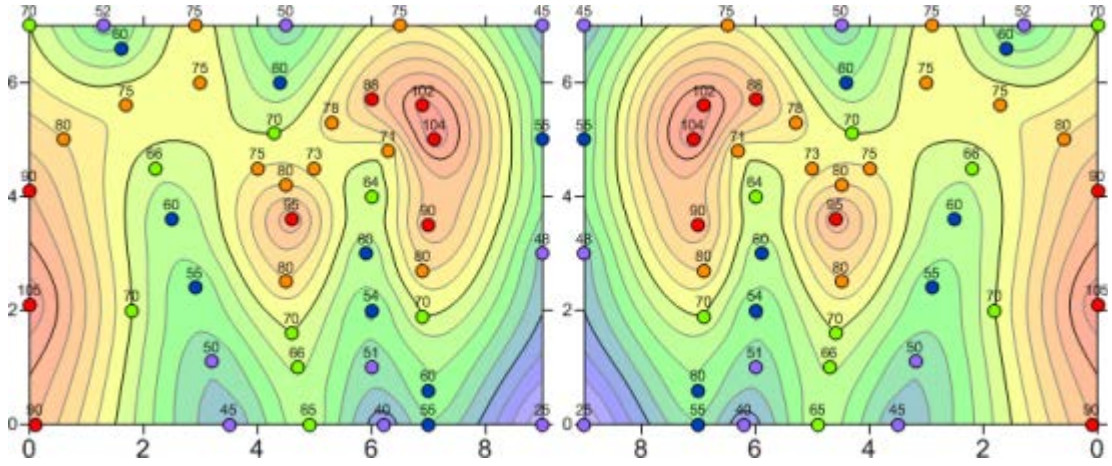
The *Cross X axis at* box is only displayed when an Y or Z axis is selected. This specifies the position of the Y or Z axis along the X axis. The value is entered in X data units. To change the value, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change.

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, profile 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.
    1. 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.
    2. The *Axis minimum* box specifies the minimum axis limit.
    3. 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

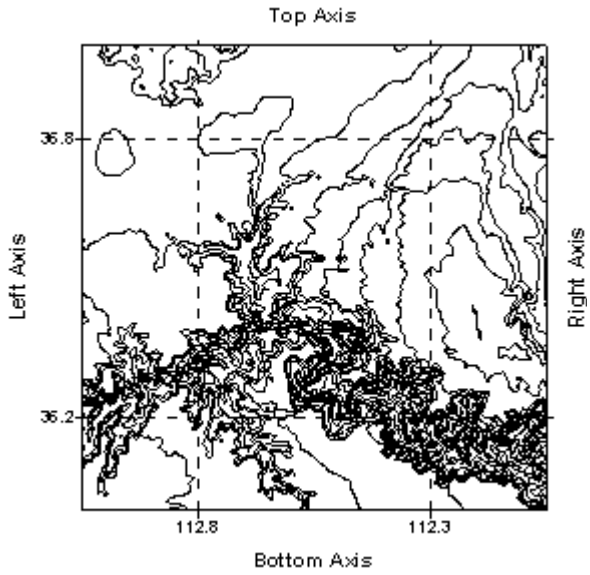
On the Scaling page in the axis properties there are three options that control the spacing of major ticks along the selected axis: the *First major tick*, *Last major tick,* and *Major interval* boxes.

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.



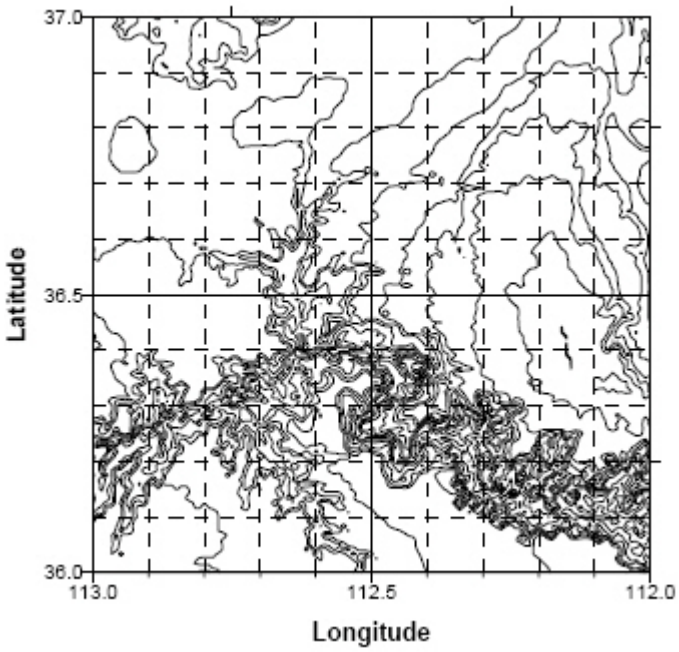
By setting the First major tick, Last major tick and Major interval values, you can control the position of major ticks along the axis.

Grid Lines Page - Axis 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.

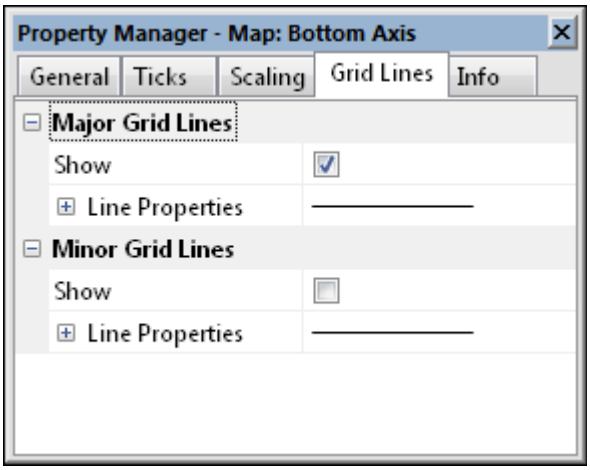
Grid lines are drawn over the extent of the map. For example, if you draw grid lines for an X axis (the Bottom Axis), the grid lines are drawn parallel to the Y axis (Left Axis) over the entire map. If your X axis is offset from the Y axis, the grid lines are not drawn beginning at the end of the X axis, 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.

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.

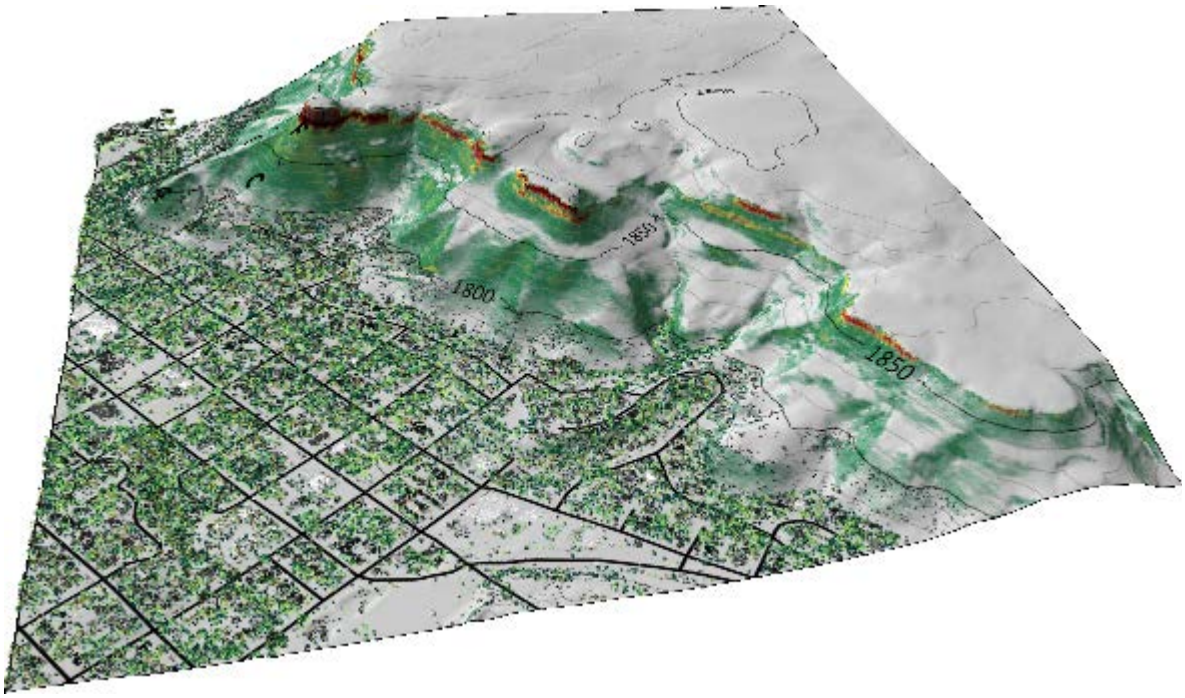
The grid lines are drawn on the map as you make changes in the **Properties** window.

Chapter 23 - 3D View

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. 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.

The 3D view window only displays layers that are visible in the map in the plot window. Show or hide surfaces, textures, and vectors in the 3D view with by selecting or clearing the visibility check boxes in the 3D view **Contents** window. Completely remove a surface, texture, or vector from the 3D view by switching to the plot window and hiding the associated map layer.

Click the **Map Tools | View | 3D View** command or *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. The map must include at least one grid-based layer or point cloud layer to create a 3D view. 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*.

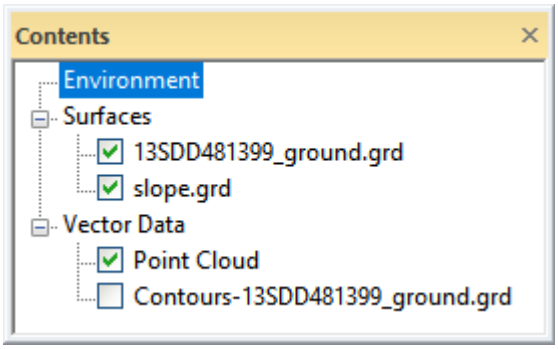


The 3D view displays the map in a three-dimensional space. This map includes contour, color relief, post, base, and point cloud layers.

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 and vectors in the 3D View*

Environment

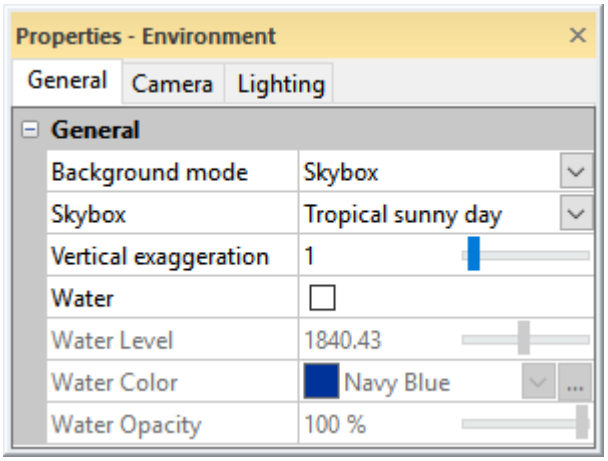
The *Environment* object in the **Contents** window contains the properties for the 3D view. Select the *Environment* to modify the 3D view background, vertical exaggeration, water level, camera, and lighting. The *Environment* object has three Properties pages:

- General
- Camera
- Lighting

The **Lighting** properties have no effect on the 3D view when a hill-shaded color relief layer or a shaded relief layer is present in the map. Turn off hill shading for the color relief layer(s) and remove any shaded relief layers to use the *Environment* **Lighting** properties.

General Page - Environment

The **Environment** page contains the 3D view window background, vertical exaggeration, and water level properties. Select the *Environment* object in the 3D view **Contents** window to modify the 3D view 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.

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.

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 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 to adjust the water level plane opacity.

Semi-Transparent Water Level and 3D Surface Layers

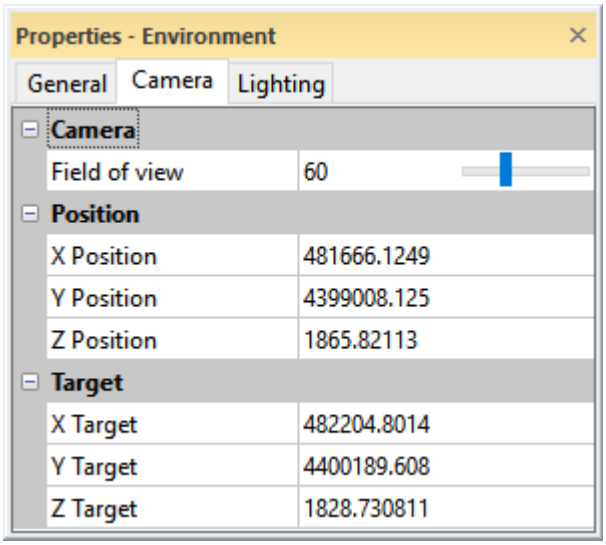
Semi-transparent water levels require certain settings. When applying a *Water opacity* less than *100%* to a 3D view, the following properties should be used:

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.

Camera Page

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 Page

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. 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 and any overlays that may have been combined with the surface. The light source is fixed relative to the surface, so if the surface is rotated, the light rotates with it.

Hill Shading and Shaded Relief in the 3D View

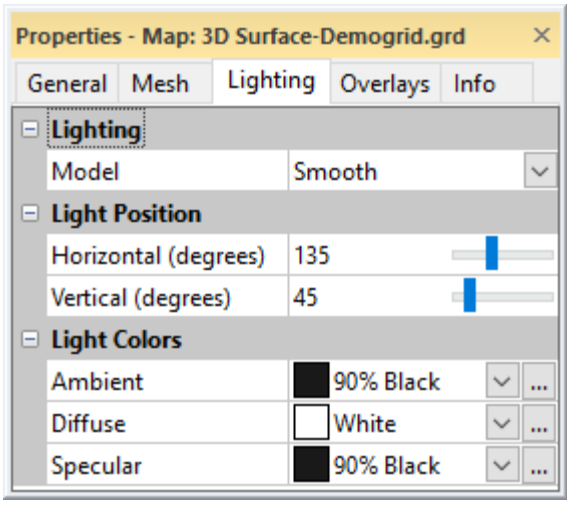
The 3D view lighting is turned off when the map in the 3D view contains a hill-shaded color relief layer or a shaded relief layer. This is because the map is already shaded by the color relief layer hill shading or shaded relief layer. The cumulative effect of both lighting systems would result in very dark regions of the map in the 3D view.

To use the 3D view **Lighting** properties instead of the shaded relief layer, the shaded relief layer must be removed from the map in the plot window. The shaded relief layer can be deleted to remove it permanently, or the shaded relief layer can be broke apart from the map to remove it temporarily.

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.

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.

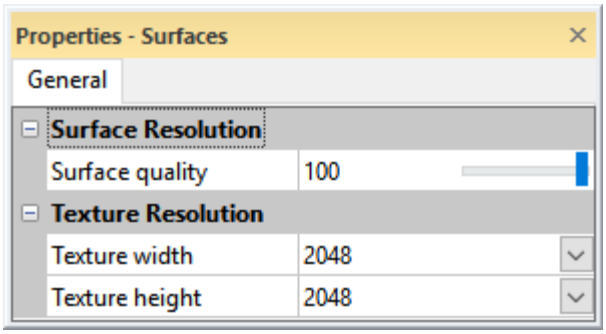
In general, these reflectivity colors should be specified as shades of gray in order to evenly reflect the surface material color components. However, special effects are possible by specifying non-gray colors for the reflectivity. For example, assume the *Ambient* reflectivity is set to pure red, and 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.

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.

General Page - Surfaces Group

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.

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.

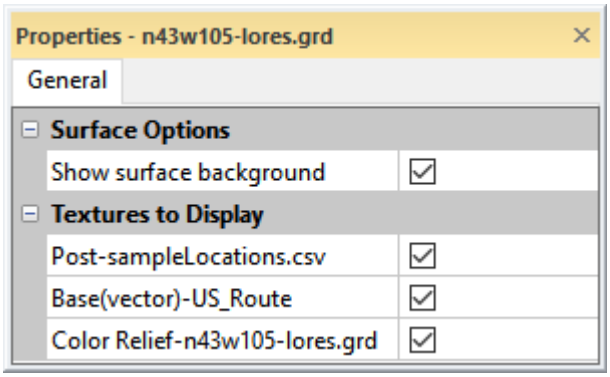
By default, each surface shows the texture from any map layer that uses the same grid file as the surface, and the textures from any base or post layers. Use the **General** page to hide or display textures.

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.

General Page - Surface

The **General** page contains the available texture overlays for the selected surface in the 3D view. Select a single grid in the *Surface* group in the 3D view **Contents** window to select the textures you wish to display on the surface. Each surface in the 3D view can have the *Textures to display* set independently.

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.



*Select the textures for the surface in the **General** page.*

Show Surface Background

By default, the surfaces in the 3D view are displayed as a lighted white surface. The textures from the layers in the map are overlaid on the surface and controlled by the *Textures to Display* section. If your textures include transparency and you wish to hide the surface, clear the *Show surface background* check box. The skybox or background will show through transparent regions of the surface when *Show surface background* is not checked.

The *Show surface background* option should be selected if you wish to add a semi-transparent water level.

Textures to Display

The available textures for 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 plot window for the texture to be included in the *Textures to display* list.

Any 3D surface textures should be hidden if you wish to add a semi-transparent water level.

Vector Data

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

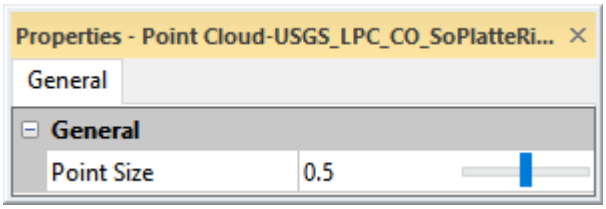
Contours are created for contour layers in the map. By default, the contour layer is displayed as a texture overlay on the surface. Vector contour lines are hidden by default. Select the check box next to the contours vector object in the **Contents** window to display the contours. There are no properties for 3D vector contours.

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.

General Page - Point Cloud

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 **Vector** 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.

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
Copy to Clipboard	Copy an image of the 3D view window to the clipboard

Walk

Click the **3D View | View | Walk** command 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 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 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

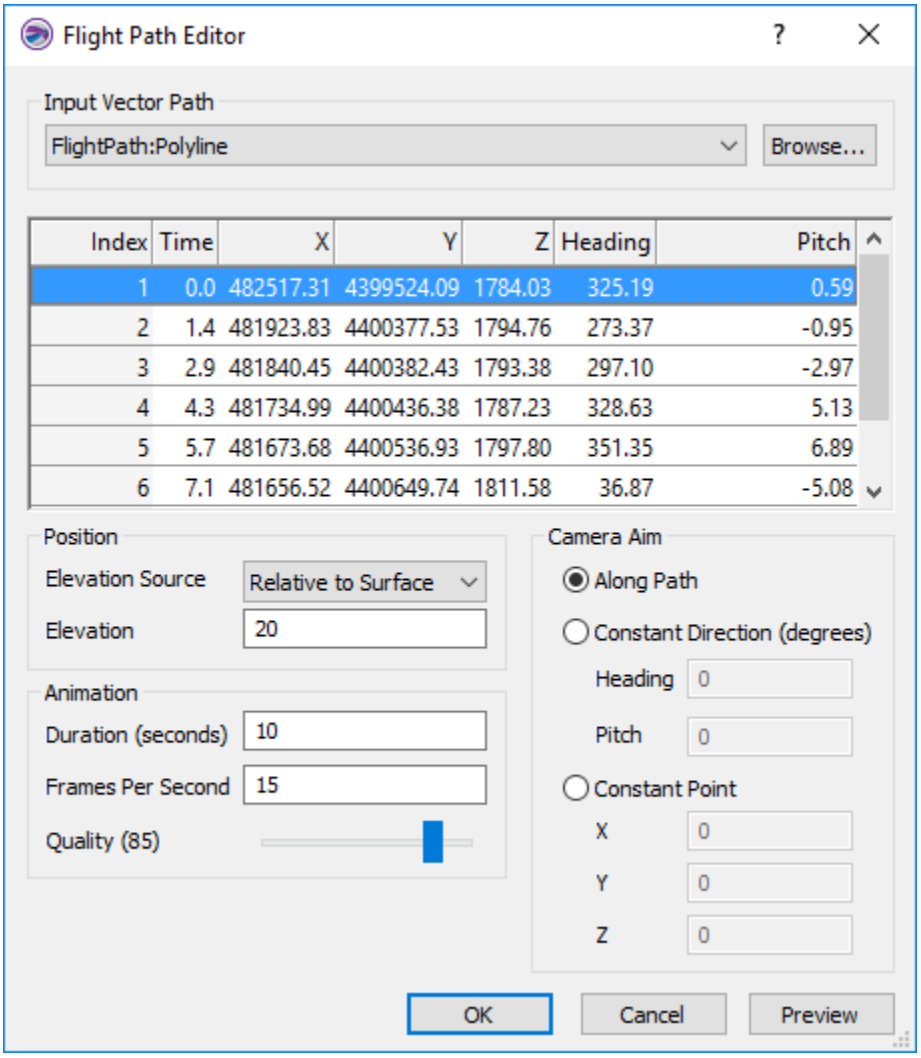
Click the **3D View | Fly-Through | Create/Edit** command to create or edit a fly-through. A fly-through moves the camera along a path specified by the vertices of a vector object. The **Flight Path Editor** is displayed when the **3D View | Fly-Through | Create/Edit** command is clicked.

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** 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** 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.

Copy to Clipboard

Click the **3D View | Tools | Copy to Clipboard** command 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.

Chapter 24 - Map Tools

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, Shaded 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
Set Limits	Set the map limits by drawing a rectangle
Trackball	Rotates and zooms the selected map in 3D
3D View	View the selected map in a new 3D view window
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
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
Open Attribute Table	Opens the Attribute Table for the objects in the selected layer
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
Export Contours	Export contours as lines in a DXF, SHP, or TXT file.
Grid	Opens the selected layer's grid in the **Grid Editor**.
Edit Contour Labels	Edits individual contour labels of the selected map
Edit Post Labels	Edits individual post map labels of the selected map

Set Limits

Click the **Map Tools | Map Tools | Set Limits** command 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.

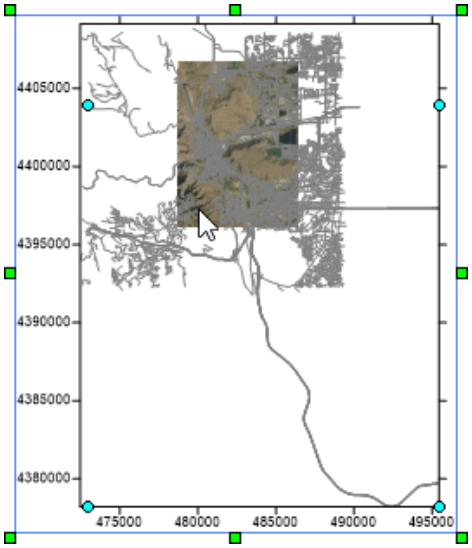
Set Limits Tips

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

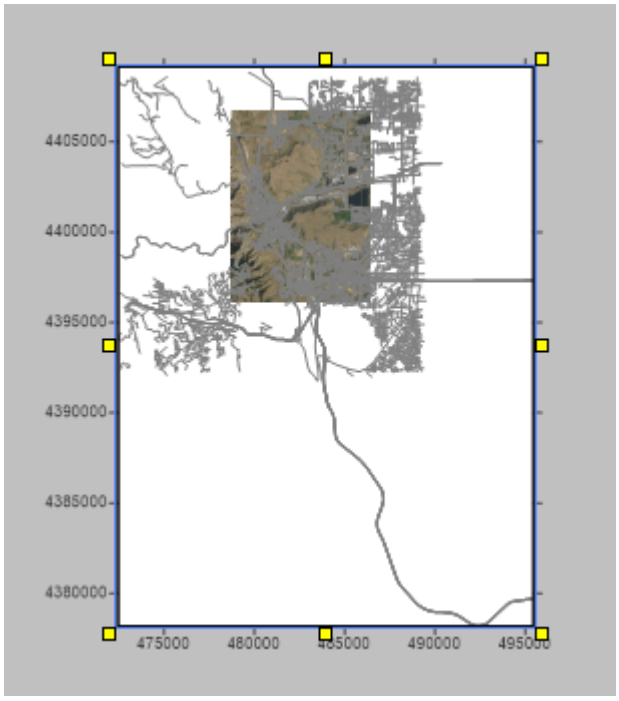
Example

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 shaded 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.



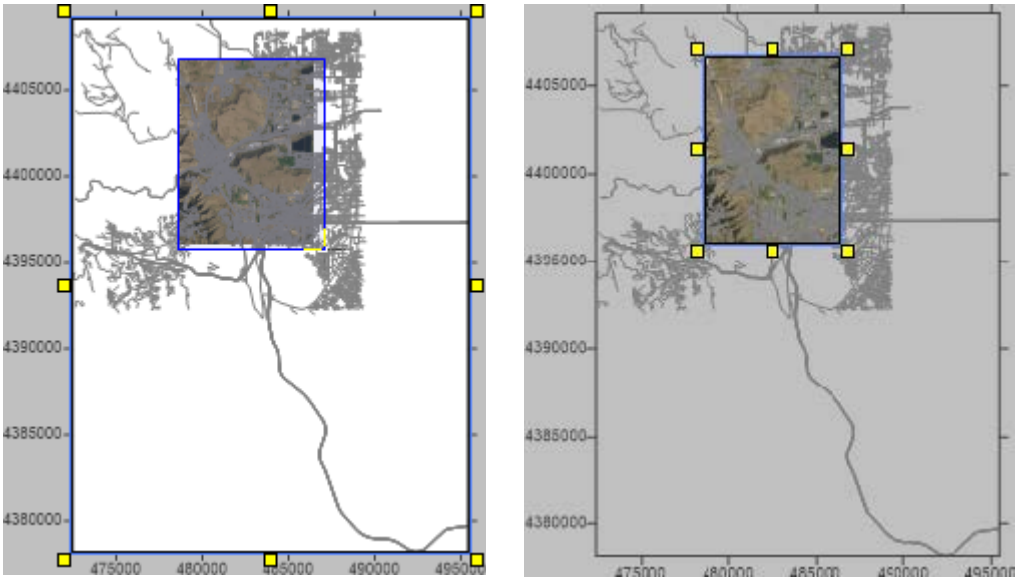
The map limits cover the entire data extents.

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.



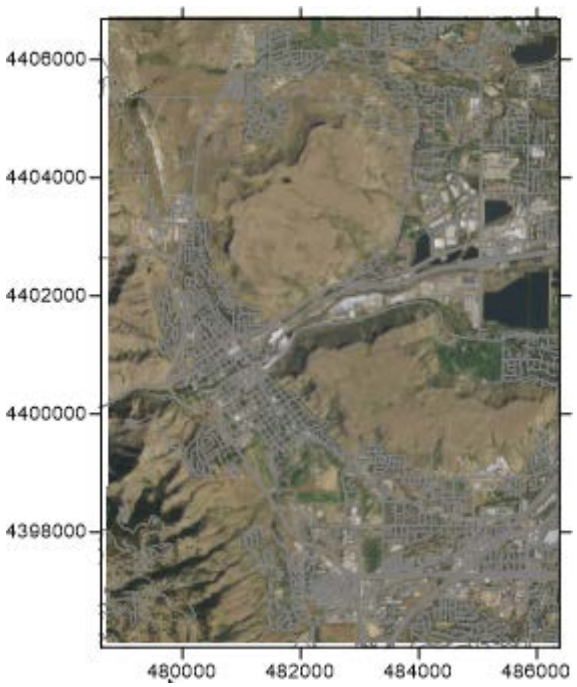
The map limits are represented by the blue rectangle.

3. Press and hold the CTRL key.
4. Click and drag in the plot window to draw the new map limits.



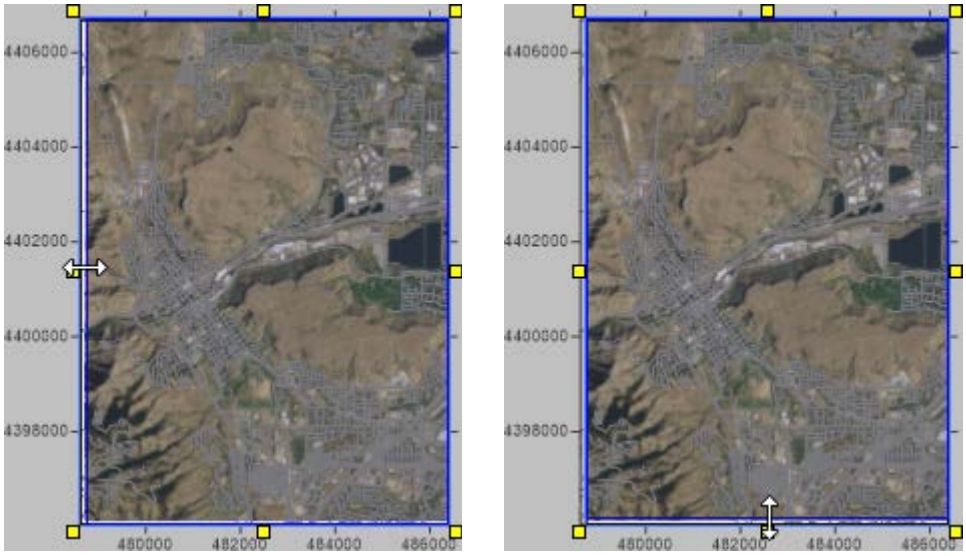
The new rectangle was drawn around the image base layer.

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.



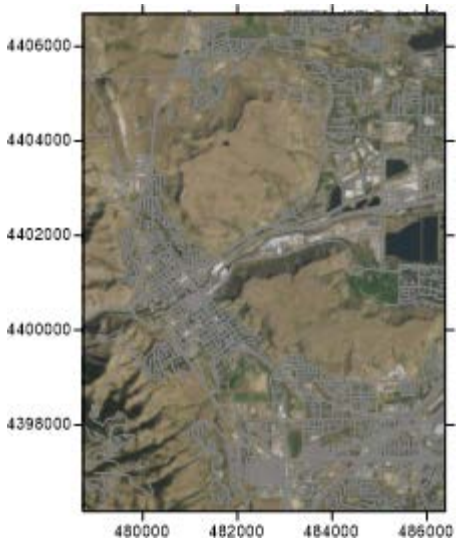
The map limits and scale have been updated.

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 rotates and tilts the view in the 3D view.

Trackball in the Plot Window

Click the **Map Tools | View | Trackball** command 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.

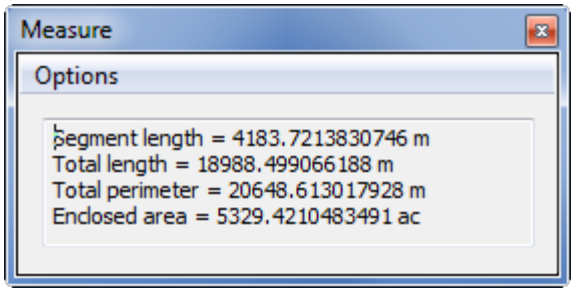
Tilt and Rotation

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 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.

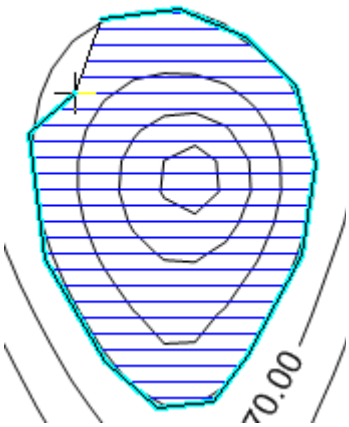
When all 3D maps have been removed, click on the *Map* in the **Contents** window to select it. Click on the View tab in the **Properties** window. Set the *Projection* to *Orthographic* and the *Tilt (degrees)* to 90.

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

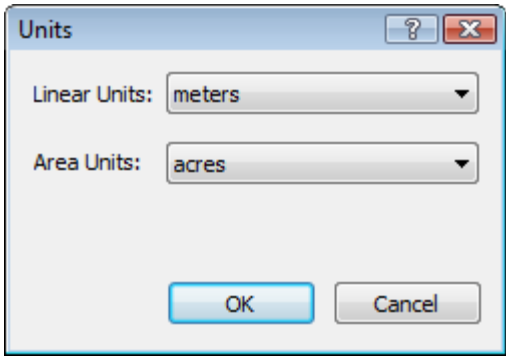
Click the **Options** menu in the **Measure** dialog to modify how the measurements are displayed and to save or export the measurement geometry.

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.

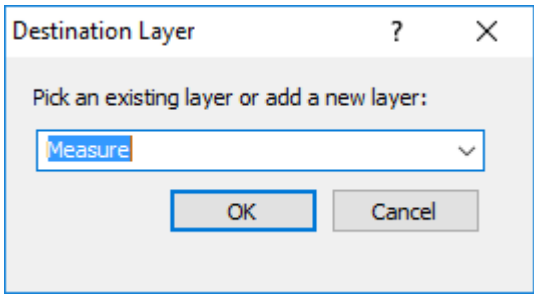
Note that the units cannot be changed if the map is not referenced or if the *Map* coordinate system is set to display latitude and longitude. Set the layer source coordinate system and the map target coordinate system and the units can be changed.

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. Click the Start Editing command to enter the base map layer. Use the Features tab commands to add polylines, spline polylines, polygons, rectangles, rounded rectangles, or ellipses to the base map layer. When all of the objects have been added, click the **Stop Editing Group** command. 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, image, shaded 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 choose **Map Tools | Map Tools | Overlay Maps**, 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 shaded relief maps, 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:
    1. Hold down the SHIFT key and click each map layer to be combined in the plot window.
    2. Hold down the CTRL key and click each map to be combined in the **Contents** window.

3. Click **Home | Selection | Select All** or press CTRL+A on the keyboard to select all the maps in the plot window.
4. 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.

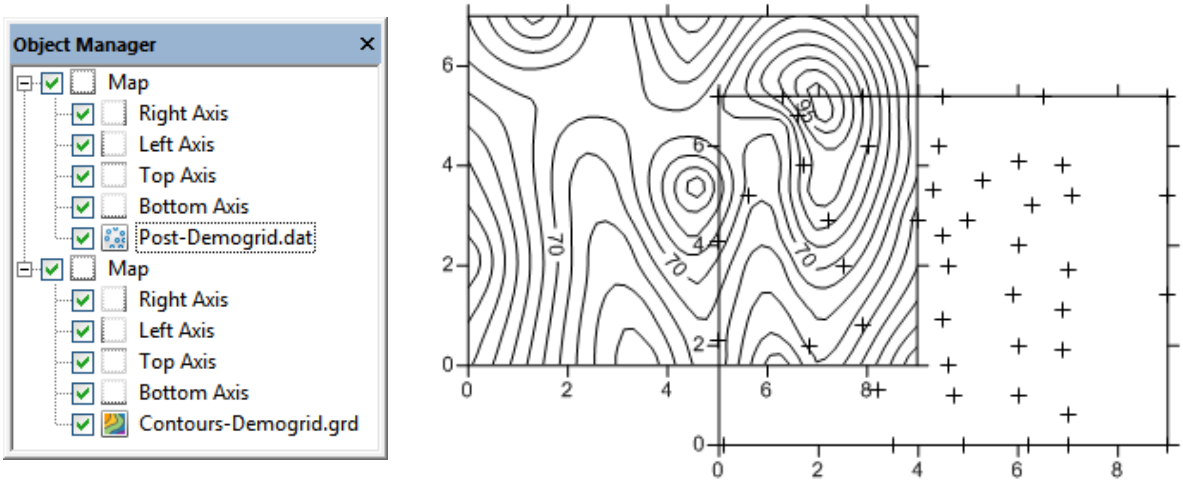
Deleting Overlays

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,

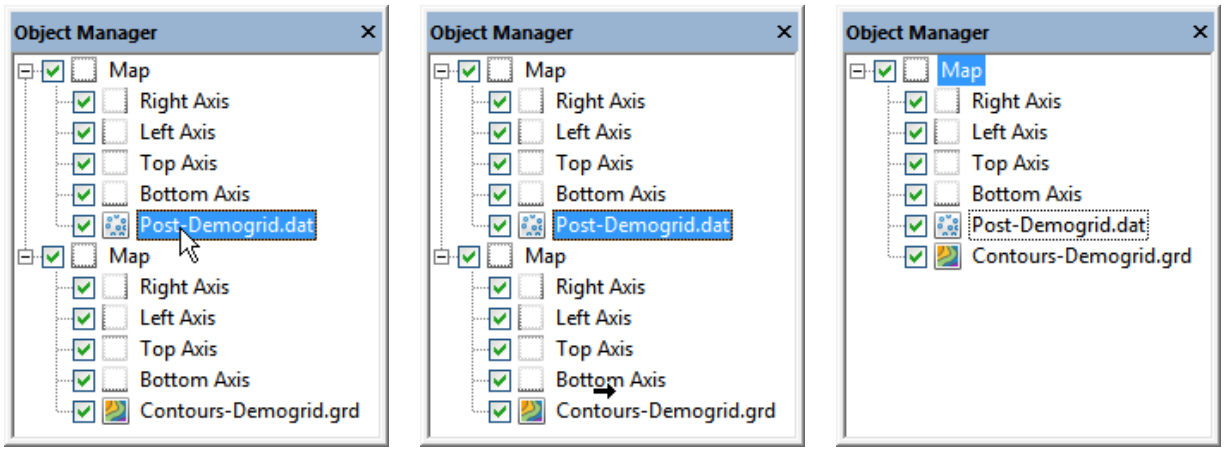
1. Start with two separate map objects. In this example, there is a post map and a contour map. Note that each map has an independent set of axes.



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

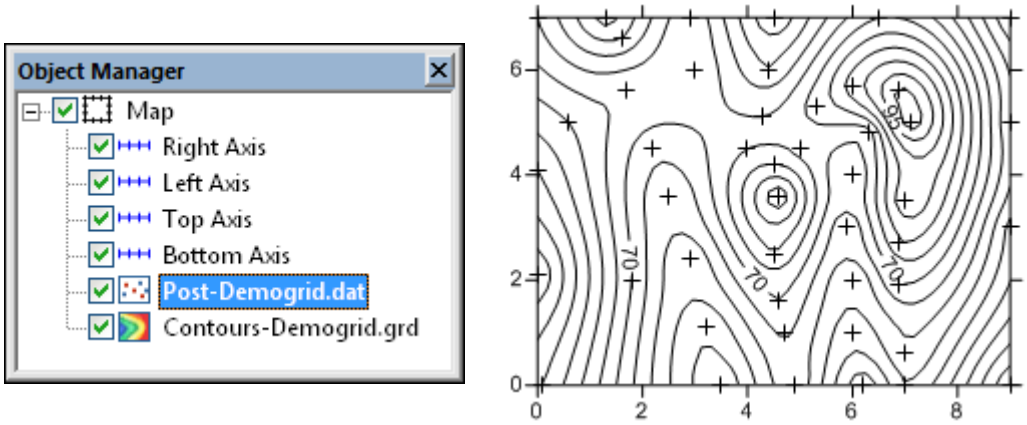
2. Select the post map layer in the **Contents** window and drag it to the contour map object. To do this, left-click and hold the left mouse button while you drag the map layer to a

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.

Combing Maps from Different Surfer Files

This method works well when you have multiple maps in different plot windows.

1. Click the **File | New | Plot** command.
2. Click the **Home | New Map | Base** command.
3. In the **Import** dialog, select the *Demoslice.bln* file from **Surfer's** *Samples* folder. If you are not in the *Samples* folder, browse to it. By default, the *Samples* folder is located in c:\Program Files\Golden Software\Surfer 14\Samples.
4. Click *Open*.
5. Leave *Unreferenced local system* selected in the **Assign Coordinate System** dialog and click *OK*. The new base map is displayed in the plot window and the **Contents** window in a separate map object.
6. Click the **File | New | Plot** command.
7. Click the **Home | New Map | Base** command.
8. In the **Import** dialog, select the *DemoText.mif* file from **Surfer's** *Samples* folder. If you are not in the *Samples* folder, browse to it. By default, the *Samples* folder is located in c:\Program Files\Golden Software\Surfer 14\Samples.
9. Click *Open*.
10. Leave *Unreferenced local system* selected in the **Assign Coordinate System** dialog and click *OK*. The new base map is displayed in the plot window.
11. Click the **Home | Selection | Select All** command to select the entire map object.
12. Click the plot tab for the first plot window or click the **Window | Plot** name.
13. Click the **Home | Clipboard | Paste** command.
14. Click the **Home | Selection | Select All** command to select both map objects.
15. Click the **Map Tools | Map Tools | Overlay Maps** command. The separate maps are combined into a single map object with 2 map layers.

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 **Map Tools | Map Tools | Stack Maps**. 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 25 - Layer Tools

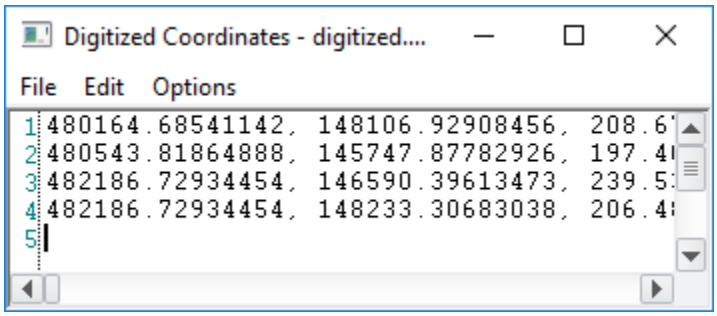
Digitize

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.

Digitized Coordinates Window

Click the **Map Tools | Layer Tools | Digitize** command 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 an Data .DAT data file or 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.

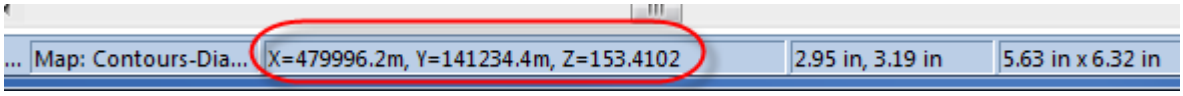
When multiple grid based maps are overlaid, the Z value is determined by the *Track map coordinates of* option in the Options dialog. When this option is set to *Topmost map/layer*, the map layer that is above all others in the Contents window is used for the Z value. When the *Track map coordinates 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 the **File | Save As** command in the **Digitized Coordinates** window. 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.

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 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)

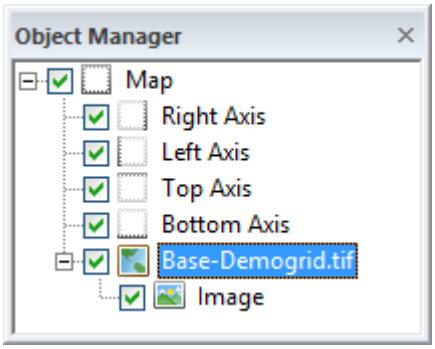
To delete a map layer (overlay) from a composite map, select the map layer in the **Contents** window and press the DELETE key on your keyboard. Alternatively, select the map in the **Contents** window, right-click and select *Delete*.

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 File | Import 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.

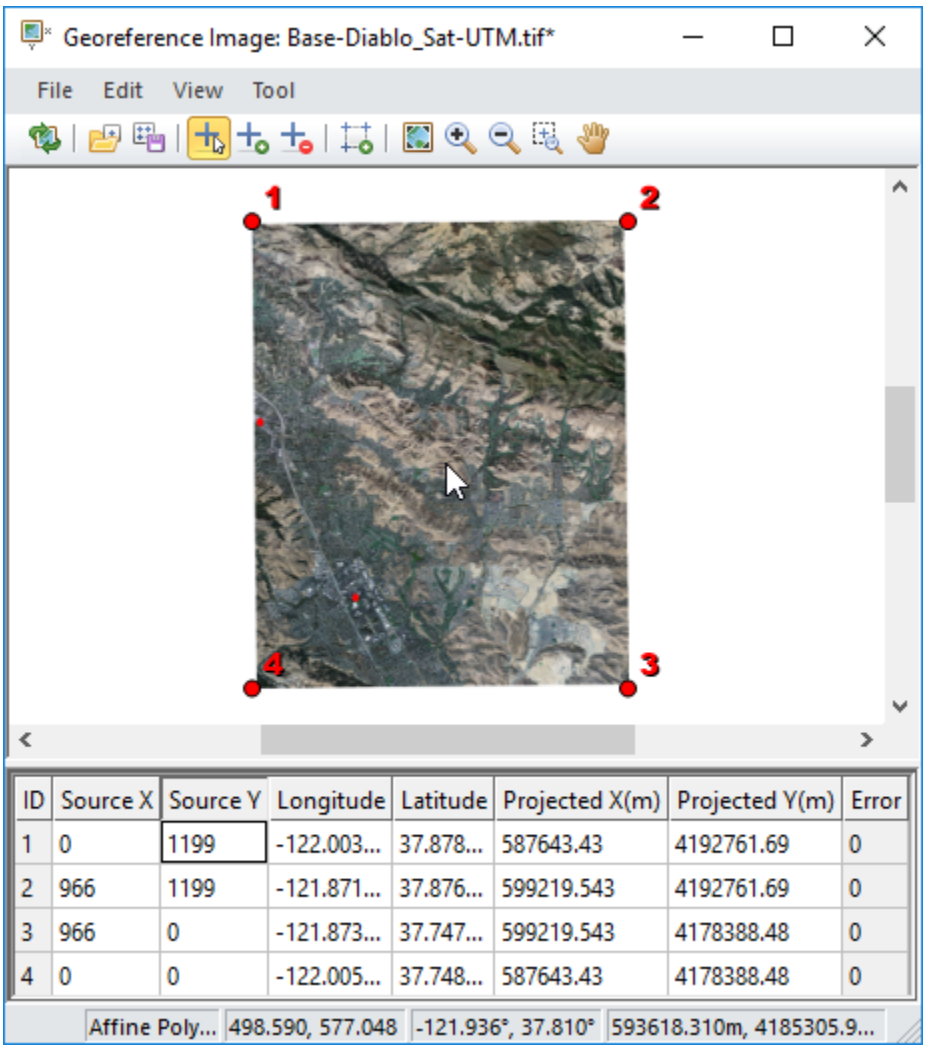
Georeference Image

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 **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 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 | Import.

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	Appends 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. All three methods add one or more points to the end of the control points table. Existing points are not replaced.

- 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.

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

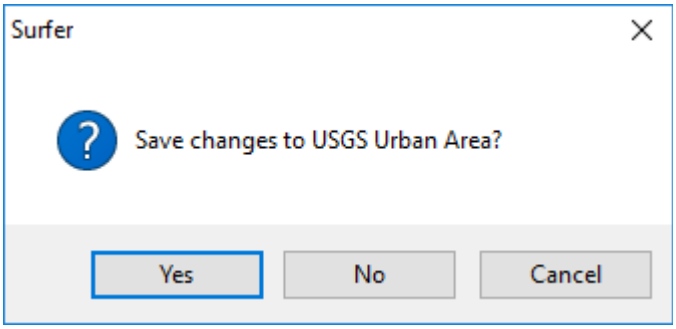
The **Edit | Set Warp Method | Automatic** option is checked by default. The default option is determined by the number of control points when the *Automatic* option is active. When there are three or more control points, the default method is *Affine Polynomial*.

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

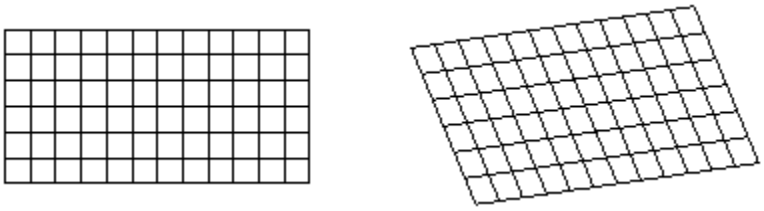
The most common transformation is *Affine Polynomial*. Affine transformations are a subset of bilinear transformations. Bilinear transformations account for rotation, shift, and differential scaling in X and Y. The X and Y axis orthogonality may change, but parallel lines remain parallel. Affine polynomial 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:

$$X = a_{0x} + a_{1x}\,x + a_{2x}\,y$$
$$Y = a_{0y} + a_{1y}\,x + a_{2y}\,y$$

This is called the six parameter affine polynomial transform.



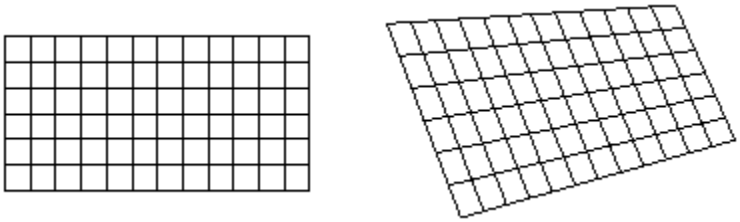
The left graphic is the file before using the affine polynomial transformation. The right graphic is the file after an affine polynomial transformation. Note that lines remain parallel.

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:

$$X = a_{0x} + a_{1x}\,x + a_{2x}\,y + a_{3x}\,xy$$
$$Y = a_{0y} + a_{1y}\,x + a_{2y}\,y + a_{3y}\,xy$$

The minimum number of control points required for this transformation is four.

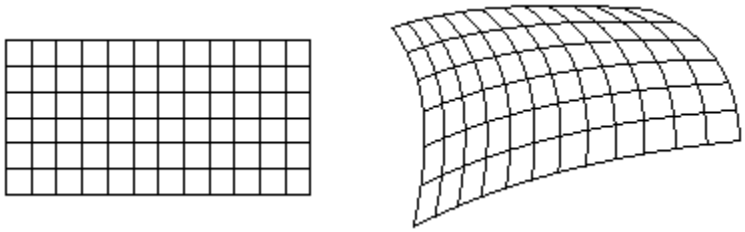


The left graphic is the file before using the first order polynomial transformation. The right graphic is the file after using a first order polynomial transformation. Note that lines are no longer parallel.

Second Order Polynomial

The minimum number of control points required for the *Second Order Polynomial* transformation is six. The equation is of the form:

$$X = a_{0x} + a_{1x}x + a_{2x}y + a_{3x}xy + a_{4x}x^2 + a_{5x}y^2$$
$$Y = a_{0y} + a_{1y}x + a_{2y}y + a_{3y}xy + a_{4y}x^2 + a_{5y}y^2$$

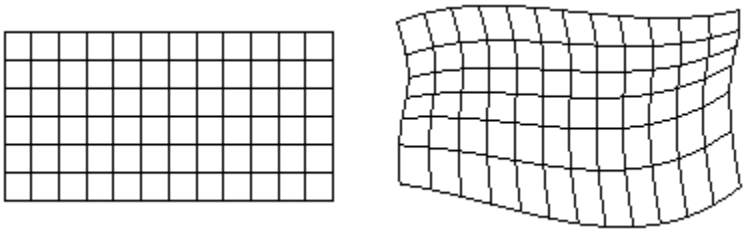


The left graphic is the file before using the second order polynomial transformation. The right graphic is the file after using second order polynomial transformation.

Third Order Polynomial

The minimum number of control points required for the *Third Order Polynomial* transformation is ten. The equation is of the form:

$$X = a_{0x} + a_{1x}x + a_{2x}y + a_{3x}x^2 + a_{4x}xy + a_{5x}y^2 + a_{6x}x^3 + a_{7x}x^2y + a_{8x}xy^2 + a_{9x}y^3$$
$$Y = a_{0y} + a_{1y}x + a_{2y}y + a_{3y}x^2 + a_{4y}xy + a_{5y}y^2 + a_{6y}x^3 + a_{7y}x^2y + a_{8y}xy^2 + a_{9y}y^3$$



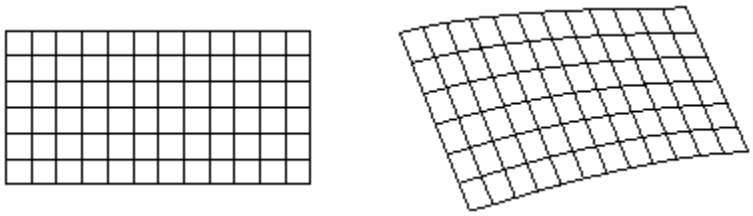
The left graphic is the file before using the third order polynomial transformation. The right graphic is the file after using third order polynomial transformation.

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:

$$\Phi(r_i) = \frac{1}{2} r_i^2 \log r_i^2$$



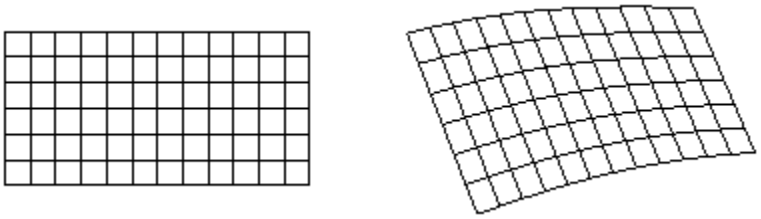
The left graphic is the file before using the thin plate spline transformation. The right graphic is the file after using a thin plate spline transformation.

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:

$$\Phi(r_i) = r_i^3$$



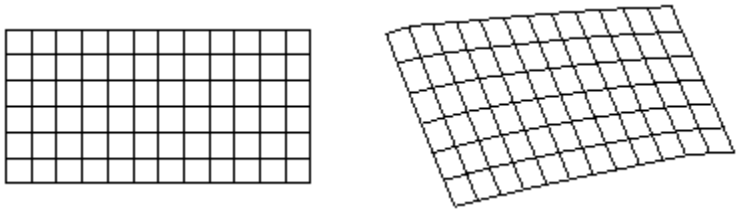
The left graphic is the file before using the natural cubic spline transformation. The right graphic is the file after using a natural cubic spline transformation.

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:

$$\Phi(r_i) = -r_i$$



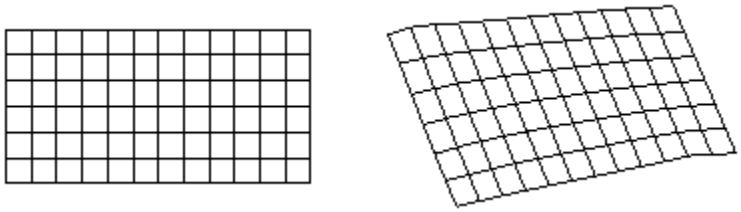
The left graphic is the file before using the Marcov spline transformation. The right graphic is the file after using a Marcov spline transformation.

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:

$$\Phi(r_i) = e^{-r_i}$$



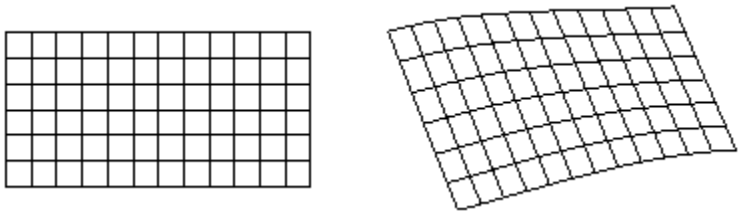
The left graphic is the file before using the exponential spline transformation. The right graphic is the file after using an exponential spline transformation.

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:

$$\Phi(r_i) = -\frac{r_i^2}{1 + r_i^2}$$



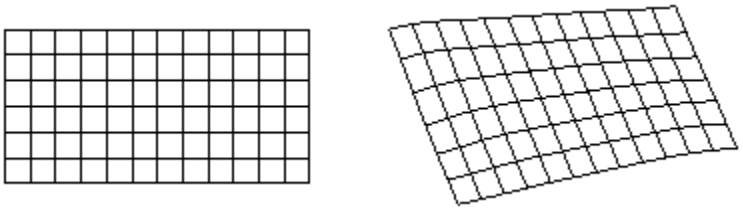
The left graphic is the file before using the rational quadratic spline transformation. The right graphic is the file after using a rational quadratic spline transformation.

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:

$$X = \underset{x}{a_0} + \underset{x}{a_1}\, x + \underset{x}{a_2}\, y + \sum_{i=0}^{N-1} \underset{x}{w_i}(x, y)\, \underset{x}{\varepsilon_i}$$

$$Y = \underset{y}{a_0} + \underset{y}{a_1}\, x + \underset{y}{a_2}\, y + \sum_{i=0}^{N-1} \underset{y}{w_i}(x, y)\, \underset{y}{\varepsilon_i}$$



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.

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**. However, the base layer does not need to be in editing mode to open the **Attribute Table**.

To open the **Attribute Table**, click the **Map Tools | Layer Tools | Open Attribute Table** command. 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**.

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**. The **Attribute Table** lists attribute fields (columns) in alphabetical order, because the attributes for each object in the **Info** page may be in a different order. The first column is always the *Id* field and contains the object name.

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**.

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 **Home | Undo | Redo** 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. The selected cells are indicated with a blue background and white text.

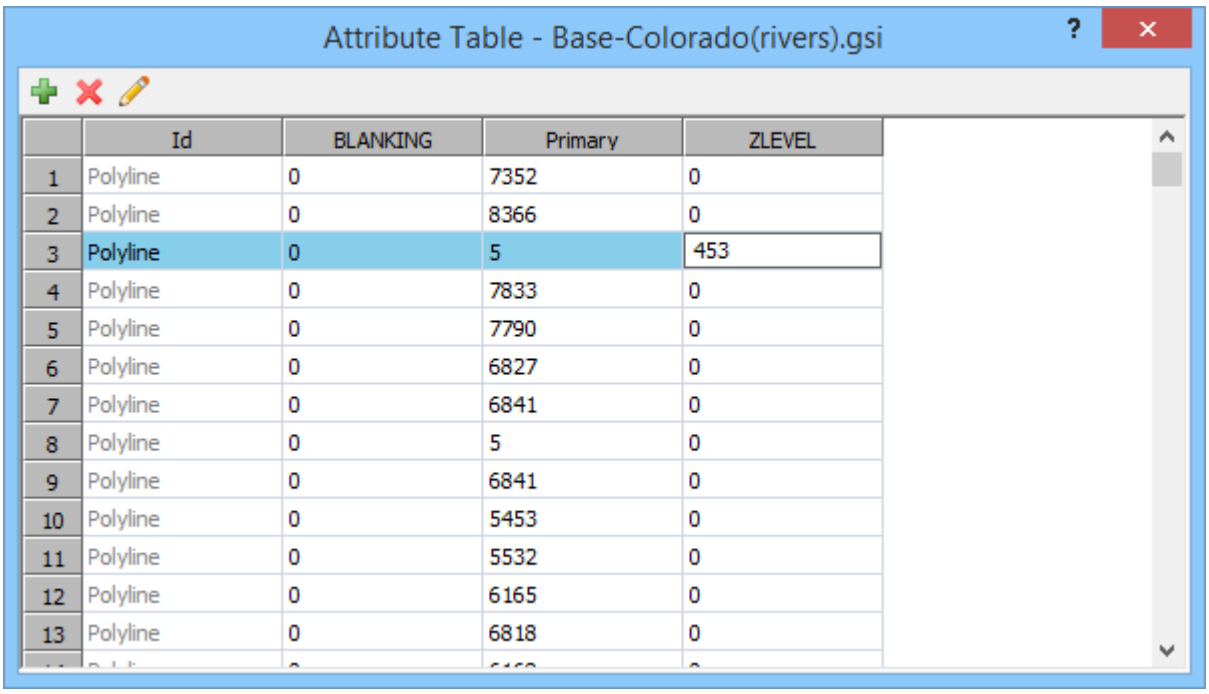
Copying and Pasting in the Attribute Table

Attribute values can be copied from the **Attribute Table** and pasted into **Surfer** or another program. Copy the selected cells by pressing CTRL+C or right clicking while in cell edit mode and selecting *Copy*. The **Home | Clipboard | Copy** command copies the selected object(s) in the plot window. The **Attribute Table** must be selected and the keyboard command or context menu must be used to copy values in the **Attribute Table**.

Text can be pasted into a single cell by double-clicking on the cell and then pressing CTRL+V or right clicking while in cell edit mode and selecting *Paste*. The **Home | Clipboard | Paste** command pastes the clipboard contents in the plot window. The **Attribute Table** must be selected and the keyboard command or context menu must be used to paste values in the **Attribute Table**.

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	BLANKING	Primary	ZLEVEL
1	Polyline	0	7352	0
2	Polyline	0	8366	0
3	Polyline	0	5	453
4	Polyline	0	7833	0
5	Polyline	0	7790	0
6	Polyline	0	6827	0
7	Polyline	0	6841	0
8	Polyline	0	5	0
9	Polyline	0	6841	0
10	Polyline	0	5453	0
11	Polyline	0	5532	0
12	Polyline	0	6165	0
13	Polyline	0	6818	0

*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 *Delete field* 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 *Attribute name* 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.

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**.

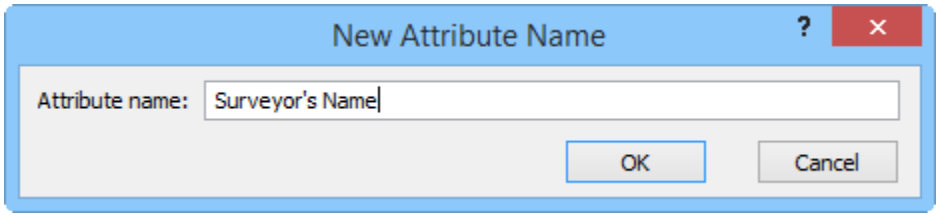
Attribute Fields

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.

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.

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.

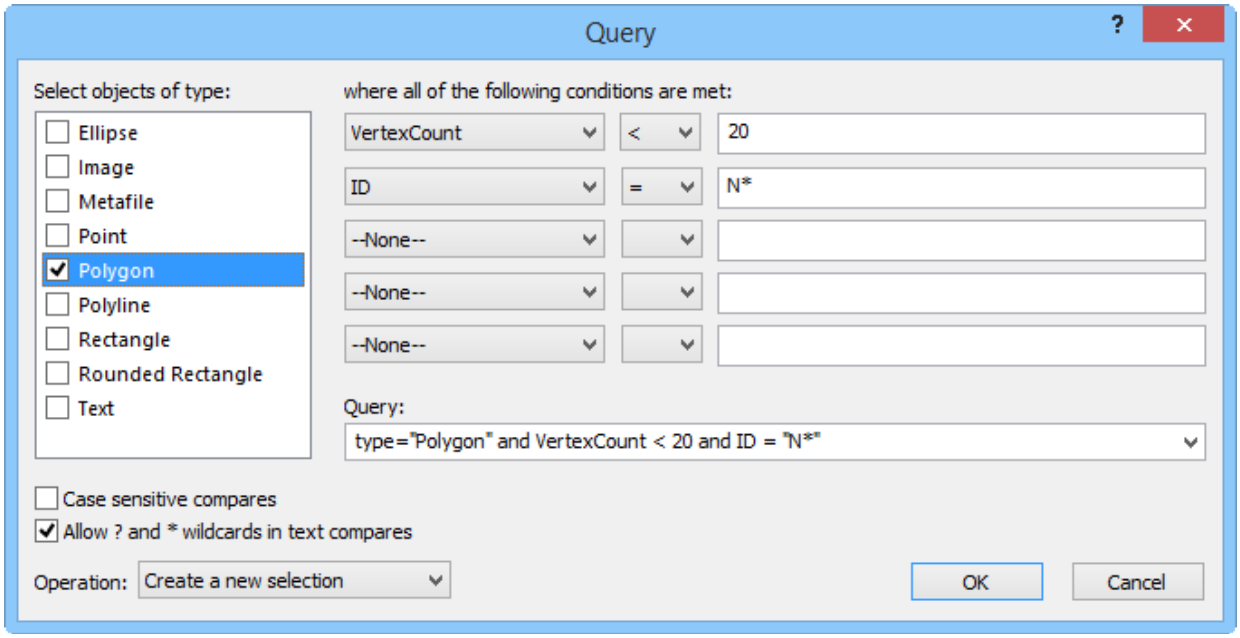
Query Objects

The **Map Tools | Layer Tools | Query Objects** command selects or deselects objects that satisfy a user-defined query. Queries are performed on the objects in a single base layer. Select a base layer in the Contents window, and click Features | Group | Start Editing. The **Start Editing** command can also be accessed in the context menu by right-clicking on the base layer in the **Contents** window.

Click the **Map Tools | Layer Tools | Query Objects** command 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.

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

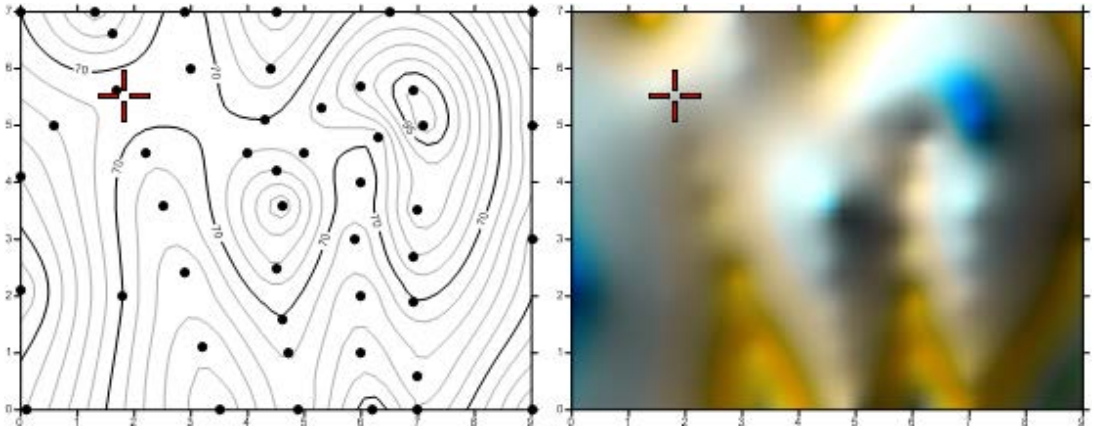
The **Map Tools | Layer Tools | Track Cursor** command allows you to enable or disable the cursor tracking between multiple maps. Tracking is done on the current map frame's coordinate system. Left-click a map to see the location of the cursor click on additional maps. The cursor tracking is updated 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.
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 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 shaded 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

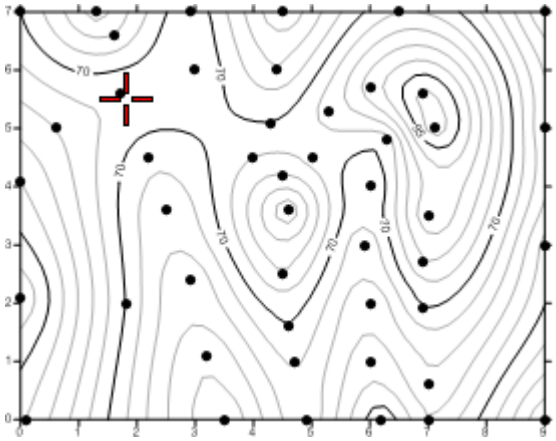
The **Data | Tools | Track Cursor** command allows you to enable or disable the cursor tracking between worksheet data, maps in the plot window and grids in the grid node editor. Select cells in the worksheet to position the tracking cursor on the map or grid. Alternatively, in the plot window or grid 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.

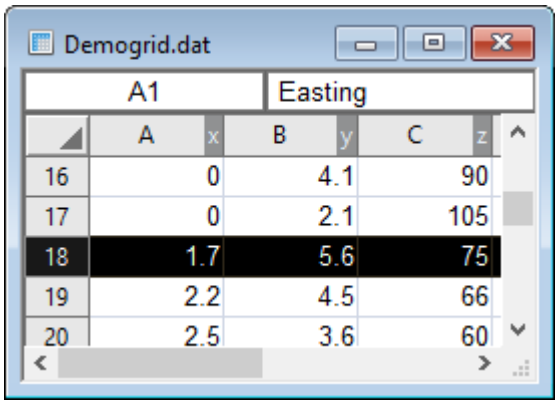
The coordinates in the worksheet row must be included within the map extents for the cursor to appear in the plot window.

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.



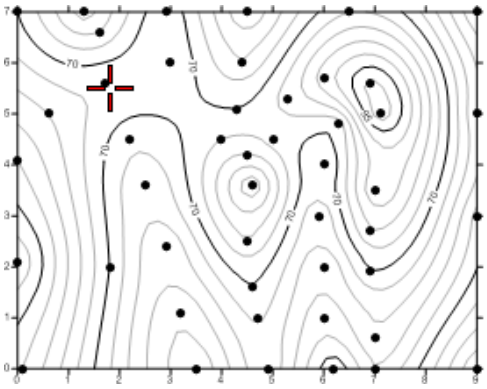
In the worksheet, the nearest post point is highlighted.

Track Cursor in the Grid Editor

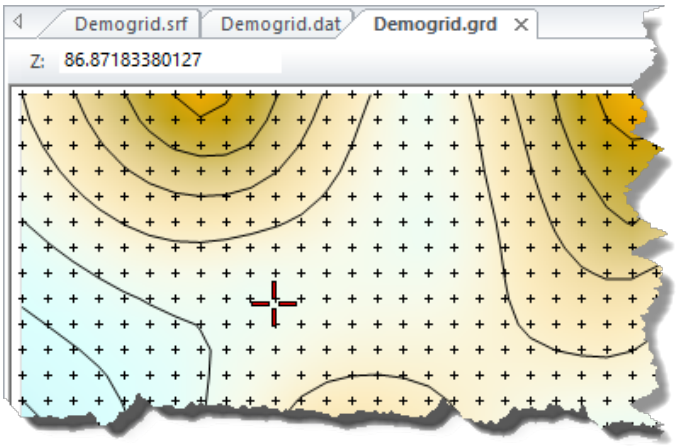
The **Grid Editor | Options | Track Cursor** command allows you to enable or disable the cursor tracking between worksheet data, maps in the plot window and grids in the grid editor. Click in the grid editor to position the tracking cursor. When clicking in the grid editor, the nearest grid node will 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. 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 26 - 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 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. Click on the map layer to select it.
3. In the Properties window, click on the Coordinate System tab.
4. 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*.
5. 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.
6. Click on the *Change* button next to *Coordinate System* to set the desired target coordinate system. When finished, click *OK*.
7. 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 projection 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 - Map Layer

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.

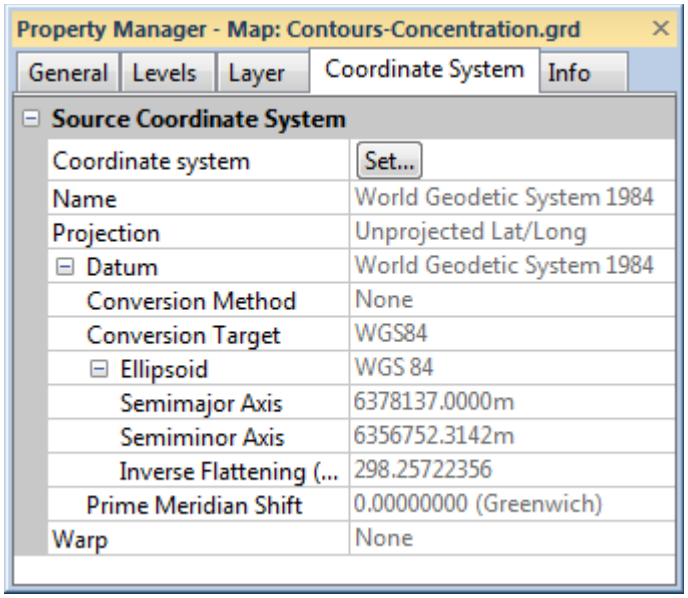
When a surface layer or wireframe layer is added to an existing map, the target coordinate system of the existing map must first be set to the same coordinate system as the surface or wireframe map to be added. To determine the coordinate system for the surface or wireframe layer, you may need to create 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.

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 - Map

Maps can be displayed in any coordinate system. The map is displayed in the coordinate system defined as the *Target Coordinate System*. A coordinate system normally has a defined projection and datum. When a map layer uses a different a different Source Coordinate System than the map's *Target Coordinate System*, the map layer is converted to the map's *Target Coordinate System*.

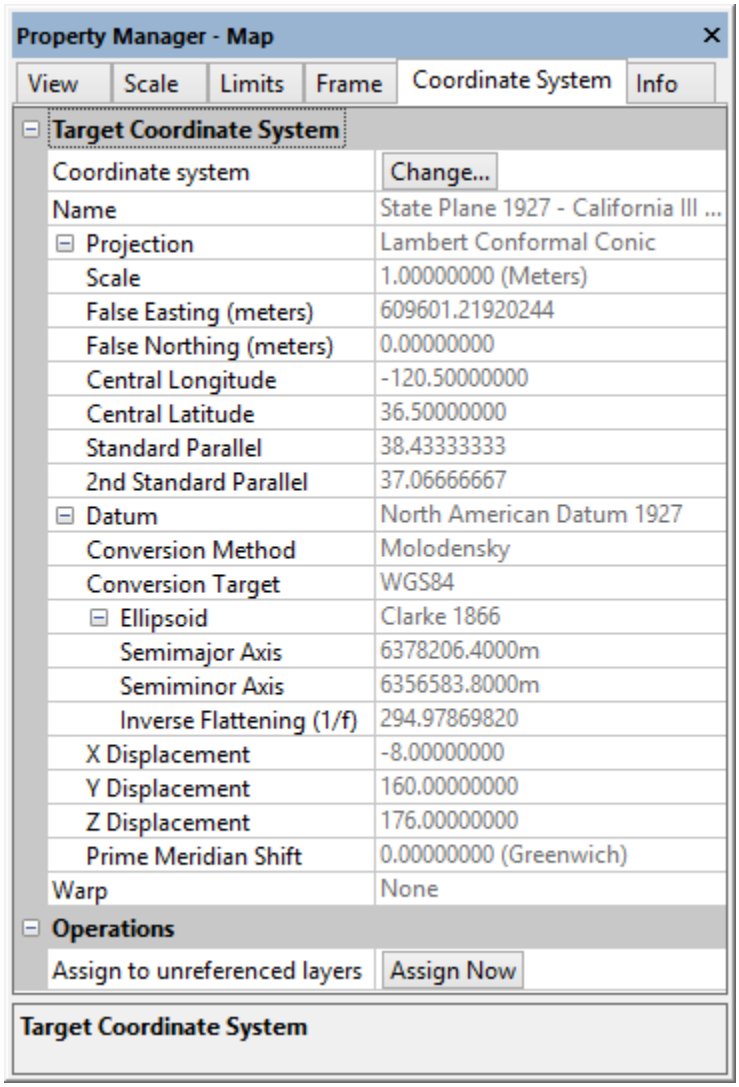
3D surface layers and wireframe layers do not have a coordinate system associated with them and cannot be converted to a different coordinate system. For this reason, there is not a **Coordinate System** tab in the **Properties** window for the surface layer, the wireframe layer, the *Map* object, or 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?

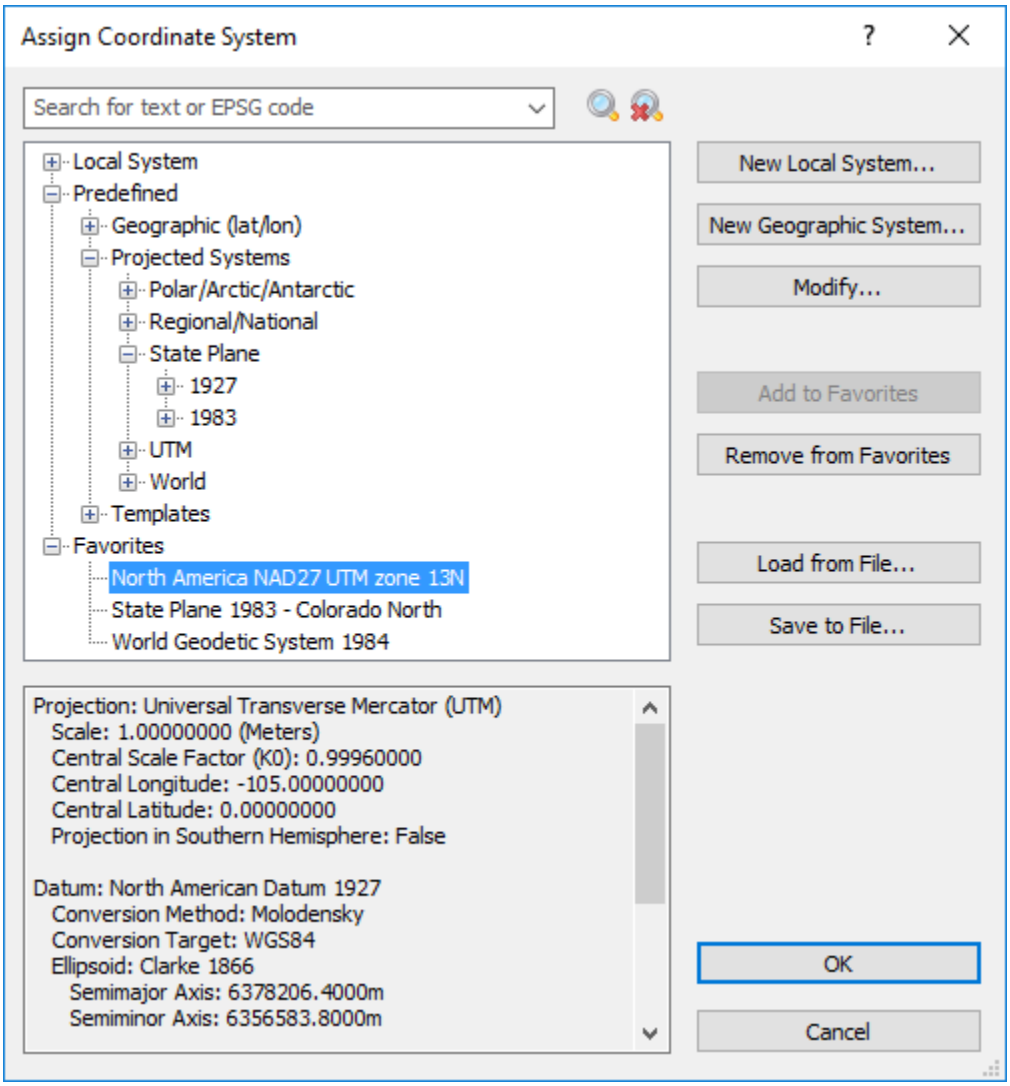
A: A good starting place is the references on the Projection References page. Many other good resources are available either online or in printed documentation.

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 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.

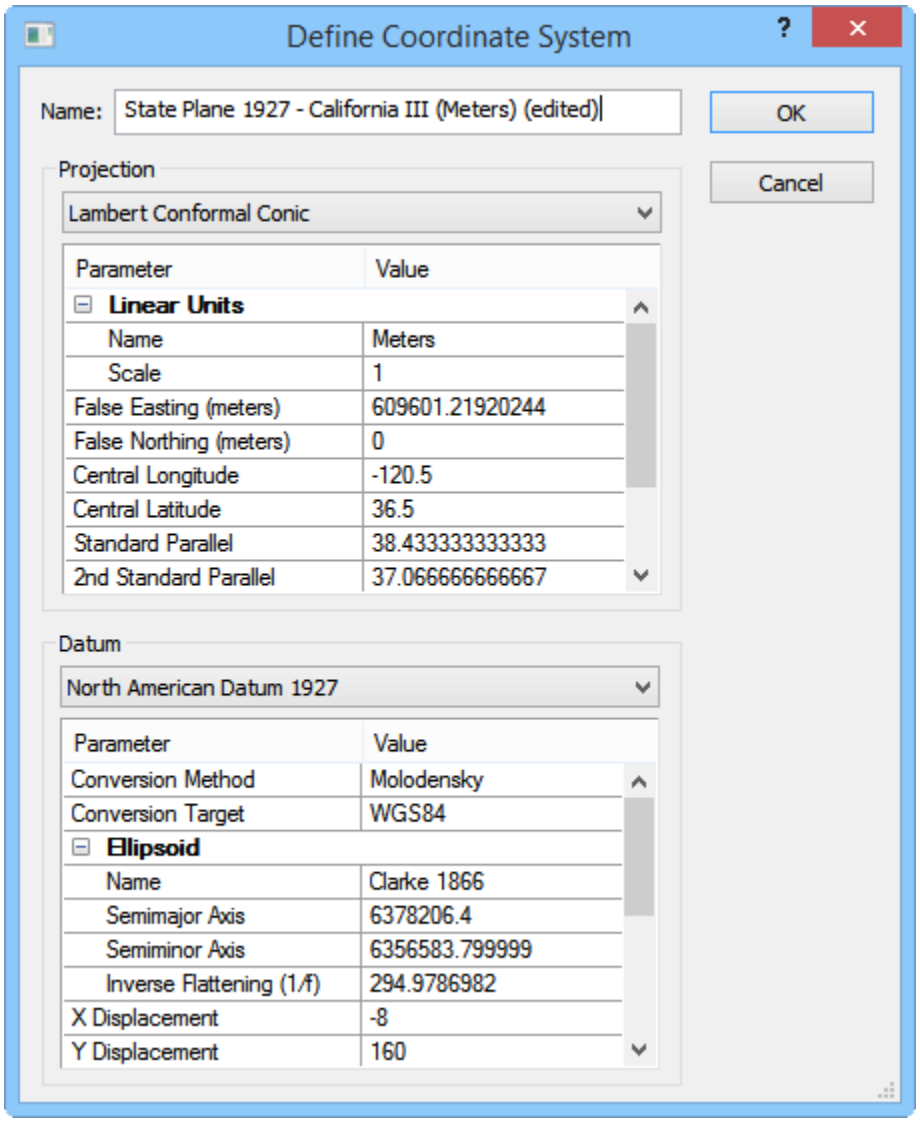
Example 3: Saving a Custom Coordinate System

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

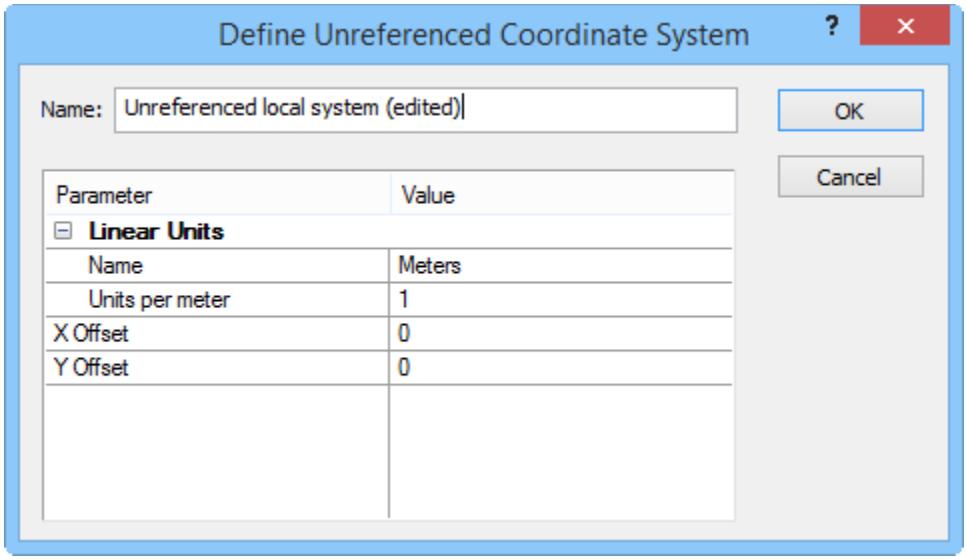
1. In the **Assign Coordinate System** dialog, click the *New Geographic System* button to open the **Define Coordinate System** dialog.
2. Change the Name to Lambert Conformal Conic Custom.
3. From the Projection list, select Lambert Conformal Conic.
4. Make any necessary changes to the *Parameter* 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 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 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.

Introduction to Map Projections

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.

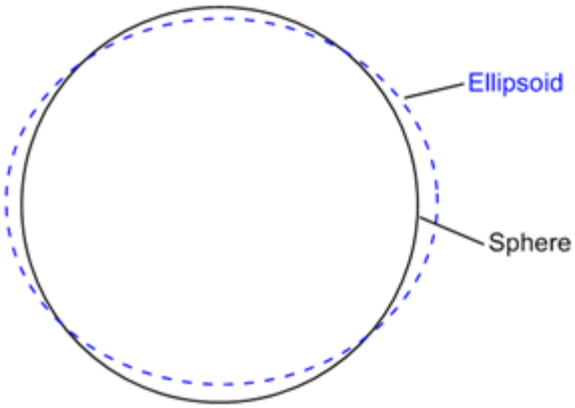
Surfer supports several of the projections that are most often used in modern cartography and related fields. The **Surfer** worksheet allows you to load data and define the projection using the Data | Coordinate System | Assign Coordinate System. You can then convert the data to another coordinate system 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. 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.

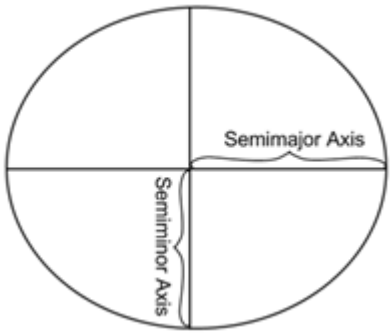
However, the Earth is actually somewhat ellipsoidal (or egg-shaped), approximately 1/300th wider than it is tall, assuming a vertical orientation with respect to the axis of rotation. This shape needs to be taken into account to produce larger scale maps of any significant accuracy.



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.

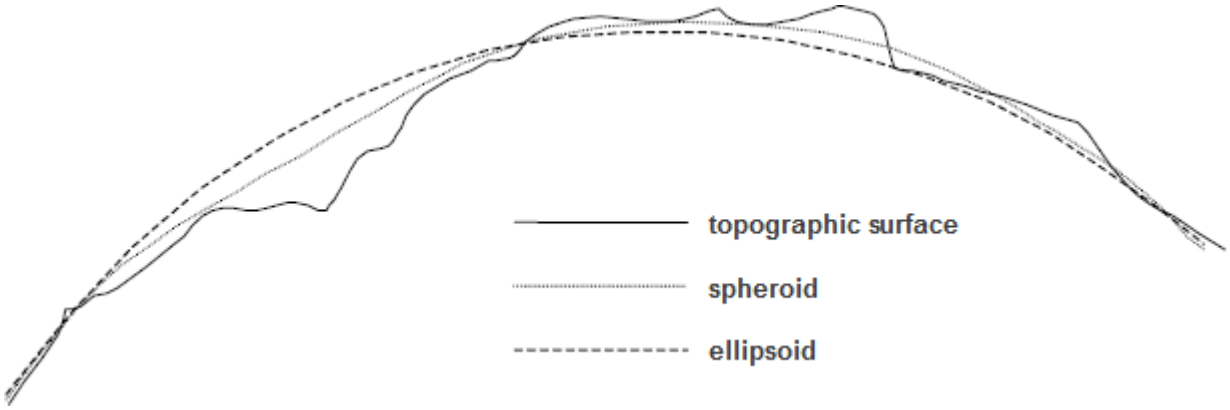
Prime Meridian

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 chaning 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	• 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

	project coordinates in polar regions.	
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 Hotine Oblique Mercator, depending on the zone.

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.	In order for a projection to be conformal, however, consistency in the surface areas, shapes, and/or scales across the	Hotine Oblique Mercator, Lambert Conformal Conic, Mercator, Oblique Mercator, State Plane

	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.	map must be sacrificed. An area enclosed by a series of arcs may be greatly distorted.	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.

Type of 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

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

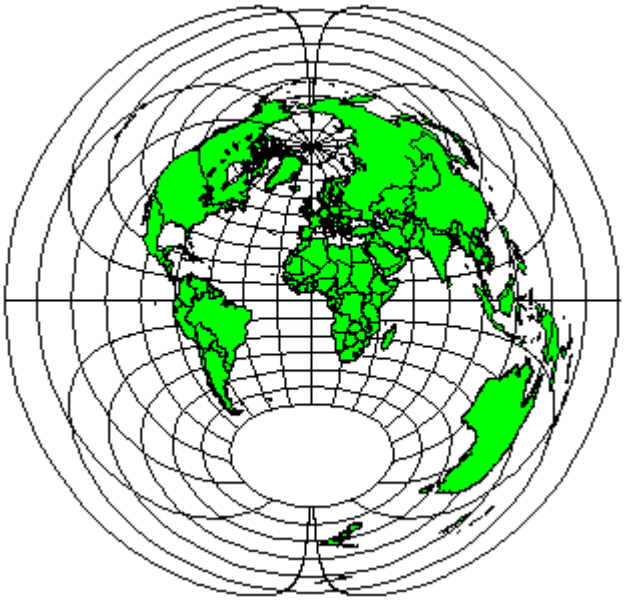
Projection Characteristics

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

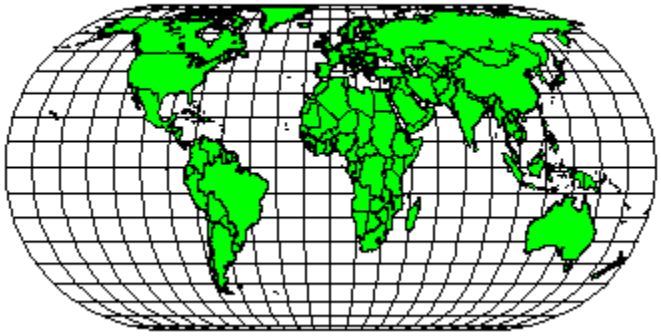
The Cassini projection is a cylindrical projection. The scale is accurate along the central longitude and along latitude lines perpendicular to the central longitude. This projection is useful for high-aspect regions (regions taller than they are wide).

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

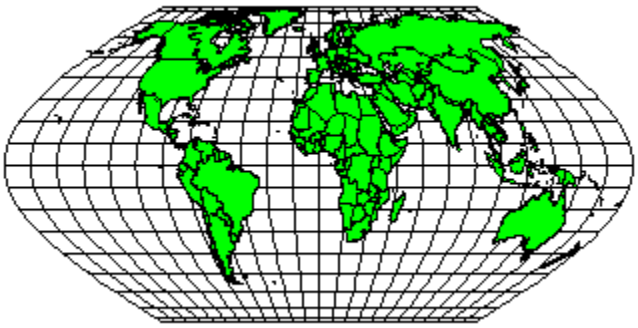
The scale is constant along any given parallel and accurate along the parallels 40°30' north and south in the Eckert IV projection. This is a pseudo-cylindrical, equal area projection with the central longitude half the length of the equator. The poles are represented by lines half the distance of the 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

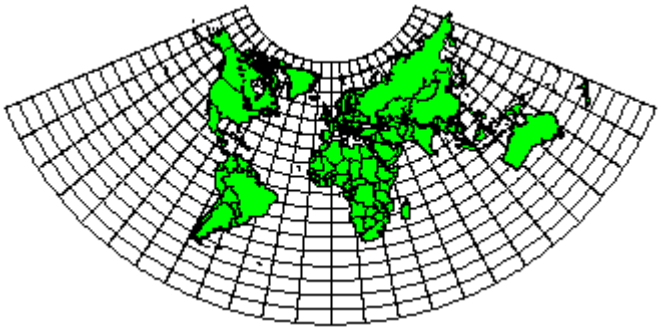
The scale is constant along any given parallel, and accurate along the parallels 49°16' north and south in the Eckert VI projection. This is a pseudo-cylindrical, equal area projection with the central longitude half the length of the equator. The poles are represented by lines half the distance of the 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

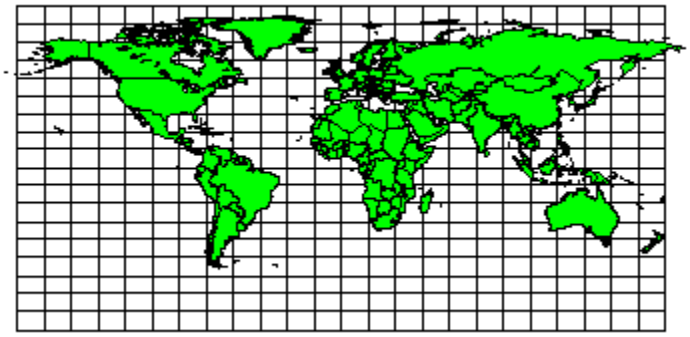
There is no distortion in scale, shape, or area along the standard parallels in an Equidistant Conic projection. Scale is true along all longitudes and along the standard parallels. Direction is locally true along the standard parallels. This projection is useful for maps of low-aspect regions (regions 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

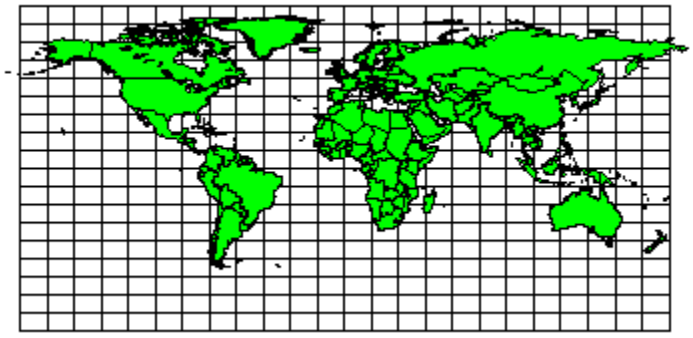
Projection Characteristics

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

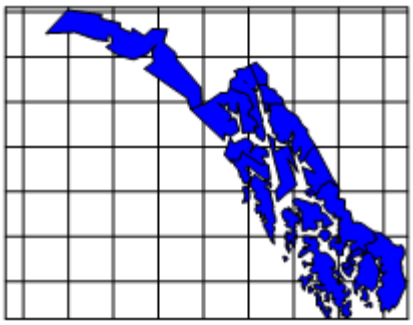
Projection Characteristics

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

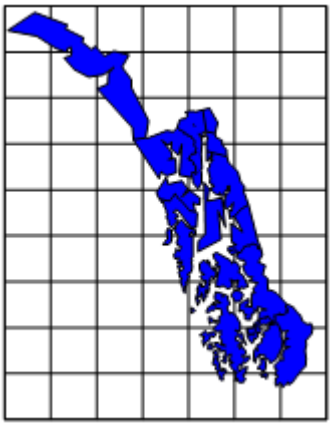
The Hotine Oblique Mercator projection is a cylindrical, conformal projection. The scale is accurate along the chosen central line by the longitude/latitude settings below. This projection is useful for oblique areas (areas that do not follow lines of latitude and longitude), such as the Alaska panhandle 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

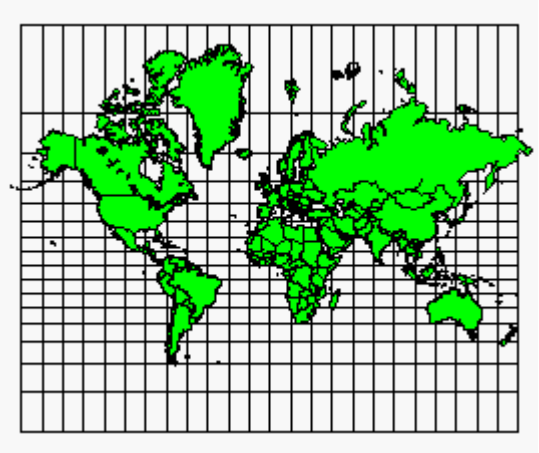
In a Lambert Conformal Conic projection, scale is constant along any given parallel and accurate along the specified standard parallels. Scale is the same in all directions at any given point. This projection is useful for equal area maps of low-aspect regions (regions that are wider than they are tall). 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

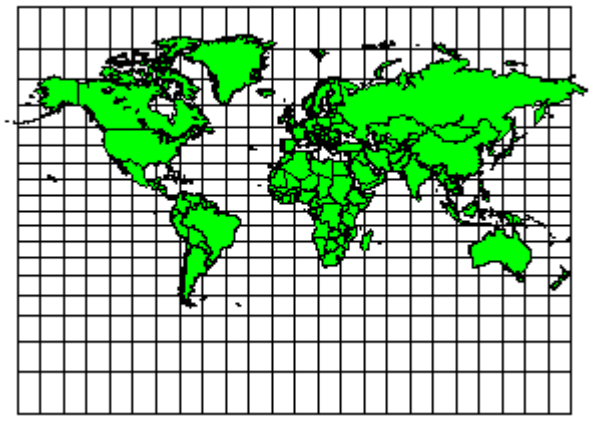
The Mercator projection is a cylindrical projection and it is conformal. In a Mercator projection, scale is constant along any given parallel and accurate along the specified center latitude. Scale is

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

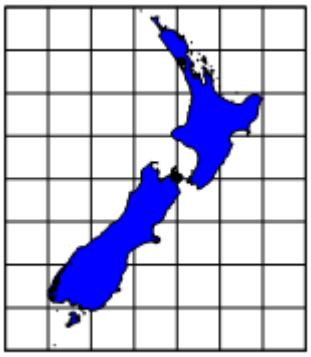
Projection Characteristics

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

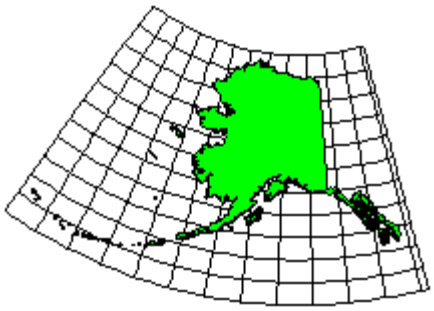
Projection Characteristics

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

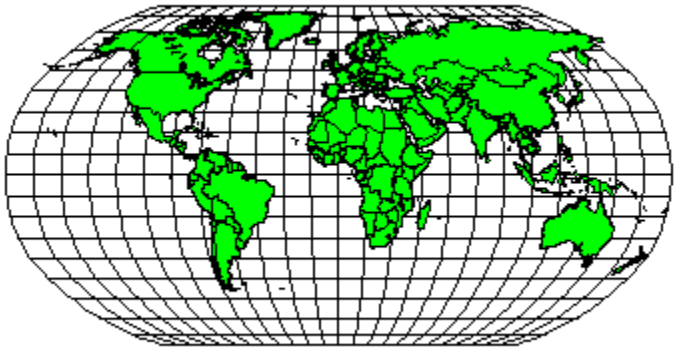
The Polyconic projection is useful for maps of continental or smaller regions. Generally, this projection is not used for world maps due to extreme distortion at any significant distance from the center of the projection. Only the central meridian is distortion-free. Notice in the sample map shown above 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

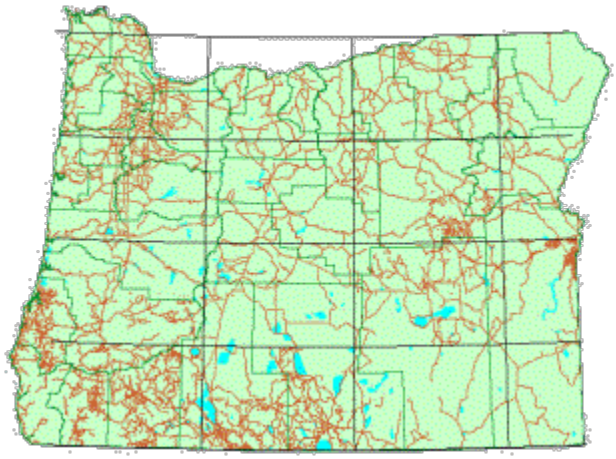
Projection Characteristics

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

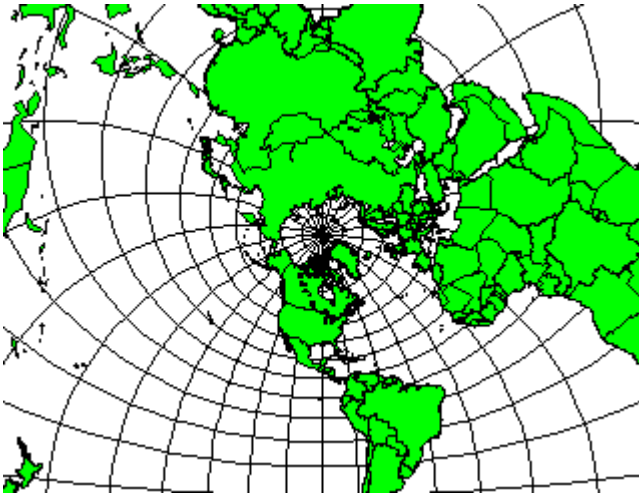
The State Plane Coordinate System (SPCS) divides the United States into a number of zones, and defines a different projection for each zone such that a suitable map of any given zone is plotted. SPCS is used mainly for intrastate views such as county or parish maps.

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

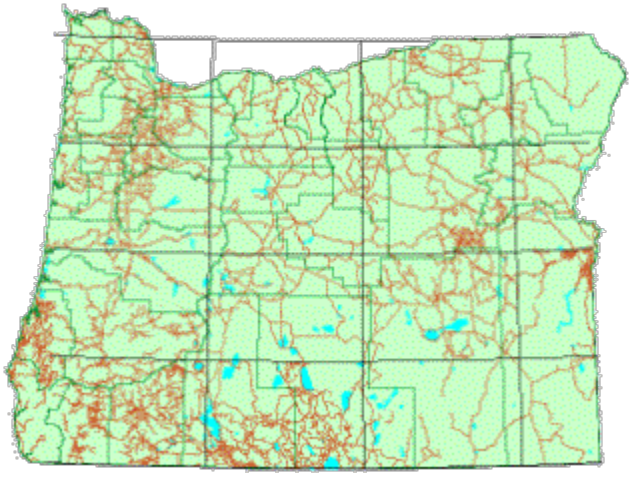
The Stereographic projection is an azimuthal, conformal projection. In a Stereographic projection, scale is constant along any circle circumscribed around the center of the projection. Distortion is minimal at the center and becomes extreme with distance from the center. Generally, this projection is 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

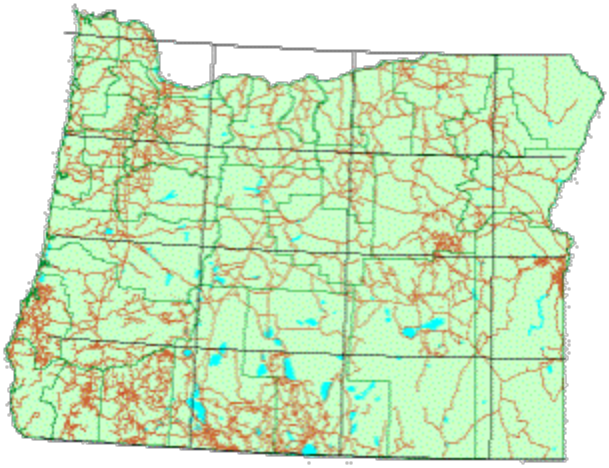
Projection Characteristics

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.

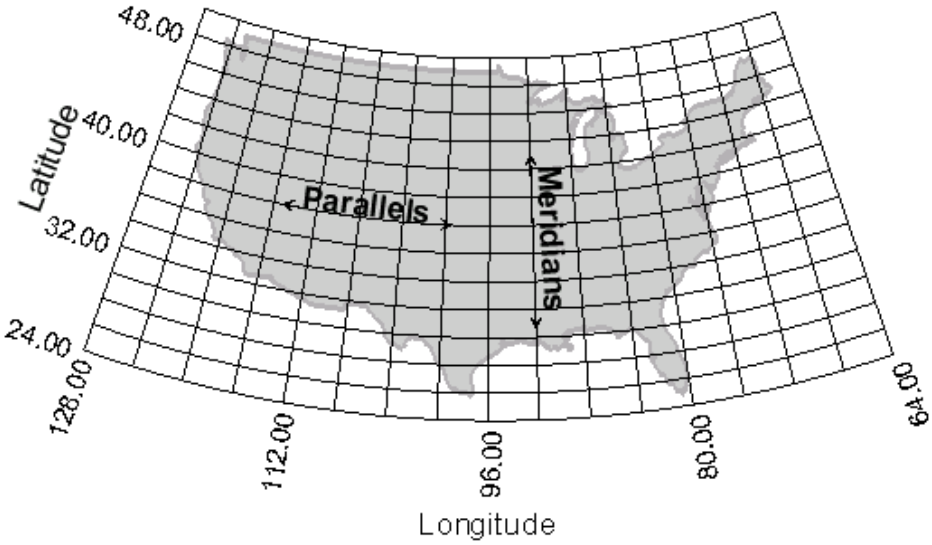
Surfer .SRF file format retains all of the information in a map, including projection information.

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.

Surfer only 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. Alternatively, you can manuall convert DMS values to decimal degrees.

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 15*).

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 27 - Creating and Editing Features

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
Start/Stop Editing	Edit features within a group or base layer
Group	Combine objects into a group
Ungroup	Split grouped objects
Change Type	Polyline to Polygon, Polygon to Polyline, Polyline to Points, and Points to Polyline converts selected objects between object types.
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
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
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** or **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

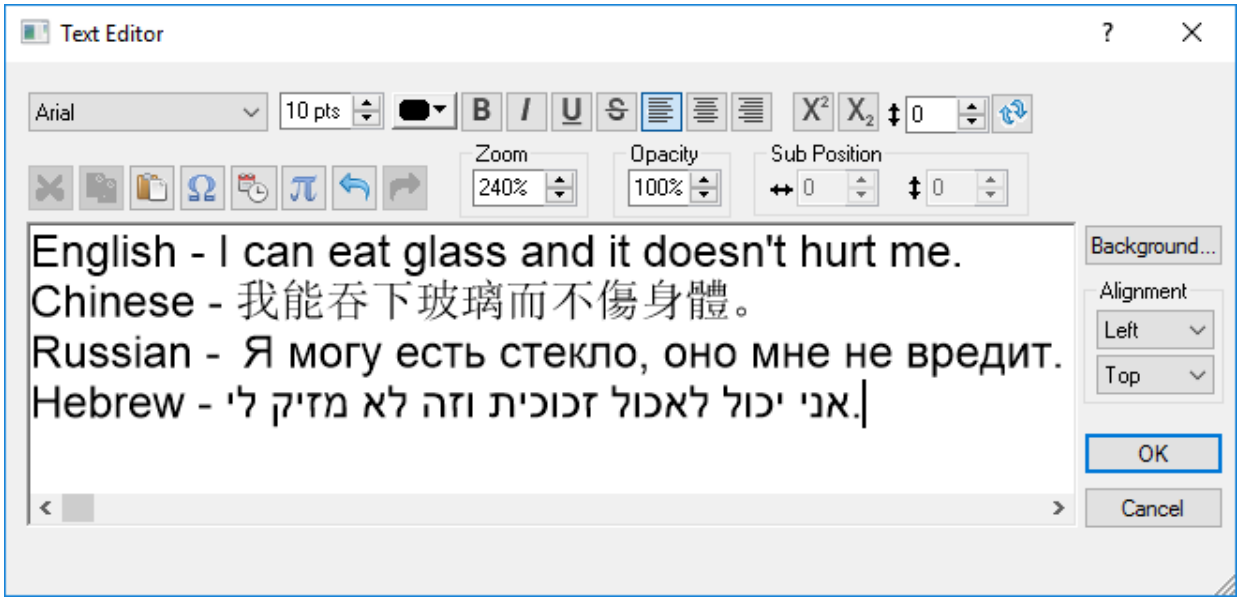
The *Font*, *Size*, *Color, Opacity*, and *Style* can be set for text blocks in the **Text Editor** dialog when typing text. These options can be set in the Properties window in the *Font Properties* section after the text is drawn.

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

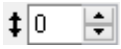
The top row of tools and buttons change the appearance of currently selected text. Additionally, subsequent text will appear with the selected options.

- 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:

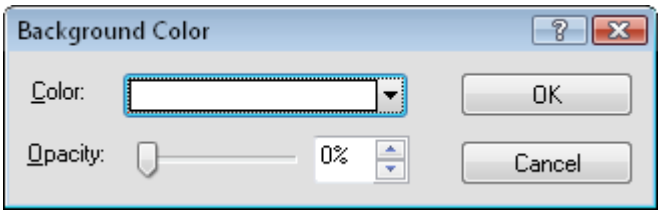
- You can cut , copy , or paste selected text, or paste objects from the clipboard into the **Text Editor**.
- Click the *Insert Symbol* button to open the Symbol Properties dialog and add a symbol to the text block.
- Click the *Insert Date/Time* button to open the Date/Time Format Builder dialog and 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 ***Background 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 Alignment* 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 button.

OK or Cancel

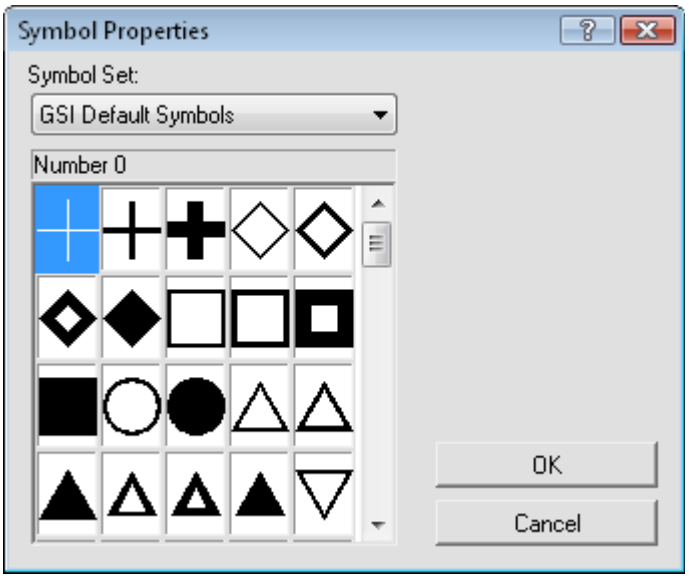
Click the *OK* button to save your changes and close the **Text Editor**. Click the *Cancel* button to exit the **Text Editor** without saving your changes.

Symbol Properties Dialog

The **Symbol Properties** 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.

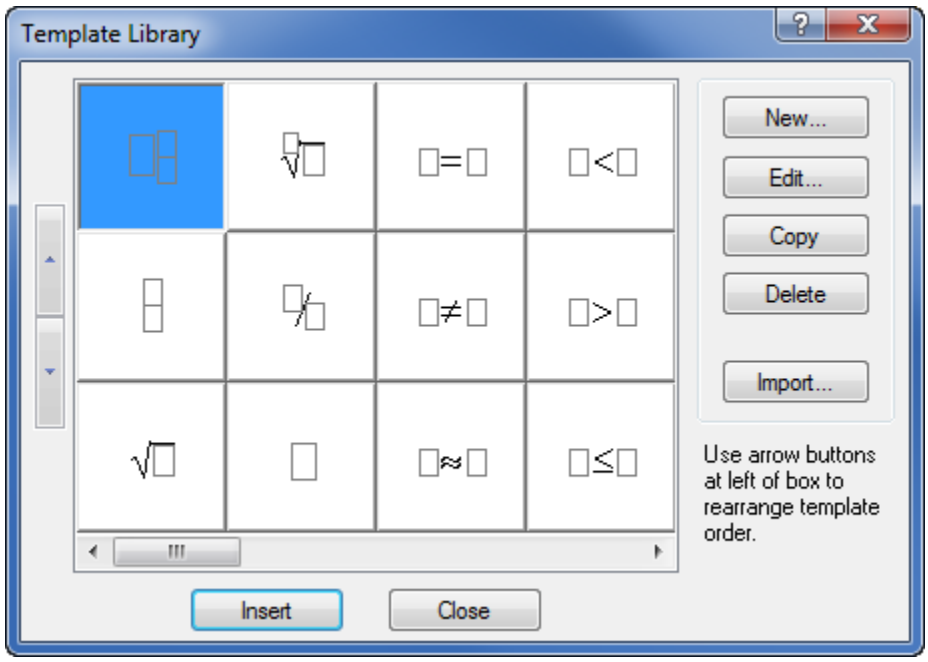
The symbol index is the symbol or glyph number as it appears in the title bar above the palette and adjacent to the symbol in the **Properties** window.

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

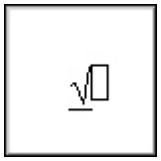
Click the *Insert* button to insert a template into the **Text Editor**.

Close

Click the *Close* button to close the template library without inserting a template into the **Text Editor**.

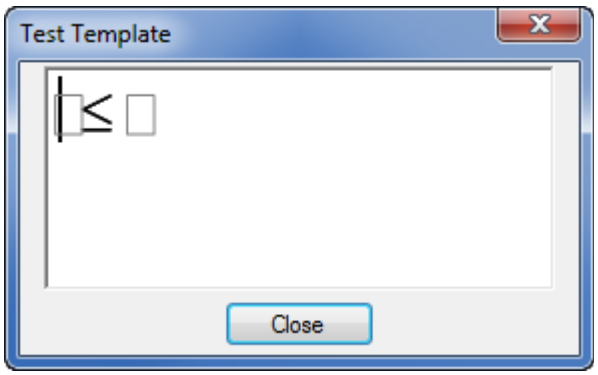
Example

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.

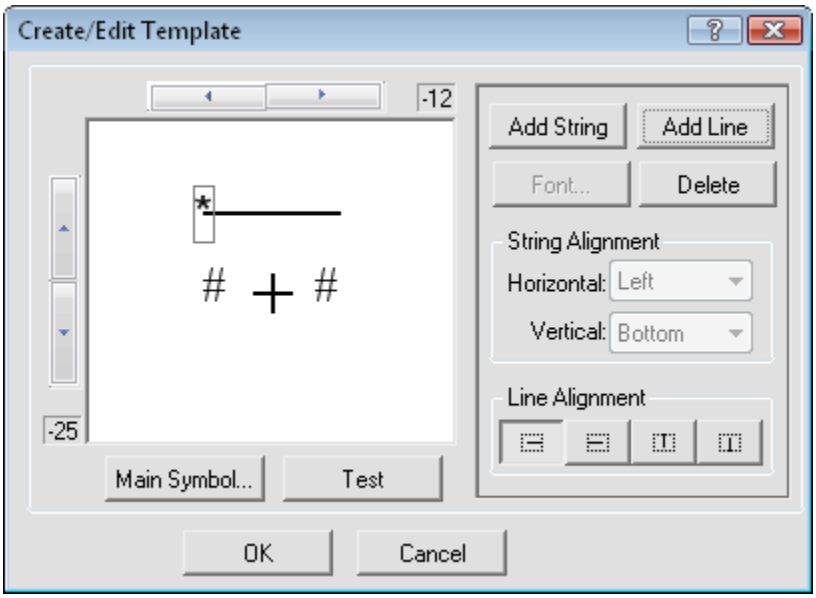


*The **Test Template** dialog displays the final template.*

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

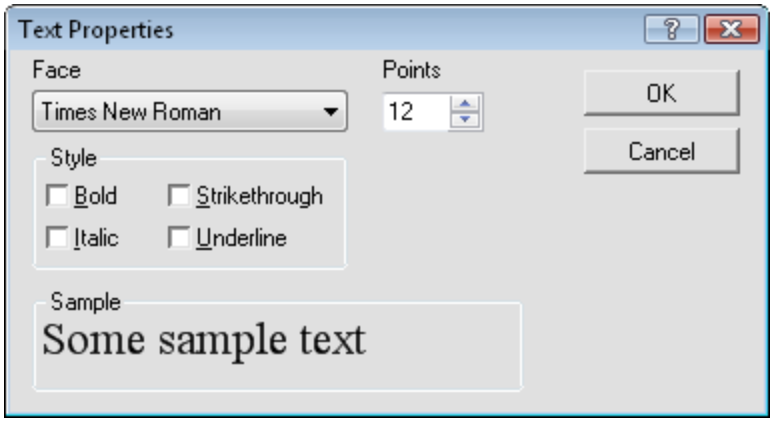
Click the *Add String* button to add a text string box. Text string boxes in the templates allow numbers or letters to be entered into the template in the **Text Editor**.

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.

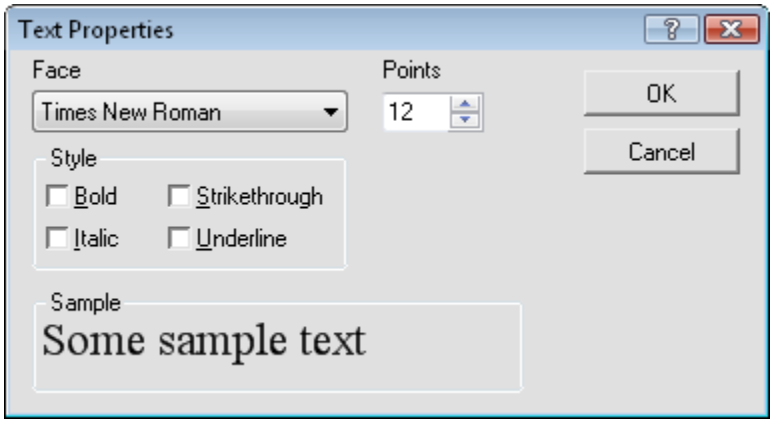
Test

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.*

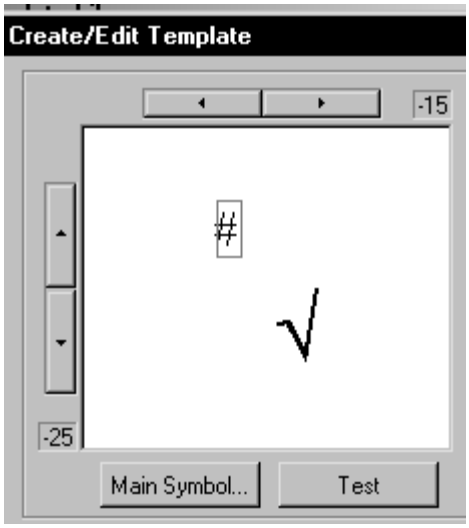
Text Editor Template Editor Example

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.

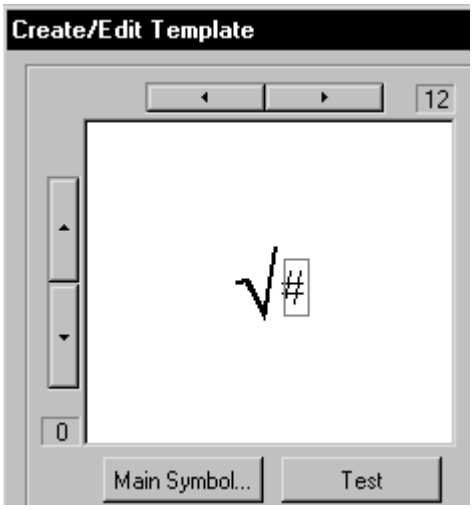
1. In the plot window, click the **Home | Insert | Text** command.
2. Click on the screen where the text should be located and the Text Editor opens.
3. Click the button in the **Text Editor** to open the Template Library dialog.
4. Click *New* in the **Template Library** dialog. The Symbol Properties 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.

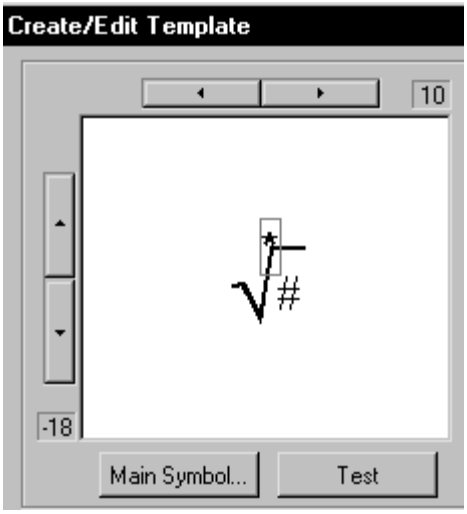


Use the up/down and left/right arrows to move the pound sign to the right of the square root symbol.

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.



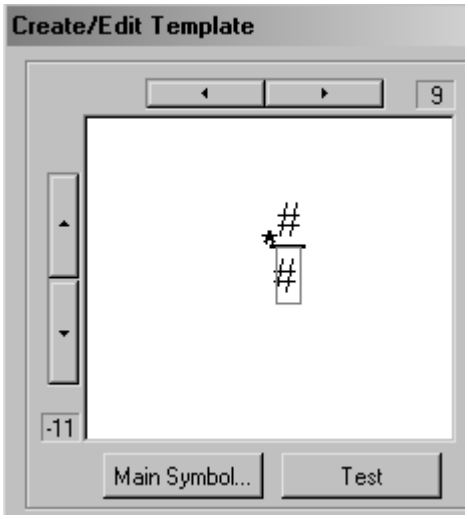
Position the line so it is next to the square root symbol.

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:

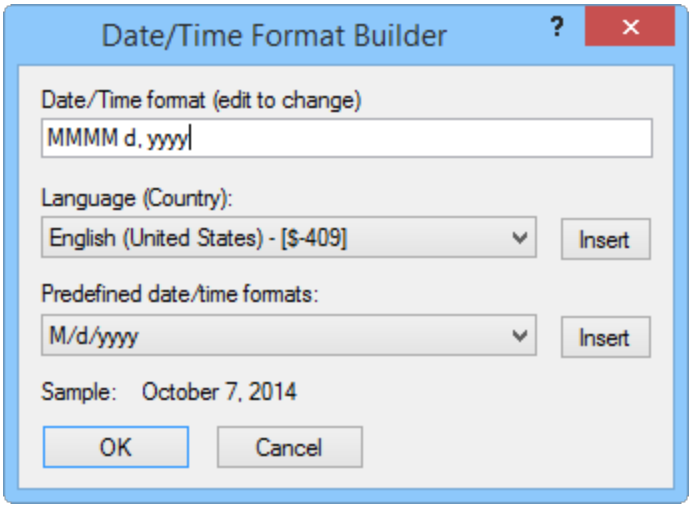
1. If the template library is not open, in the **Text Editor** click the button to open the **Template Library**.
2. Highlight the new square root template in the **Template Library**, and click *Insert*.
3. You will be asked to save the changes in the template library. Click *Yes.* The square root symbol 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 the **Create/Edit Template** dialog. Use the *Sub Position* boxes in the **Text Editor** to 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 dialog 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 section 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 instruction 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** or **Features | Insert | Polyline** command to draw a polyline.

To draw a polyline:

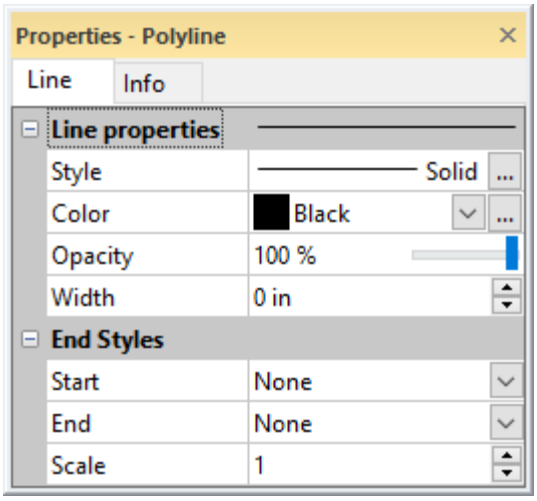
1. Click the **Home | Insert | Polyline** command to begin drawing a polyline.
2. The cursor changes to a cross hair cursor to indicate drawing mode.
3. Move the cursor over the location for the start of the polyline and click the left mouse button.
4. Move the cursor to the next position along the line 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.
- 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.



*Customize polyline properties in the **Properties** window.*

Line Page

The **Line** page allows you to customize the polyline *Style*, *Color*, *Opacity*, *Width*, and *End Styles*. The *Style*, *Color*, *Width*, *Opacity*, and *Sample* are identical to the Line Properties. The *End Styles* section is unique to polyline properties.

End Styles

The ends of the lines can have arrowheads on them as defined in the *End Styles* section. The *Start* style is placed at the first vertex of the line. The *End* style is placed at the last vertex. The *Scale* determines the size of the arrowhead. The *End Styles* section is not present and/or active when editing some objects.

Info Page

The Info page for a polyline displays the *Coordinate System*, *Number of vertices*, and *Length*.

Polygon

Click the **Home | Insert | Polygon** or **Features | Insert | Polygon** command to draw an irregularly shaped area. Polygons must have at least three vertices (points).

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.
- 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 Page

The **Fill** page contains options that allow you to customize the polygon Fill properties.

Line Page

The **Line** page allows you to customize the polygon Line properties.

Info Page

The Info page displays the Coordinate System, Total number of vertices, Number of sub-polygons (rings), Perimeter, and Area.

Point

Points are markers that use symbols to indicate 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** or **Features | Insert | Point** command to place symbols in the plot document.

The *Symbol Set*, *Size*, *Fill color*, *Fill opacity*, *Line color*, *Line 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 Page

The Symbol page allows you to edit the *Symbol, Symbol Set, Size, Fill color, Fill opacity, Line color,* and *Line opacity* for the symbol.

Info Page

The Info page for a symbol displays the *Coordinate System* and *Position* of the symbol.

Spline Polyline

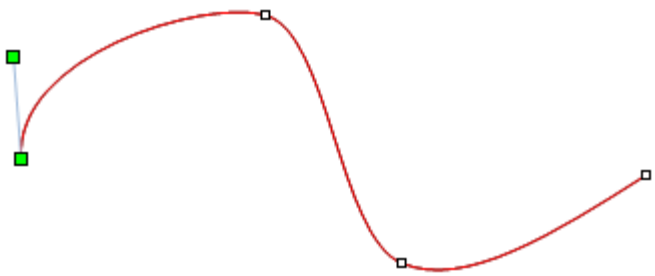
A spline polyline is a smooth, flowing polyline with no sharp or distinct angles. Click the **Home | Insert | Spline Polyline** or **Features | Insert | Spline Polyline** command to draw a spline polyline. You can change the line style, color, opacity, width, and end styles for the spline polyline display.

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.

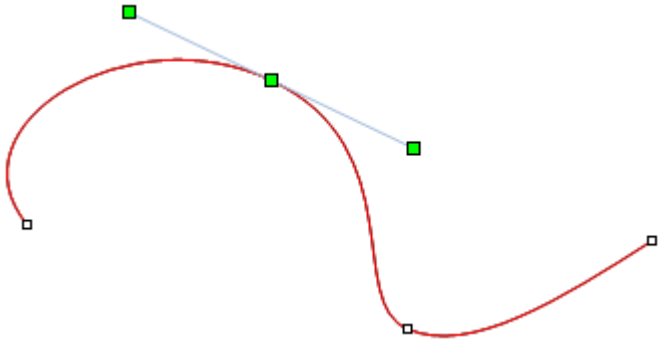
To edit a Spline Polyline

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 contain the following pages.

Line Page

The **Line** page allows you to customize the spline polyline Line properties.

Info Page

The Info page for a spline polyline displays the *Coordinate System, Number of curves,* and *Length.*

Range Ring

Click the **Home | Insert | Range Ring** or **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 Page

The General page contains general options for the range ring display, such as number, size, and shape.

Fill Page

The Fill page contains options for the range ring fill properties. 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.

Range ring fill opacities are additive. Multiple rings can be used with a semi-transparent fill to create a bulls-eye effect. For example, consider a range ring with three rings and a fill opacity of 20%. The largest ring will have an effective opacity of 20%. The middle ring is stacked on the large ring, and therefore has an effective opacity of 36%, i.e. $1 - 0.80^2$. The smallest ring is stacked on both larger rings and has an effective opacity of 48.8%, i.e. $1 - 0.80^3$.

Line Page

The Line page contains options for the range ring line properties. 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 Page

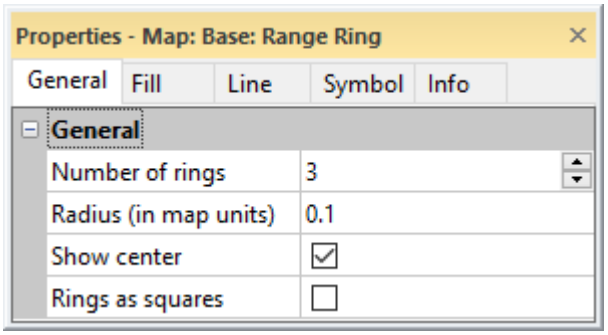
The Symbol page contains options for the range ring center symbol properties. The *Show center* property on the **General** page must be checked to display a symbol in the center of the range ring.

Info Page

The Info page displays general information about the range ring and its attributes.

General Page - Range Ring

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** or **Features | Insert | Rectangle** command to create a rectangle or square in the plot document.

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 Page

The **Fill** page allows you to customize the rectangle Fill properties.

Line Page

The **Line** page allows you to customize the rectangle Line properties.

Info Page

The Info page for a rectangle displays the *Coordinate System, Perimeter,* and *Area.*

Rounded Rectangle

You can use the **Home | Insert | Rounded Rectangle** or **Features | Insert | Rounded Rectangle** command to create a rounded rectangle or square in the plot document.

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. You can draw a rounded rectangle out from the center rather than corner to corner by holding down the SHIFT key.
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.

To draw a rounded square:

- Hold down the CTRL key while dragging the mouse to draw a rounded square rather than a rounded rectangle.
- Hold down the SHIFT and CTRL keys while dragging the mouse to draw the rounded square from a center point rather than from end to end.

Drawing Tips

- 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 Page

The **Fill** page allows you to customize the rounded rectangle Fill properties.

Line Page

The **Line** page allows you to customize the rounded rectangle Line properties.

Info Page

The Info page for a rounded rectangle displays the *Coordinate System*, *Perimeter,* and *Area.*

Ellipse

You can use the **Home | Insert | Ellipse** or **Features | Insert | Ellipse** command to create an ellipse or circle in the plot document.

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. You can draw an ellipse out from the center rather than corner to corner by holding down the SHIFT key.
5. Release the left mouse button when the ellipse is the preferred size and shape.
6. Press the ESC key to end draw mode.

To draw a circle:

- 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.

Drawing Tips

- 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 Page

The **Fill** page allows you to customize the ellipse Fill properties.

Line Page

The **Line** page allows you to customize the ellipse Line properties.

Info Page

The Info page for an ellipse displays the *Coordinate System, Perimeter,* and *Area.*

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.

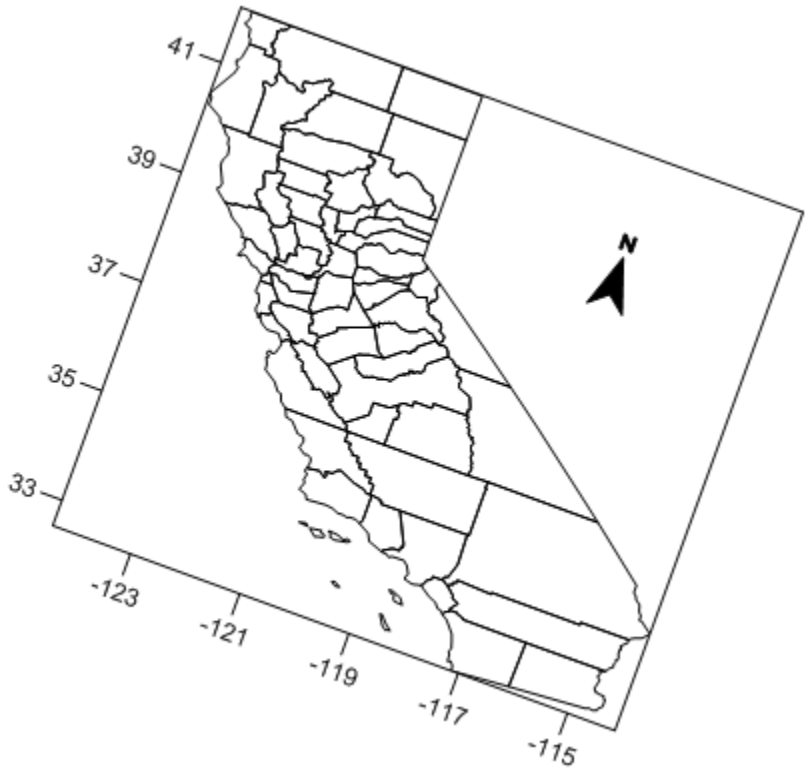
This North arrow is rotated to 25 degrees. The

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.

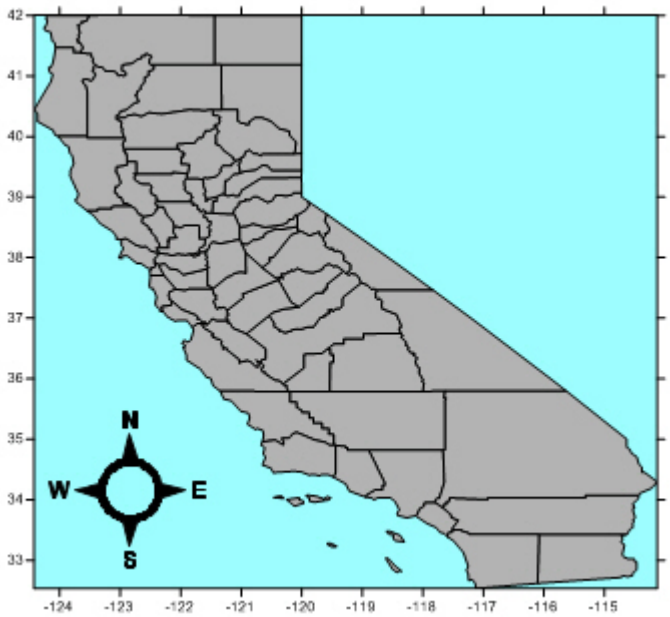


This North arrow is a post map layer on a base map of California. The GSI North Arrows symbol set Number 6 is used. The map is rotated to -20 degrees.

To Add a North Arrow as a Base Map

This method uses the **Start Editing** command to enter the base map. Once you are in the base map group, you can draw a North arrow symbol that will be included in the base map coordinates. Exit the base map group to complete the process. 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. Select the *Base* map layer in the **Contents** window, and use the Start Editing command. Alternatively, right-click on the *Base* map layer in the **Contents** window and select **Start Editing** from the context menu. Be sure to select the "Base" map layer, not the "Map" object.
3. Create a new point object using the Home | Insert | Point command.
4. 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.
5. After the point is drawn, click on the symbol to open the symbol properties in the **Properties** window.
6. Change the *Symbol Set* to either *GSI Default Symbols* or *GSI North Arrows*.
7. Select a North arrow Symbol (i.e. GSI Default Symbols Number 64 - 71, or GSI North Arrows Number 1 - 25).
8. 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.
9. 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.
10. Use the Stop Editing command to exit the base map group.



*This North arrow was added to the base map with the **Start Editing** command. The base map polygons are filled in gray. The base map background is blue.*

Polyline to Polygon

Click the **Features | Change Type | Change Type | 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 Type | Change Type | 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.

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 | Change Type | Polyline to Polygon** command. The polyline is changed to a polygon. Click the **Features | Group | Stop Editing** to end object editing mode in the base map.

Polygon to Polyline

Click the **Features | Change Type | Change Type | 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.

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 | Change Type | Polygon to Polyline** command. The polygon is changed to a polyline. Click the **Features | Group | Stop Editing** to end object editing mode in the base map.

Polyline to Points

The **Features | Change Type | Change Type | 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.

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 | Change Type | Polyline to Points** command. The polyline is changed to points. Click the **Features | Group | Stop Editing** to end object editing mode in the base map.

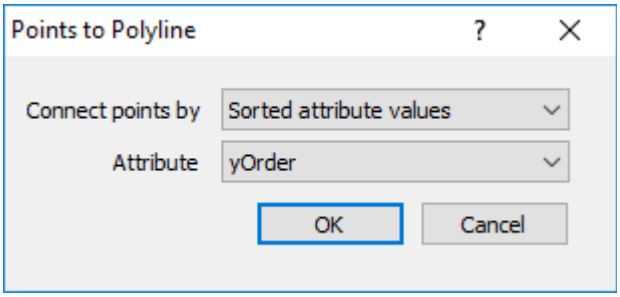
Points to Polyline

The **Features | Change Type | Change Type | 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 Type | Change Type | 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 to move, add, and delete vertices within a selected polyline, polygon, or spline polyline. Objects such as metafiles, composites, base maps, polygons, and polylines 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

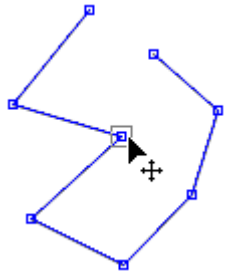
To exit reshape mode, press the ESC key on the keyboard. Alternatively, select another command mode.

Select Vertices

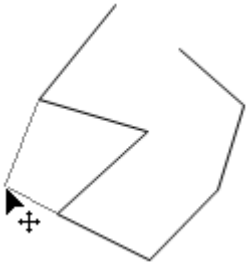
- Left-click on a vertex to select it. The selected vertex is indicated by a solid green square.
- To select the first vertex, press the HOME key. To select the last vertex, press the END key.
- To shift the selected vertices forward by one position, press the TAB key. To shift the selected vertices 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

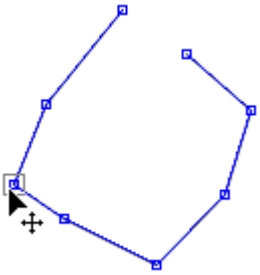
Once a vertex or vertices are selected and the cursor display is a , hold the left mouse button and drag the vertex to a new location. Release the left mouse button to place the vertex in the new location.



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

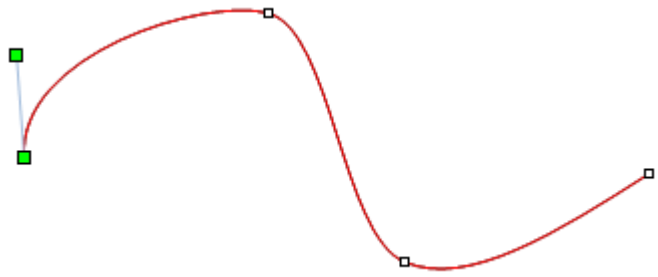
When the reshape tool is active, the status bar updates with current position in world coordinates and map coordinates (if available).

To edit a polyline or 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.

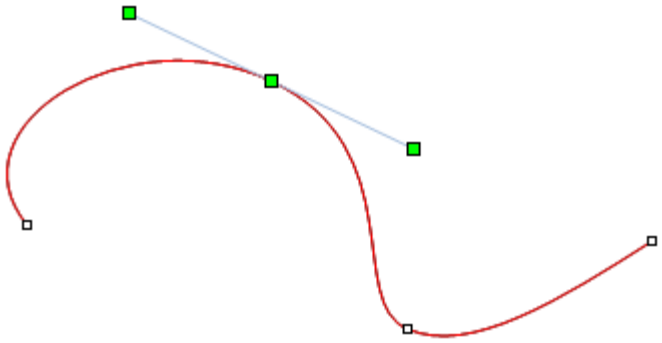
To edit a spline polyline

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.

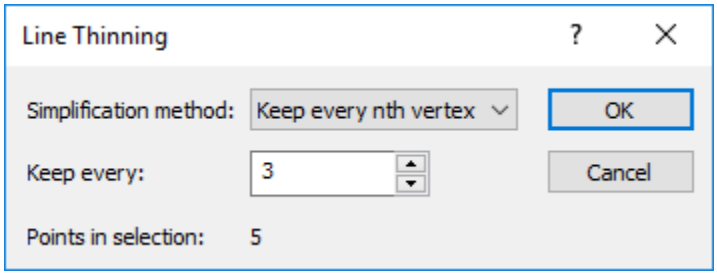
Thin

Click on a polyline or polygon and click the **Features | Edit Features | Thin** command or 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. To smooth polylines or polygons in a base map, enter the group first by clicking the Start Editing command. Then, click on the polylines or polygons, and click the **Features | Edit Features | Thin** 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.*

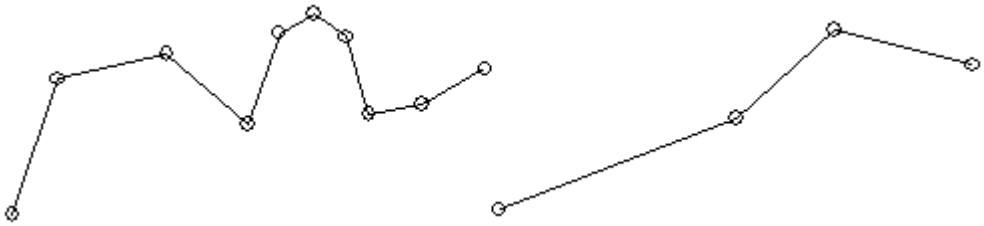
Simplification Method

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.

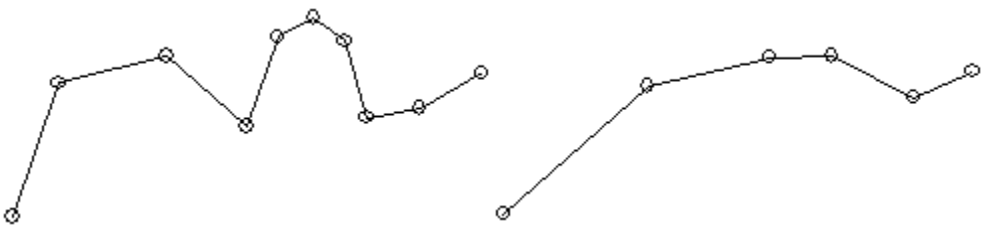


The original polyline is on the left. The polyline after a Keep every of 3 was applied is shown on the right.

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.

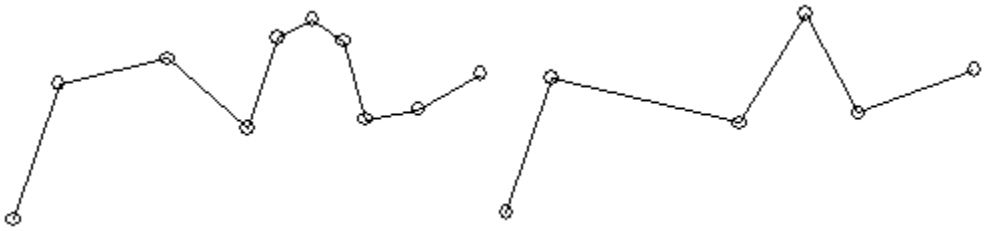


The original polyline is on the left. The polyline after an Average every value of 3 was applied is shown on the right.

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 units of the selected polyline or polygon. If multiple polylines or polygons are selected, all of the objects should use the same coordinates, otherwise the *Deviation Distance* will be incorrect.



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

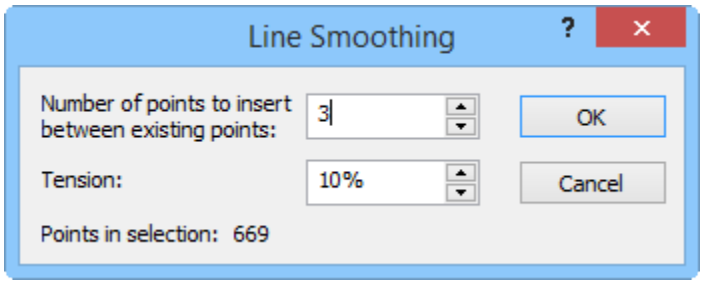
Click on a polyline or polygon and click the **Features | Edit Features | Smooth** command or right-click on the polyline or polygon and click **Smooth** to open the **Line Smoothing** dialog. The **Line Smoothing** dialog adds points to a selected polyline or polygon to smooth the line. Smoothing removes 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. To smooth polylines or polygons in a base map, enter the group first by clicking the Start Editing command. Then, click on the polylines or polygons and click the **Features | Edit Features | Smooth** 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

Connect Polylines

Click the **Features | Edit Polylines | Connect Polylines** command 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 or spline polylines must be selected.

The new 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.

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 | Edit Polylines | Connect Polylines** command. The polylines are connected. Click the **Features | Group | Stop Editing** to end object editing mode in the base map.

Break Polyline

Click the **Features | Edit Polylines | Break Polyline** command or right-click a polyline and click **Break Polyline** to break the selected polyline or spline polyline into multiple polylines. The cursor changes to to indicate break mode. Click on the polyline or spline polyline in the location where the line should be broken. For the command to be available, a single polyline or spline polyline must be selected. Spline polylines are converted to regular polylines after being broken apart.

The new polylines retain the line properties from the original polyline. If the object name has not been changed from the default *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.

Note about Base Maps

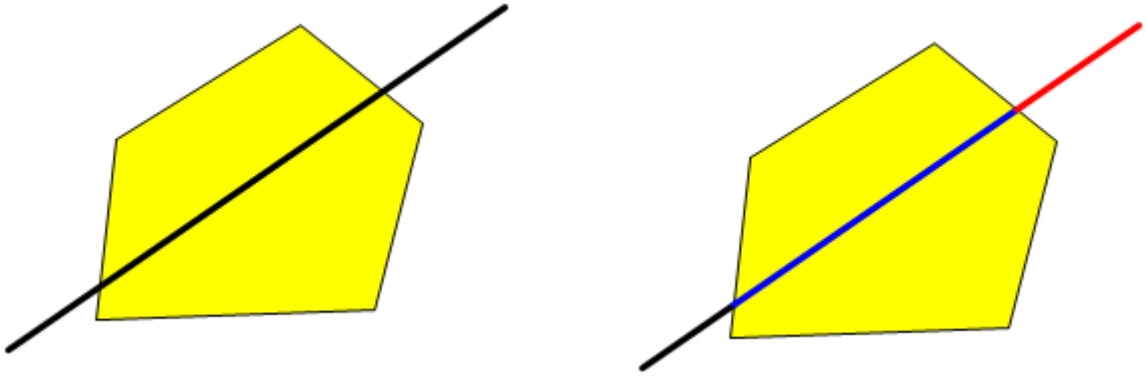
If objects should be edited in a base map, click on the object in the *Base* layer to select it. Click the **Features | Group | Start Editing** command to enter editing mode. Then, click the **Features | Edit Polylines | Break Polyline** command. Click on the polyline and two polylines are created. Click the **Features | Group | Stop Editing** to end object editing mode in the base map.

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. 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.

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. If the objects you wish to combine are part of a group or base map, select the group or base map and click the Start Editing command.
2. Select all the objects that you wish to combine into a single polygon.
3. Click the **Features | New Features | Union of Polygons** command.
4. Select whether or not to keep the original objects in the Union Polygons dialog.
5. 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.
6. If the objects were part of a group or base map, click the Stop Editing command.

*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.

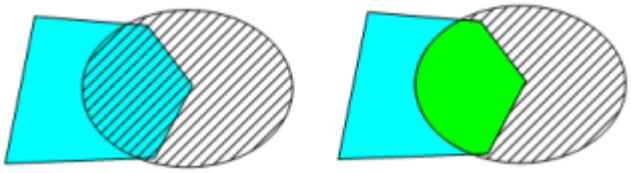
- Contiguous objects are part of the created polygon with the **Union Polygons** command. With the **Combine Islands/Lakes** command, contiguous objects are separate islands in a single complex polygon.
- Intersecting areas of the selected objects are treated as part of the new polygon when the **Union 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).*

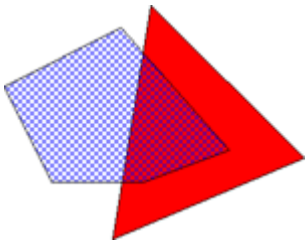
To use the **Intersect Polygons** command:

1. Select two or more overlapping objects.
2. Click the **Features | New Features | Intersect Polygons** command, or 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.

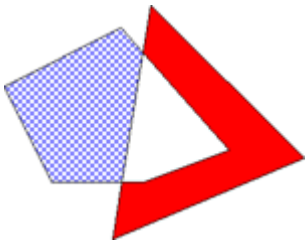
Difference of Polygons

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:

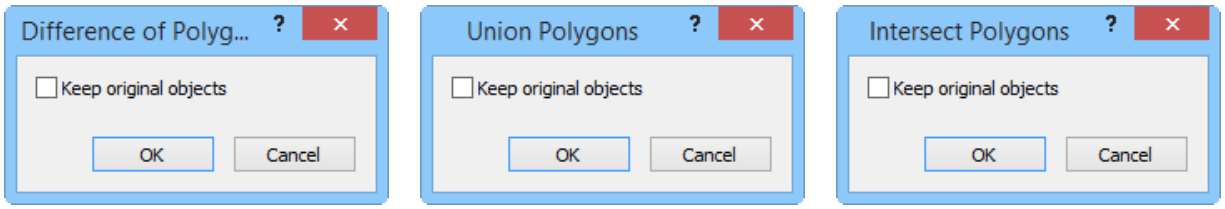
To create a difference of polygons:

1. Select two or more polygons.
2. Click the **Features | New Features | Difference of Polygons** command, or 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.*

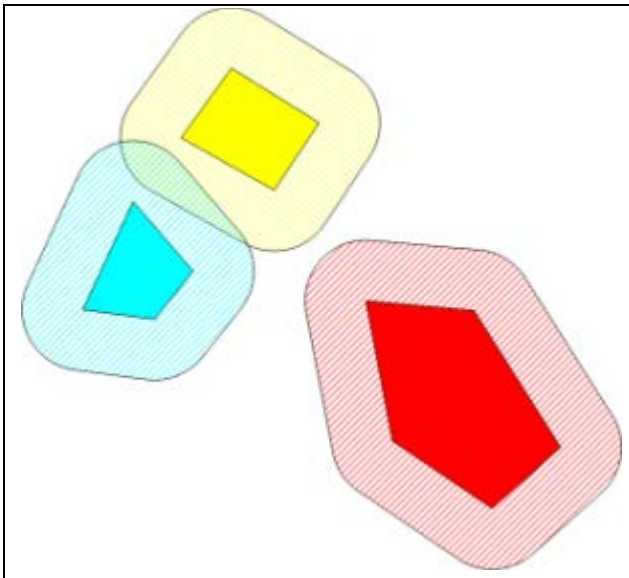
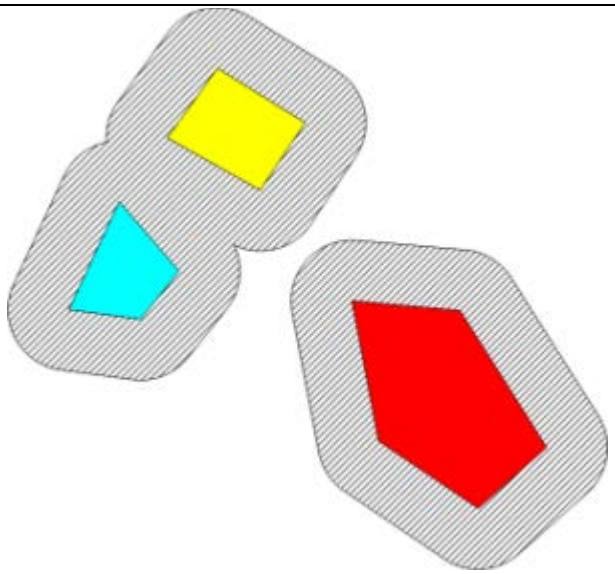
Keep Original Objects

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.

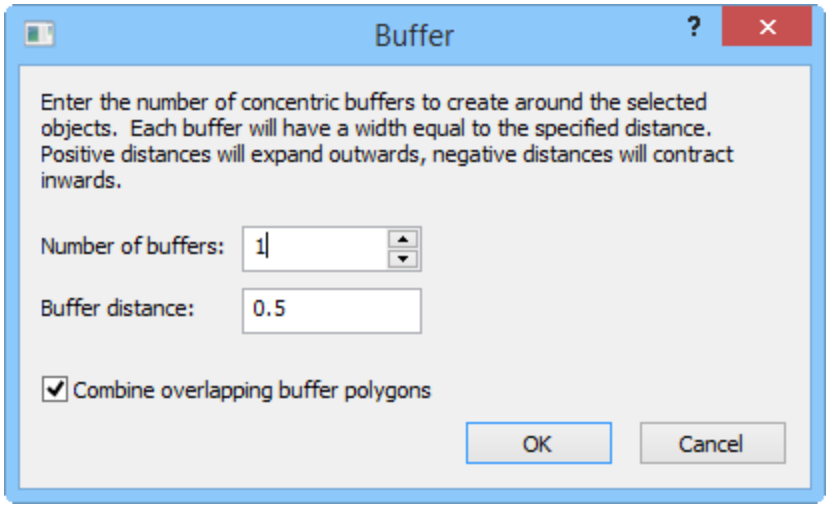
	
The Combine overlapping buffer polygons option was not checked in this example. Notice the overlapping region between the yellow and blue buffer polygons. Three new polygons were added to the plot window and ***Contents*** *window.*	*The Combine overlapping buffer polygons option was checked for this 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. If the object around which the buffer should be created are part of a group or base map, click the Start Editing command.
2. Select the object around which to create a buffer polygon by clicking the object in the plot window or Contents window.
3. Click the **Features | New Features | Buffer** command.
4. Specify the Number of buffers and Buffer distance and select whether to Combine overlapping buffer polygons in the **Buffer** dialog.
5. Click *OK*, and the buffer polygon is created.
6. If the object was part of a group or base map, click the Stop Editing command.

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).

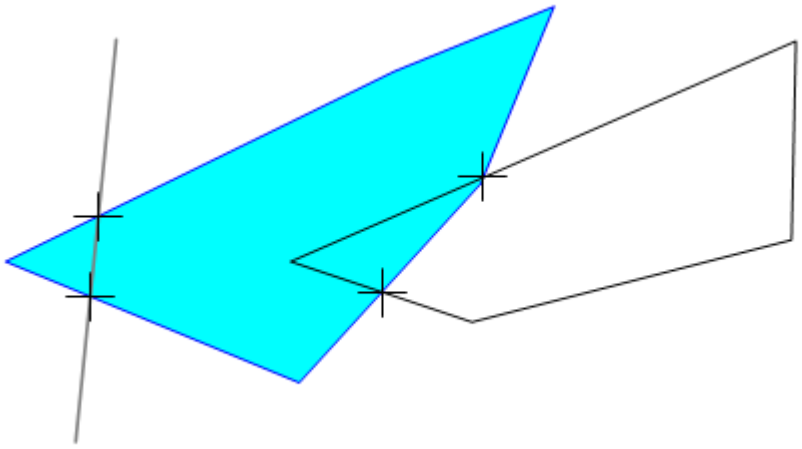
Combining Buffer Polygons

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

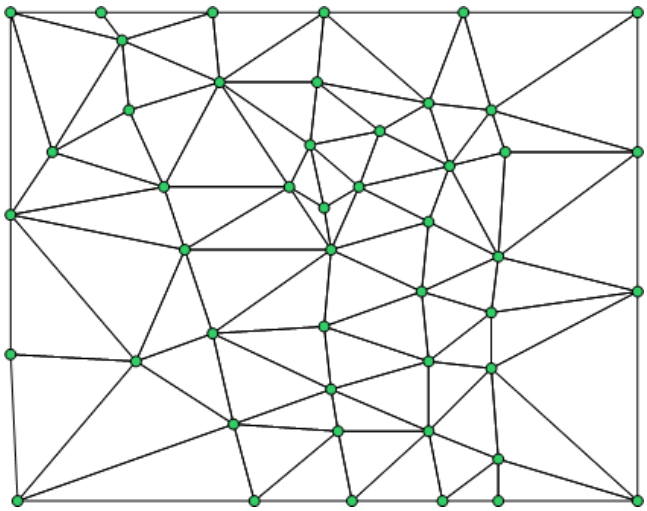
To create points at intersections:

1. Select two or more intersecting objects.
2. Click the **Features | New Features | Intersection Points** command.
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.*

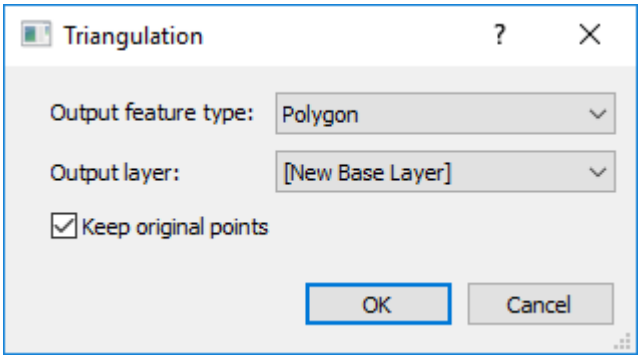
Creating a Triangulation Diagram

The following steps describe how to create a triangulation diagram:

1. If you wish to create a triangulation diagram from points in a base layer, select the base layer and click Features | Group | Start Editing.
2. Select three or more points from which to create the diagram.
3. Click the **Features | New Features | Triangulation** command.
4. Select the desired options in the **Triangulation** dialog.
5. 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. 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.

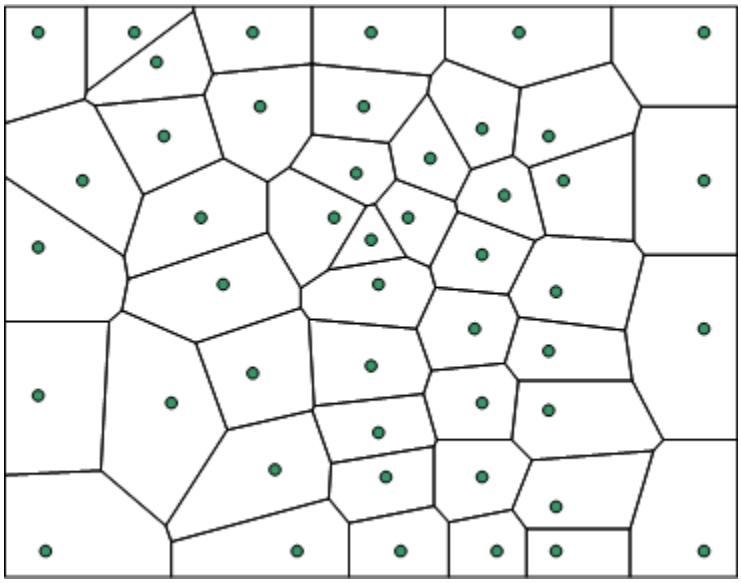
Keep Original Objects

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.*

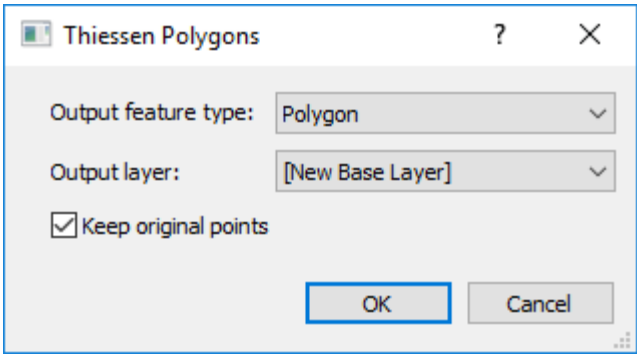
Creating a Thiessen Polygon Diagram

The following steps describe how to create a Thiessen polygon (Voronoi) diagram:

1. If you wish to create a Thiessen polgyon diagram from points in a base layer, select the base layer and click Features | Group | Start Editing.
2. Select three or more points from which to create the diagram.
3. Click the **Features | New Features | Thiessen Polygons** command.
4. Select the desired options in the **Thiessen Polygons** dialog.
5. 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. 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 | Islands/Lakes | 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.

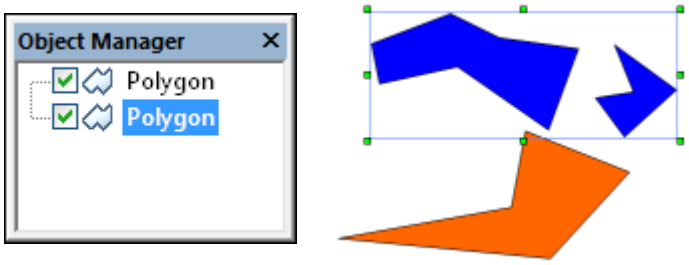
Complex polygons can contain polygons that are touching, intersecting, completely separate, or contained completely within other polygons. Each individual polygon in the group is called an island. An example of islands that can form a complex area is the Hawaiian islands. When a polygon is completely 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.

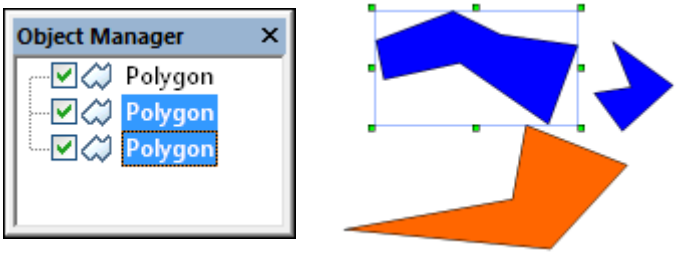
Splitting a Complex Polygon into Multiple Polygons

1. Click on a complex polygon in the **Contents** window. The polygon can be imported with the **File | Import** 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 | Islands/Lakes | Split** command. A 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 | Islands/Lakes | 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

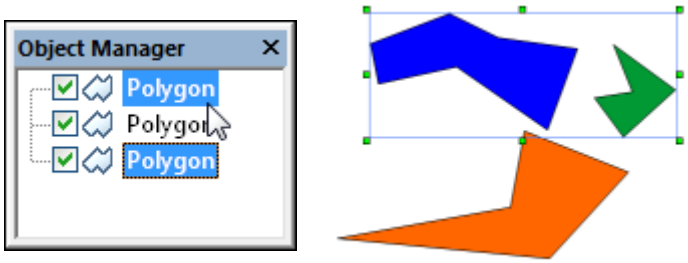
islands. When a polygon is completely contained inside another polygon, a lake is formed. Lakes are the holes in a complex polygon.

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 all of the polygons to be combined. Then, click the **Features | Islands/Lakes | Combine** command. The polygons are combined into a single complex polygon. Click the **Features | Group | Stop Editing** command to end object editing mode in the base map.

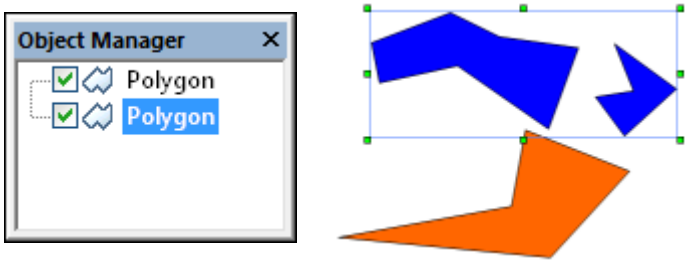
Combining Multiple Drawn Polygons

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.
9. Click the **Features | Islands/Lakes | Combine** command. 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.

Undo

Click the **Home | Undo | Undo** command 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 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 press CTRL+V on the keyboard to copy the clipboard contents into the current document. The objects to be copied 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

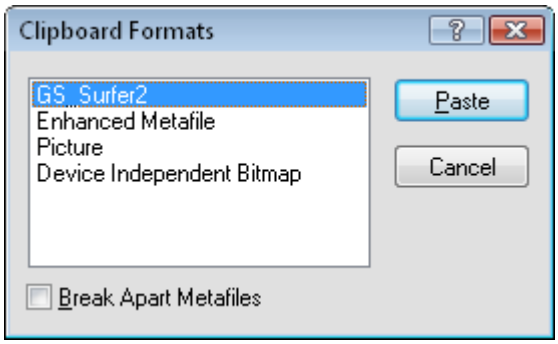
In the plot window, the clipboard contents are pasted in the center of the window. The pasted contents are automatically selected, and can be dragged to a new location. Alternatively, change the position or size of the selected object with the Position/Size group.

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 **Surfer**.

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 **Surfer** map or drawing to the clipboard. It is stored in the clipboard as a native **Surfer** object. You can paste this map or drawing into another plot window in **Surfer**. 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 **Surfer** 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 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 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 removes an object from the document. An object can sometimes be restored with the Undo command.

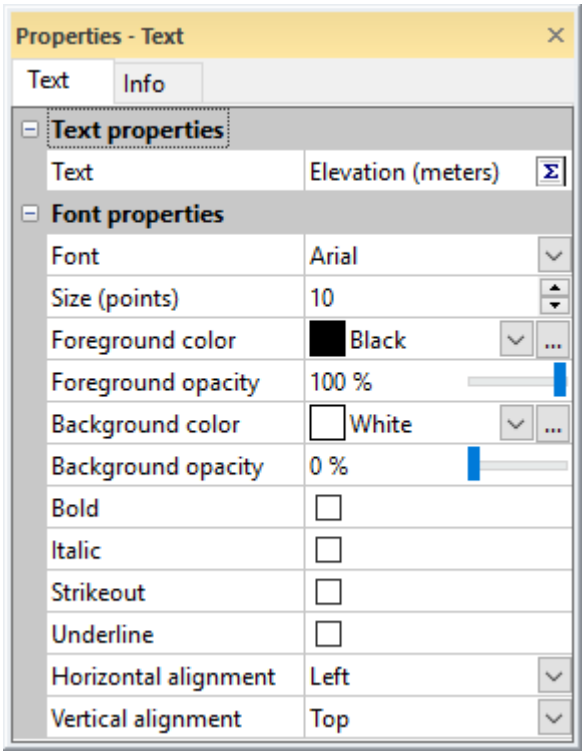
To delete an object, select the object in the plot window or Contents window and press the DELETE key or click the **Delete** command.

Chapter 28 - Common Properties

Many properties are the same for various features and map layers in **Surfer**. These common properties are explained in the following chapter.

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.

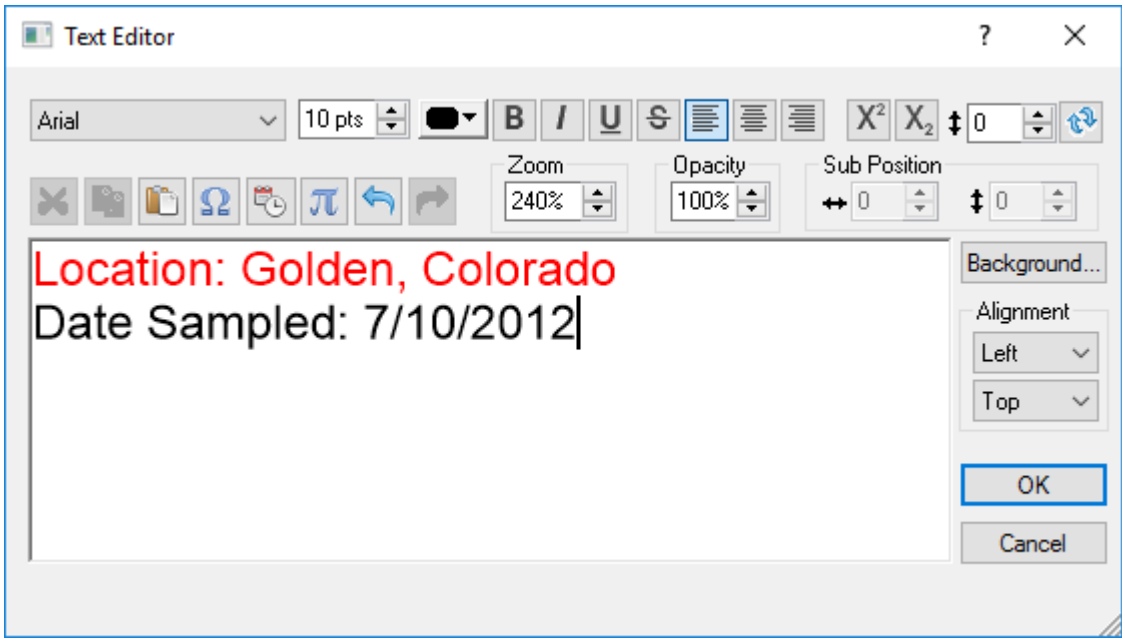
For complex text or multiple lines of text, click the Σ button to open the Text Editor. Enter text, edit text properties for individual characters, and add templates, symbols and timestamps with the **Text Editor**. The text you enter into the **Text Editor** is converted into math text instructions 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:

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:

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 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 oblique 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).

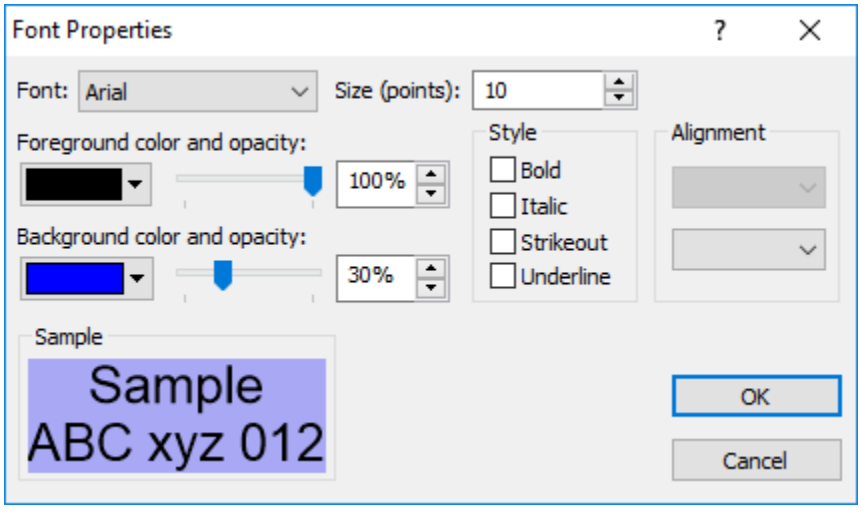
- *Left* horizontally aligns the text box so that the reference point is to the left of the text box.
- *Center* horizontally centers the text box on the reference point.
- *Right* horizontally aligns the text box so that the reference point is to the right of the text box.
- *Top* vertically aligns 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.



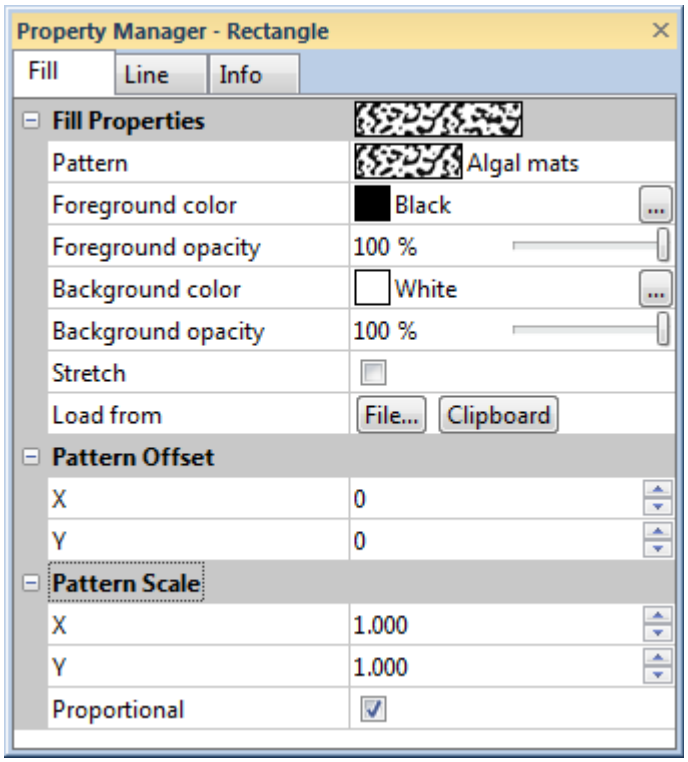
*Set the font properties for contour labels and grid node editor labels in the **Font Properties** dialog.*

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 on the button at the right of the color sample to open the **Colors** dialog, where you can specify a custom color.

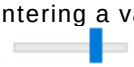
Foreground Opacity

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.

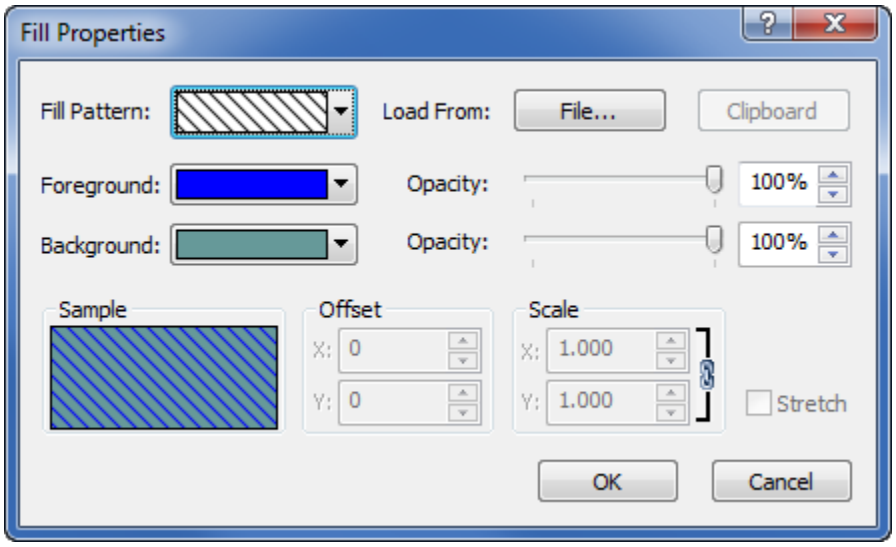
When the scale is proportional, the current aspect ratio is maintained. Changing the X or Y scale will cause the other scale to be adjusted proportionally to maintain the aspect ratio at the time the scale was set to proportional.

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

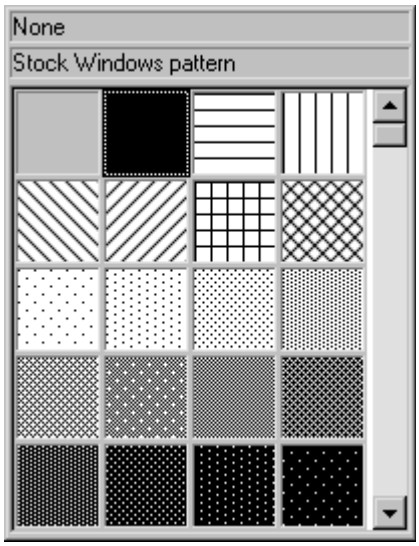
The *Sample* box shows the selected pattern, foreground color, and background color.

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 limestone
87.5% Black	USGS 217	Limestone w/ saccharoidal
Sand	USGS 218	Crossbedded limestone
Swamp	USGS 219	Cherty crossbedded limestone
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 limestone
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	Bedded chert 2
Andesite 2	Granodiorite 3	Fossiliferous bedded chert
Andesite 3	Hornfels	Fossiliferous rock
Basalt 1	Hornfels 2	Diatomaceous rock
Basalt 2	Carbonatite	Subgraywacke
Conglomerate	Kimberlite	Crosbedded subgraywacke
Dolomite	Dendrites	Ripple-bedded graywacke
Fish Fossil	Calc-silicate	Peat
Gneiss 1	Oolite	Bony coal
Gneiss 2	Med unconsolidated sediment	Underclay
Granite1	Fine unconsolidated sediment	Flint clay
Granite 2	Jointed/Fractured rock	Bentonite
Gravel	Gneiss USGS 2	Glaconite

Limestone - Fossil 1
Limestone - Fossil 2
Limestone - Oolitic
Limestone - Stomatolitic
Limetsone
Mudstone
Quartzite
Rhyolite
Rocks
Sand 1
Sand 2
Sandstone 1
Sandstone 2
Schist
Slate
Stones
White Marble
BIF
Black Marble
Gneiss 3
Granite 3
Granite 4
Granite 5
Green Marble
Orbicular Granite
Oxidized Zone
Pahoehoe
Pink Marble
Pumice
Ruby Zoisite
Scoria
Zebra Dolomite
Sand and Gravel
Coarse sand
Well graded coarse sand
Coarse, medium-graded sand
Coarse, well-graded sand
Coarse, well-graded sand 2
Fine, well-graded sand
Fine, poorly-graded sand
Medium, poorly-graded sand
Coarse, crossbedded sand
Medium, poorly-graded sand 2
Gneiss USGS
Clastic dike
Glacial-1
Glacial-2
Rhyolite USGS
Rhyolite USGS 2
Dacite
Salt 1
Salt 2
Volcanic A
Volcanic B
Bedded tilted sandstone
Sandy gravel
Volcanic tuff
Rhyolitic tuff
Algal mats
Lignite
Fine, poorly-graded sand 2
claystone
Marsh
Periglacial-1
Periglacial-2
Periglacial-3
Periglacial-4
Periglacial-5
Gravel or conglomerate 1
Gravel or conglomerate 2
Crosbedded gravel
Till or diamicton
Breccia 1
Breccia 2
Massive sand
Bedded sand
Crossbedded sand
Crossbedded sand 2
Ripple-bedded sand
Argillaceous sandstone
Calcareous sandstone
Dolomitic sandstone
Loess
Silty shale
Calcareous siltstone
Dolomitic siltstone
Limonite
Siderite
Phosphatic rock
Gypsum
Salt 3
Interbedded sandstone
Interbedded sandstone 2
Ripple-bedded sandstone
Interbedded shale
Interbedded shale 2
Interbedded shale 3
Interbedded calcareous shale
Interbedded silty limestone
Interbedded limestone 1
Interbedded limestone 2
Interbedded limestone 3
Interbedded limestone 4
Metamorphism
Quartzite USGS
Slate USGS
Schistonse
Schist USGS
Contorted schist
Schist and gneiss
Gneiss
Contorted gneiss
Soapstone
Tuffaceous rock
Crystal tuff
Devitrified tuff
Volcanic breccia and tuff
Volcanic breccia
Zeolitic rock
Basaltic flows
Banded igneous rock
Granite 2 USGS
Igneous rock 1
Igneous rock 2
Igneous rock 3
Igneous rock 4
Igneous rock 5
Igneous rock 6
Igneous rock 7

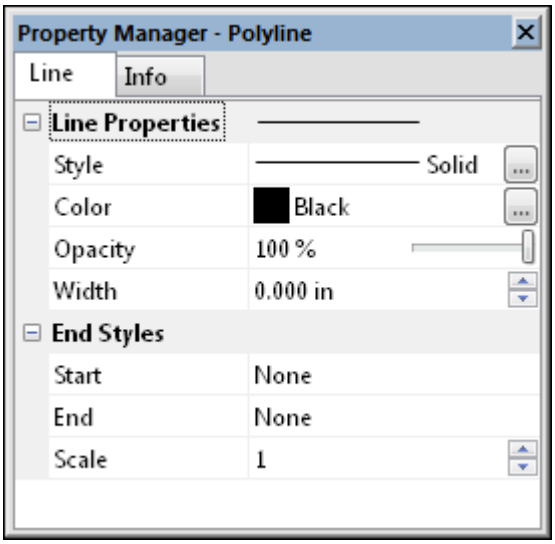
Loose, medium-graded gravel	Silty shale 2	Igneous rock 8
Medium graded sand	Clay shale	Pophyritic rock 1
Coarse crossbedded gravel	Cherty shale	Porphyritic rock 2
Medium, well-sorted sand	Dolomitic shale	Vitrophyre
Inorganic silt 1	Calcareous shale or marl	Quartz
Inorganic silt 2	Carbonaceous shale	Ore
Inorganic silt 3	Oil shale	

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*.

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

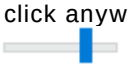
Click the line next *Style* to open the line style palette. Click on a style to use it for the selected line. The line style sample updates to show the new selection. Click on the ... button at the right of the line style to open the **Custom Line** dialog, where you can specify a custom line style.

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.

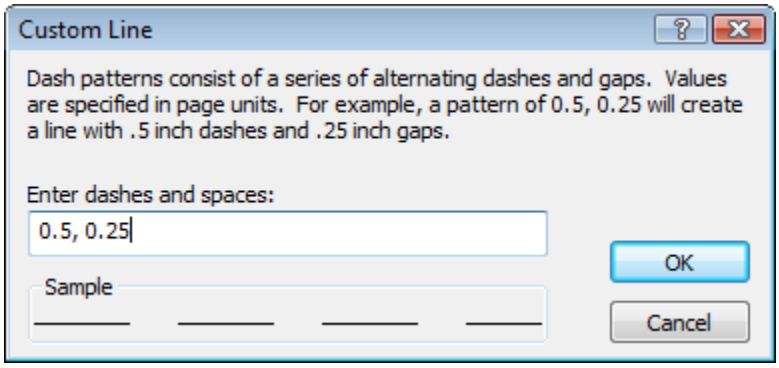
The *Scale* determines the scale factor of the arrowhead. To change the size, highlight the existing value and type a new value. Press ENTER on the keyboard to make the change. Alternatively, click the buttons to increase or decrease the values. A value of 1 makes the arrow the default size.

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

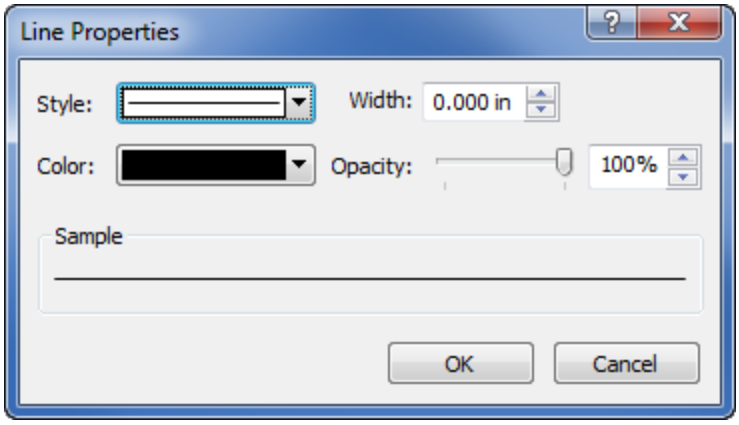
The *Sample* box shows the new line style.

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.

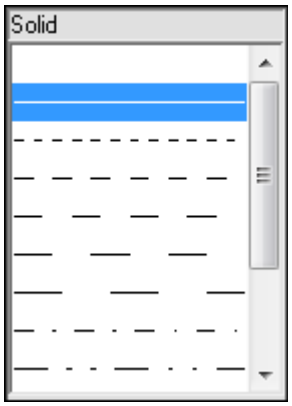
Sample

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

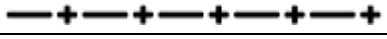
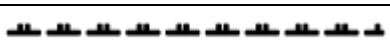
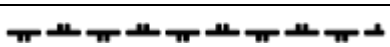
Select a line style from the line palette.

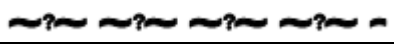
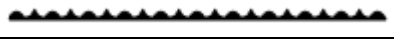
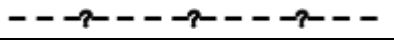
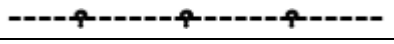
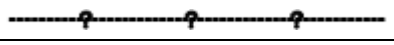
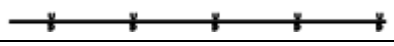
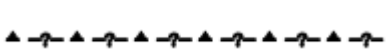
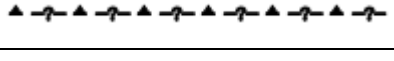
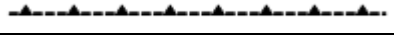
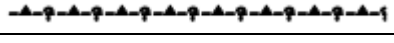
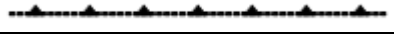
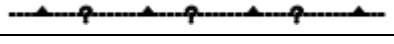
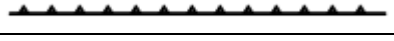
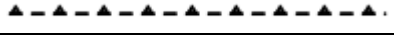
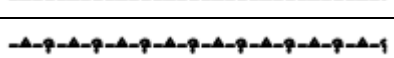
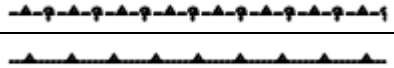
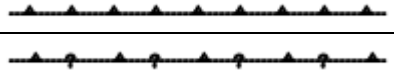
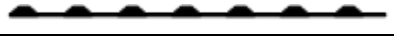
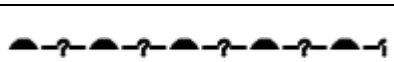
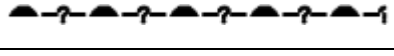
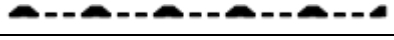
Line Styles

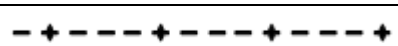
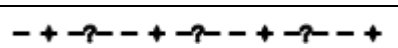
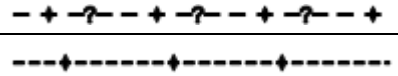
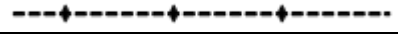
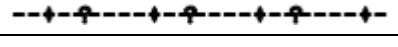
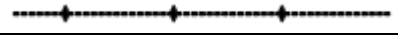
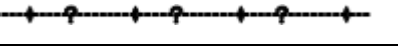
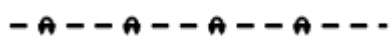
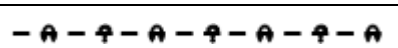
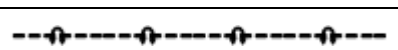
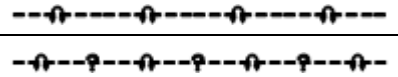
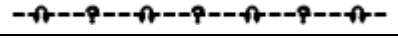
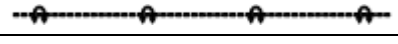
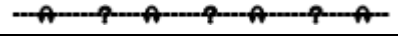
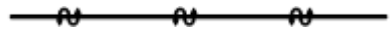
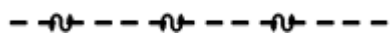
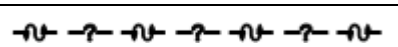
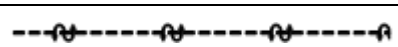
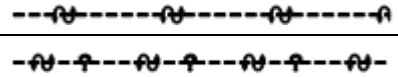
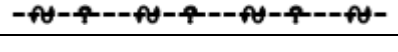
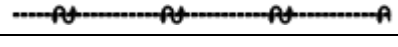
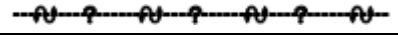
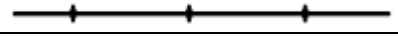
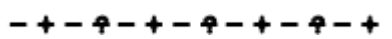
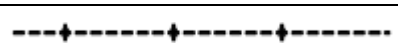
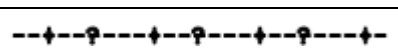
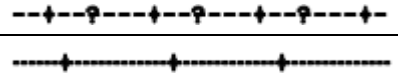
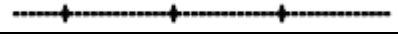
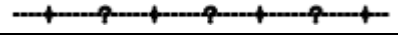
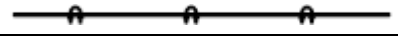
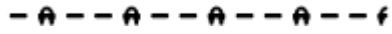
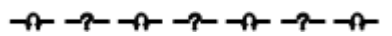
Invisible
Solid
.1 in. Dash
.2 in. Dash
.3 in. Dash
.4 in. Dash
.5 in. Dash
Dash Dot
Dash Dot Dot
Dash Dot Dot Dot
Dash Dash Dot
Dash Dash Dot Dot
Dash Dash Dot Dot Dot

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	

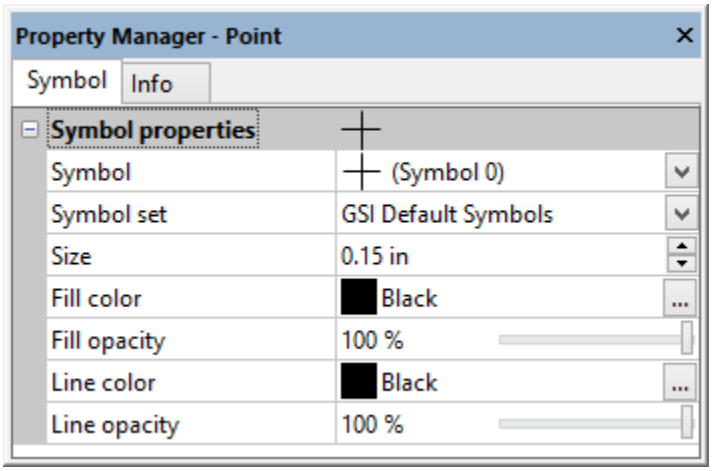
Inverted syncline - Approximately located	
Inverted syncline - Approximately located, queried	
Inverted syncline - Inferred	
Inverted syncline - Inferred, queried	
Inverted syncline - Concealed	
Inverted syncline - Concealed, queried	
Dipping Bed - Certain	
Dipping Bed - Approximately located	
Dipping 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*.

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.

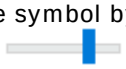
Size

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.

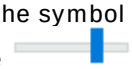
Fill Opacity

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

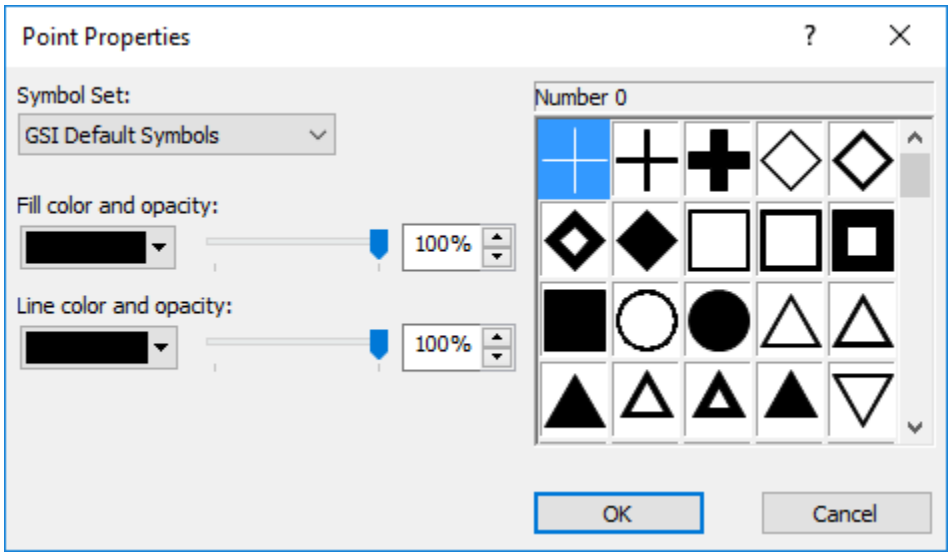
The Info page in the **Symbol Properties** dialog displays the *Position* and any attributes associated with the point.

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.

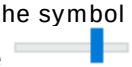
Fill Color and Opacity

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.

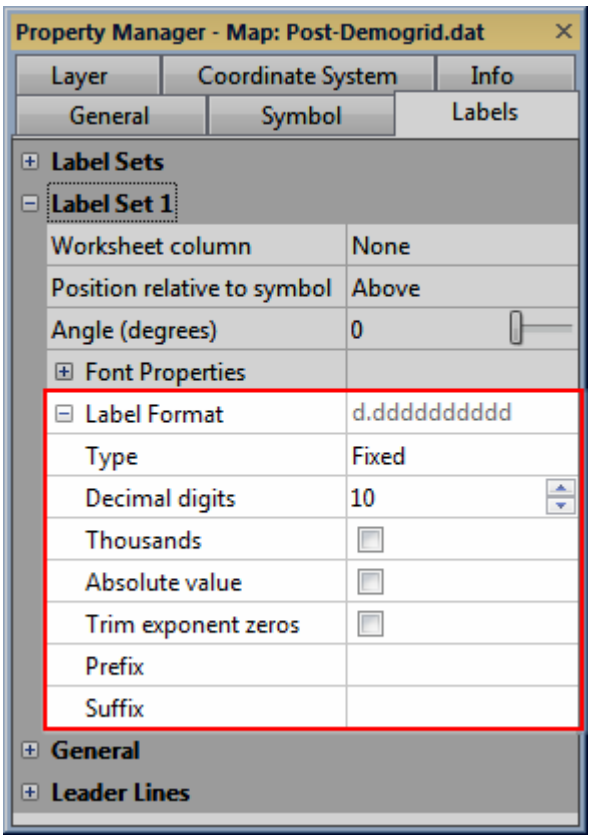
Info Page

The Info page in the **Point Properties** dialog displays the *Position* in terms of X, Y.

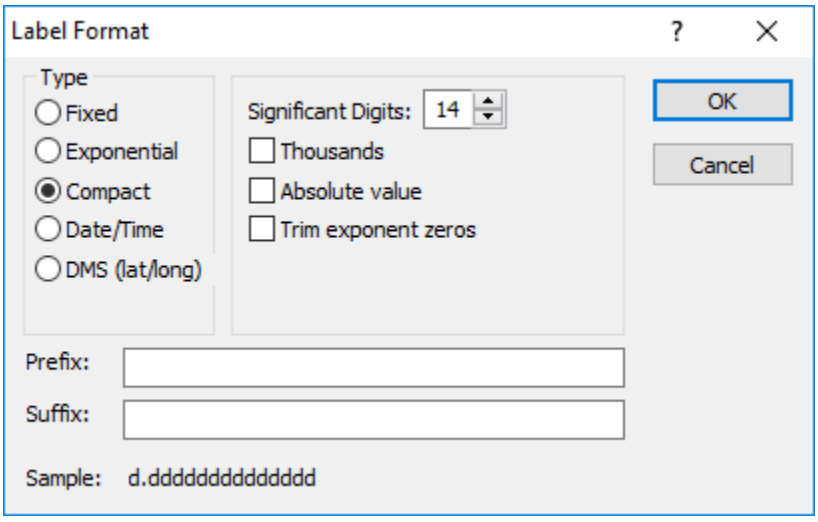
Label Formats

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 the *Decimal digits* box. For example, if the numeric format is set to *Fixed* with 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 the *Decimal digits* property. The +dd represents a sign and at least two digits for the exponent. At least two digits are displayed unless the *Trim exponent zeros* option is selected. For example, if the numeric format is set to *Exponential* with two *Decimal 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 *Significant 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).*

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

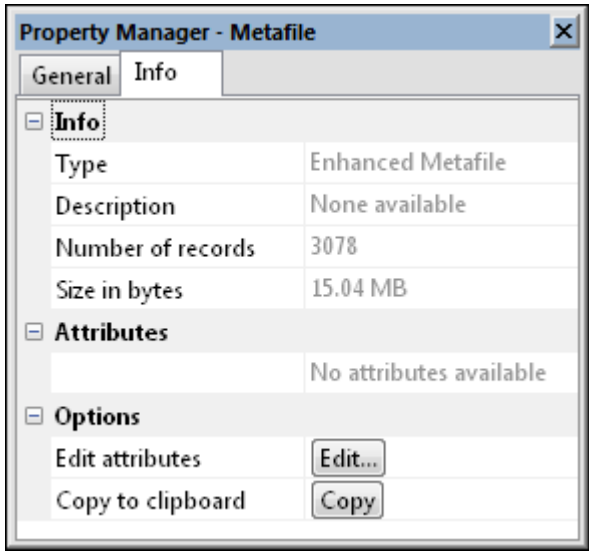
You can add a text string before each label using the *Prefix* box. For example, a "$" could be used as a prefix. Type the text exactly as you want it to appear in the *Prefix* box.

Suffix

You can add a text string after each label using the *Suffix* box. For example, a unit of measure " ppm" could be used as a suffix. Type the text exactly as you want it to appear in the *Prefix* box.

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 **File | Import** 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

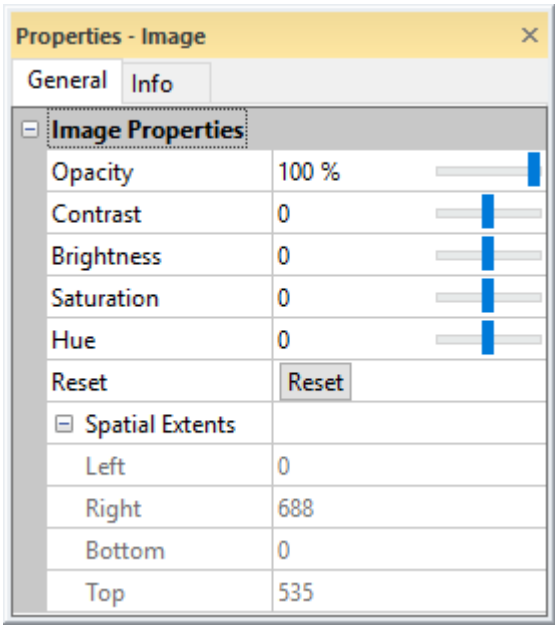
The **General** page controls the metafile opacity. Change the *Opacity* of the metafile by entering a value from 0% (completely transparent) to 100% (completely opaque) or dragging the slider to change the opacity percentage.

Info Page

The Info page for a metafile displays the *type* of metafile, *description, number of records,* and *size in bytes*.

General Page - Image Properties

Image files can be loaded into **Surfer** with the File | Import 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.*

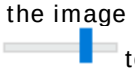
Image Properties

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.

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

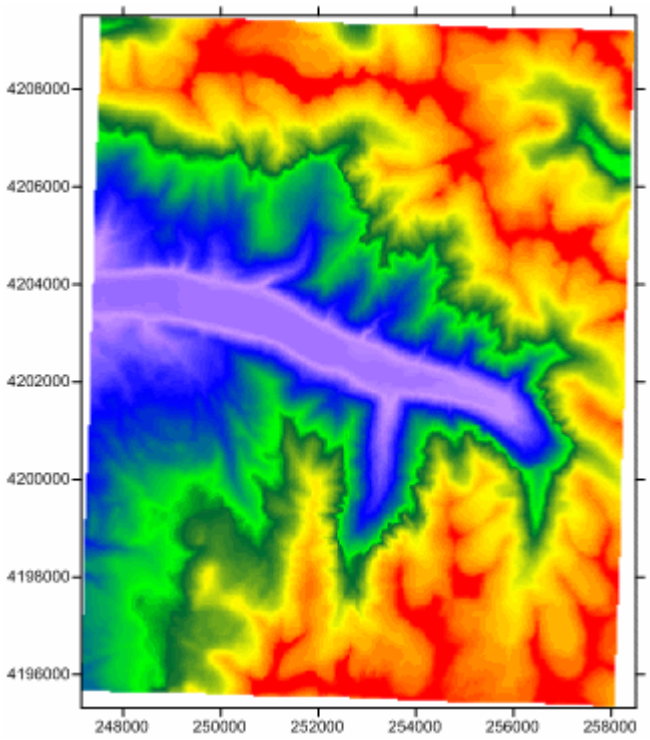
When an image is included in a base (vector) layer, the *Georeference* property is available. Click *Georeference image* in the *Georeference* field to apply control points to the image and reference it to your map's coordinate system. Clicking *Georeference image* opens the Georeference Image window. 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.

Adding Color to Maps

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, shaded 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 Dialog

The Colormap dialog 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:
    1. Click the color bar next to *Fill colors* and select a new color scale.
    2. To open the **Colormap** dialog, click the button to the right of the color.
5. If the Level method is set to Advanced:
    1. Click the *Edit Levels* button next to *Contour levels*.
    2. Click the *Fill* button.
    3. In the Fill dialog, click the *Foreground Color* button. The **Colormap** dialog opens.

Contour map lines also use the **Colormap** dialog 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** dialog opens.

Color Relief and Shaded Relief Maps

To open the **Colormap** dialog for a color relief or shaded relief map,

1. Click on the image or shaded 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** dialog, 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 tab.
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** dialog, click the button to the right of the colormap.

3D Surface Maps

To open the **Colormap** dialog for a surface map,

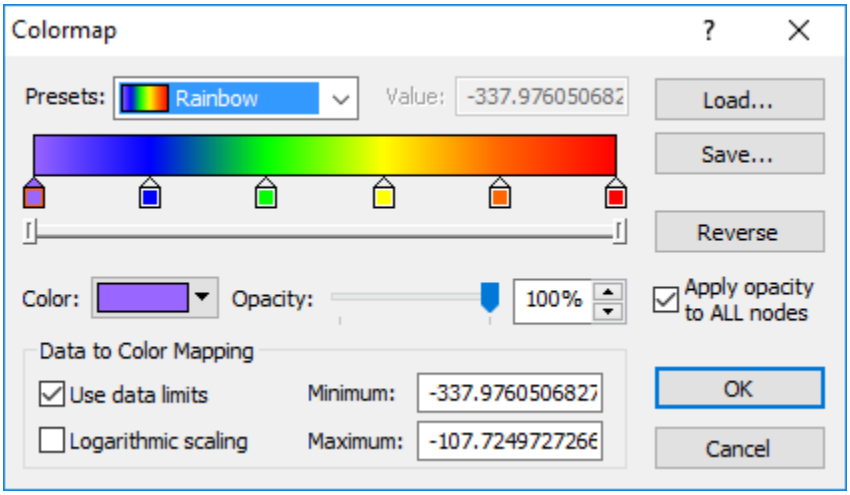
1. Click on the surface map to select it.
2. In the **Properties** window, click on the **General** tab.
3. Open the *Material Color* section.
4. Click the color bar next to *Upper* to select a new color scale.
5. To open the **Colormap** dialog, click the button to the right of the colormap.

Colormap Dialog

The **Colormap** dialog is used to assign either *Preset* or custom color spectrums to a map. Contour maps, post maps, classed post maps, color relief maps, shaded relief maps, vector maps, surface maps, and base map symbology all use the **Colormap** dialog to assign colors to the map.

The colormap has specific colors assigned to nodes along the spectrum. **Surfer** automatically blends colors between the nodes to produce a smooth color gradation over your map. After you create a colormap, you can save the spectrum as a .CLR file for later use with other maps. Although the colors are assigned to specific values on a map, the spectrum can be used with maps containing different Z ranges because the node values are stored as percentages. Several predefined colormaps are available in the SAMPLES directory and are shown in the *Presets* list.

The **Colormap** dialog is opened differently depending on the map type you are using to access the dialog.



*Use the **Colormap** dialog to assign colors to a map.*

Presets

The *Presets* list allows you to select a predefined .CLR file created by Golden Software. Custom colormaps cannot be added to the preset list.

Data Value

The *Value* displays the data value of the node selected. The selected node may be accurately repositioned by entering a new value in the *Value* box. The first and last nodes cannot be changed and this control is disabled when an end node is selected.

The *Value* option is not used the same way with the classed post map. 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.

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** dialog 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.

Reversing a Colormap

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.

Colormap Example

The sample of the generated colormap is display directly above the anchor nodes.

Assigning Colors

The **Colormap** dialog allows you to specify the colors to associate with a range of data within the map. Blended colors are achieved by creating anchor nodes along a color bar and assigning specific colors to the specified anchor nodes. Colors are linearly interpolated between adjacent anchors.

Adding a Node

You can add additional anchor nodes at any position along the colormap. To create a new anchor node, left-click below or on the colormap where you want the new node added. The new anchor is automatically assigned a color in the color palette, and the value is displayed in the *Value* box. You can add as many anchor nodes as you want. This lets you blend colors in many different ways on the colormap.

Deleting a Node

Left-click on an anchor node and press the DELETE key on the keyboard to delete a node. Nodes can also be dragged outside the top or bottom of the dialog to remove it from the spectrum. The first (far left) and last (far right) anchor nodes cannot be deleted.

Scroll Control

The scroll control appears as a horizontal bar with draggable end handles. Drag a handle left or right to zoom the colormap 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.

Use the scroll control to zoom the colormap in or out.

Color

Click the *Color* box to specify the color of the currently selected anchor node. The color palette opens.

Changing the Opacity

Change the *Opacity* of a node by selecting the node and 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.

3D surface maps cannot be made transparent.

Changing the Opacity of All Nodes

Click the *Apply opacity to ALL nodes* option to change the opacity for all nodes. Uncheck the *Apply opacity to ALL nodes* to change individual node opacity.

Colormap Data Range

The *Data to Color Mapping* group contains options for setting the minimum and maximum data values 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.

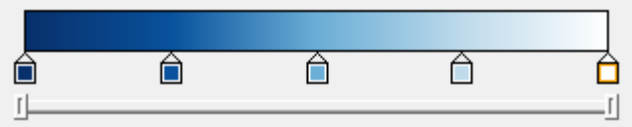
Colormap Data Minimum and Maximum

If you would prefer to set the colormap minimum and maximum values to other values, enter the new numbers into the *Minimum* and *Maximum* 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 each time. If a data value within the map falls outside this range, it is assigned the minimum or maximum color, whichever is closest.

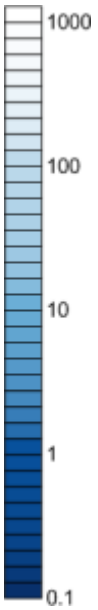
The minimum and maximum values cannot be set for the classed post map. For the classed post map, the far left *Minimum* node is always the median value for the first class. The far right *Maximum* node is always the median value for the last class. This cannot be changed.

Logarithmic Scaling

Check the box next to the *Logarithmic scaling* to set the intervals between the nodes to a log(10) scale. The *Minimum* and *Maximum* 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, uncheck the box next to *Logarithmic scaling.*



When the Logarithmic scaling option is checked, the color scale bar uses a Log(10) scale between nodes. This colormap displays nodes at 0.1, 1, 10, 100, and 1000.



The associated color scale bar for the colormap above is shown with the major log intervals displayed.

Anchor Nodes

The color spectrum is defined by anchor nodes at user-defined points along a color spectrum in the Colormap dialog. Colors are automatically blended between adjacent anchors.

Anchor nodes are represented by a slider button below the color spectrum. A node with a colored arrow and an orange boundary indicates the node is currently selected.

Anchor Node Value

The data value for the node is displayed in the *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 *Value* box.

Initially, the minimum grid value and the maximum grid value anchor node colors are defined as black and white, respectively. You can change either of the colors by clicking once on the anchor slider button and then clicking on any color in the color palette. The color spectrum is updated to show the change.

Add Anchor Node

You can add additional anchor nodes at any position along the color spectrum. To create a new anchor node, left-click below or on the color spectrum where you want the new node added. The new anchor is automatically assigned a color in the color palette, and the value is displayed in the *Value* box. You can add as many anchor nodes as you want. This lets you blend colors in many different ways on the color spectrum.

Delete Anchor Node

Left-click on an anchor node and press the DELETE key on the keyboard to delete a node. Nodes can also be dragged outside the top or bottom of the dialog to remove it from the spectrum. The first (far left) and last (far right) anchor nodes cannot be deleted.

Positioning an 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, the data value is indicated in the *Value* box. Alternatively, you can set the anchor to a specific data value by entering a number into the *Value* box. Note that the beginning and ending anchors cannot be moved or deleted.
4. Reset the *Minimum* and *Maximum* 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:

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.

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 *Data to Color Mapping* group.

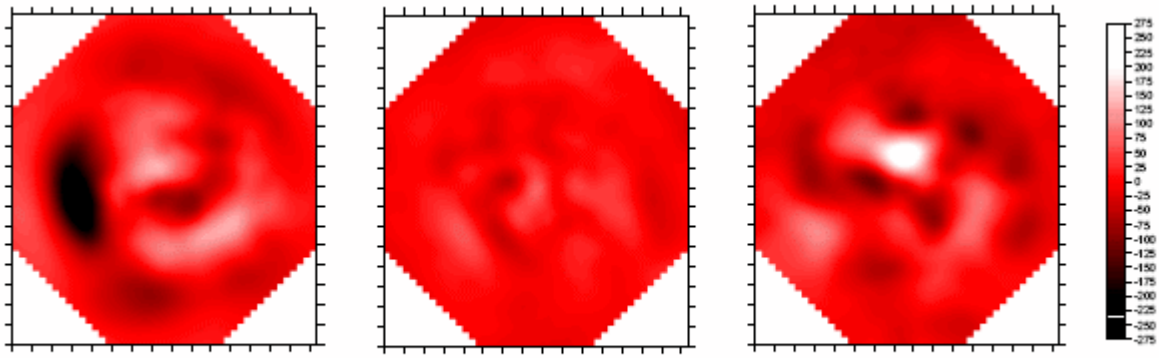
Example

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, shaded relief 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.
6. In the *Data to Color Mapping* 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** dialog and open the color spectrum .CLR file you just created.
7. In the *Data to Color Mapping* 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

Introduction to 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.

Level Files - Wireframe

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

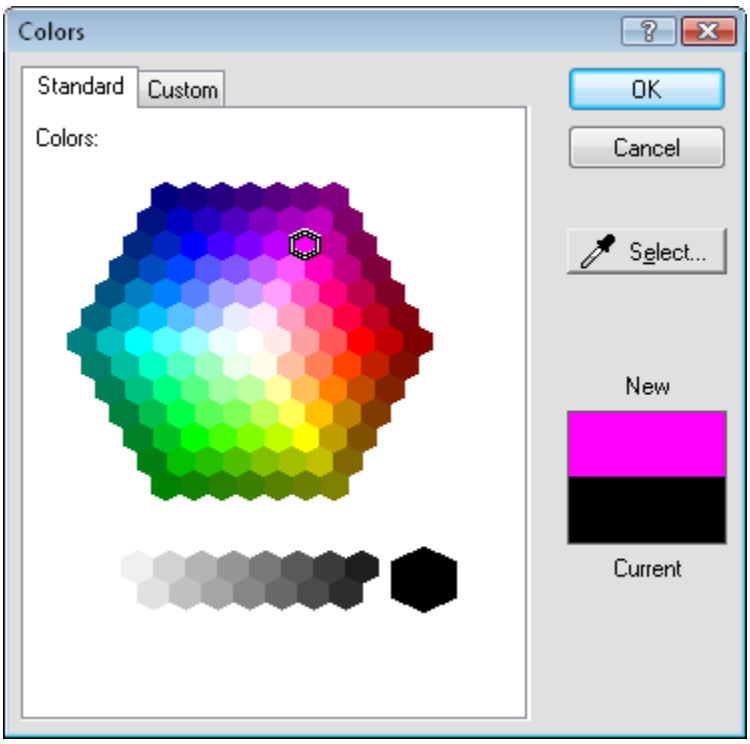
Click a color in the standard palette spectrum.

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

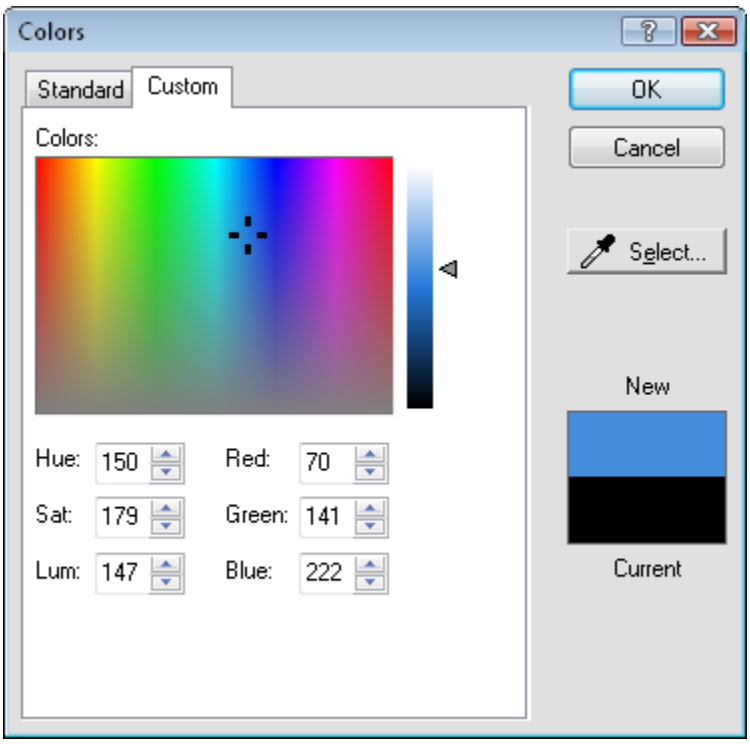
Click anywhere in the color spectrum to choose a new color, or enter new values in the *Hue, Sat, Lum, Red, Green,* and *Blue* boxes. You can drag the slider next to the color spectrum to adjust the new color's intensity.

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

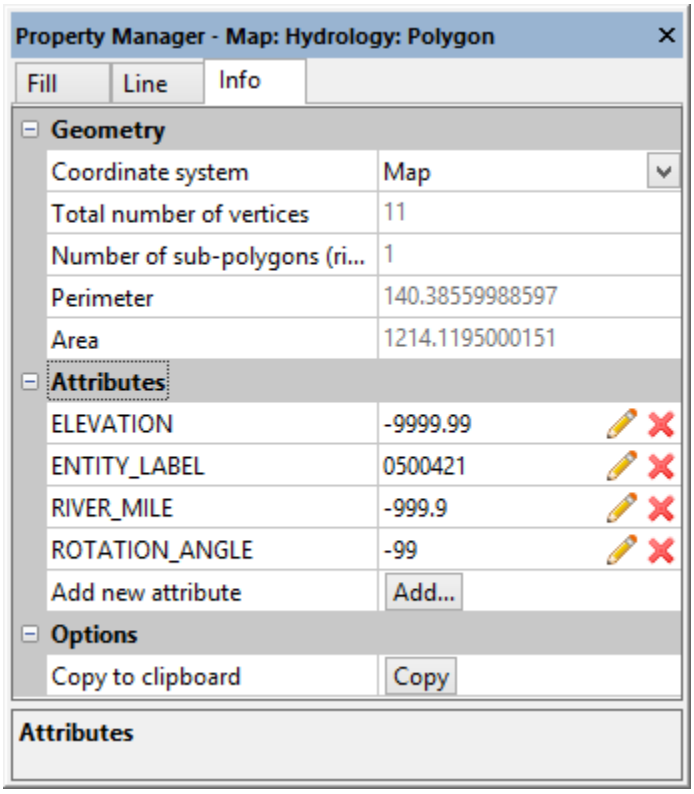
Color Names

Black	Moss Green	Sand
90% Black	Dark Green	Walnut
80% Black	Forest Green	Ruby Red
70% Black	Grass Green	Brick Red
60% Black	Kentucky Green	Tropical Pink
50% Black	Light Green	Soft Pink
40% Black	Spring Green	Faded Pink
30% Black	Turquoise	Crimson
20% Black	Sea Green	Regal Red
10% Black	Faded Green	Deep Rose
White	Ghost Green	Neon Red
Blue	Mint Green	Deep Pink
Cyan	Army Green	Hot Pink
Green	Avocado Green	Dusty Rose
Yellow	Martian Green	Plum
Red	Dull Green	Deep Violet
Magenta	Chartreuse	Light Violet
Purple	Moon Green	Violet
Orange	Murky Green	Dusty Plum
Pink	Olive Drab	Pale Purple
Dark Brown	Khaki	Majestic Purple
Powder Blue	Olive	Neon Purple
Pastel Blue	Banana Yellow	Light Purple
Baby Blue	Light Yellow	Twilight Violet
Electric Blue	Chalk	Easter Purple
Twilight Blue	Pale Yellow	Deep Purple
Navy Blue	Brown	Grape
Deep Navy Blue	Red Brown	Blue Violet
Desert Blue	Gold	Blue Purple
Sky Blue	Autumn Orange	Deep River
Ice Blue	Light Orange	Deep Azure
Light BlueGreen	Peach	Storm Blue
Ocean Green	Deep Yellow	Deep Blue

Info Page

The **Info** page contains information about the selected object. The **Info** page is 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.

The **Info** page is located in the Properties window for the selected object. A single object must be selected to display the **Info** page.



*The **Info** page displays information for the selected object.*

Geometry Section

The **Geometry** section allows you to set the *Coordinate system* 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.

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.

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.

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 only available for text, images, metafiles, and grouped objects.

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.

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 for an object,

1. Click on the object to select it.
2. In the **Properties** window, click on the **Info** tab.
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 add an attribute for multiple objects in a base layer, 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.

Rename Attributes

Rename the attribute by clicking the button next to the attribute value. Type the new name in the New Attribute Name dialog.

Delete Attributes

Delete the attribute by clicking the button next to the attribute value.

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** page for Polyline Properties displays the *Coordinate system, Number of vertices,* and *Length.*

Polygon

The **Info** page for Polygon Properties displays the *Coordinate system, Total number of vertices, Number of sub-polygons (rings), Perimeter,* and *Area.*

Point

The **Info** page for Symbol Properties displays the *Coordinate system* and *Position* in X and Y units.

Rectangle

The **Info** page for Rectangle Properties displays the *Coordinate system, Perimeter* and *Area.*

Rounded Rectangle

The **Info** page for Rounded Rectangle Properties displays the *Coordinate system, Perimeter* and *Area.*

Ellipse

The **Info** page for Ellipse Properties displays the *Coordinate system, Perimeter* and *Area.*

Spline Polyline

The **Info** page for Spline Polyline Properties displays the *Coordinate system, Number of curves* and *Length*.

Range Ring

The **Info** page for Range Ring Properties 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

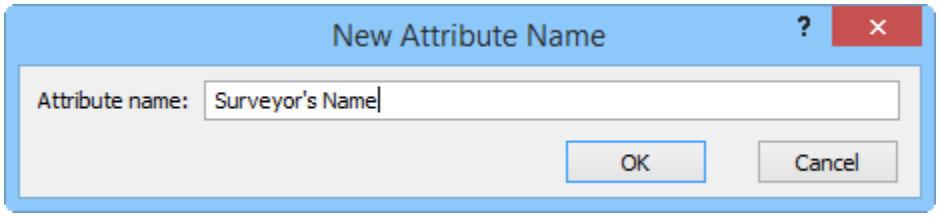
The **Info** page for Metafile Properties displays the *Type* of metafile, *Description, Number of records,* and *Size in bytes*.

Image

The **Info** page for Image Properties 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.

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.

Chapter 29 - Selecting and Arranging 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.

Several procedures are available to select objects in a plot window:

- Objects may be selected using the Contents window. To display the **Contents** window, use the **View | Show/Hide | Contents** command. Once displayed, click on the object in the list you wish to select. The object is then selected in the plot window. The object 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.
- 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.

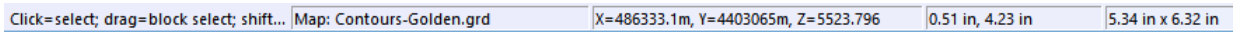
Selecting a Map Layer and Changing the Object ID

After creating a multi-layer map with a post map layer and a contour map layer, you can still modify the individual map layers.

Selecting Map Layers

An individual map layer can be selected in the multi-layer map by clicking on the layer in the plot window or clicking on the map layer in the Contents window. The easiest way to select a map layer in a multi-layer map is to click on the layer name in the **Contents** window.

However, you can also select the layer in the plot window with the mouse. Whenever two or more objects occupy the same position in the plot window, use the CTRL key and the left mouse button to select the desired object. The CTRL key allows you to cycle through the selection of overlapping objects. For example, if you want to select a contour map layer behind a post map layer, hold down the CTRL key and click until the contour map layer is selected. You can use the status bar to help you to determine which object is selected.



*The status bar at the bottom of the **Surfer** window indicates the selected object in the plot window. In this example, the status bar reports that the Map:Contours-Colorado.grd is selected.*

Creating Multiple Map Layers

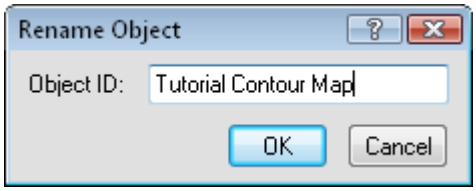
To create a map with two map layers:

1. Click the **Home | New Map | Contour** command.
2. In the **Open Grid** dialog, select a grid file, such as *Demogrid.grd*.
3. Click *Open* and the contour map is created.
4. Click anywhere on the contour map to select it.
5. Click the **Map Tools | Add to Map | Post Layer** command.
6. In the **Open Data** dialog, select a data file, such as *TutorWS.dat*.
7. Click *Open* and the post map layer is added to the contour map layer.

Renaming the Map Layers

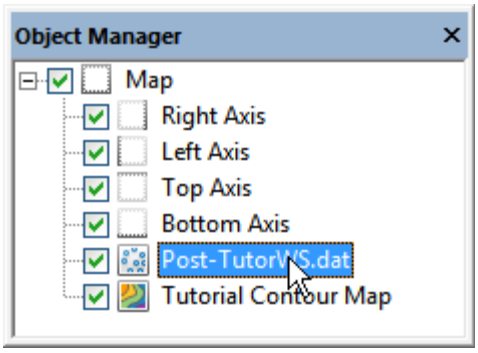
To select a map layer and assign or change the Object ID:

1. Click the contour map layer name in the **Contents** window. In this case, click the word *Contours-Demogrid.grd*. The status bar should now report *Map: Contours-Demogrid.grd*.
2. Click the Rename Object command or press the F2 key on the keyboard.
3. In the **Rename Object** dialog, type the name *Tutorial Contour Map* and click *OK*. The status bar and **Contents** window update to reflect the name change.



*Enter a new object name in the **Rename Object** dialog.*

4. Repeat steps 1-3 and rename the post map layer to *Tutorial Post Map*.



*Left-click the word Post-TutorWS.dat in the **Contents** window to select the post map layer.*

If you click on the *Tutorial Contour Map* layer in the **Contents** window, notice that the properties in the **Properties** window changes the title to **Properies - Map: Tutorial Contour Map**. When you rename an object in **Surfer**, the **Properties** window reflects the name change making it easier for you to keep track of the object you are editing. For example, if you have eight maps in the plot window, it is beneficial to change the map names to something meaningful to save time when trying to edit them.

Select Tool

Click the **Home | Selection | Select** command 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. 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. To select multiple objects in a base layer or group, first select the base layer or group and click the Features | Group | Start Editing command.

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, 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 All

Use the **Home | Selection | Select All** command, or press CTRL+A on the keyboard, to select all objects in the document. A bounding box surrounds all selected objects. To select all the objects in a base layer or group, first select the base layer or group and click the Start Editing command.

Deselect All

Use the **Home | Selection | Select All | Deselect All** command, 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

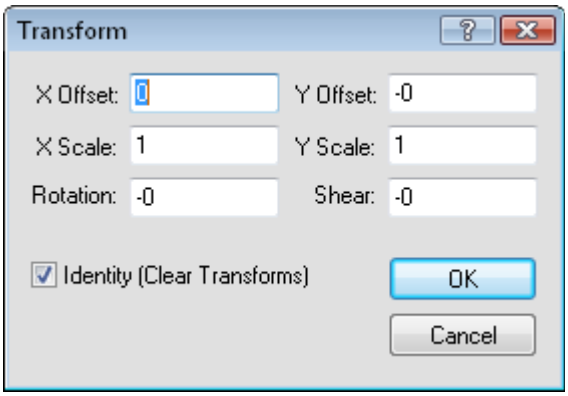
The **Home | Selection | Select All | Invert** command reverses the selected and deselected objects. All previously selected objects become deselected. All previously deselected objects become selected. A bounding box surrounds all selected objects.

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 opens 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

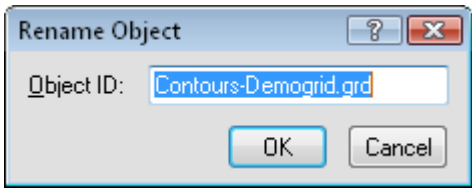
Foley, James D. and Andries van Dam (1990), *Computer Graphics: Principles and Practice*, Addison-Wesley, Reading, MA, 1174 pp.

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 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 the **Contents** window,

- 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

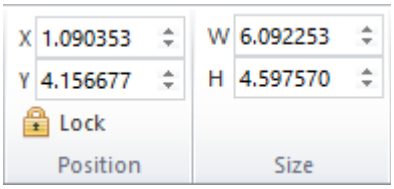
The **Layout** tab in the ribbon has the following commands:

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 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.

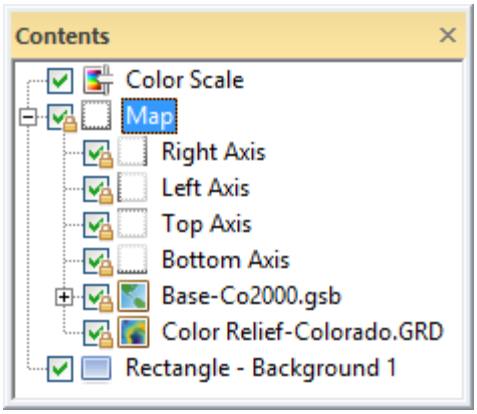
- To drag a handle with the mouse, move the cursor over the handle and then press and hold the left mouse button. Move the cursor to a new position. Release the left mouse button and the object is resized.
- To drag a handle with the keyboard, move the cursor over the handle, press and hold the SPACEBAR, 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

Objects in base layer can be moved when the Start Editing command is used on the base layer. When objects in a base layer are individually locked, that object cannot be moved individually. To move the object, unlock the object, click the **Features | Group | Start Editing** command, 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 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.

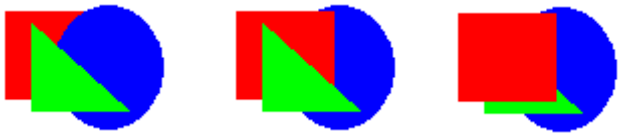
Bring Forward

The **Layout | Arrange | Bring to Front | Bring Forward** command 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 moves 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.

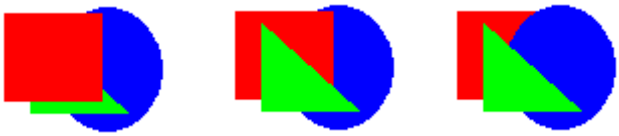
Send Backward

The **Layout | Arrange | Send to Back | Send Backward** command moves 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 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** to set the width off all selected objects equal to the width of the reference object.

Height

Click **Layout | Arrange | Size Both | Height** 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 resize. 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 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

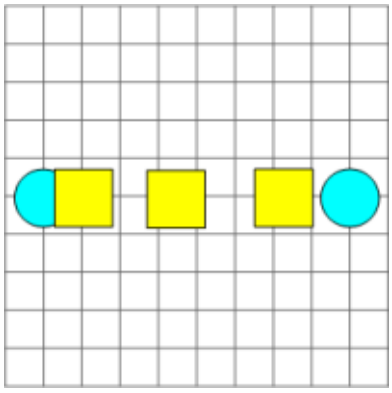
Click the **Distribute Horizontally** command when Align to Margins is active to distribute two or more objects from left to right with equal spacing between the page margins. All objects are aligned horizontally between the page margins. This means the bounding box size may change, and the left and 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

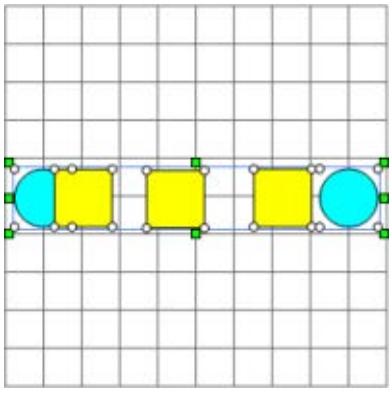
The following steps demonstrate how to distribute objects horizontally:

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.

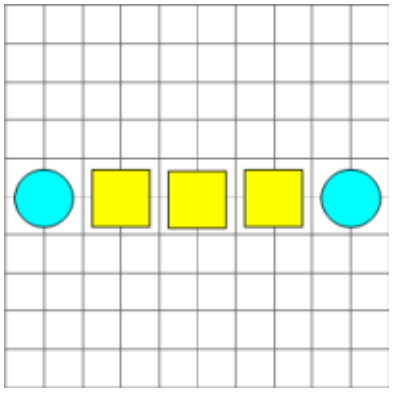
The interior objects are arranged between the outer most objects with equal spacing.



*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 **Distribute Horizontally** 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 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

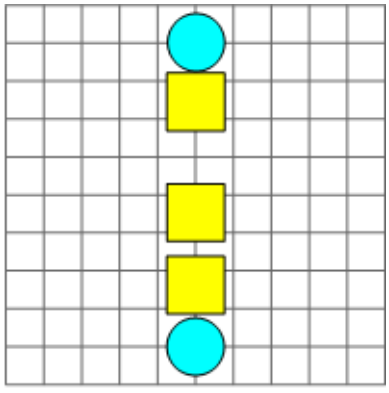
Click the **Distribute Vertically** command when Align to Margins is active to distribute two or more objects from top to bottom with equal spacing between the page margins. All objects are aligned vertically between the page margins. This means the bounding box size may change, and the top and bottom 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

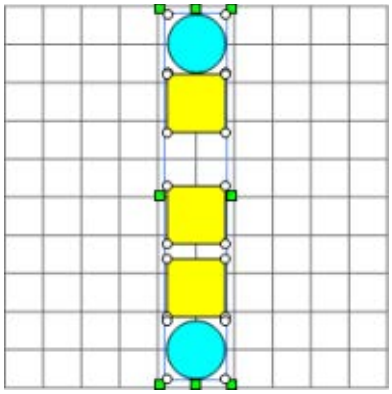
The following steps demonstrate how to distribute objects vertically:

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.

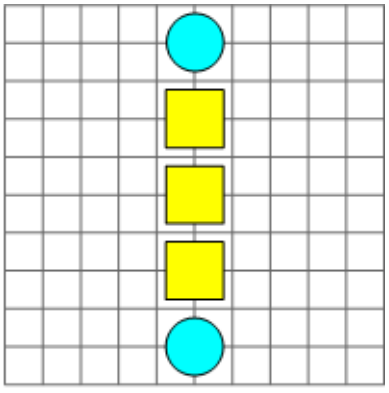
The interior objects are arranged between the outer most objects with equal spacing.



*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 **Distribute Vertically** 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

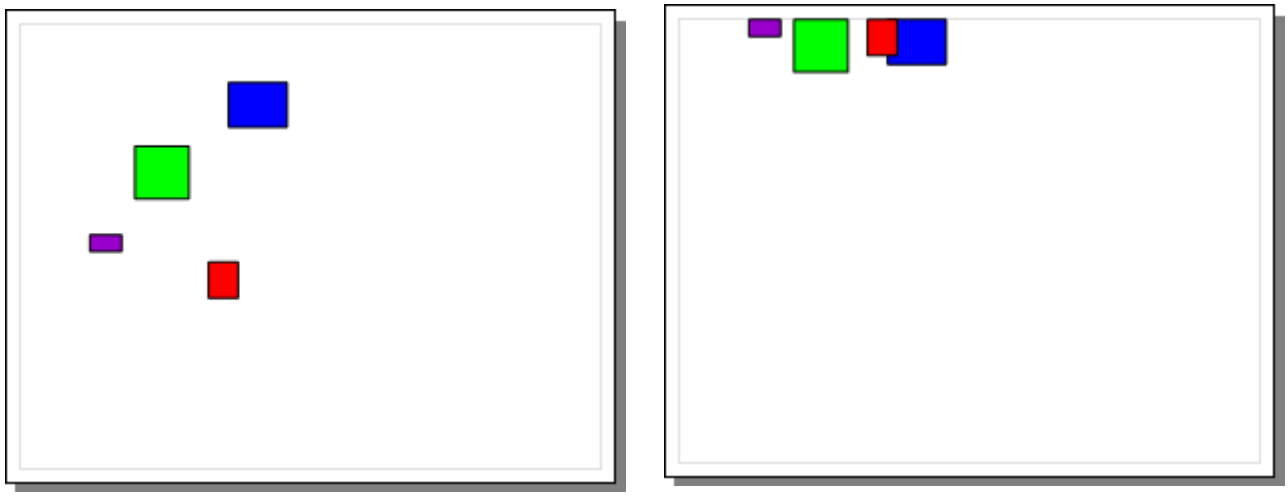
The **Align to Margins** option determines whether objects are aligned and/or distributed relative to the bounding box or relative to the page margins. Click the **Layout | Align | Align to Margins** command to turn the **Align to Margins** mode on or off. When **Align to Margins** is turned on, the 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.

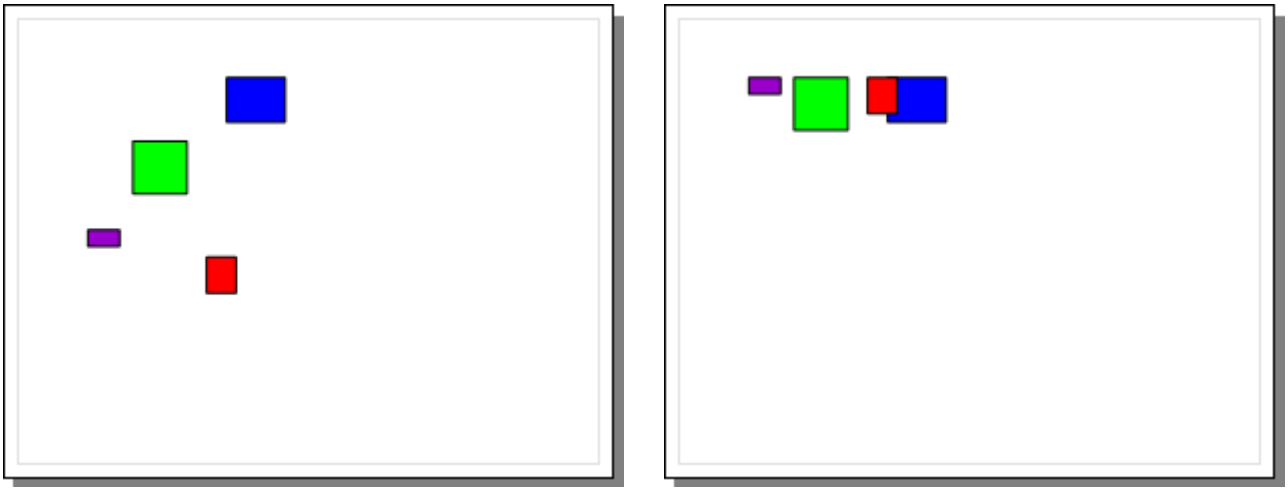
Align to Margins Example

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.



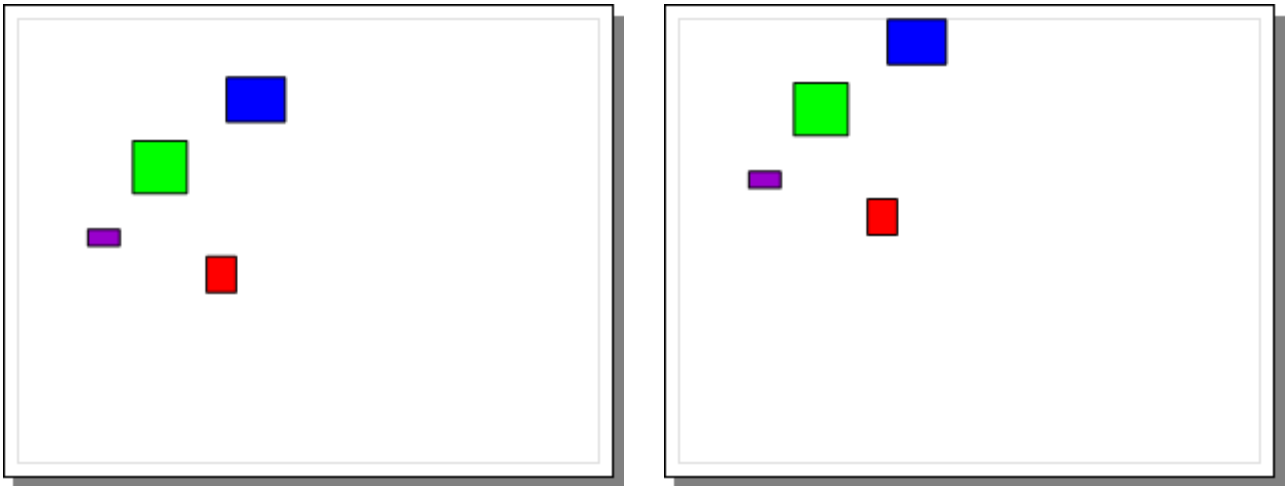
*The four rectangles were selected and **Layout | Arrange | Align | Top** was clicked while **Align to Margins** was checked. The four rectangles are aligned along the top margin.*

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 **Layout | Arrange | Free Rotate** 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

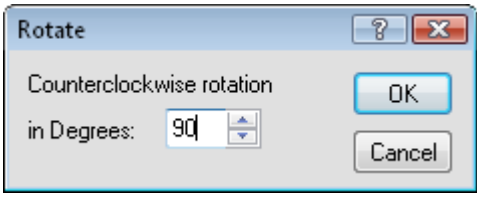
To clear rotation, use Home | Selection | Transform.

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 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.

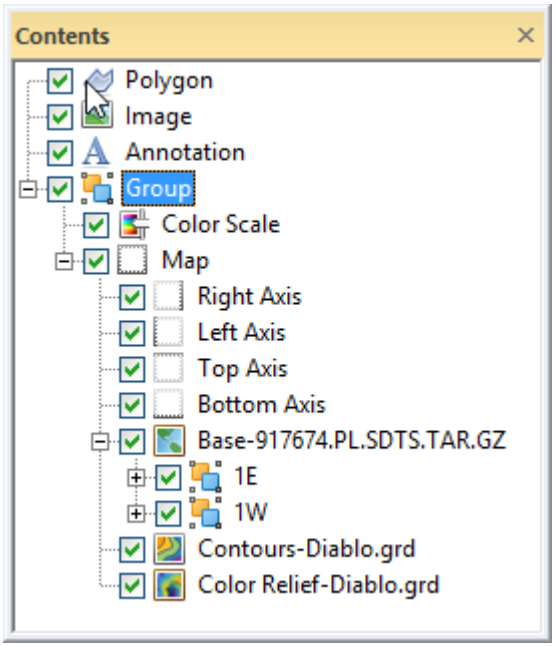


*Specify the degree of Counterclockwise rotation in the **Rotate** dialog.*

To clear rotation, use Home | Selection | Transform.

Group

The **Features | Group | Group** command 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.

To create a subgroup within a group or base layer, first select the base layer or group, and next click the Start Editing command. The objects within the group or base layer can then be combined into subgroups. Once you have finished, click the **Stop Editing** command.

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.

To move the location of individual items in the group object, click the Features | Group | Start Editing command. Move the item to the new desired location. Once you have completed moving the objects, click the Features | Group | Stop Editing command to exit the group.

A group object can be transformed with the **Home | Selection | Transform** command.

Ungroup

The **Features | Group | Ungroup** command 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.

Golden Software Boundary [.GSB]

Atlas Boundary [.BNA]

USGS [.DLG], [.LGO], [.LGS]

AutoCAD [.DXF]

Golden Software Blanking [.BLN]

Metafiles

Pasted objects

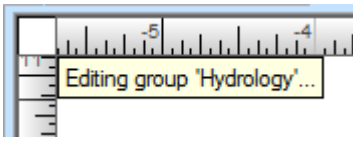
Editing Groups

Click the **Features | Group | Start Editing** command to manually add new objects to a base layer, an empty base layer, or a grouped object. Alternatively, right-click on a group or base layer and select **Start Editing** from the context menu.

Base layers and groups can be edited. The **Start Editing** command is enabled when a base (vector) layer or group is selected. Base (raster) layers cannot be edited. If you need to add vector features to a map with a base (raster) layer, add an empty base layer to the map.

Edited Group

The edited group is indicated in two ways. First, a message is displayed in the top-left corner of the plot window indicating which group is currently being edited. Second, the sizing handle color changes from a green to a gray. When the group is being edited, the editing commands operate within the group only. New objects can be added to the group. Multiple objects can be selected, deleted, and edited. Queries are performed on the objects in the group.



The message indicating the open group is displayed in the top-left corner of the plot window.

Stop Editing

The **Stop Editing** command replaces the **Start Editing** command while a group is being edited. Click the **Features | Group | Stop Editing** command to stop editing the group. Exit the group to select or modify objects outside of the group.

Editing Groups with Coordinate Systems

When the base layer and map have different coordinate systems, some editing functionality is disabled. This is because a geotransformation is applied to the base layer to display it in the map's target coordinate system. The following processes CANNOT be applied to a geotransformed base layer or objects within the geotransformed base layer:

- Create text
- Create a range ring
- Create a rectangle or rounded rectangle
- Create an ellipse
- Move or resize an object (including by using the size/position group or clicking and dragging the object or selection handles, or the Align commands)
- Rotate an object (with either Free Rotate or Rotate)
- Copy a layer or object

Removing the Geotransformation

If you want to perform one of the processes listed above, you must first change the map target coordinate system to the same coordinate system as the base layer you wish to edit. Alternatively, you can change the map to use an unreferenced local system. To do this:

1. Click on the *Map* object in the Contents window.
2. In the Properties window, click on the **Coordinate System** tab.
3. Click the *Change* button.
4. In the **Assign Coordinate System** dialog, note the current coordinate system.
5. Click the ⊞ next to *Local System*.
6. Click the Unreferenced local system and click OK.
7. Click on the *Base* map in the **Contents** window.
8. Click the **Features | Group | Start Editing** command.
9. Make any necessary adjustments.
10. Click the **Features | Group | Stop Editing** command.
11. Click on the *Map* object in the **Contents** window.
12. In the Properties window, click on the **Coordinate System** tab.
13. Click the *Change* button.
14. In the **Assign Coordinate System** dialog, define the coordinate system as it was assigned in step 4 and click *OK*.

Chapter 30 - Changing the View

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 rectangles, actual size, or to the full screen
Pan	Scrolls the window by "grabbing" the contents and dragging them
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 overlapping tiles
Tile Vertical	Arranges windows vertically so there are no overlapping 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, 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 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 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 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 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. 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.

Enter Zoom Rectangle Mode

Select the **View | Zoom | Rectangle** command 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 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 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 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 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.

Redraw

Click the **View | Redraw | Redraw** command 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** 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 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 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 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 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 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 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 to arrange multiple windows in a non-overlapped fashion side by side.

Reset Windows

Click the **View | Windows | Reset Windows** command 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 31 - Importing, Exporting, and Printing

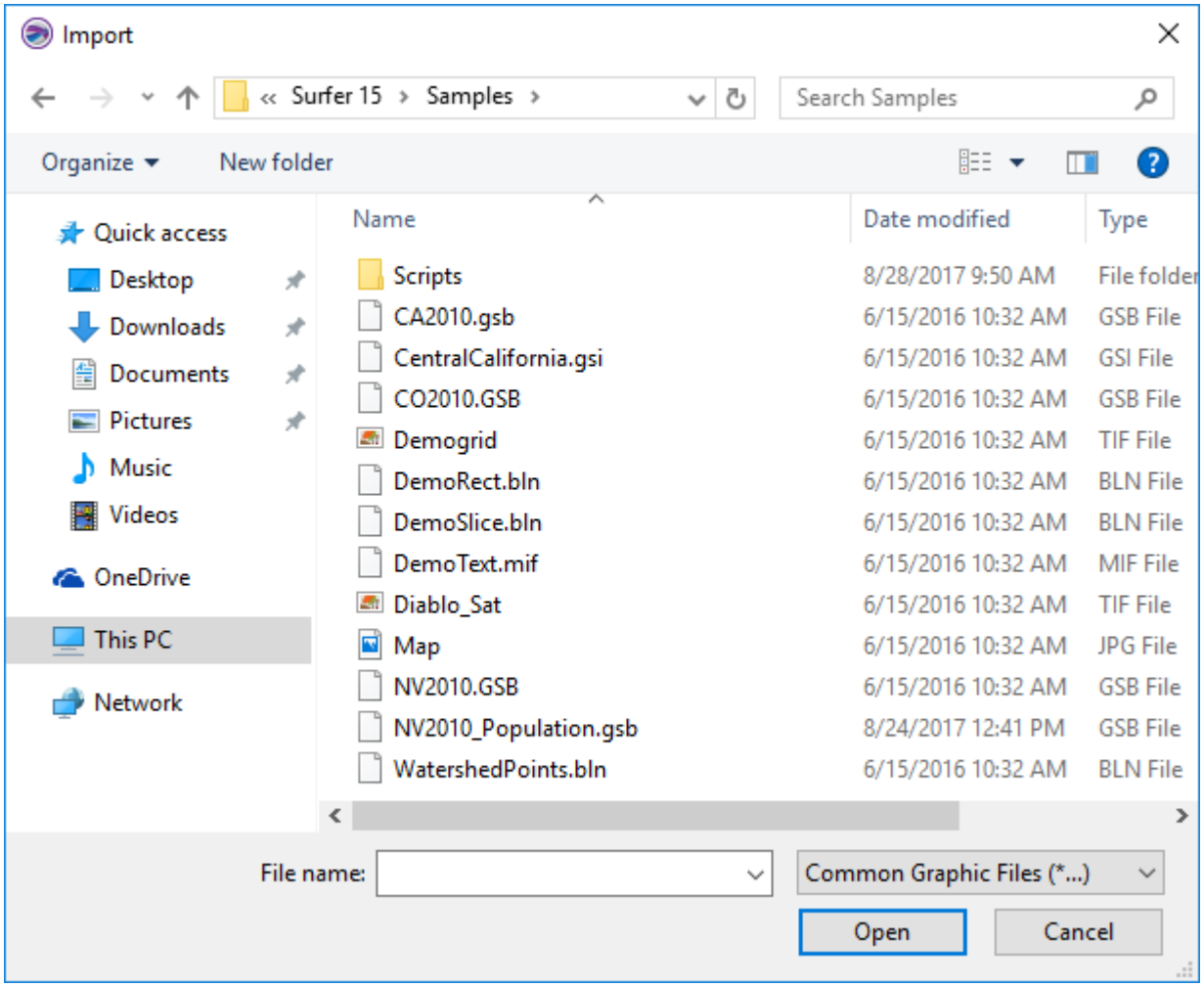
Import

You can import graphic files in **Surfer** with the **File | Import** command. The **File | Import** command is similar to the **Home | New Map | Base** command except that the file is imported as a grouped graphic object rather than as a map layer.

Click the **File | Import** command, or click the button on the Quick Access Toolbar, to import graphic "trim" objects, background graphics, or some form of annotation. Use Home | New Map | Base to import boundary information suitable for overlaying on top of other map types.

Import Dialog

The **File | Import** command in the plot opens 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.

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.

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 **File | Import** 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 **File | Import** 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)

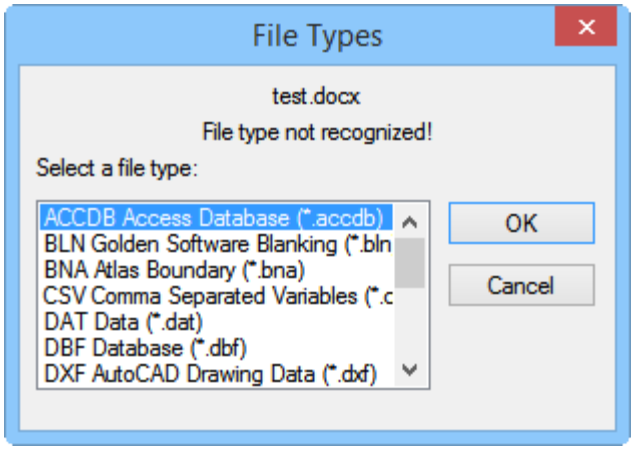
- EMF Windows Enhanced Metafile (*.emf)
- GEOJSON JSON Data Interchange (*.geojson, *.json)
- GIF Image (*.gif)
- GML Geograpy Markup Language (*.gml)
- GPX GPS Exchange Format (*.gpx)
- GSB Golden Software Boundary (*.gsb)
- GSI Golden Software Interchange (*.gsi)
- JPG Compressed BItmap (*.jpg, *.jpeg)
- 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)

Remarks

- To open Golden Software Blanking .BLN and Atlas Boundary .BNA files in the worksheet use File | Open rather than **File | Import**.
- 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

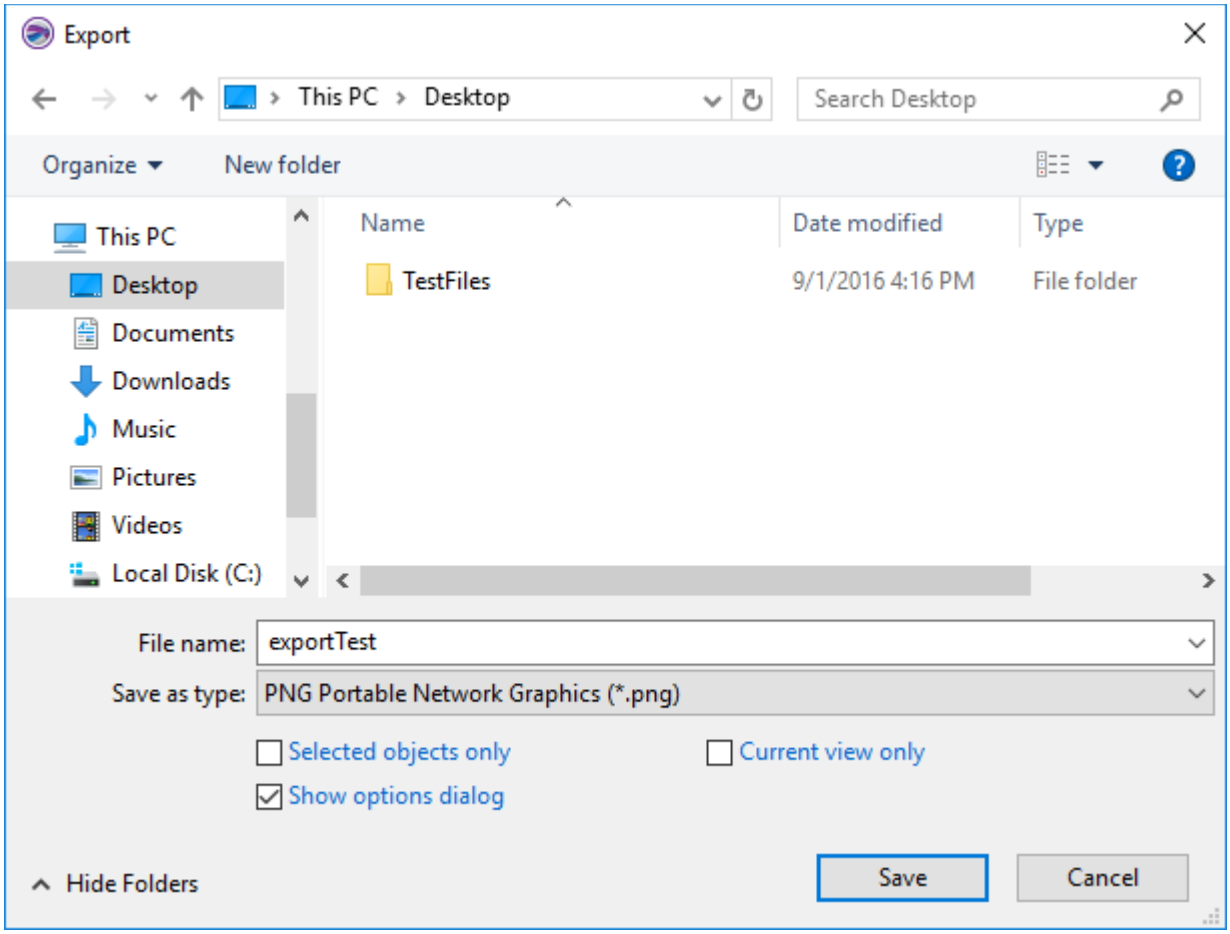
The **File | Export** command saves files as graphic files to use in other programs. Click the **File | Export** command, or the button on the quick access toolbar, to open the **Export** dialog. The **File | Export** command is disabled if there are no objects in the **Surfer** document.

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. There is no need to type in an extension because it is automatically added. If a file extension is typed in the box along with the file name, the file type is determined by the typed extension. For example, if MYPLOT.DXF is typed in the *File name* box, the resulting file is in the AutoCAD DXF format, no matter what is set in the *Save as type* field.

Projected Coordinates

If the map that you are exporting is in a defined coordinate system, the Spatial References tab will appear after clicking *Save*. This dialog allows you to specify the file format to which you want to save the projected information. 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.

Export Format Types

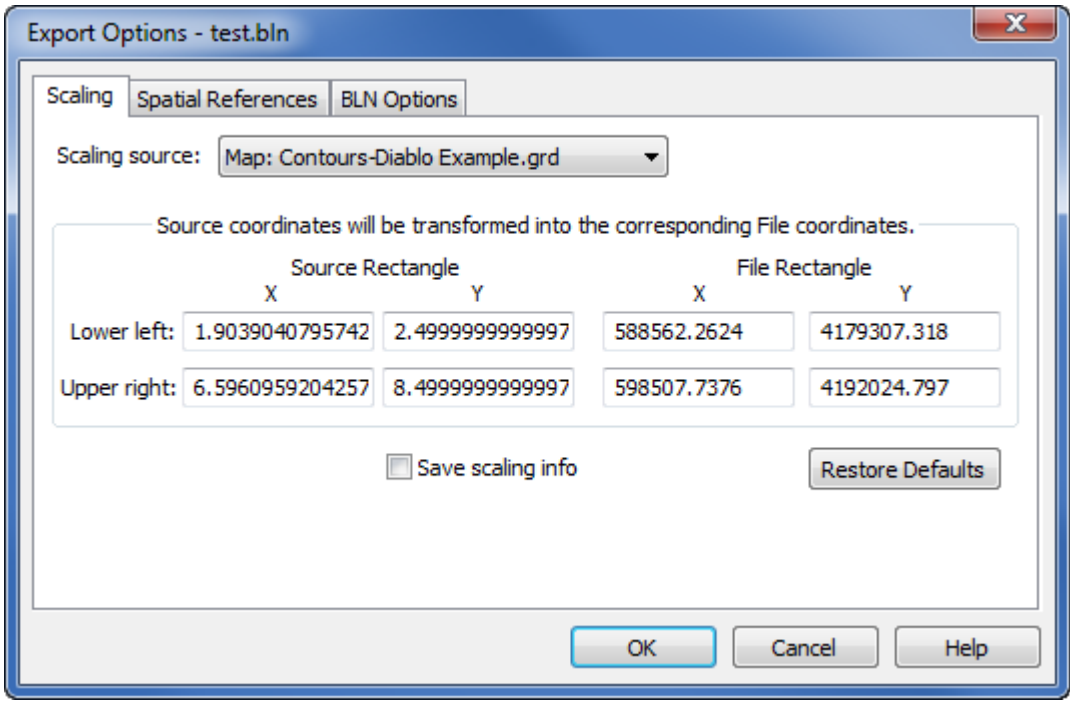
The file specific page in the **Export Options** dialog is specific to the export type you selected.

Surfer supports the following export format types:

- BLN Golden Software Blanking (*.bln)
- BMP Windows Bitmap Image (*.bmp)
- BNA Atlas Boundary (*.bna)
- CSV XYZ points (*.csv)
- DAT XYZ points (*.dat)
- DXF AutoCAD Drawing (*.dxf)
- EMF Windows Enhanced Metafile (*.emf)
- EPS Encapsulated Postscript (*.eps)
- GIF Image (*.gif)
- GSB Golden Software Boundary (*.gsb)
- GSI Golden Software Interchange (*.gsi)
- JPG Compressed Bitmap Image (*.jpg, *.jpeg)
- KML Google Earth KML (*.kml)
- KMZ Google Earth KMZ (*.kmz)
- MIF MapInfo Interchange Format (*.mif)
- PDF (Vector) (*.pdf)
- PDF (Raster) (*.pdf)
- PNG Portable Network Graphics (*.png)
- PNM Image (*.pnm)
- RGB SGI-RGB Image (*.rgb, *.rgba, *.bw)
- SHP Esri Shapefile (*.shp)
- SUN Sun Raster Image (*.ras, *.sun)
- TGA Targa (TrueVision) (*.tga)
- TIF Tagged Image (*.tif, *.tiff)
- WMF Windows Metafile (*.wmf)
- X AVS X-Image (*.x, *.ximg)

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

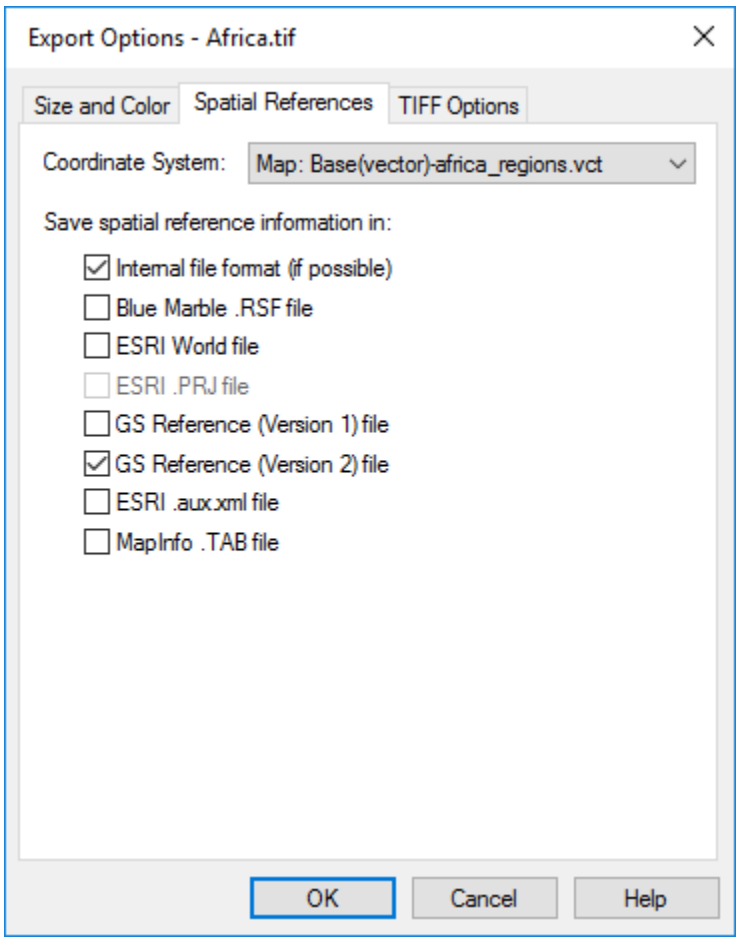
Check the box next to the *Save scaling info* option to save the scaling information to be stored for future use.

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

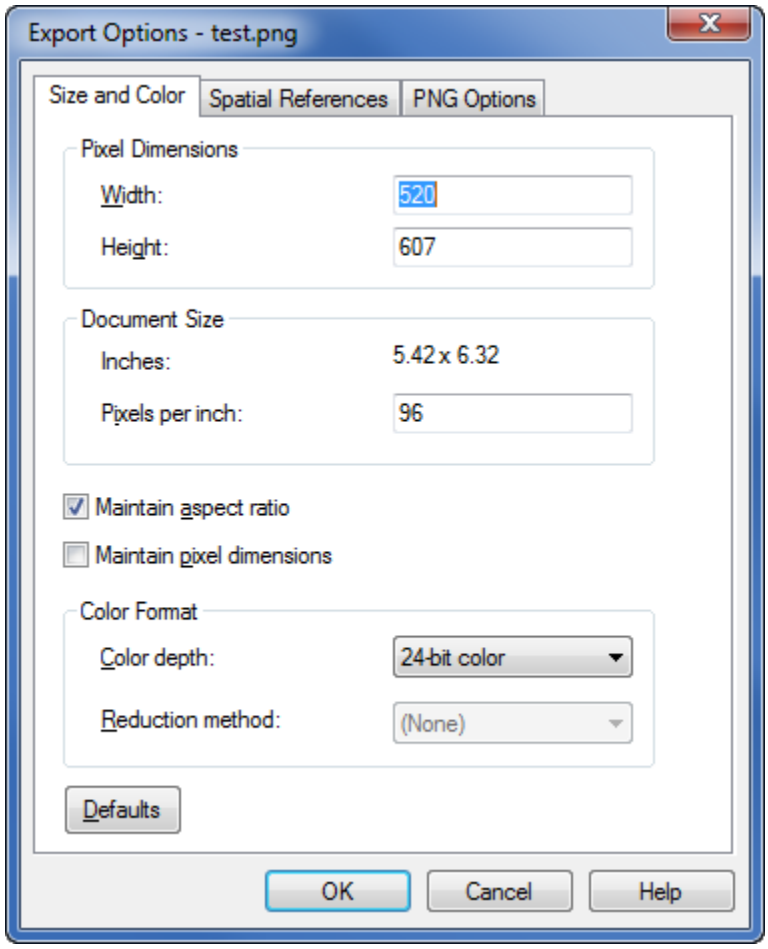
Environmental Sciences Research Institute (Esri) supports an xml-based format for storing georeference information in a separate file. Various Esri programs will import the information in this file when importing grid files, such as Surfer .GRD files.

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.

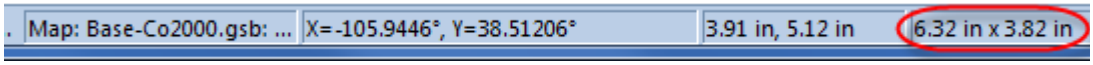
Pixels Per Inch

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.

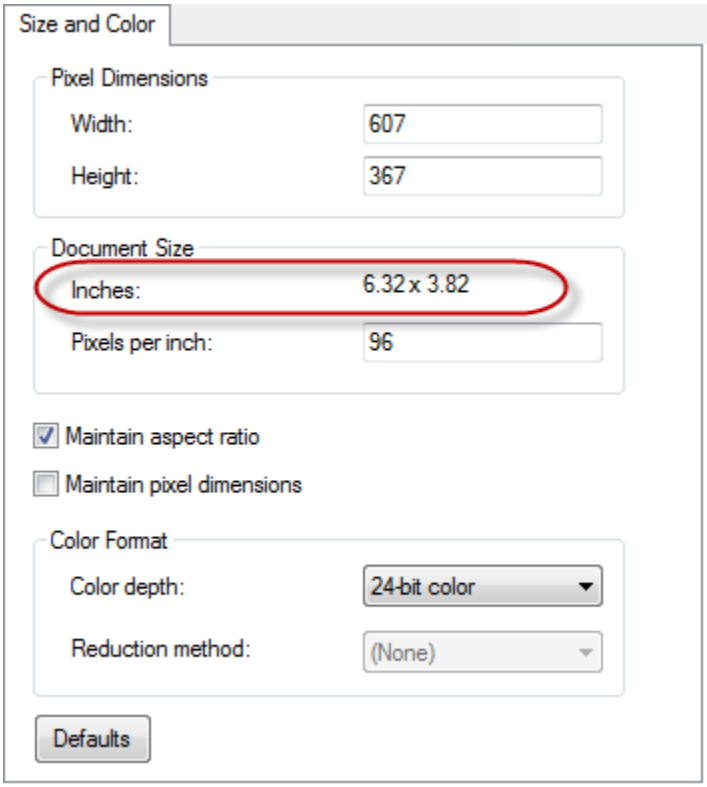
Example

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 Colo*r. 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.

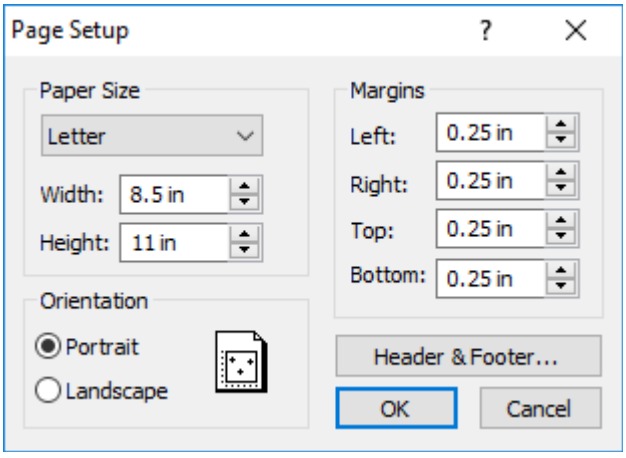
Page Setup - Plot

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
- Right-click on the plot window and click **Page Setup**
- Click the **Layout | Page Setup | Margins | Custom** command.
- 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.

Quickly 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.

Quickly 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*.

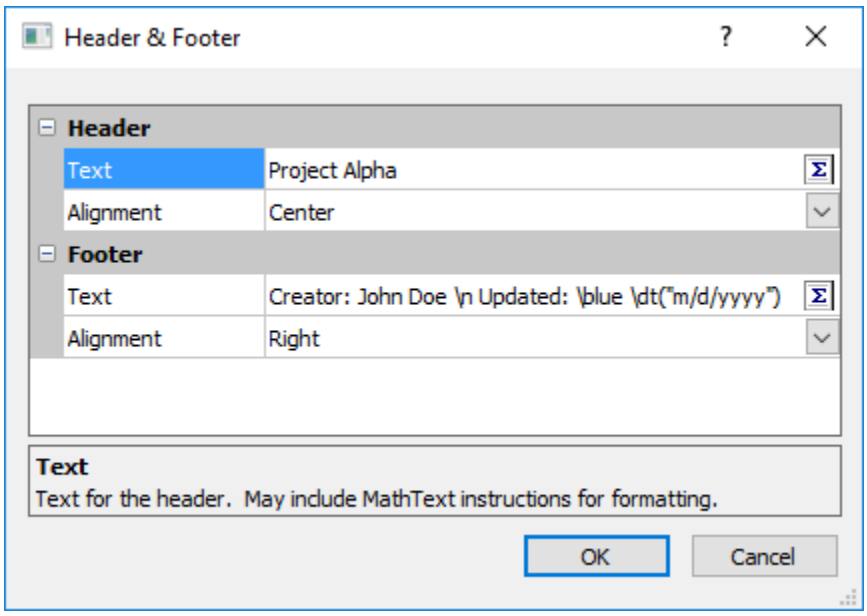
Setting the margins does not move an existing map on the page. Margins can be displayed on the page by checking the *Show margins* option on the User Interface page of the **Options** dialog.

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 **Layout | Display | Footer** command 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 Σ 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

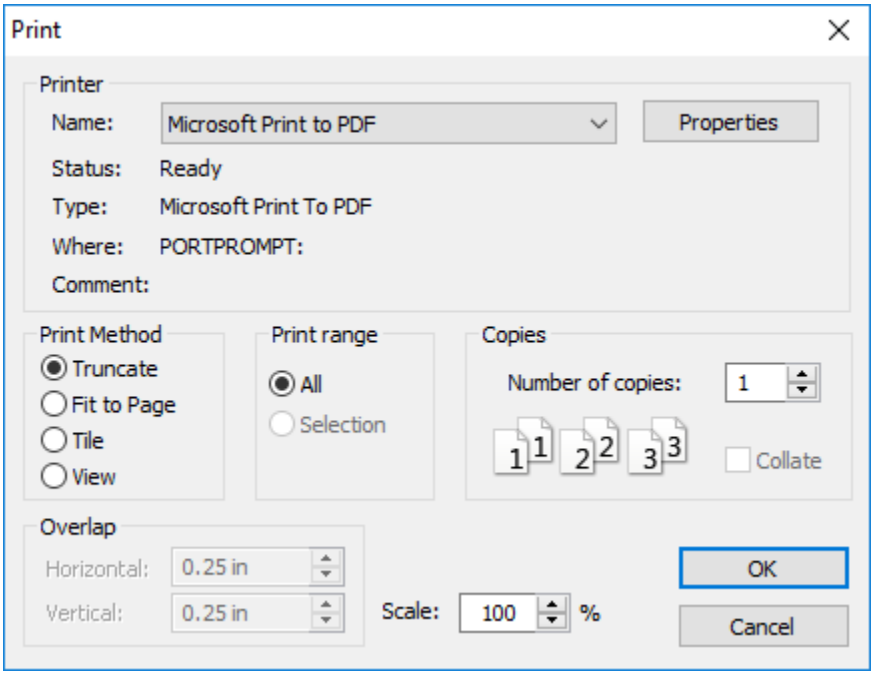
Set the header or footer text alignment relative to the page in the *Alignment* field. Click the current option and select *Left*, *Center*, or *Right* from the list. Use the Text Editor to set the text justification to the left, center, or right.

Print - Plot

Click the **File | Print** command 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 multiple page documents are printed two or more times.

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

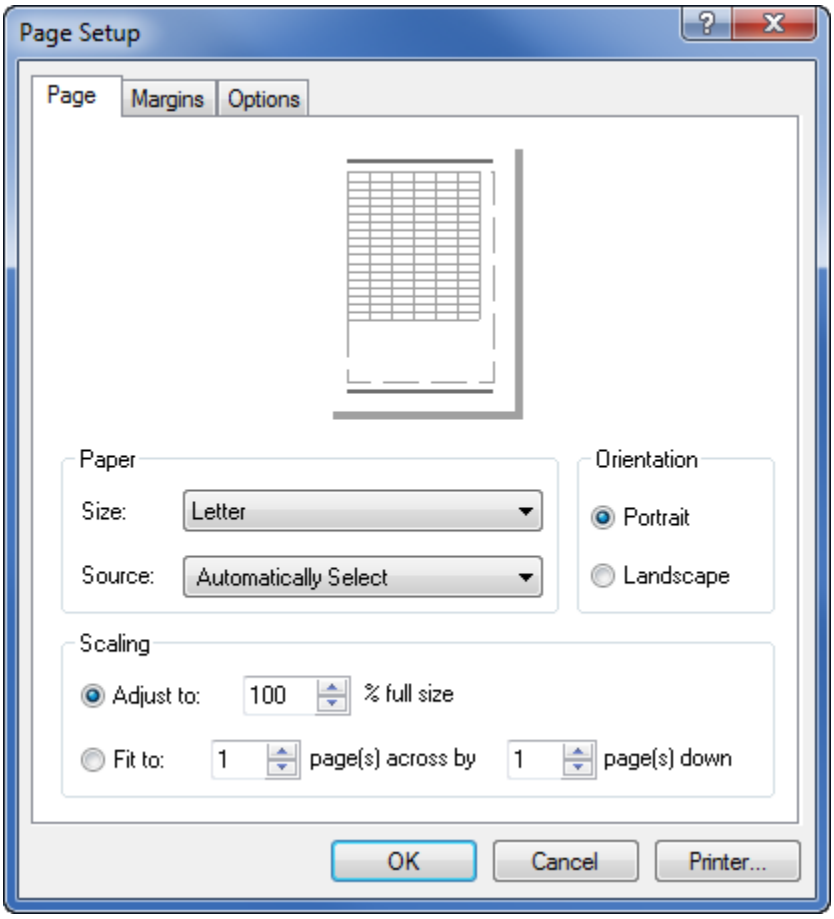
Before printing the worksheet, the page format of the worksheet can be set by clicking **File | Page Setup**. The **Page Setup** dialog controls items such as printing headers and footers, centering the data on the page, showing grid lines, etc. There are three pages in the **Page Setup** dialog: **Page**, **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

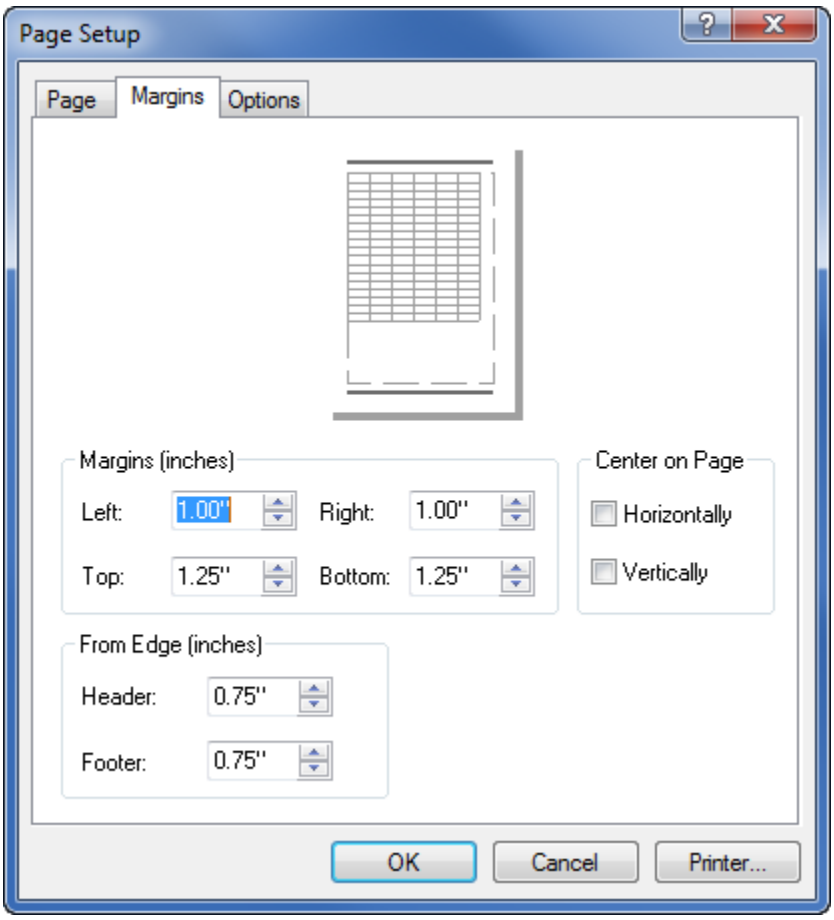
The amount of data in the worksheet determines how many pages are required to print the worksheet. The *Fit to __ page(s) across by __ page(s) down* option tells the program to print the worksheet at 100% scale or less. This option does not automatically scale the printed worksheet greater than 100%. 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)

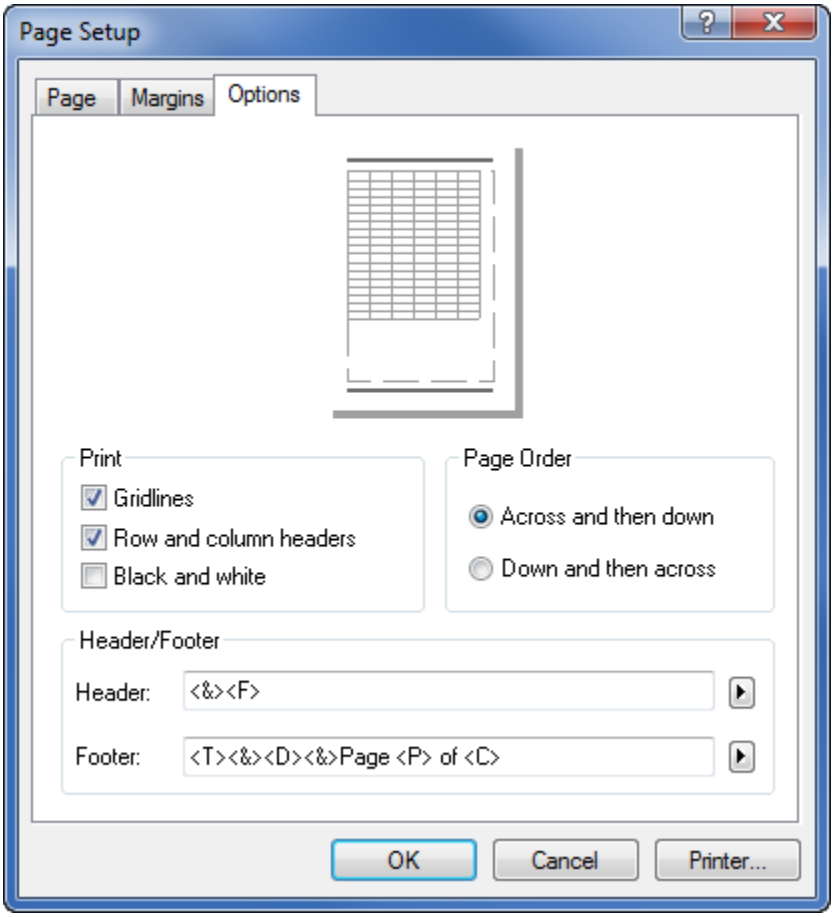
Headers and *Footers* are the only items that print outside of the margins. The *From Edge (inches)* group options controls how far the *Header* or *Footer* is printed from the edge of the page. If these values are greater than the *Top* or *Bottom* margins, it is possible that the worksheet data can 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<&>< F><&><D> to print out a name, file name, and date:

```
Joe
Smith        COLORADO.DAT        01/05/10
```

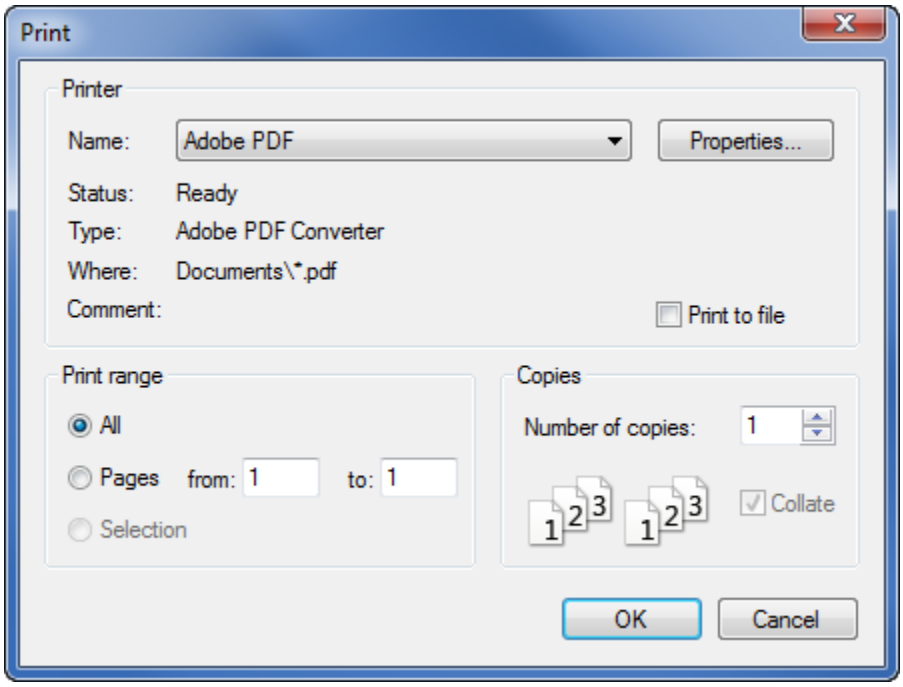
To print centered text use the "&" operator one time, such as <&>< F>.

```
FILENAME.DAT
```

Print - Worksheet

Click the **File | Print** command 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 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 32 - Options, Defaults, and Customizations

Surfer's default settings and advanced options are customized by clicking the **File | Options** command. 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 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.
Default Properties	Set line, fill, symbol, font, and label format default attributes.

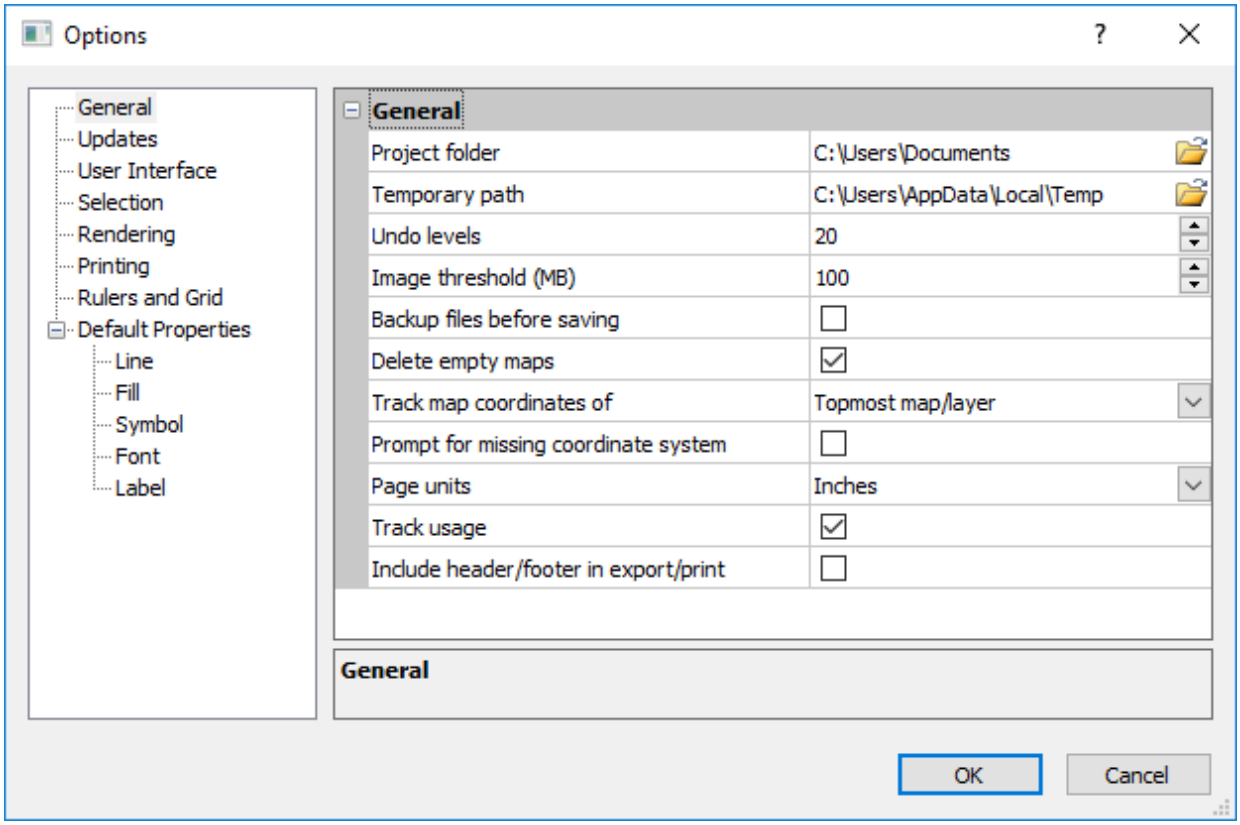
Advanced default options are available for most menu commands by clicking the File | Defaults command.

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 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.

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.

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 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; it's 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

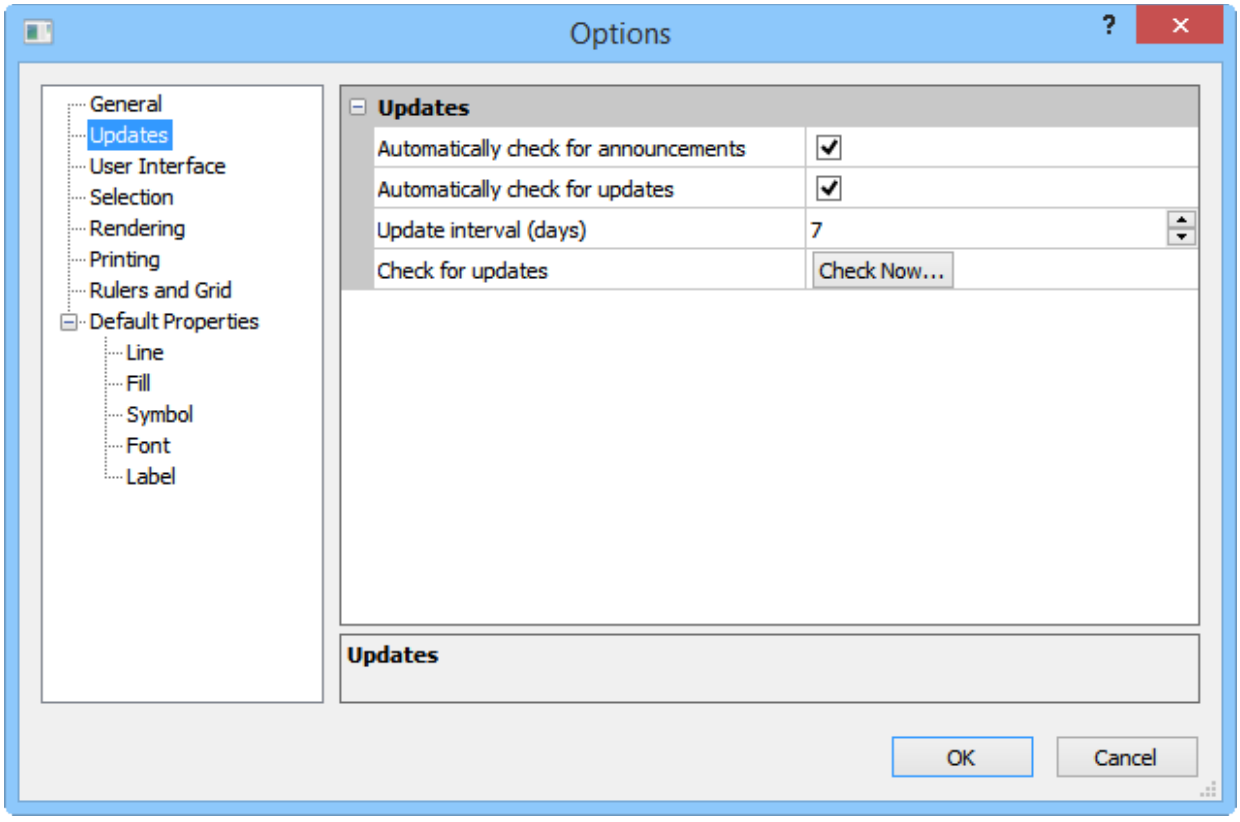
Check the *Include header/footer in export/print* check box to include the header/footer when printing or exporting the plot window. By default the header and footer are not included in printed and exported output. Clear the *Include header/footer in export print* check box to only display the header/footer in the plot window.

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.

Automatically Check for Announcements

Check the *Automatically check for announcements* 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.

Automatically Check for Updates

Check *Automatically check for updates* to allow Golden Software to automatically check for program updates (i.e. **Surfer** 11.0 to **Surfer** 11.1) according to the user defined time interval. After the specified time, 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.

Update Interval (days)

Surfer will automatically check for updates after the specified interval of time has passed. Specify the update time interval in days by highlighting the existing value and typing a number in the *Update interval (days)* box or clicking the to increase or decrease the number of days. This option can be set to a value between one and 90.

Check For Updates

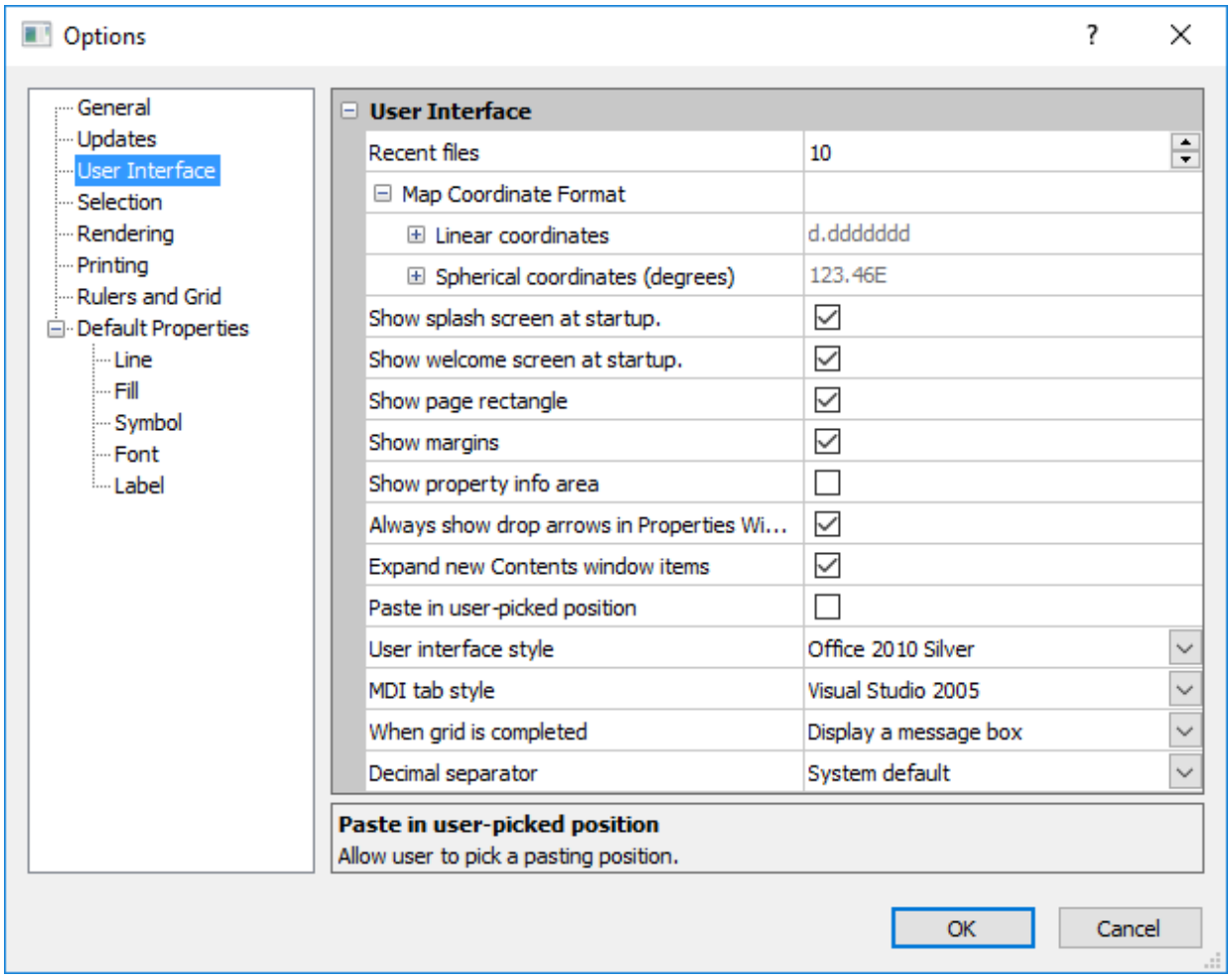
Click the next to the *Check for updates* command to check for program updates. Before using this command, make sure your computer is connected to the Internet. Follow the directions in the **Internet Update** dialog to complete the update if an update is available.

Options - User Interface

Set the user interface environment on the **User Interface** page in the Options dialog.

Options Dialog

Click the **File | Options** command 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. To change the number of files, click the to increase or decrease the value or highlight the value and type a new number.

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.

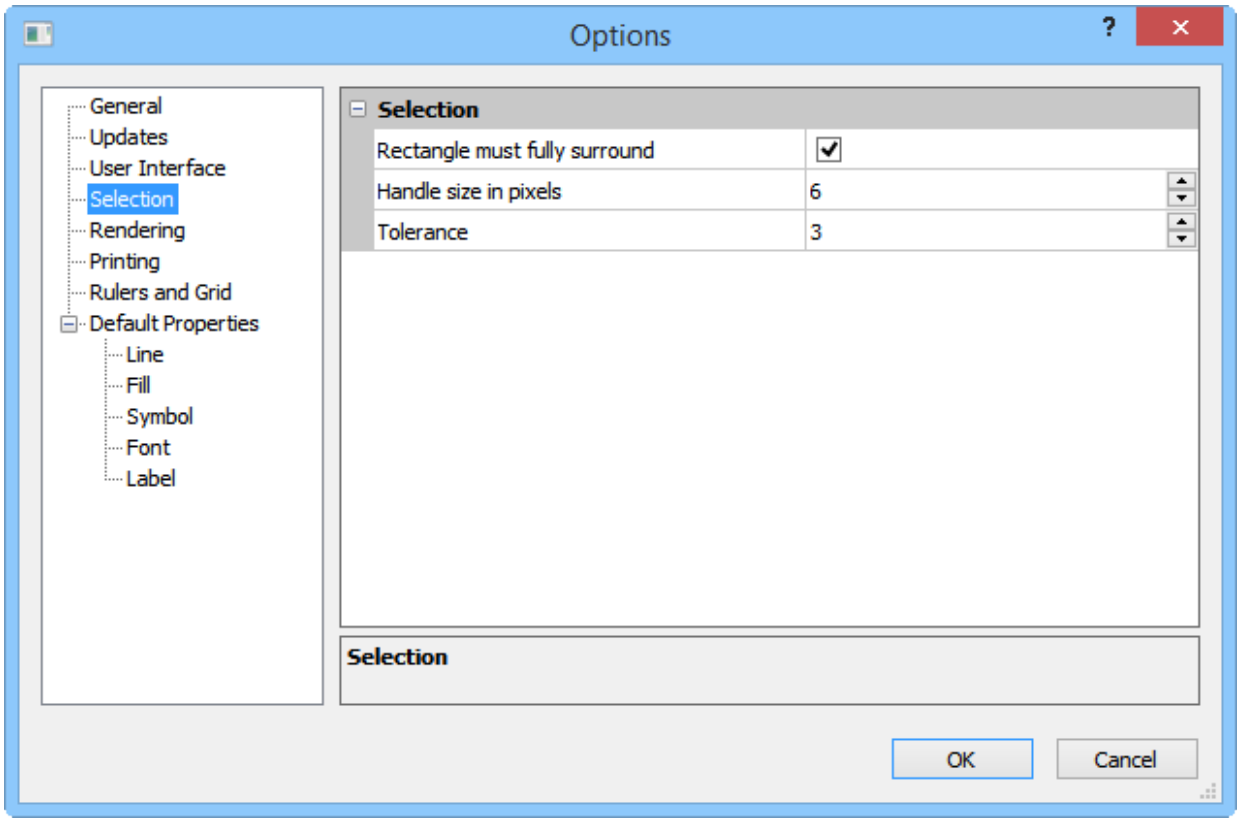
The *Decimal separator* setting is mainly used to import data files from different locales correctly in your instance of **Surfer**. For example, a CSV file uses the period (.) for the decimal separator and a comma (,) for the data delimiter in the English locale. However a CSV file uses the comma (,) 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 - Selection

Set the options for how objects are selected on the **Selection** page in the Options dialog.

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

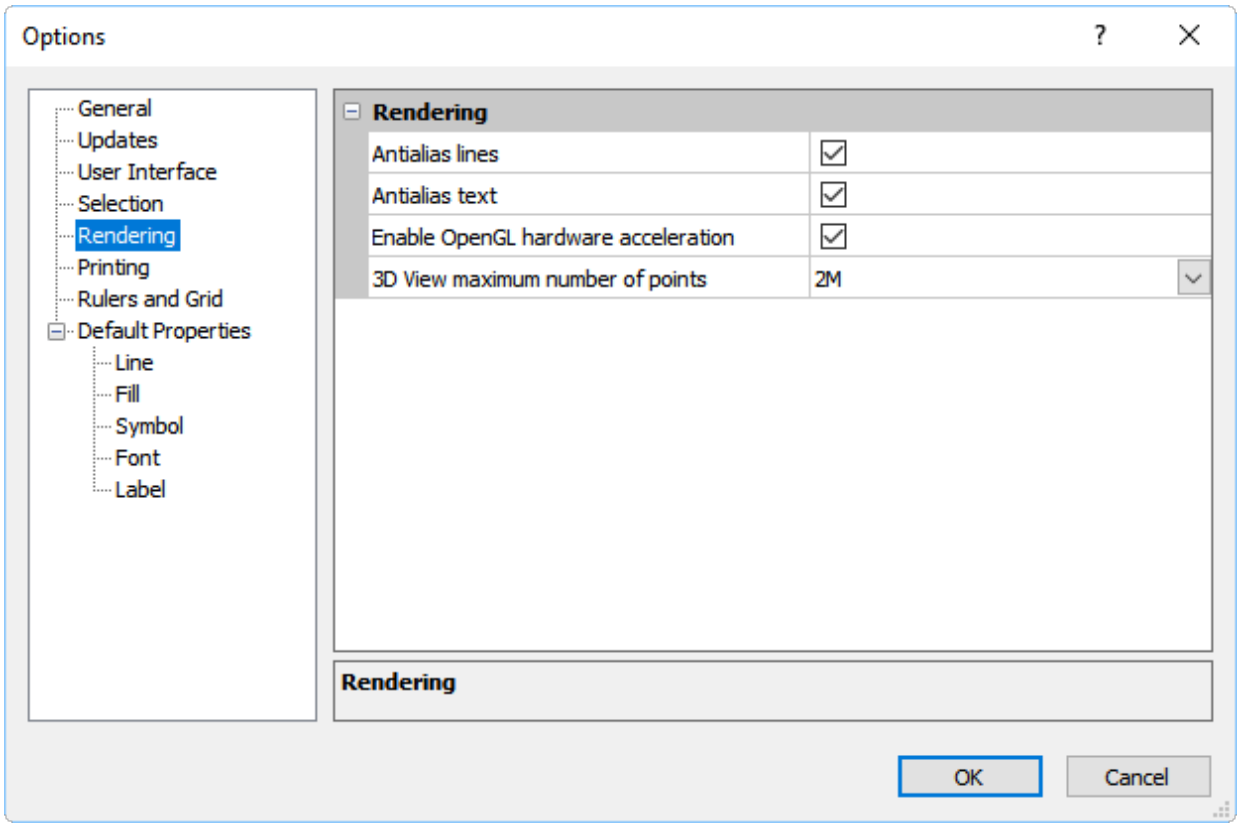
The *Tolerance* option controls how far away the cursor can be from an object when clicking to select the object. The tolerance setting range is from zero to 25 pixels. When the tolerance size is zero, the cursor must be directly on the object to select it. The tolerance size does not affect the size 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.

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. 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.

3D View Maximum Number of Points

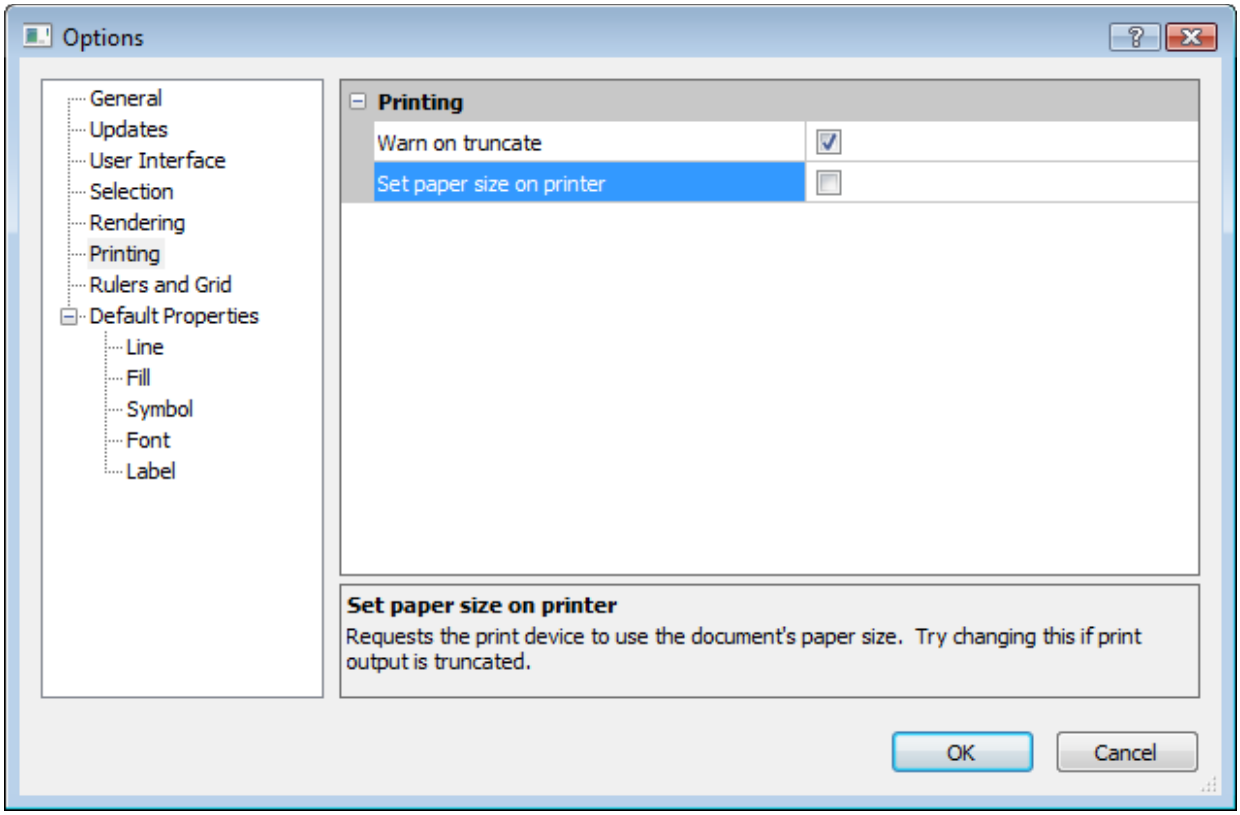
Large point cloud layers can use a lot of resources to render in the 3D view. The number of points displayed in the 3D view is limited by the *3D View maximum number of points* option. If Surfer is hanging or slow to respond when viewing a point cloud in the 3D view, reduce the *3D View maximum number of points* option. The default is 4 million (4M) 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 *3D View maximum number of points* value in the **Options** dialog to apply the change.

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.

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

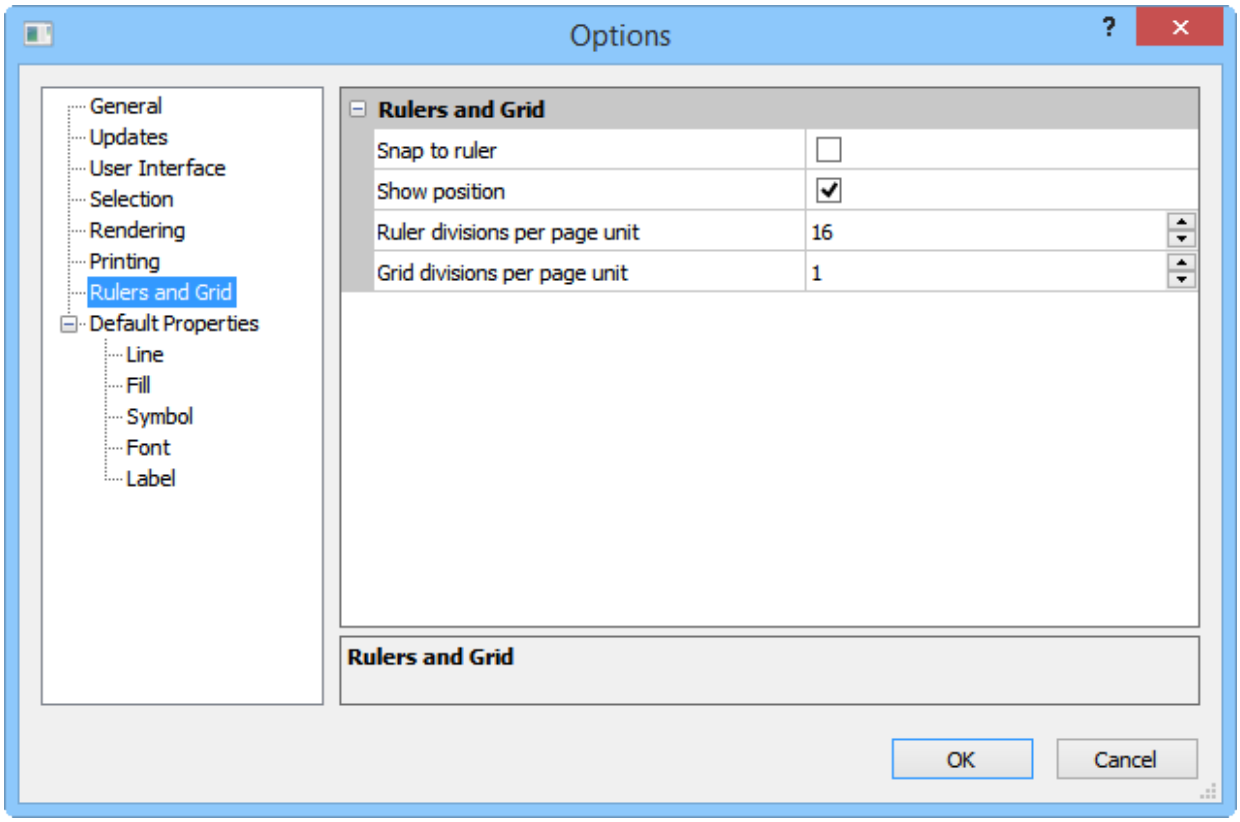
The *Set paper size on printer* option controls the page sizes available for printing. By default, this option is unchecked. When unchecked, the paper size on the **Page Setup** dialog is not automatically sent to the printer. The printer paper size will need to be set separately in the File | Print 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.

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

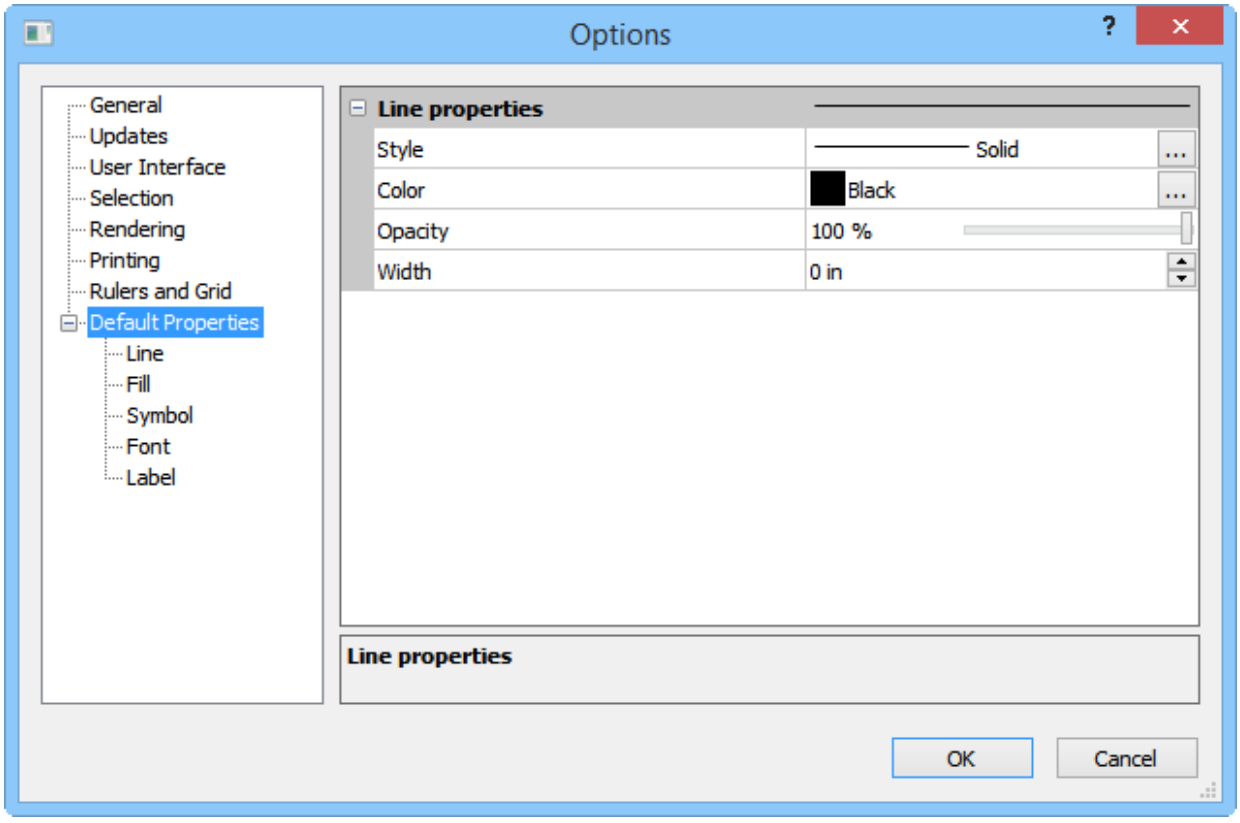
The *Grid divisions per page unit* option controls the number of divisions per page unit on the grid. Select a value between 1 and 99. To change the value, highlight the existing value and type a new number or click the to increase or decrease the number of divisions. The number of divisions are in page units. Set the page units on the General page of the **Options** dialog.

Options - Default Properties

Use the **Default Properties** pages in the Options dialog to specify the default line, fill, symbol, font, and label properties used when creating new objects.

Options Dialog

Click the **File | Options** command to open the **Options** dialog. Click on the *Line, Fill, Symbol, Font,* or *Label* link under the *Default Properties* option on the left side of the dialog to open the appropriate default setting page.



*Customize the Defaults options 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.

Default Settings

The **File | Defaults** command opens the **Defaults** dialog. The **Defaults** dialog controls most of the default settings used in **Surfer's** commands. 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.

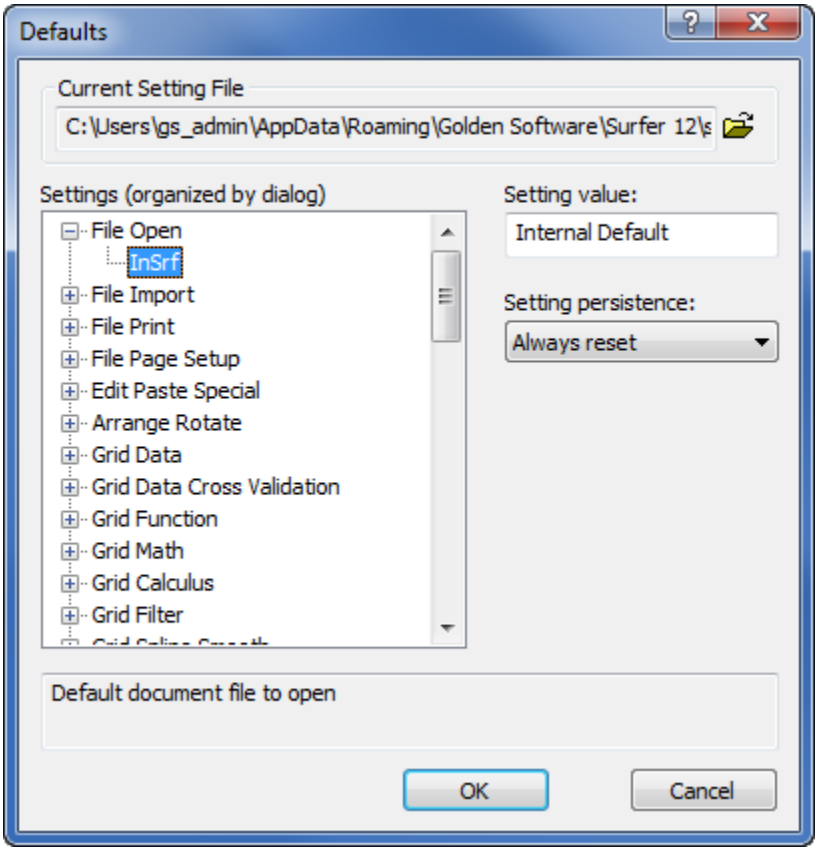
Changing Defaults in the Dialog

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 15 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 15 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 15\Surfer.exe" /SET "C:\Documents and Settings\<User Name>\Application Data\Golden Software\Surfer 15\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 15.

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 Surfer 15, 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)

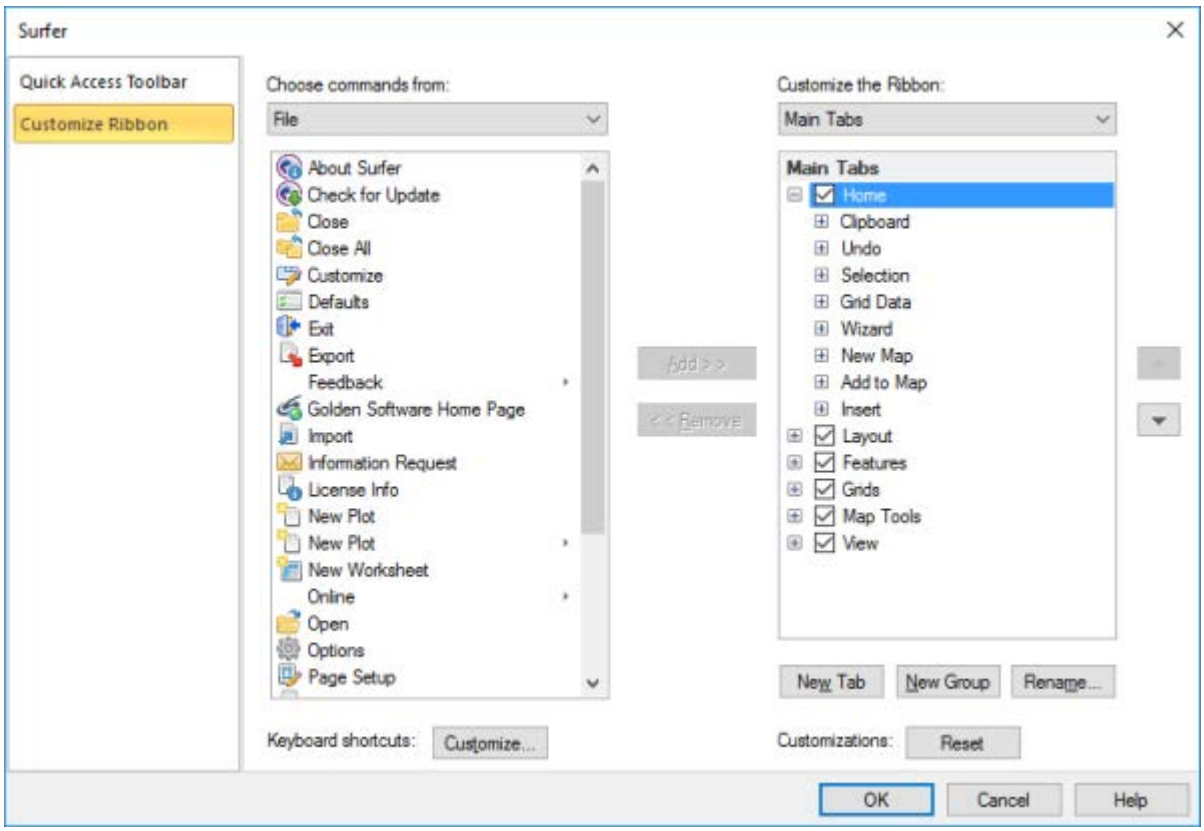
	3 = Thousands and Absolute Value (use thousands and absolute values)
DateFormat	The date format is entered as a number and is not enclosed in double quotes. The number values are listed on the wksDateFmtType Values page. Select the desired format and use the *Value* from this page in the label property 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 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 Shortucts

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

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 SHIFT+DEL	Cut the selected objects to the clipboard

CTRL+C or CTRL+INSERT	Copy the selected objects to the clipboard
CTRL+V or SHIFT+INSERT	Paste the clipboard contents into the plot window
CTRL+Y	Redo the previous Undo command
CTRL+Z or ALT+BACKSPACE	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

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 ALT+BACKSPACE	Undo the last command
CTRL+Y	Redo the last undo
CTRL+X	Cut the selected text to the clipboard

CTRL+C or CTRL+INSERT	Copy the selected text to the clipboard
CTRL+V or SHIFT+INSERT	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

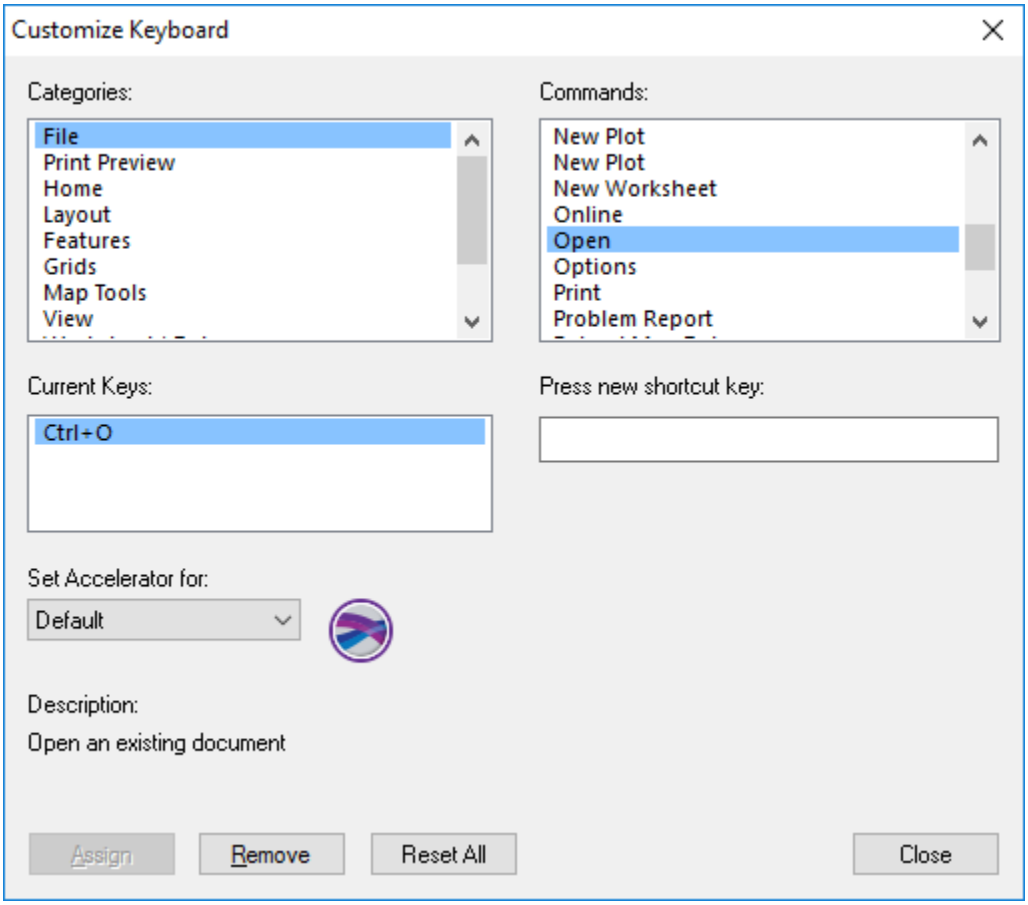
Worksheet Window

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 SHIFT+INSERT	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 *Current Keys* 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 33 - Automating Surfer

Golden Software **Scripter**™ is a program for developing and running scripts. **Surfer** operations 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.

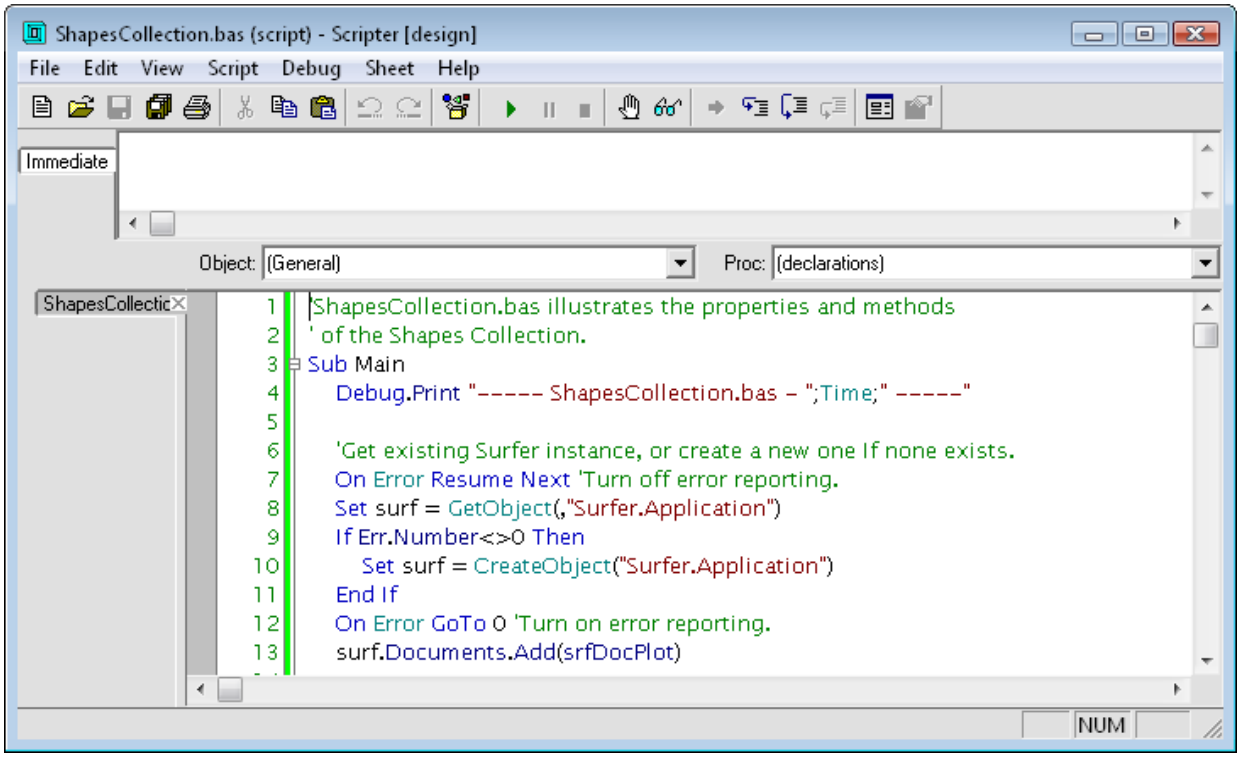
As an example, suppose you are monitoring groundwater levels in an area. Each week you collect XYZ data, create a contour map from this data, print the map, and save it in a **Surfer** file [.SRF]. The example scripts provide examples that automate these tasks. The sample script asks for the XYZ data 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** CD-ROM or download instructions. If you need new download instructions, contact technical support.

Scripter will open whichever version of **Surfer** was opened last. For example, If you want to run a script in **Surfer 8**, open **Surfer 8**, close it, then run the script and it will use **Surfer 8**. If you want to run a script in **Surfer 15**, open **Surfer 15**, close it, run the script and it will use **Surfer 15**.



*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.

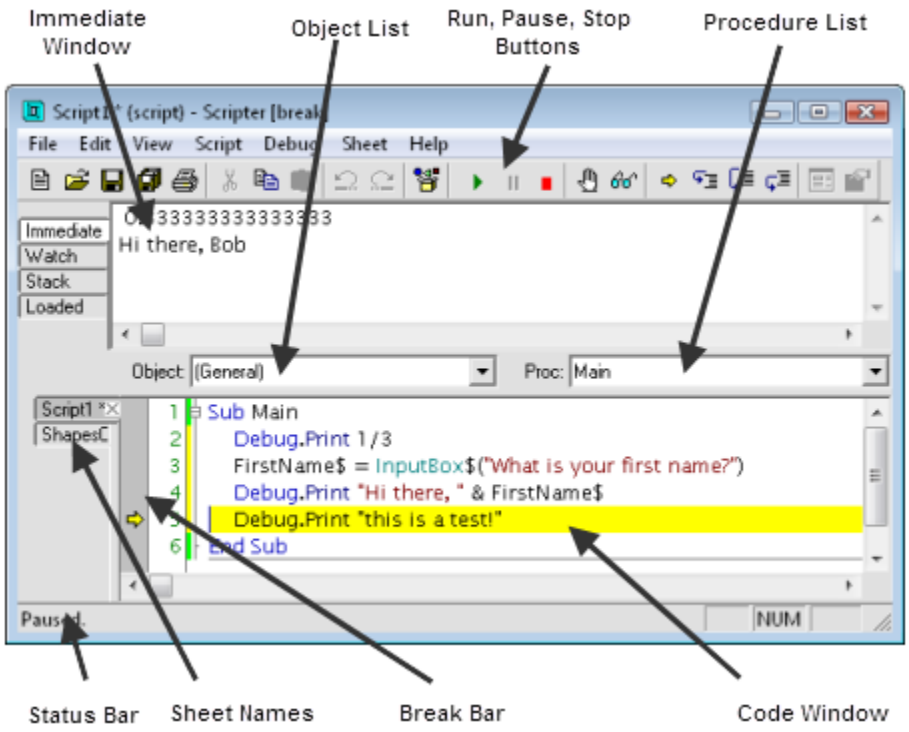
In addition to being a scratch area for evaluating language statements, the immediate window shows debugging information. The output from the `Debug.Print` statement and the value of

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

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* help book 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 15\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 15\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 15\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 15\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 the **Scripter** BASIC 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), *VB and VBA in a Nutshell: The Languages*, O'Reilly & Associates, Inc., Sebastapol, CA, 633 pp.

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 15\Scripter\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 15\scripter\scripter.exe" -x
"c:\temp\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
Software\Surfer\Samples\Demogrid.dat", _
ExclusionFilter:="x<4", xMin:=4, Algorithm:=srfKriging, DupMethod:= srfDupFirst)
```

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 window"
```

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, DupMethod:=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: E") is a one argument range of columns
[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:

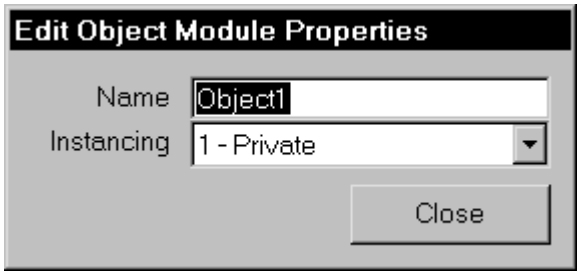
```
Public universal_data As String
```

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 XE "examples:class module" &<<![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.

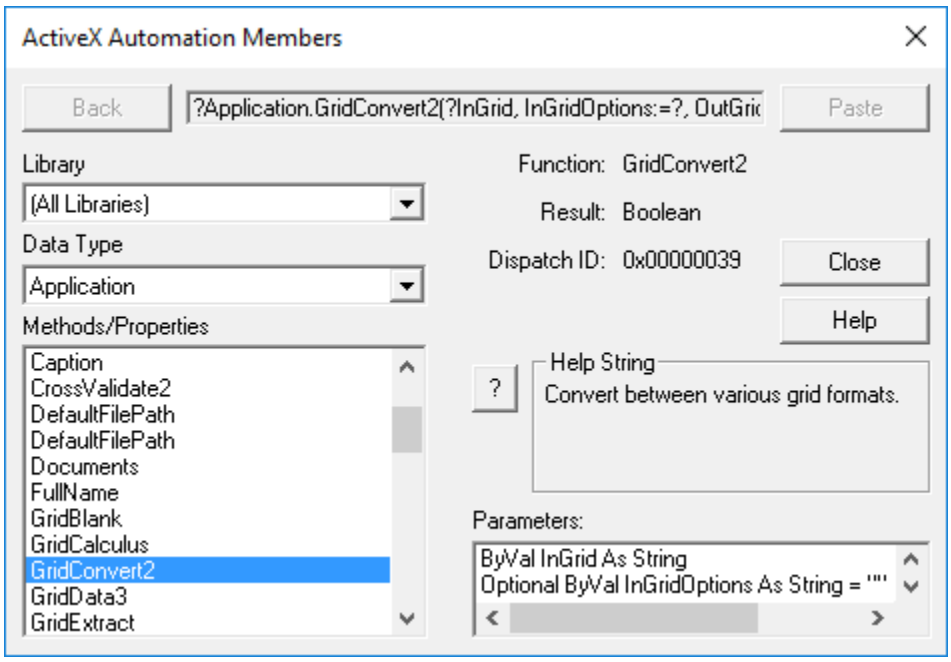
To add a type library reference to the current script module:

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` statment 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) = coordinates(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.

To add a component to a dialog, first select from the palette of components at the left side of the **UserDialog Editor**. After clicking a palette button, drag the mouse pointer diagonally in the dialog design area where you want to place the component. As you design the dialog, you can edit the properties of components you have placed in the dialog. To edit the properties of a component, double-click the item, click the right mouse button on the item, or select the component and click the button. Every dialog must include an OKButton or a CancelButton, or both. See the *Dialog Controls* table below for a list of dialog components and their properties.

When you have finished designing the dialog, click the button. The code for the dialog is inserted into the script. To edit the dialog template after it has been inserted into the script, first move the cursor in the code window to any line between the `BEGIN DIALOG` statement and the `END DIALOG` statement. Next, select the **Edit | UserDialog Editor** command. The previously saved state of the dialog is shown in the dialog editor. When you save the dialog again, the previous dialog template is replaced with your changes.

To show your custom dialog in a script, first use the `DIM` statement to declare a variable as the `UserDialog` type, and then call the `DIALOG` function to display the dialog (see the example). The `DIALOG` function takes a user dialog variable as its argument and returns a number indicating which button was clicked to end the dialog. The `DIALOG` function returns -1 if the *OK* button was clicked, 0 if the *Cancel* button was clicked, or an integer greater than zero if a push button was clicked (1 for

the first push button listed in the dialog template, 2 for the second push button in the dialog template, and so forth).

If the return value is not needed, the DIALOG instruction may be called as a subroutine rather than as a function. In this case, do not enclose the dialog variable in parentheses. If the DIALOG instruction is called as a subroutine, however, the script will end with a run-time error if a cancel button is clicked.

To define more than one custom dialog in a script, you must place each dialog template in its own subroutine or function. If you try to define more than one custom dialog in the same procedure, **Scripter** will show an error indicating that the UserDialog type has already been defined.

The values contained by dialog controls are accessed the same the way the fields of user-defined variable types are accessed. Type the dialog variable name, followed by a period, followed by the field name of the dialog component. Option button values cannot be accessed directly, but are accessed via the field name of their associated option group. The value of an OptionGroup is the number of the selected option button (the first option button in the group is 0, the second option button is 1, and so forth). You can initialize the values contained by dialog controls prior to showing the dialog, and retrieve the values entered in the dialog after it has been invoked.

Dialog Controls

Button	Item	Description	Properties
	GroupBox	A rectangle used to group related controls.	Field, Caption
	Text	A text label, requires no response from the user	Field, Caption
	TextBox	An edit box used to enter and edit text	Field, Type (single line, multiple line, password)
	CheckBox	A box which is checked and unchecked as the user clicks on it, a three-state check box has a disabled (grayed) state	Field, Caption, Type (2 state, 3 state, 3 state auto check)
	OptionButton	A round button for choosing from a set of options, only one of a group of option buttons may be checked	Field, Caption, Option Group
	ListBox	A window that contains a list of items that can be selected by the user	Field, Array of Items, Type (list, sorted list)
	DropListBox	A list that is visible when opened by the user, the text may be editable or not and the list may be sorted or not	Field, Array of Items, Type (list, text, sorted list, sorted text)
	ComboBox	A text box with an attached list box, the list may be sorted or not	Field, Array of Items, Type (text, sorted)
	Picture	Displays a bitmap (image) in the dialog	Field, Caption, Type (from file, from clipboard)

	PushButton	A push button	Field, Caption
	OKButton	Push button with the *OK* caption	Field
	CancelButton	Push button with *Cancel* caption	Field
n/a	Dialog	Definition of dialog	Dialog Function, Caption, Centered

Properties:

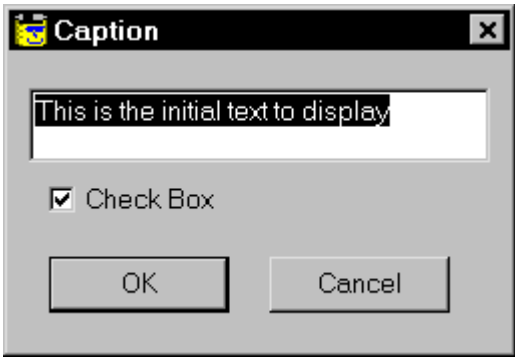
- **Dialog Function:** The dialog function is the name of a special function that is called when various events happen in a dialog. Define a dialog function to control the behavior of a dialog and to retrieve its input.
- **Field Name:** The field name is the name used to refer to a component.
- **Caption:** 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 the **Help | BASIC Language Help** for 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 VB `CreateObject` function. The `CreateObject` function finds the name `Surfer.Application` in the system registry, automatically activates **Surfer**, and returns a reference to the **Surfer** *Application* object. For an introduction to the **Scripter** programming language, see Scripter BASIC Language.

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 Writing Scripts 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 `CreateObject` 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, GridBlank, 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 the Documents property 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).

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.Type` property to determine if the returned object is a *PlotWindow*, *WksWindow*, or *GridWindow* object.

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 Application 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 Application 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 Application 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**.

Also 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 **Surfer 15**, be sure to dispatch the current interface.

The current interfaces include all of the properties and methods available in **Surfer 15**. The properties and methods specific to the former interfaces can be found in the **Surfer 14** and older application help files.

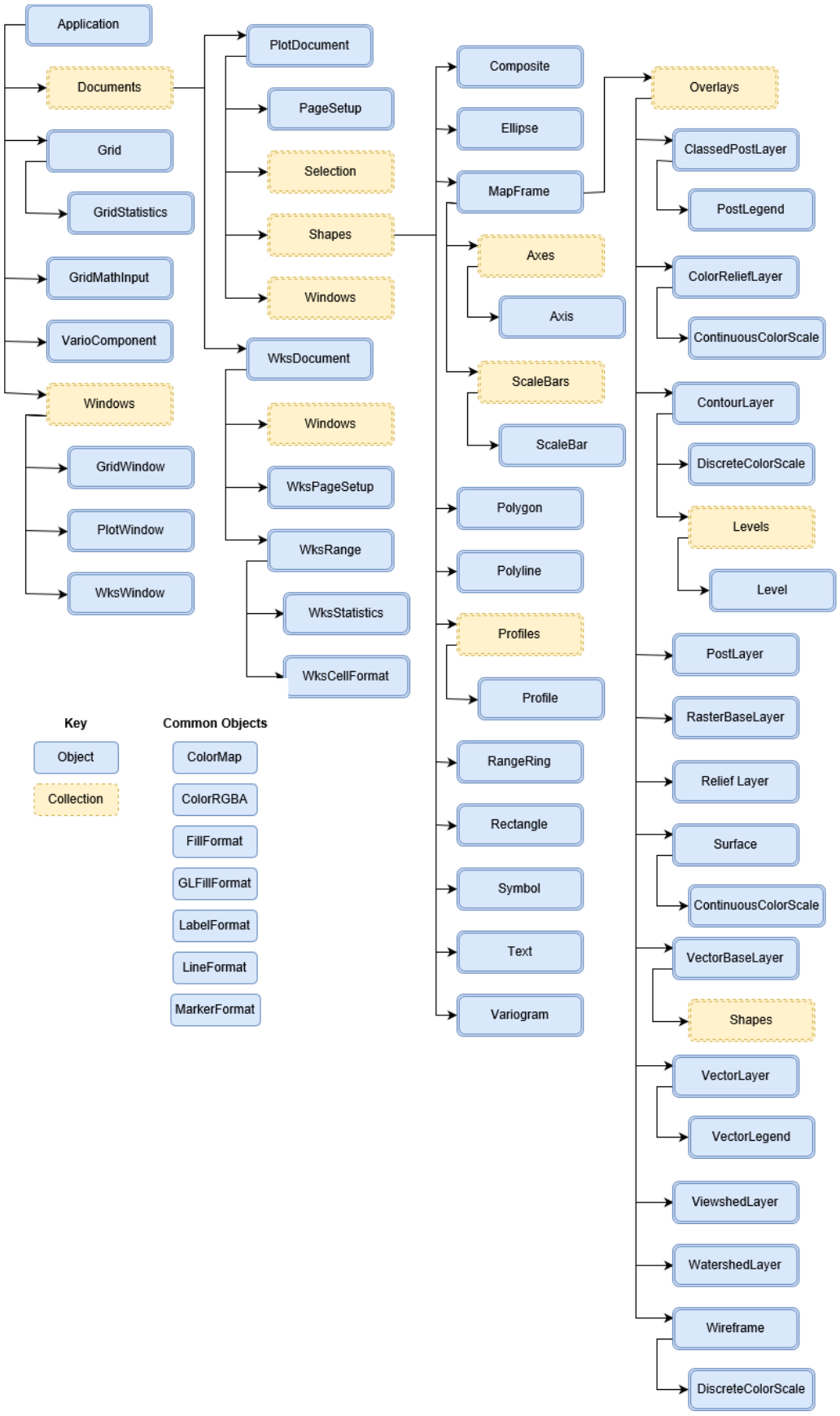
Object	Interface (current)	Interface (former)
Application	IApplication2	IApplication
Axes	IAxes	
Axis	IAxis2	IAxis
ClassedPostLayer	IClassedPostlayer	IClassedPostMap
ColorMap	IColorMap2	IColorMap
ColorNode	IColorNode	
ColorNodes	IColorNodes	
ColorReliefLayer	IColorReliefLayer	IImageLayer, IImageMap
ColorRGBA	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	
Grid	IGrid2	IGrid
GridMathInput	IGridMathInput	
GridStatistics	IGridStatistics	
GridWindow	IGridWindow	
LabelFormat	ILabelFormat2	ILabelFormat

Layer	ILayer	
Legend	ILegend	
Level	ILevel	
Levels	ILevels	
LineFormat	ILineFormat	
MapFrame	IMapFrame2	IMapFrame
MarkerFormat	IMarkerFormat	
Overlays	IOverlays	
PageSetup	IPageSetup	
PlotDocument	IPlotDocument2	IPlotDocument
PlotWindow	IPlotWindow	
Polygon	IPolygon	
Polyline	IPolyline	
PostLayer	IPostLayer	IPostMap
PostLegend	IPostLegend	
Profile	IProfile	
Profiles	IProfiles	
RangeRing	IRangeRing	
RasterBaseLayer	IRasterBaseLayer	
Rectangle	IRectangle	
ReliefLayer	IReliefLayer	IReliefMap
Ruler	IRuler	
ScaleBar	IScaleBar2	IScaleBar
ScaleBars	IScaleBars	
Selection	ISelection2	ISelection
Shape	IShape	
Shapes	IShapes4	IShapes3, IShapes2, IShapes
Surface	ISurface	
Symbol	ISymbol	
Text	IText	
VarioComponent	IVarioComponent	
Variogram	IVariogram	
VectorBaseLayer	IVectorBaseLayer	
VectorLayer	IVectorLayer	IVectorMap
VectorLegend	IVectorLegend	

ViewshedLayer	IViewshedLayer
WatershedLayer	IWatershedLayer
Window	IWindow
Windows	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**.


Application
Documents
Grid
GridStatistics
GridMathInput
VarioComponent
Windows
GridWindow
PlotWindow
WksWindow
PlotDocument
PageSetup
Selection
Shapes
Windows
WksDocument
Windows
WksPageSetup
WksRange
WksStatistics
WksCellFormat
Composite
Ellipse
MapFrame
Axes
Axis
ScaleBars
ScaleBar
Polygon
Polyline
Profiles
Profile
RangeRing
Rectangle
Symbol
Text
Variogram
Overlays
ClassedPostLayer
PostLegend
ColorReliefLayer
ContinuousColorScale
ContourLayer
DiscreteColorScale
Levels
Level
PostLayer
RasterBaseLayer
Relief Layer
Surface
ContinuousColorScale
VectorBaseLayer
Shapes
VectorLayer
VectorLegend
ViewshedLayer
WatershedLayer
Wireframe
DiscreteColorScale
Key
Object
Collection
Common Objects
ColorMap
ColorRGBA
FillFormat
GLFillFormat
LabelFormat
LineFormat
MarkerFormat

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 15\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.
- The online Support Forum is a location to search previous script discussions for examples. You may also post specific script questions on the forum.

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, ContourLayer As Object
    Dim InFile, GridFile, BaseName As String

    '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 BaseName = Left(InFile,ExtStart-1)

    'Create a grid from the specified data file using the Kriging algorithm
    GridFile = BaseName + ". grd"
    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 = Plot.Shapes.AddContourMap(GridFile)

    'Assign the contour map properties to the variable named "ContourLayer"
    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, datafile As String, grid As String)
    ' Create a variogram.
    Set v = plot.Shapes.AddVariogram ( datafile, ShowReport:= False)
```

```
    ' Create a grid using the computed variogram
    plot.Application.GridData DataFile:=datafile, Algorithm := SurferAppKriging,
_
    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(SurferAppVarNugget,1.0,2.0,3.0)
    Set components(2) =
SurferApp.NewVarioComponent(SurferAppVarLinear,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 intended for backward compatibility with **Surfer 8**. In **Surfer 9** use add map layers.

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
  mapframe.Application.ScreenUpdating = False

 ' Adjust the left and bottom axes
 For Each axis In mapframe.Axes
  If axis.AxisType = SurferAppATLeft Or axis.AxisType = SurferAppATBottom Then
   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 = old_update_setting
End Sub
```

Appendix A - 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 π/2
asin(x)	Arcsine in the range -π/2 to π/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 -π/2 to π/2
atan2(y,x)	Arctangent of y/x in the range -π 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.

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).

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:

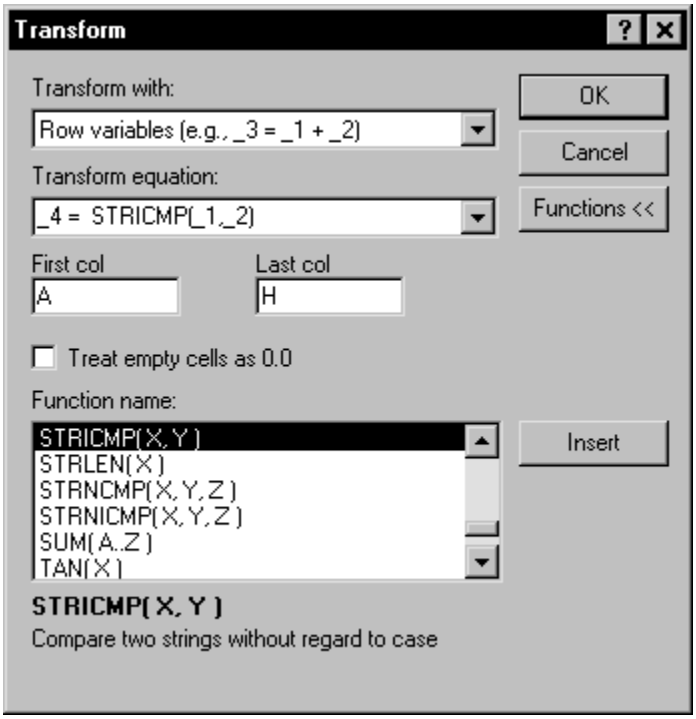
1. numeric values (disregarded in string comparisons as mentioned above)
2. cells starting with a space character
3. common punctuation
4. numeric text (numbers entered as text)
5. less common punctuation
6. uppercase letters
7. even less common punctuation
8. lower case letters
9. uncommon punctuation
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 -** IF(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.
NOT	Logical NOT	Inverts the boolean value. True becomes false, false becomes true.
&	AND	The result is true if both operands are true.
|	OR	The result is true if either of the two operands is true
XOR	Exclusive-OR (XOR)	The result is true if both operands differ.
||	OR	The result is true if either of the two operands is true.
OR	OR	The result is true if either of the two operands is true.

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, true_value, false_value)

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

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
```

$\delta^{234}U$

Appendix C - File Formats

File Format Chart

There are a number of important distinctions that can be made for different file types because each file format uses a unique set of characteristics. The important file format characteristics are outlined in the following table:

File Format	File Name	Import	Export	Coordinate Type[1]	Drawing Properties[2]	Text Blocks[3]	Content Type[4]
?10G	Globe DEM	X [13]					G
000	S-57 Electronic Navigation Chart	X [12]		R			V
ACCDB	Microsoft Access	X [14]					D
ADF	Esri ArcInfo Binary Grid	X [13]	X [10]				G
AGR	ArcInfo ASCII Grid	X [13]	X [10]				G
AIG	ArcInfo ASCII Grid	X [13]	X [10]				G
AM	Amira Mesh	X [13]	X [10]				G
AN?	ACR-NEMA Medical Image	X [12, 13]		P			R
ASC	ArcInfo ASCII Grid	X [13]	X [10]				G
ASC	CPS-3 Grid Format	X [13]	X [10]				G
ASC	Z-Map Plus Grid Format	X [13]					G
ASCII Data	ASCII Data Files	X [14]	X [9]	R			D
ASI	Amira Stacked Images	X [13]					R and V
BIL	Banded Interleave Grid By Line	X [13]	X [10]				G
BIN	Raw Binary Grid	X [13]	X [10]				G
BIP	Banded Interleave Grid By Pixel	X [13]	X [10]				G
BLN	Golden Software Blanking	X [12, 14]	X [9, 11]	R			V
BMP	Windows Bitmap	X [12, 13]	X [11]	P			

File Format	File Name	Import	Export	Coordinate Type[1]	Drawing Properties[2]	Text Blocks[3]	Content Type[4]
BNA	Atlas Boundary	X [12, 14]	X [9, 11]	R			V, D
BSQ	Banded Sequential Grid	X [13]	X [10]				G
BW	SGI-RGB Image	X [12, 13]		P			G
CLP	Windows Clipboard	Paste	Copy	O	X	X	R and V
COL	Amira Mesh	X [13]	X [10]				G
CPS	CPS-3 Grid	X [13]	X [10]				G
CPS3	CPS-3 Grid	X [13]	X [10]				G
CSV	Comma Separated Variable	X [14]	X [9.11]				D
CSV	Formatted Text Grid	X [13]					G
DAT	ETOPO-5	X [13]					G
DAT	Z-Map Plus Grid	X [13]					G
DAT	Data	X [14]	X [9, 10, 11]				D
DAT	Formatted Text Grid	X [13]					G
DBF	Database	X [14]					D
DCM	DICOM 3 Medical Image	X [12, 13]		P			
DDF	SDTS Topological Vector Profile	X [12]		R			V
DDF	SDTS Digital Elevation Model	X [13]					G
DEM	USGS Digital Elevation Model	X [13]	X [10]				G
DGN	Microstation Design v7	X [12]		O	some		V
DIC	DICOM3 Medical Image	X [12, 13]		P			R and V
DLG	USGS Digital Line Graph	X [12]		R			V
DOS	ETOPO-5	X [13]					G
DTED	Digital Terrain Elevation Data	X [13]					G
DXF	AutoCAD DXF	X [12,14]	X [11]	R O	some	some	V

File Format	File Name	Import	Export	Coordinate Type[1]	Drawing Properties[2]	Text Blocks[3]	Content Type[4]
E00[8]	Esri ArcInfo Export Format	X [12]		R		X	V
E00[8]	Esri Grid Format	X [13]		R			G
ECW	ER Mapper	X [12, 13]		R			R
EMF	Windows Enhanced Metafile	X [12]	X [11]	O	X	X	R and V
EPS	Encapsulated Postscript		X [11]	P	X		R
ERS	ER Mapper Grid	X [13]	X [10]				G
FLD	AVS Field	X [13]	X [10]				G
FLT	Esri Float Grid Format	X [13]	X [10]				G
FTG	Formatted Text Grid	X [13]					G
GGF	Geosoft Binary Grid		X [10]				G
GIF	Image	X [12, 13]	X [11]	P			R
GEOJSON	GeoJSON Data Interchange	X [12]		R			V
GML	Geography Markup Language	X [12]		R			V
GPX	GPS Exchange File	X [12]		R			V
GRB	World Meteorological Organization Grid	X [13]					G
GRD	ArcInfo ASCII Grid	X [13]	X [10]				G
GRD	CPS-3 Grid Format	X [13]	X [10]				G
GRD	NetCDF Grid File	X [13]					G
GRD	Surfer Grid	X [13]	X [10]				G
GRD	Geosoft Binary Grid	X [13]	X [10]				G
GRD	Z-Map Plus Grid Format	X [13]					G
GSB	Golden Software Boundary	X [12]	X [11]	R			V
GSI	Golden Software Interchange	X [12]	X [11]	R	X	X	R and V

File Format	File Name	Import	Export	Coordinate Type[1]	Drawing Properties[2]	Text Blocks[3]	Content Type[4]
GXF	Grid Exchange	X [13]	X [10]				G
HDF	Hierarchical Data	X [13]	X [10]				R
HDR	GTopo-30	X [13]		R			G
HGT	SRTM Elevation Data	X [13]		R			G
HTM	HTML Image Map		X [8]	P			V
IMG	Analyze 7.5 Medical Image	X [13]	X[10]				R
IMG	ERDAS Imagine Image	X [13]		P, R			R and/or G
IMG	Idrisi Raster Image	X [13]					R
INFO	Leica Confocal Raw Slices	X [13]					R
JPG	JPG Compressed Bitmap	X [12, 13]	X [11]	P, R			R
KML	Google Earth KML	X [12]	X [11]	P, R	X		V
KMZ	Google Earth KMZ	X [12]	X [11]	P, R	X		V
LAS	LiDAR LAS Binary Data	X [14]		R			D
LAT	Iris Explorer	X [13]	X [10]				G
LAZ	LiDAR LAS Binary Data	X [14]		R			D
LGO	USGS Digital Line Graph	X [12]		R			V
LGS	USGS Digital Line Graph	X [12]		R			V
MDB	Microsoft Access						D
MID	MapInfo Data	X [14]	X [9]				D
MIF	MapInfo Interchange	X [12]	X [11]	R	some		V
NC	NetCDF Grid File	X [13]	X [10]				G
PBM	Image	X [12, 13]		P			R
PDF	Portable Document File (Raster)	X [12, 13]	X [11]	P			R
PDF	Portable Document File (Vector)		X [11]	O, R			V

File Format	File Name	Import	Export	Coordinate Type[1]	Drawing Properties[2]	Text Blocks[3]	Content Type[4]
PGM	Image	X [12, 13]		P			
PLT	Golden Software PlotCall	X [12]		O		X	V
PLY	Stanford Polygon	X [12]		R			V
PNG	Portable Network Graphics	X [12, 13]	X [11]	P			
PNM	Image	X [12, 13]	X [11]	P			
PPM	Image	X [12, 13]		P			
RAS	Sun Raster Image	X [12, 13]	X [11]	P			R and V
RAW	Binary Grid	X [13]	X [10]				
RGB	SGI-RGB Image	X [12, 13]	X [11]	P			
RGBA	SGI-RGB Image	X [12, 13]	X [11]	P			
RST	Idrisi Raster Image	X [13]		P			R
RT*	TIGER/Line File	X [12]		R			V
SEG	SEG-P1 Exchange Format	X [12, 14]	X [11]	R			D V
SHP	Esri Shapefile	X [12]	X [11]	R			V
SID	LizardTech MrSID Image	X [12, 13]		P, R			
SLK	Sylk Spreadsheet	X [14]	X [9]	R			D
SP1	SEG-P1 Exchange Format	X [12, 14]	X [11]	R			D V
STK	Metamorph	X [13]					R
STL	Stereolithography 3D Mesh		X [10]				
SUN	Sun Raster Image	X [12, 13]	X [11]	P			R
SVG	Scalable Vector Graphics		X [11]	R	X		R and V
TAB	MapInfo Table	X [12]		R	some		V
TAR	SDTS Topological Vector Profile	X [12]		R			V
TAR	SDTS Digital Elevation Model	X [13]					G
TAR.GZ	SDTS Topological Vector Profile	X [12]		R			V

File Format	File Name	Import	Export	Coordinate Type[1]	Drawing Properties[2]	Text Blocks[3]	Content Type[4]
TAR.GZ	SDTS Digital Elevation Model	X [13]					G
TGA	Targa (TrueVision)	X [12, 13]	X [11]	P			
TGZ	SDTS Topological Vector Profile	X [12]		R			V
TGZ	SDTS Digital Elevation Model	X [13]					G
TIF	Tagged Image	X [12, 13]	X [11]	P, R			
TXT	Text Data	X [14]	X [9]				D, G
TXT	Formatted Text Grid	X [13]					G
VCT	IDRISI Binary Vector File	X [12]		R			V
VTK	Visualization Toolkit	X [12, 13]	X [10]	P			V
WMF	Windows Metafile	X [12]	X [11]	O	X	X	R and V
X	AVS X-Image	X [12, 13]	X [11]	P			R
XIMG	AVS X-Image	X [12, 13]	X [11]	P			R
XLS	Excel Spreadsheet	X [14]	X [9]				D
XLSM	Excel Spreadsheet 2007	X [14]					D
XLSX	Excel Spreadsheet 2007	X [14]	X [9]				D
XYZ	Z-Map Plus Grid Format	X [13]					G
ZIP	SDTS Topological Vector Profile	X [12]		R			V
ZIP	SDTS Digital Elevation Model	X [13]					G
ZMAP	Z-Map Plus Grid Format	X [13]	X [10]				G
ZYC	Z-Map Plus Grid Format	X [13]	X [10]				G
ZYCOR	Z-Map Plus Grid Format	X [13]	X [10]				G

1 When imported with Home | New Map | Base (Import column above must contain 12) or when exported (Export column above must contain 11):

R = real world coordinates such as latitude/longitude, UTM, etc.

P = pixel coordinates

O = other (i.e. page coordinates, metafile coordinates, etc.)

2 Contains properties such as line color, fill, etc.

3 Text blocks are editable. (When the *All text as lines* option is chosen in the export options, the text is not editable.)

4 Files can contain raster (image) information, vector (lines) information, or both. There is no raster to vector conversion in **Surfer**. If you have an image on screen and try to export it as a vector format, such as a DXF, the resulting file will be blank. If you have an image on the screen and have drawn polylines or polygons and export as a vector format, only the drawing objects are included in the file.

V = contains vector information only

R = contains raster information only

R and V = contains raster and vector information

D = contains data information

G = contains grid information

5 Not Used

6 Not Used

7 Not Used

8 The .E00 file format is not publicly documented by Esri. Although Golden Software has tested this .E00 import filter software with a number of publicly available .E00 files, it may not be compatible with all .E00 files created by all versions of Esri application programs. Golden Software is not affiliated with Esri, and this import filter software is not a product of, nor endorsed by, Esri.

9 The File | Save As command in the worksheet can be used to save this format.

10 The File | Save As command in the grid editor or one of the grid menu commands in the plot window can be used to save this format with the **Save Grid As** dialog.

11 The File | Export command in the plot window can be used to save this format.

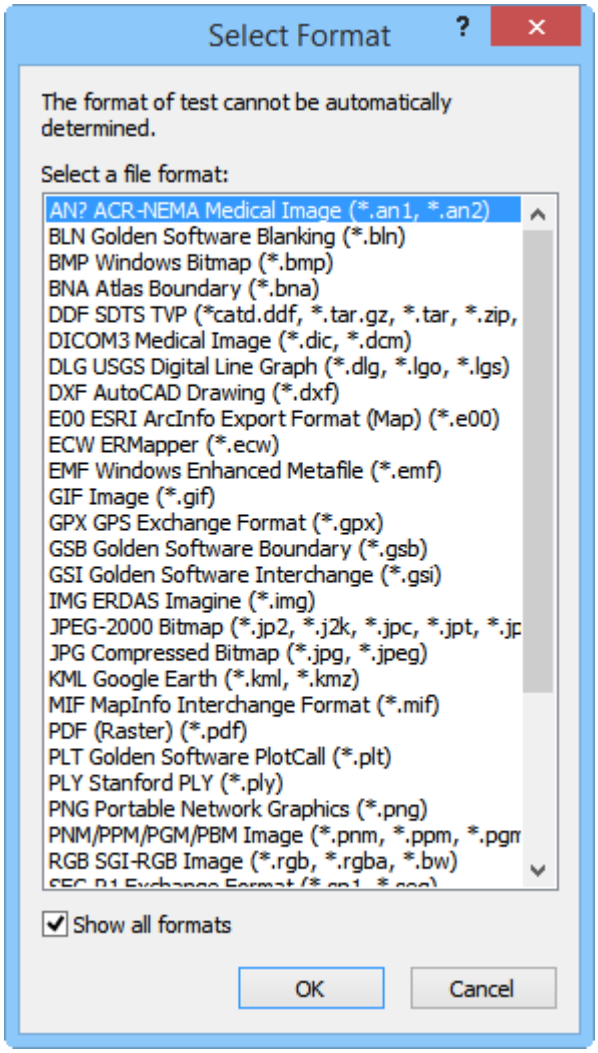
12 The File | Import command in the plot window or Import dialog (i.e. **Home | New Map | Base**) can be used to open this format.

13 The Open Grid dialog (i.e. **Home | New Map | Contour**) can be used to open this format.

14 The File | Open or Data | Merge command in the worksheet can be used to open this format.

File Type Not Recognized

The **Select Format** dialog appears if a file type that is not recognized is imported into **Surfer**. This error occurs when the file data does not match the format specified by the file extension.



*The **Select Format** dialog appears if you import a file type that is not recognized.*

Select a File Type

Select a recognized file type from the *Select a file type* list.

Show All Formats

When the **Select Format** dialog first opens, **Surfer** appends the list of supported file types to the most likely file type for the selection. If the desired file type is not in the short list, check the *Show all formats* box and the list will include all supported file types. If the file type is not in the list, you will not be able to use the file directly. Save the file in another program as a different format.

ASCII .DAT, .TXT, .CSV Data Files

ASCII files are generic format files that can be read or produced by most applications. There are three common ASCII data formats: .DAT, .CSV, and .TXT. ASCII files are generic format files read or produced by most applications. These files can also be imported into most applications, including word processors, spreadsheets, and ASCII editors.

Worksheet Formatting

ASCII files do not contain any worksheet formatting information such as row height, column width, or cell formatting. When ASCII files are loaded into the worksheet, the default column formatting parameters are applied to the data. This does not result in any change to data, but might result in rounding of values in the data display. There is no limitation on the number of rows or columns in an ASCII format. ASCII formats save and load slowly because there is a conversion from binary numbers to character representation.

Format

There are some distinctions in formatting of ASCII files. Here are some brief notes that outline the usefulness of the ASCII file features.

- *Delimiters* control the separation between cell entries in a file. Spaces, tabs, semi-colons, or commas can be used to separate cells. If cell entries contain spaces in text, the comma or semi-colon delimiters are necessary if quotes are not used to qualify the text. Otherwise, the text string would be interpreted as two cell entries rather than a single entry.
- *Placing Quotes Around Text* - There are two types of entries in an ASCII file, values and text. Values are actual numbers, while text can be any type of character, including numbers and text characters. Single or double quotes can be placed around text strings. If a number should be interpreted as text, surround it with double quotes. When text strings contain spaces, it is recommended to use single or double quotes around text cell entries.
- *Using Commas or Semicolons in Addition to Quotes* - Although double quotes are not required around text strings, they are useful when creating a space-delimited file that contains text. Often there are text strings that contain spaces, as in a date containing month name, day and year. With space delimited files this single entry is interpreted as more than one cell when loading this file into the worksheet. The safest way to eliminate this problem is to place double quotes around all text strings and use comma delimiting between variables.

Comma Separated Variables

Comma separated variable .CSV files are comma delimited with double-quotes around text strings (non-numeric or mixed alpha numeric).

When the computer's locale setting has the *Decimal separator* as *comma*, .CSV files are imported with commas as the decimal separators and semi-colons as column separators. When the File | Options | User Interface *Decimal separator* is manually set to comma, .CSV files also import with commas as decimal separators and semi-colons as column separators.

ASCII Text

ASCII text files .TXT are normally tab delimited ASCII text files with no quotes around the text strings. After selecting .TXT as the format, the **Data Export Options** dialog is displayed.

Golden Software DAT Files

ASCII .DAT files are ASCII files with no set format. Any delimiter or text qualifier can be set. When a file is saved in the .DAT format, the **Data Export Options** dialog is displayed.

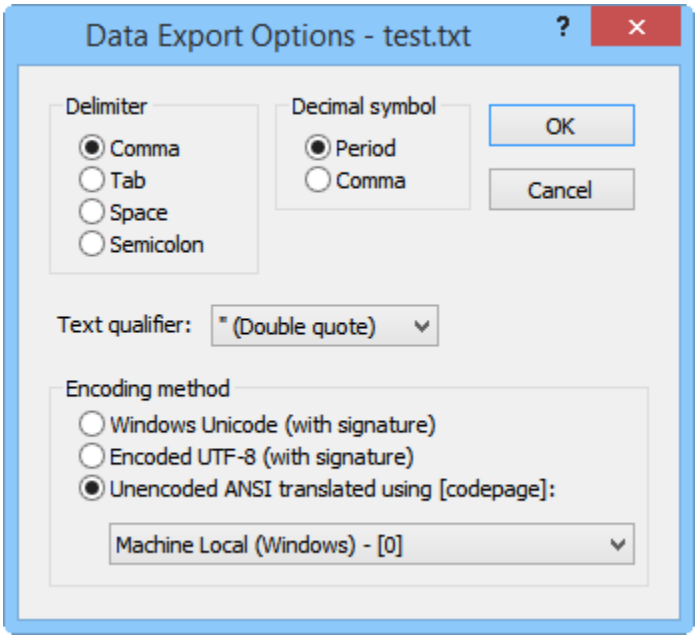
XYZ Points Files (.DAT, .CSV)

See XYZ Points [.CSV, .DAT] File Description

Data Export Options Dialog

The **Data Export Options** dialog is displayed when saving a .TXT file or .DAT file.

- *Delimiters* are the characters used between cells in a single row (fields in a record), and can be commas, spaces, semicolons, or tabs.
- *Text Qualifiers* are double-quotes, single-quotes, or none. For example, if double-quotes are chosen all non-numeric or mixed alpha numeric cell entries are surrounded by double-quotes in the file.
- Decimal symbol is the symbol used as the decimal point. This can be a comma or period. This option is only available with .TXT files.
- Encoding method determines the format of the data. Windows Unicode and Encoded UTF-8 data are often referred to as international data. It would include character sets from Russia, Israel, China, Greece, Hungary, among others. If the data does not appear correctly in the exported file, the Encoding method may be specified incorrectly. ANSI encoding contains characters within the first 256 characters of a font. These are normally in English. After selecting Unencoded ANSI translated using [codepage], select the codepage from the list that will read the data correctly.



*Choose one Delimiter and a Text Qualifier for .TXT files in the **Data Export Options** dialog.*

Import Options Dialog

Data Import Options Dialog

Import Automation Options

Data Import Automation Options

Export Options Dialog

Data Export Options Dialog

Export Automation Options

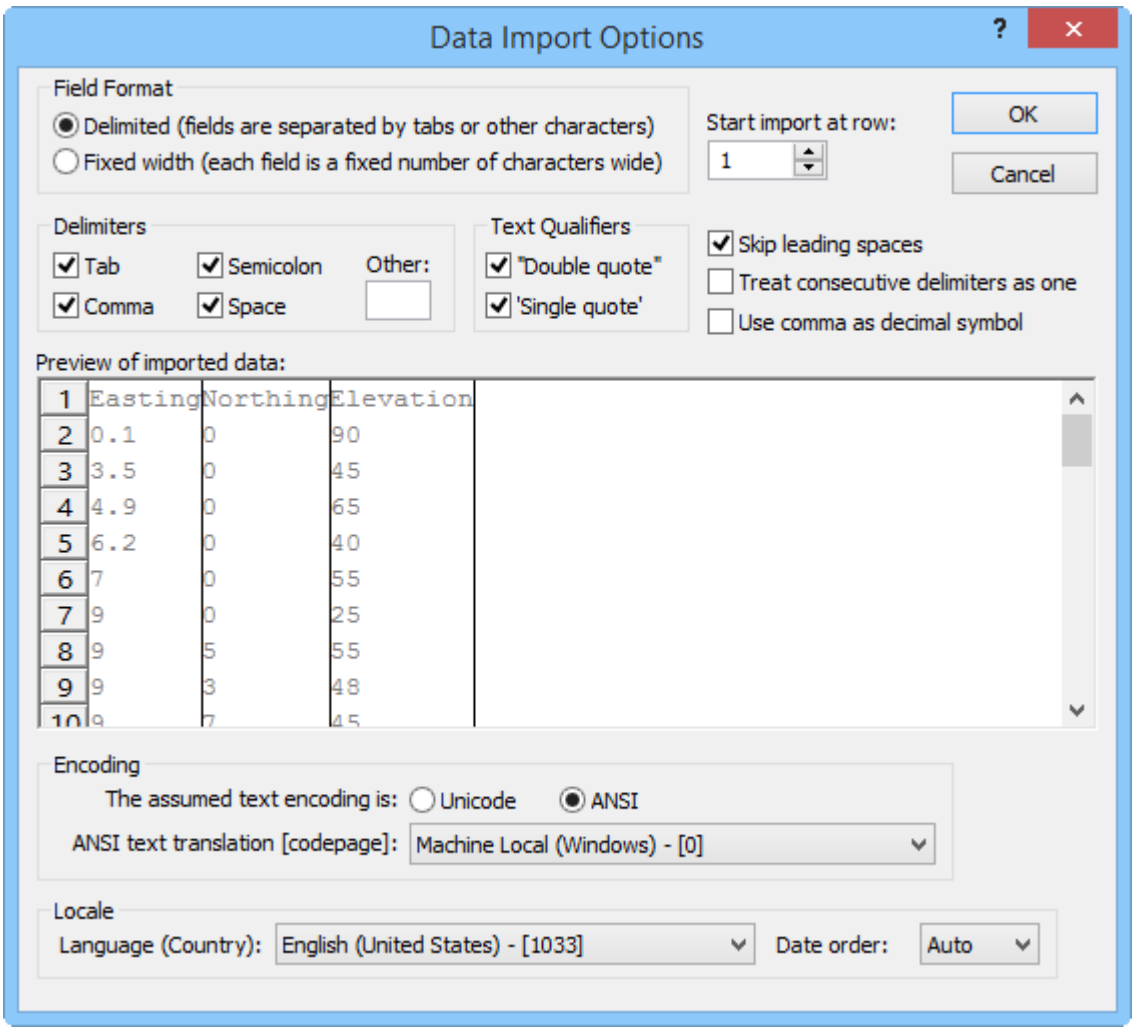
Data Export Automation Options

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.

Import Options String

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of a series of semicolon-separated keywords that specify the behavior of the import. See the *Worksheet FilterID* topic for links to file-type specific import options.

Examples:

Text file: "Delimiter=comma; TextQualifier=singlequote;SkipExtraDelimiters=0"

This would set the delimiter character to a comma and the text qualifier character to the single-quote mark (‘).

Database: “SQLSTR=, Table=table1”

This uses *Table1* from the database.

Excel File: "Sheet=Sheet1"

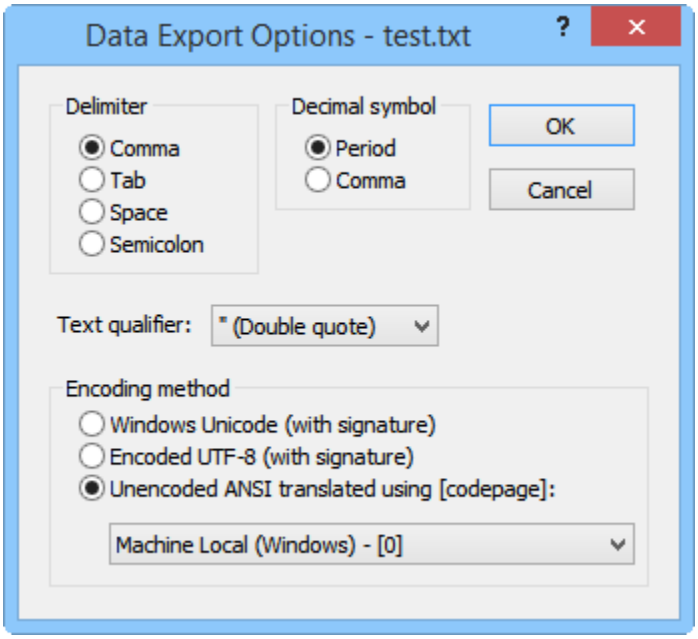
This uses *Sheet1* from the XLS, XLSX, or XLSM file.

Option	Action	Default	Description
ColumnBreaks	N,N,N...	no option string	*ColumnBreaks* is used when *UsedFixedWidth* is set to 1. The field separators are a series separated by commas or spaces, for example: 6 12 20 25.
DecimalSymbol	period = period comma = comma	period	*DecimalSymbol* specifies the character that separates the integer and fractional components of the data values.
Delimiter	tab = tab comma = comma semicolon = semicolon space = space equals = equals other = enter the delimiter character		The *Delimeter* is the character that separates data cell values. You can use more than one delimiter in the string. For example, " Delimiter=comma,%,tab"
EatWhiteSpace	0 = Do not skip extra space and tab in the data 1= Skip extra space and tab in the data	0	*EatWhiteSpace* removes extra space and tab characters preceding and following data. A 0 means not to skip the extra space. For example, " 1 2" would leave the column A blank and import data into the columns B and C. A 1 means to skip the extra space. For example, " 1 2" would import data into the columns A and B.
FilterID	String	""	FilterId specifies the file format extension if the extension entered for File Name is unknown. If the command calling the import options string has a *FilterID* property, you do not need to use this *FilterID*.
ImportCodePage	String	""	The name of the ANSI code page to use when importing Unicode data
Sheet	sheet name		*Sheet* opens the specified sheet in an Excel workbook.
SkipExtraDelimiters	0 = Do not skip extra delimiters in the data	1	*SkipExtraDelimiters* specifies whether multiple delimiters are treated as one delimiter.

	1= Skip extra delimiters in the data		"SkipExtraDelimiters=0" means "don't skip the extra delimiters" For example, 1,,2,,,3 would import into columns A,C, and F. "SkipExtraDelimiters=1" means "do skip extra delimiters". For example, 1,,2,,,3 would import into columns A, B and C.
SQLSTR	SQL string	No	Structured Query Language access string.
StartRow	N	1	*StartRow* is the row number at which to begin importing.
Table	table or query name		*Table* opens the specified table in an Access database.
TextQualifier	doublequote = doublequote(") singlequote = singlequote(') none = none other = enter the text qualifier		The *TextQualifier* specifies the character that surrounds cells containing text. You can use more than one text qualifier in the string. For example, " TextQualifier=doublequote,%"
UseFixedWidth	0 = No 1 = Yes	0	*UseFixedWidth* uses a fixed width for columns during import.
Locale	String	System Locale	*Locale* is the locale ID in decimal values. The default locale is determined by the locale setting in the Windows Control Panel.
DateOrder	0 = Auto 1 = MDY (Month, Day, Year) 2 = DMY 3 = YMD 4 = MYD 5 = DYM 6 = YDM	0	The *DateOrder* specifies the order in which dates are written in the data file. When *DateOrder* is set to 0, the standard date order from the *Locale* is used.

Data Export Options Dialog

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



*Specify the delimiter and text qualifier in the **Data Export Options** dialog.*

Delimiter

Choose *Comma*, *Tab*, *Space*, or *Semicolon* as the character to use to delimit fields in the saved .DAT, or .TXT file.

Decimal Symbol

Select the option in the *Decimal symbol* group that should be used to as the decimal symbol. When comma is selected, the number 123 and 45 thousands would appear in the file as 123,45. When *Period* is selected, the number would appear in the file as 123.45. This option is only available with .TXT files.

Text Qualifier

Select *(none)* for no qualifiers in the export file. Select *Double quotes* to place double quote characters "" around fields in the export file. Select *Single quotes* to place single quote characters '' around fields in the export file.

Encoding Method

The *Encoding method* section allows the choice of *Windows Unicode* data, *Encoded UTF-8* data, or *Unencoded ANSI translated using* data when exporting or saving an ASCII data file. *Windows Unicode* and *Encoded UTF-8 data* are often referred to as international data. It would include character sets from Russia, Israel, China, Greece, Hungary, among others. If the data does not appear correctly in the exported file, the *Encoding method* may be specified incorrectly.

ANSI encoding contains characters within the first 256 characters of a font. These are normally in English. After selecting *Unencoded ANSI translated using [codepage]*, select the *codepage* from the list that will read the data correctly.

OK and Cancel Buttons

Click the *OK* button to proceed with the export process, or click the *Cancel* button to close the dialog without exporting the data set.

Export Options String

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters which specify the behavior of the various export options. A typical example would be:

" Delimiter=comma; TextQualifier=singlequote "

This would set the delimiter character to a comma and the text qualifier character to the single-quote mark (‘).

Option	Action	Default	Description
DecimalSymbol	period = period comma = comma	period	The character to use as the decimal symbol. This option is only available with .TXT files.
Delimiter	tab = tab comma = comma semicolon = semicolon space = space other = enter the delimiter character	comma	Character that separates data cell values. Only one delimiter character should be specified on export. For example, if you want to export a data file using a percent signs to delimit data, add a "Delimiter=%" to the export options string.
ExportCodePage	String	""	The name of the ANSI code page to use when exporting Unicode data.
ExportEncodingMethod	0 = unencoded ANSI translated using codepage 1 = Windows Unicode (with signature) 2= encoded UTF-8 (with signature)	0	The text encoding method. 0 is normal text. 1 and 2 are Unicode text.
FileFormat	BIFF8 = Excel-97	BIFF8	Determines the file format when saving an XLS file.

	BIFF5 = Excel-95		
TextQualifier	doublequote = doublequote (") singlequote = singlequote (') none = none other = enter the text qualifier	doublequote	Specifies the character that surrounds cells containing text. Only one text qualifier character should be specified on export. For example, if you want to export a data file using a single quote mark as the text qualifier, add "TextQualifier=singlequote" to the export options string.

Image (Bitmap) File Descriptions

A bitmap is an image displayed as an array of dots or "bits." In **Surfer**, images are typically imported as a base map.

Device-independent bitmaps and BMP file format

A typical BMP file usually contains the following blocks of data:

BMP File Header	Stores general information about the BMP file
Bitmap Information (DIB header)	Stores detailed information about the bitmap image.
Color Palette	Stores the definition of the colros being used for indexed color bitmaps.
Bitmap Data	Stores the actual image, pixel by pixel

The following sections discuss the data stored in the BMP file or DIB in details. This is the standard BMP file format.[2] Some bitmap images may be stored using a slightly different format, depending on the application that creates it. Also, not all fields are used; a value of 0 will be found in these unused fields.

DIBs in memory

A BMP file is loaded into memory as a DIB data structure, an important component of the Windows GDI API. The DIB data structure is the same as the BMP file format, but without the 14-byte BMP header.

BMP file header

This block of bytes is at the start of the file and is used to identify the file. A typical application reads this block first to ensure that the file is actually a BMP file and that it is not damaged. Note that the first two bytes of the BMP file format (thus the BMP header) are stored in big-endian order. This is the magic number 'BM'. All of the other integer values are stored in little-endian format (i.e. least-significant byte first).

Offset#	Size	Purpose
0	2	the magic number used to identify the BMP file: 0x42 0x4D (Hex code points for **B** and **M**)
2	4	the size of the BMP file in bytes
6	2	reserved; actual value depends on the application that creates the image
8	2	reserved; actual value depends on the application that creates the image
10	4	the offset, i.e. starting address, of the byte where the bitmap data can be found.

Bitmap information (DIB header)

This block of bytes tells the application detailed information about the image, which will be used to display the image on the screen. The block also matches the header used internally by Windows and OS/2 and has several different variants. All of them contain a dword field, specifying their size, so that an application can easily determine which header is used in the image. The reason that there are different headers is that Microsoft extended the DIB format several times. The new extended headers can be used with some GDI functions instead of the older ones, providing more functionality. Since the GDI supports a function for loading bitmap files, typical Windows applications use that functionality. One consequence of this is that for such applications, the BMP formats that they support match the formats supported by the Windows version being run. See the table below for more information.

Size	Header	Identified by	Supported by the GDI of
40	Windows V3	BITMAPINFOHEADER	all Windows versions since Windows 3.0
12	OS/2 V1	BITMAPCOREHEADER	OS/2 and also all Windows versions since Windows 3.0
64	OS/2 V2		
108	Windows V4	BITMAPV4HEADER	all Windows versions since WIndows 95/NT4
124	Windows V5	BITMAPV5HEADER	WIndows 98/2000 and newer

For compatibility reasons, most applications use the older DIB headers for saving files. With OS/2 being obsolete, for now the only common format is the V3 header.

Color Palette

The palette occurs in the BMP file directly after the BMP header and the DIB header. Therefore, its offset is the size of the BMP header plus the size of the DIB header.

The palette is a block of bytes (a table) listing the colors available for use in a particular indexed-color image. Each pixel in the image is described by a number of bits (1, 4, or 8) which index a single color in this table. The purpose of the color palette in indexed-color bitmaps is to tell the application the actual color that each of these index values corresponds to.

A DIB always uses the RGB color model. In this model, a color is terms of different intensities (from 0 to 255) of the additive primary colors red (R), green (G), and blue (B). A color is thus defined using the 3 values for R, G and B (though stored in backwards order in each palette entry).

The number of entries in the palette is either 2n or a smaller number specified in the header (in the OS/2 V1 format, only the full-size palette is supported).[2][4] Each entry contains four bytes, except in the case of the OS/2 V1 versions, in which case there are only three bytes per entry.[4] The first (and only for OS/2 V1) three bytes store the values for blue, green, and red, respectively,[2] while the last one is unused and is filled with 0 by most applications.

As mentioned above, the color palette is not used when the bitmap is 16-bit or higher; there are no palette bytes in those BMP files.

Bitmap Data

This block of bytes describes the image, pixel by pixel. Pixels are stored "upside-down" with respect to normal image raster scan order, starting in the lower left corner, going from left to right, and then row by row from the bottom to the top of the image.[2] Uncompressed Windows bitmaps can also be stored from the top row to the bottom, if the image height value is negative.

RGB color (24-bit) pixel values are stored with bytes in the same order (blue, green, red) as in the color table.[2]

If the number of bytes matching a row (scanline) in the image is not divisible by 4, the line is padded with one to three additional bytes of unspecified value (not necessarily 0) so that the next row will start on a multiple of 4 byte location in memory or in the file. (the total number of bytes in a row can be calculated as the image size/bitmap height in pixels) Following these rules there are several ways to store the pixel data depending on the color depth and the compression type of the bitmap.

File Formats

The term image (bitmap) includes the following file formats:

- BMP Bitmap .BMP
- ECW ERMapper Images .ECW
- EPS Encapsulated Postcript .EPS
- GIF Graphics Interchange Format .GIF
- JPEG Compressed Bitmap .JPG
- PICT MacIntosh .PCT
- PDF Raster
- PNG Portable Network Graphics .PNG
- PNM Portable Any Map Image .PNM, .PPM, .PGM, .PBM
- RAS Sun Raster Image .RAS, .SUN
- RGB Silicon Graphics RGB Image .RGB, .RGBA, .BW
- SID LizardTech MrSID Image .SID
- TGA TrueVision Targa .TGA
- TIF Tagged Image File Format .TIF, .TIFF

Loading a BMP

Use the **File | Import** or **Home | New Map | Base** to load a .BMP file. Images can be opened in the Grid Editor with **File | Open**.

Import Options Dialog

No import options dialog is displayed for most image file formats.

See MrSID Import Options Dialog.

See .ECW Image Import Options Dialog.

Import Automation Options

No import automation options are available for most image file formats.

See MrSID Import Automation Options.

See ER Mapper .ECW Import Automation Options.

Export Options Dialog

See Image (Bitmap) Export Options Dialog

Export Automation Options

See Image (Bitmap) Export Automation Options

Image (Bitmap) Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,Width=640,Height=480,ColorDepth=4"

This would set all export options to their default values, then set the bitmap width to 640 pixels, the bitmap height to 480 pixels and the color depth to 4 (i.e., 256 colors).

Note: A script writer is allowed to change the size of a bitmap, even if it has associated georeference parameters. New (but possibly not minimal RMS) georeference parameters will be automatically calculated and saved. Avoid using the "Width=", "Height=", "HDPI=" or "VDPI=" automation options if you do not want associated georeference parameters recalculated.

Option	Action	Default	Description
Width	N		Sets the width of the exported image in pixels. Either a Width or an HDPI option may be specified, but not both.
Height	N		Sets the height of the exported image in pixels. Either a Height or a VDPI option may be specified, but not both
HDPI	N	96	Sets the number of horizontal pixels in the exported image that corresponds to a distance one horizontal inch

			on the Surfer plot. Either a Width or an HDPI option may be specified, but not both. The default HDPI is the resolution of the display device driver on your computer (this is 96 dots per inch on many Windows display devices, but number can very from device to device).
VDPI	N	96	Sets the number of vertical pixels in the exported image that corresponds to a distance of one vertical inch on the Surfer plot. Either a Height or a VDPI option may be specified, but not both. The default VDPI is the resolution of the display device driver on your computer (this is 96 dots per inch on many Windows display devices, but number can very from device to device)
KeepAspect	0 = No 1 = Yes	1	If set to 1, KeepAspect adjusts the aspect ratio (the width versus height) of the exported image to match the aspect ratio of the Surfer plot that is being exported. If an option is given to set the Width or HDPI of an image, KeepAspect automatically assigns the appropriate Height or VDPI to maintain the same aspect ratio as the Surfer plot. Likewise, if an option is given to set the Height or VDPI of an image, KeepAspect automatically assigns the appropriate Width or HDPI to maintain the same aspect ratio of the Surfer plot. Note that the KeepAspect option is ignored if both the Width and Height options or the HDPI and VDPI options are specified. If set to 0, the aspect ratio of the exported image is determined solely by the Width and Height or HDPI and VDPI options. The size is not adjusted to match the aspect ratio of the Surfer plot.

KeepPixelSize	0 = No 1 = Yes		Locks the pixel dimension width and height. Changes to the VDPI and HDPI will affect the document size, but not the pixel size, when this is set to 1. Changes to the VDPI and HDPI will affect the pixel dimensions, but not the document size, when this is set to 0. Note that the KeepPixelSize option is ignored if the either the Width or Height option is specified.
ColorDepth	-32 = 32-bit grayscale -16 = 16-bit grayscale -8 = 8-bit grayscale -4 = 4-bit grayscale -1 = Monochrome 1 = 1-bit color indexed 4 = 4-bit color indexed 8 = 8-bit color indexed 24 = 24-bit true color 32 = 32-bit true color with alpha	24 for most formats 8 if the format doesn't support 24-bit true color	Specifies the color depth (or pixel format) of the exported image. The greater the color depth, the more faithfully the image will represent the colors assigned to objects in your document. Note that not all color depths are supported by all export filters. In cases where the requested format is not supported by a particular export filter, the export procedure will automatically select an alternate format that is compatible with the export filter.
ColorReductionMethod	1 = Ordered Dither 2 = Diffuse Dither 3 = Popularity 4 = MedianCut555 5 = MedianCut888	5	In cases where an indexed ColorDepth has been selected for the exported image, the ColorReductionMethod option selects which algorithm is used to reduce the 16 million possible colors in the Surfer plot down to the actual number of colors available in the image's color table.
Quality	N	100	For JPG and JPG2 export only. Set compression quality 0 - 100. A value of 100 is highest quality.
Compress	0 = No compression 1 = Packbits	0	For TIF export only. Set the compression type of an exported TIF file.

	2 = Deflate		
CompressImages	0 = False 1 = True	1	For PDF Raster export only. Compresses images in the PDF to reduce file size.
FitPage	0 = Use application page size 1 = Fit page around exported objects	0	For PDF Raster export only. Sets the size of the PDF to the application page size, if true. Otherwise, PDF page size is set to the size of the exported objects.
Format	0 = Monolithic (one strip) 1 = One row per strip 2 = 16 rows per strip 3 = 64 rows per strip 4 = 64x64-pixel tiles 5 = 256x256-pixel tiles	3	For TIF export only. Choose from a variety of strip and tile output formats when exporting TIF files. The option *In one strip containing the entire image* (0) is the least efficient option but has the highest degree of compatibility with other software.
FormatJ2K	0 = JP2 container 1 = J2K container	0	Sets the Container Format for JPG-2000 export only.
BackgroundColor	1677216 = no transparency 16777217 = background transparency 16777215 = white transparency 0 = black transparency less than 16777216 = custom color transparency	16777216	Color value for transparency for GIF export only. If the value is less than 16777216, the GIF transparent color is composed of red, green, and blue and is set to a custom color. The BackgroundColor value is calculated by, Red + Green*256 + Blue*65536 where *Red*, *Green*, and *Blue* are values between 0 and 255.
UseTransparency	0 = No 1 = Yes	0	Use application background transparency for PNG export only. If set to 1, background is transparent.
SaveRefInfoAsInternal	0 = No 1 = Yes	0	Save spatial references to internal format (if possible)

SaveRefInfoAsBlueMarbleRSF	0 = No 1 = Yes	0	Save spatial references as Blue Marble .RSF file
SaveRefInfoAsESRIprj	0 = No 1 = Yes	0	Save spatial references as Esri .prj file
SaveRefInfoAsESRIWorld	0 = No 1 = Yes	0	Save spatial references as Esri World file
SaveRefInfoAsGSIREF	0 = No 1 = Yes	0	Save spatial references as Golden Software Reference (version 1) file
SaveRefInfoAsGSIREF2	0 = No 1 = Yes	0	Save spatial references as Golden Software Reference (version 2) file
SaveRefInfoAsAUXXML	0 = No 1 = Yes	0	Save spatial references as Esri AUX.XML file
SaveRefInfoAsTAB	0 = No 1 = Yes	0	Save spatial references as MapInfo TAB file

?10G Globe DEM data File Description

Surfer can import GLOBE DEM data ?10g files only. GLOBE DEM data ?10s files are not supported.

GLOBE DEM data ?10g files are the Global Land One-Kilometer Base Elevation (GLOBE) digital elevation model (DEM).

This is 16-bit signed integer data in a simple binary raster. There are no header or trailer bytes embedded in the image. The data are stored in row major order (all the data for row 1, followed by all the data for row 2, etc.). All files have 10800 columns, and either 4800 or 6000 rows."

GLOBE is a data base, a data management philosophy, a working environment, and a file format. GLOBE began with the conceptual opening of a two-dimensional thirty-arc-second (30") latitude-longitude digital data array, and the hope to populate it with both the Best Available Data (B.A.D.), and the Globally Only Open-Access Data (G.O.O.D.). The former could include copyright data that might be made available for distribution by GLOBE with minimal restrictions, while the latter could not contain any restricted data. Allowing for both options has enabled GLOBE, for example, to work with the Australian Surveying and Land Information Group to develop a DEM much better than could otherwise be included while respecting the intellectual property rights of the Australian government.

Digital Elevation Data File Format

Files ?10G and ?10B ("?" is the wildcard notation for tile letters "A" through "P") are provided as 16-bit signed integer data in a simple binary raster. There are no header or trailer bytes embedded in the image. The data are stored in row major order (all the data for row 1, followed by all the data for row 2, etc.). All files have 10800 columns, and either 4800 or 6000 rows. The following diagram depicts the organization of the files:

```
bytes1/2.................................bytes21599/21600
       bytes21601/21602.....................bytes43199/43200
       ....................................................
       ....................................................
       etc.
       ..............................(last byte-1)/(last byte)
```

The data are in little-endian byte order (that is, for IBM-compatible PCs, Digital Equipment VAXes, etc.) UNIX workstations using big-endian byte order can swap bytes using the command:

```
dd if=inputfilename of=outputfilename conv=swab
```

where "inputfilename" and "outputfilename" are replaced with the user's selection of input and output file names

Projection Information

Listed below is the projection information for each data file in GLOBE.

```
Projection       Geographic (latitude/longitude)
        Datum            WGS84
        Zunits           Meters above mean sea level
        Hunits           30 arc-seconds of latitude and longitude
        Spheroid         WGS84
        Xshift           0.0000000000
        Yshift           0.0000000000
        Cell Referencing Each cell is nominally bound by 30" intervals
                         of latitude and longitude, beginning with any
                         whole degree (e.g. 0.0000 degrees)
        Parameters       NONE other than those above
```

Loading ?10G Files

Globe DEM .?10G files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

000 IHO S-57 Navigation Chart File Description

Surfer imports International Hydrographic Organization S-57 Electronic Navigation Chart (ENC) .000 files. S-57 ENC files include point, polyline, and polygon features with attributes. S-57 files are typically used to view hydrographic data. S-57 .000 files may include feature update files with extensions .001, .002, etc. and text files with information about the data or collecting agency.

For detailed file structure information and for attribute definitions, see the *IHO Transfer Standard for Digital Hydrographic Data Publication S-57 Edition 3.1*.

Loading S-57 .000 Files

S-57 .000 files can be loaded to create a base map (Home | New Map | Base) with the **Import** dialog.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

References:

IHO Transfer Standard for Digital Hydrographic Data. Publication S-57 Ed 3.1. Nov 2000. International Hydrographic Bureau. www.iho.int

IHO S-57 (ENC). Geospatial Data Abstraction Library. www.gdal.org

ACCDB Microsoft Access File Description

Microsoft Access .MDB is a binary database file format used by pre-2007 versions of Microsoft Access. The .ACCDB format is used in Access 2007 and 2010. **Surfer** can import data from tables and queries in both Access .MDB and .ACCDB formats.

Import Options

See Database Tables and Fields Dialog.

Import Automation Options

Import Automation Options

Microsoft Access (MDB) Import Options Dialog

If the .MDB file contains multiple tables, you can select which table to load in the *Choose Table To Load* list. The other controls in the dialog show previews of the data that will be imported. See Microsoft Access .MDB Import Options Dialog.

64-Bit Access Driver

In order to import Microsoft Access Database (*.mdb, *.accdb) files, you must have the Microsoft Access Database driver installed on your machine. It's shipped as part of the Microsoft Office suite and comes in 32-bit and 64-bit versions. Installing the 32-bit Microsoft Office suite will install the 32-bit Access Database driver. Installing the 64-bit Microsoft Office suite will install the 64-bit Access Database driver. Unfortunately, Microsoft doesn't allow BOTH to be installed simultaneously on a 64-bit Windows platform. If you need to import data from Microsoft Access Database files into

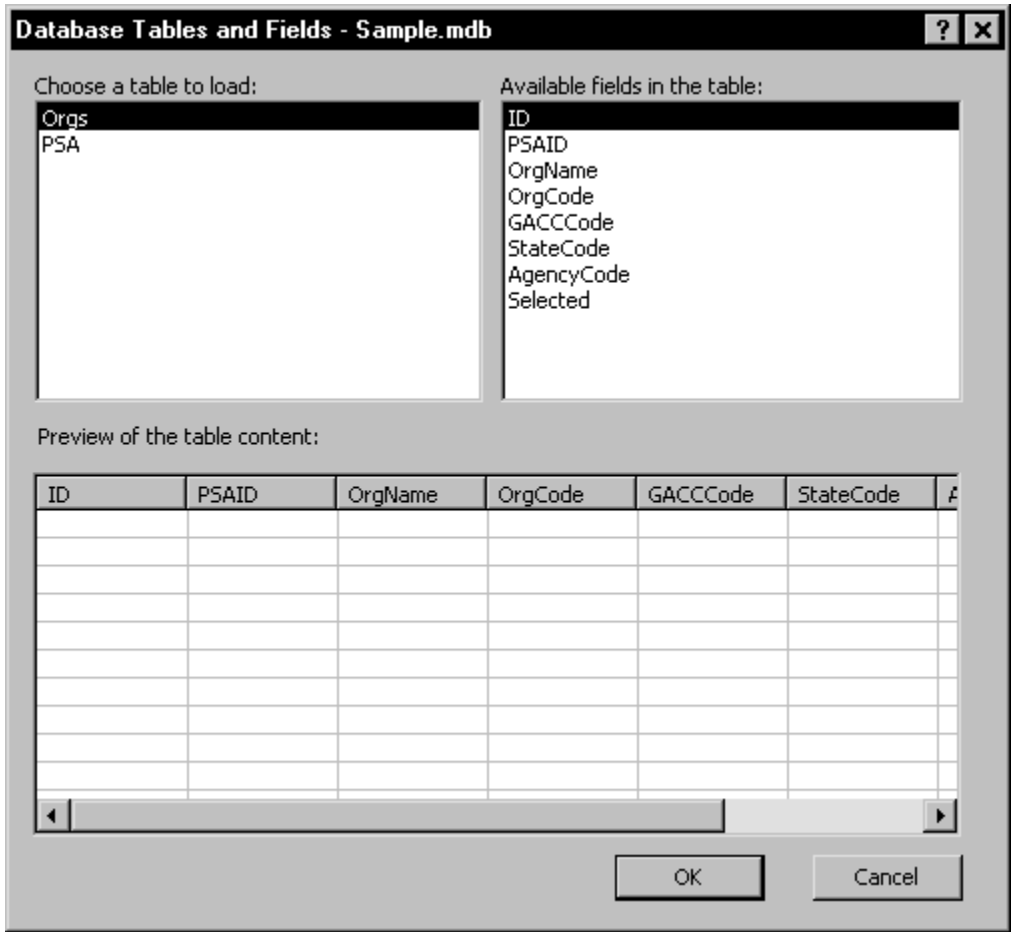
Golden Software products, you must install the 32-bit version of our product if you have a 32-bit Access Database driver. You must install the 64-bit version of our product if you have a 64-bit Access Database driver. If you don't need to import Access Database data with our product, you may install either version on a 64-bit Windows platform.

Microsoft Access .MDB and .ACCDB Import Options Dialog

Microsoft Access .MDB is a binary database file format used by pre-2007 versions of Microsoft Access. The .ACCDB format is used in Access 2007 and 2010. **Surfer** can import data from tables and queries in both Access .MDB and .ACCDB formats.

Database Tables and Fields Dialog

In the worksheet, select **File | Import** to load a .MDB or .ACCDB file. The **Database Tables and Fields** dialog allows you to choose what table or query to load and preview the data that will be imported.



*Specify .MBD or .ACCDB import options in the **Database Tables and Fields** dialog.*

Choose a Table to Load

If the .MDB or .ACCDB file contains multiple tables and queries, you can select which table or query to load in the *Choose Table To Load* list.

Available Fields in the Table

The *Available fields in the table* displays all of the available fields in the table.

Preview of the Table Content

A preview of the selected table content is shown in the *Preview of the table content* section.

ADF Esri ArcInfo Binary Grid File Description

Surfer can import and export Esri Arc/Info binary grid files .ADF.

The Arc/Info Binary Grid format is the internal working format of the Arc/Info Grid product. This format should not be confused with the Arc/Info ASCII Grid format which is the interchange format for grids.

A grid coverage (or data set) actually consists of a number of files. A grid normally lives in its own directory named after the grid. For example, the grid nwgrd1 lives in the directory nwgrd1, and has the following component files:

- nwgrd1/dblbnd.adf - file containing the bounds of the grid.
- nwgrd1/hdr.adf - header file containing grid cell resolution, type, compression, and tile information.
- nwgrd1/prj.adf - optional projection file containing the known map projection information of the grid.
- nwgrd1/sta.adf - statistics file containing minimum, maximum, mean, and standard deviation of the cell values.
- nwgrd1/vat.adf - value attributes file containing the attribute data associated with the zones of the zones of the grid.
- nwgrd1/w001001.adf - the actual raster data file containing the value of each cell in the first (base) tile.
- nwgrd1/w001001x.adf - index file containing file offset and size of each block in the raster data file.

Note: If the grid has more than one tile, the value and index .adf file names will increment correspondingly. For example, the data files for tile 2 could be w001002.adf and w001002x.adf, tile 3 could be w001003.adf and w001003x.adf, and so on. The actual numbering of additional tiles is automatic and is based on their spatial relationship to the first tile. **Surfer** automatically reads all tiles and locates each appropriately.

Loading Files

Esri ArcInfo Binary Grid .ADF files are similar to grid files and can be loaded to create a grid based map (i.e. **Home | New Map | Contour**), or when using the functions in the Grid menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or the **File | Save A**s command in the grid node editor to export an Esri ArcInfo binary grid .ADF file.

Note: Esri ArcMap or ArcCatalog requires that .ADF files do not have spaces in the file names. If you want to include a space, it is recommended that you replace all spaces with an underscore (_).

Export Options Dialog

See ArcInfo ADF Export Options.

Export Automation Options

See ADF Esri ArcInfo Binary Grid Export Options

ADF Esri ArcInfo Binary Grid File Format Export Options

Surfer imports and exports Esri ArcInfo Binary Grid files. These files are uniform grids.

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .ADF files. The Spatial References page is displayed when exporting a .ADF file when the grid file contains a coordinate system. The dialog controls how spatial reference information is stored.

AM Amira Mesh [.AM, .COL] Files

Surfer imports and exports Amira Mesh files. These files can contain uniform lattices, point data, or geometric figures.

File name extensions

.AM, .COL

Format(s) Supported for Import

- 3D uniform lattice; 8-, 16-, 32-bit integer, float, double
- point set; double

Format(s) Supported for Export

- 3D uniform lattice; 8-, 16-, 32-bit integer, float, double

Loading Files

Amira Mesh files are similar to grid files and can be loaded to create a grid based map (i.e. **Home | New Map | Contour**), or when using the functions in the Grid menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Import Restrictions/Limitations

Amira data sets that contain geometric figures are imported as point sets, i.e. "HxLineSet" and "Tetrahedra."

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .AM, .COL files. Alternatively, the Amira Mesh file format is available when prompted to save a grid file.

Export Options

See Amira Mesh Export Options.

Export Restrictions/Limitations

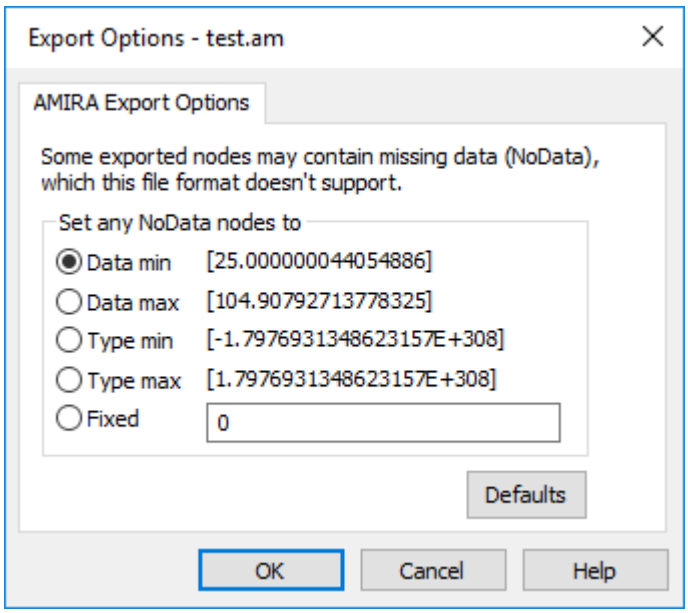
The current implementation does not support exporting of geometric data.

Amira Mesh .AM, .COL Export Options

Surfer imports and exports Amira Mesh files. These files can contain uniform lattices, rectilinear lattices, curvilinear lattices, point data, or geometric figures.

Export Options Dialog

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .AM, .COL files. The **Export Options** dialog is only displayed if the lattice contains one or more nodes assigned the NoData value. The dialog controls determine how NoData values are treated in the export file.



*Control how the NoData nodes are exported in the AMIRA **Export Options** dialog.*

Data Min

Choose *Data min* to export NoData nodes with the indicated minimum value (varies depending on contents of lattice).

Data Max

Choose *Data max* to export NoData nodes with the indicated maximum value (varies depending on contents of lattice).

Type Min

Choose *Type min* to export NoData nodes with the indicated minimum value for the lattice's data type (usually zero).

Type Max

Choose *Type max* to export NoData nodes with the indicated maximum value of the lattice's data type.

Fixed

Choose *Fixed* to export NoData nodes with some other value, and enter the desired value in the adjacent edit control.

Defaults

Return all settings to the default values by clicking the *Defaults* button.

Amira Mesh .AM, .COL Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"BlankMethod=BM_FIXED, FixedBlankValue=1.70141E+038"

This would indicate that the NoData value for a lattice or grid node is a custom value, and that value is equal to **Surfer's** NoData value.

Option	Action	Default
BlankMethod	Method for assigning values to blanked nodes: BM_DATA_MIN = Minimum value of the lattice BM_DATA_MAX = Maximum value of the lattice. BM_TYPE_MIN = Minimum value of the inherent data type. BM_TYPE_MAX = Maximum value of the inherent data type. BM_FIXED = A user specified value	BM_TYPE_MAX
FixedBlankValue	Used if BlankMethod is BM_FIXED	No

ASC Arc/Info ASCII Grid

Surfer imports and exports ASC Arc/Info ASCII grid files.

An Esri grid is a raster GIS file format developed by Esri. The ASCII format is also known as Arc/Info ASCII grid. The ASCII format is used as an exchange, or export format, with a simple ASCII file structure.

File Format

The first six lines of the file indicate the reference of the grid, followed by the values listed in the order they would appear (left to right and top to bottom).

Example

```
ncols         4
nrows         6
xllcorner     0.0
yllcorner     0.0
cellsize      50.0
NODATA_value -9999
 -9999 -9999 5 2
-9999 20 100 36
3 8 35 10
```

32 42 50 6

88 75 27 9

13 5 1 -9999

Where:

- *ncols* is the numbers of rows (represented as integers);
- *nrows* is the numbers of rows (represented as integers);
- *xllcorner* and *yllcorner* are the western (left) X coordinate and southern (bottom) Y coordinates, such as easting and northing (represented as real numbers with an optional decimal point)
- *cellsize* is the length of one side of a square cell (a real number)
- *nodata_value* is the value that is regarded as "missing" or "not applicable"; this line is optional, but highly recommended as some programs expect this line to be declared (a real number).

The remainder of the file lists the raster values for each cell, starting at the upper-left corner. These real numbers (with optional decimal point, if needed) and are delimited using a single space character.

File name extensions

.AIG, .ASC, .AGR, .GRD

Loading Files

ArcInfo ASCII grid files are similar to grid files and can be loaded to create a grid based map (i.e. **Home | New Map | Contour**), or when using the functions in the Grid menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .AIG, .ASC, .AGR, and .GRD files. Alternatively, the ArcInfo ASCII Grid file format is available when prompted to save a grid file.

Export Options

See Spatial References

ASI Amira Stacked Slices [.ASI] File Description

Surfer imports images in the Amira Stacked Slices format. One or more images are imported from separate files as specified by a description file. The description file can be created by the user or by another application. The recommended (though not mandatory) file name extension is .ASI.

File Description

The description file is an ordinary text file. Any lines in the file that begin with a pound (#) character are assumed to be comments and are ignored. Any other lines must adhere to the format described below.

Typically the file begins with a comment line that reads

```
# Amira Stacked Slices
```

Next comes the optional "pathname" line. If present, this line specifies the pathname of the directory where the image files are located. If the pathname line is not present, the software assumes the image files are located in the same directory as the description file.

Example 1

```
pathname c:\MyPictures
```

Next comes the optional "pixelsize" line. If present, this line specifies the spacing of the lattice nodes for each pixel in application units.

Example 2

```
pixelsize 0.1. 0.1
```

If the pixelsize line is not present, the default size is one unit per pixel.

Next comes the image list consisting of one or more lines, each of which specifies the file name of an image file to load. Each of these lines can also optionally specify an application Z position for each image slice.

Example 3

```
picture1.jpg 10.0
picture2.bmp 11.0
picture3.jpg 12.0
picture4.tif 15.0
```

The above example loads four image slices positioned at 10, 11, 12, and 15 units along the Z axis.

If the z positions are not given with the image file names, the default z positioning, which uses the larger of the x pixel size and y pixel size as the amount to space between each slice on the z axis, is used.

If the z positions are uniformly spaced, the imported lattice is of the "uniform" lattice type.

Finally, the file ends with a line containing the word "end."

Example 4: Description File

```
# Amira Stacked Slices

# Directory where image files live
pathname c:\MyPictures

# Pixel Size
pixelsize 0.5 0.5

# Image List
slice1.jpg 1.6
slice2.jpg 1.8
slice3.jpg 2.0

# Mark the end of the description file
end
```

The image files to which the description file refers may include any of the image file formats supported by the other filters, such as .JPG, .BMP, .PNG, .TGA, and so on.

Loading .ASI Files

Amira Stacked Slices .ASI files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Import Restrictions/Limitations

Note that the images referenced by any particular description file must all be of the same width and height (in pixels) and have the same color depth (in bits per pixel). If any referenced image differs in dimensions or color depth from the others, an error message displays and the import does not complete.

BIL Band Interleaved (BIL, BIP, BSQ) File Description

Surfer can import and export lattice data as .BIL, .BIP, and .BSQ raster data sets.

Band Interleaved by line .BIL format is a common format for distributing satellite image data and organizing image data for multiband images. Band Interleaved by Pixel .BIP, and Band Sequential .BSQ. The .BIL, .BIP, and .BSQ are formats produced by remote-sensing systems. The primary difference among them is the technique used to store brightness values captured simultaneously in each of several colors or spectral bands. The data values are stored in a simple binary raster.

The .BIL, .BIP, and .BSQ files are binary files, and they must have an associated ASCII header .HDR file. This header file must have the same file name as the .BIL, .BIP or .BSQ, and only its file extension differs. This header file contains data about the image such as the number of rows and columns in the image and latitude and longitude.

There are no header or trailer bytes imbedded in the image. The data are stored in row major (all the data for row 1, followed by all the data for row 2, etc.). A header file is an ASCII text file containing the size and coordinate information for the raster data.

The .BIL, .BIP, and .BSQ files are not in themselves image formats but are schemes for storing the actual pixel values of an image in a file. These files support the display of single and multiband images and handle black-and-white, grayscale, pseudo color, true color, and multispectral image data.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

See Spatial References

BIL, .BIP, .BSQ File Format Export Options

Surfer can import and export lattice data as .BIL, .BIP, and .BSQ raster data sets.

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .BIL, .BIP, or .BSQ files. The Spatial References page is displayed when exporting a .BIL, .BIP, or .BSQ file when the grid file contains a coordinate system. The dialog controls how spatial reference information is stored.

BLN Golden Software Blanking 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.

Example 1

This example shows a simple .BLN file, with a single polygon:

```
5 0
```

```
1 1
1 3
4 3
4 1
1 1
```

Example 2

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
```

Loading a BLN

Use the **File | Open**, **File | Import**, or the **Open Data** dialog (i.e. **Home | New Map | Base**, **Home | New Map | Post**) to load a .BLN file.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

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.

Export Options Dialog

See BLN Golden Software Blanking Export Options Dialog

Export Automation Options

See BLN Golden Software Blanking Export Automation Options

Golden Software Blanking [.BLN] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

```
"Defaults=1,AreasToCurves=1"
```

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to closed curves.

Option	Action	Default
`Defaults=1`	Set all options to their default values	No
`ForgetOptions=1`	Don't remember options for later use	No
`AreasToCurves=1`	Convert area objects to closed curves	No

Golden Software Blanking .BLN Export Options Dialog

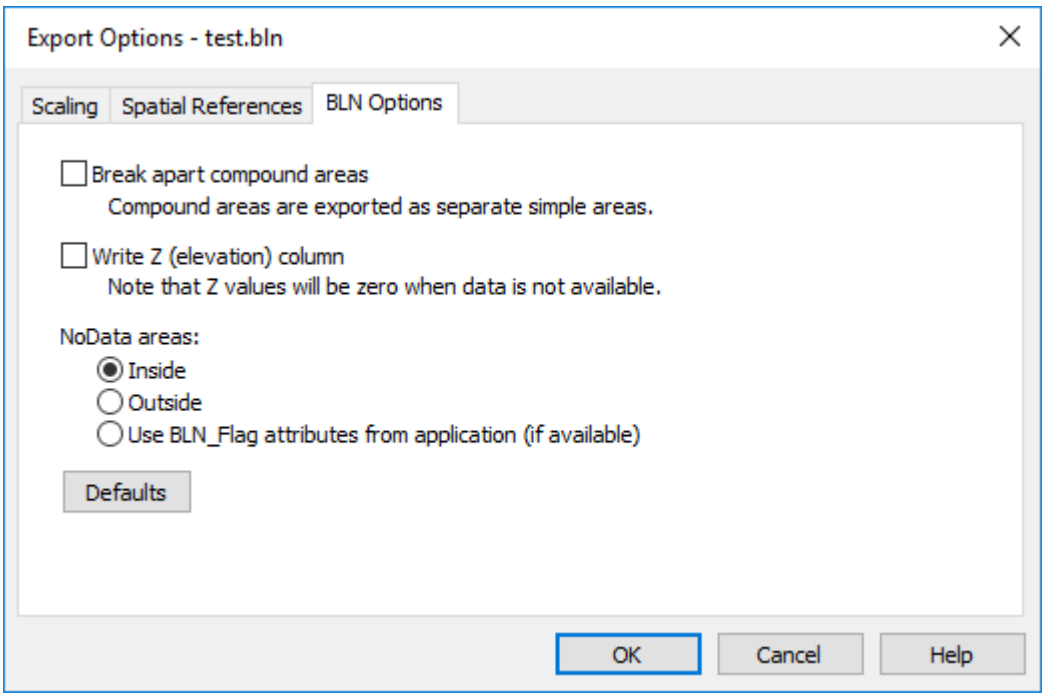
The **Export Options** dialog allows you to specify options which determine how information in the file is exported.

When exporting, the application specifies coordinates in *Page Units* (inches or centimeters) as indicated on the document rulers. You may want these values offset and/or scaled back to original map coordinates in the Golden Software Blanking .BLN file.

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.

BLN Options Page



Specify the BLN export options in the ***Export Options*** *dialog.*

Break Apart Compound Areas

Choose *Break apart compound areas* to have compound areas (those containing islands or lakes) output as separate area entities. This option should be chosen if the .BLN file is to be used as a boundary file. Do not choose this option if the .BLN file is to be used as a blanking file.

Write Z (elevation) column

Choose *Write Z (elevation) column* to export the .BLN file with the Z values. If ZLEVEL values are not available for the object, 0 will be written in the Z column.

NoData Areas

Select whether to assign NoData values to areas *Inside*, *Outside*, or *Use BLN_Flag attributes from application (if available)*. Selecting *Inside* writes the blanking flag as 1 in the .BLN file. Selecting *Outside* writes the blanking flag as 0 in the .BLN file. Selecting *Use BLN_Flag attributes from application (if available)* sets the blanking flag for each object to its BLN_Flag attribute value.

The blanking flag can be overridden in the Assign NoData to Grid dialog. However, the default option is to use the blanking flag from the BLN file to define the NoData direction.

Defaults

The *Defaults* button sets all options to default conditions.

Scaling Page

See the Scaling Page for detailed information.

Golden Software Blanking [.BLN] Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,ScalingSourceApp=0"

This would set all export options to their default values, then indicate the scaling source information should not be taken from the application, but from previously saved values.

Option	Action	Default	Description
BreakApartCompoundAreas	0 = No 1 = Yes	0	Compound areas (multi-ring polygons) will be split apart into multiple non-compound areas (simple polygons) during export.
WriteZ	0 = No 1 = Yes	0	Z values (elevation) will be included in the .BLN file. When Z values are not available for an object, 0 will be written in the Z column.
BlankMode	0 = Outside 1 = Inside 2 = Use BLN_Flag attribute	1	When set to 0, sets the blanking flag to 0 to assign NoData values outside areas. When set to 1, sets the blanking flag to 1 to assign NoData values inside areas. When set to 2, the BLN_Flag attribute determines the blanking flag for each object.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.

FileLLY	N.N	Set scaling rectangle lower left Y value.
FileURX	N.N	Set scaling rectangle upper right X value.
FileURY	N.N	Set scaling rectangle upper right Y value.

Remarks

Scaling rectangle parameters changed between **Surfer 8** and **Surfer 9** and newer versions. For example:

```
Use FileURX instead of BLNUpperRightX.
Use PageLLY instead of APPLowerLeftY.
```

Surfer 8 and older rectangle-coordinate parameters are still supported, but in new scripts use of the current parameter names is recommended.

Boolean values within options strings are not equivalent to Booleans in **Scripter** BASIC. Use “1” instead of “True” and “0” instead of “False”.

Create a BLN File to Display Multiple Polylines

You may want to create a BLN file with multiple lines for use as a fault file, a breakline file, to slice a grid file, or to use as a base map file. To create a BLN file containing multiple polylines, there are a few options:

Digitize from Surfer

It is easy to digitize line segments from an existing map in **Surfer** (contour, base, etc.). When creating a blanking file in this manner the header information is added to the file automatically. To create a blanking file with the Digitize command:

1. Create a map in the plot window, and then select the map. This can be an overlay, although the overlay cannot contain a wireframe or surface, or be tilted at any angle other than 90 degrees.
2. Click **Map Tools | Layer Tools | Digitize** and the pointer becomes a cross hair.
3. Click the left mouse button in the plot window to write the current coordinates to the **Digitized Coordinates** editor window.
4. If you wish to create multiple polylines, click in the **Digitized Coordinates** window and 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.
5. When you are finished defining the polyline(s), click **File | Save As** in the **Digitized Coordinates** window, and save the file as a BLN file. Press ESC or choose any tool button to exit digitize mode.

Draw it in the Plot Window

You can also draw the objects you want over your map and export them to a BLN file. To do this:

1. Select your existing map and click Map Tools | Add to Map| Layer | Empty Base. That will add an empty base map layer, called *Base*, to your map. You can see this in the **Contents** window.
2. Select the new empty base map layer in the **Contents** window and click Features | Group | Start Editing. You are now "inside" the base layer group.
3. You may want to zoom in a few times so you can see the map clearly.
4. Click Features | Insert | Polyline and draw the polyline over the map. You can use the scroll bars on the sides of the window when drawing the polyline.
5. Double click when you are finished to end the polyline. Now the base map layer has one polyline object in it. Draw as many polylines as needed.
6. Press the ESC key on the keyboard to exit draw mode.
7. Click **Features | Group | Stop Editing** to exit the base layer group.
8. Turn off the four axes in the **Contents** window (clear the visibility check boxes) and any other map layers (e.g. the contour layer).
9. Go to File | Export.
10. In the **Export** dialog, choose a location to save the file, give the file a name, and choose to save it as *BLN Golden Software Blanking (*.bln)*.
11. Click *Save*.
12. In the **Export Options** dialog, click *Save* again and the file is created.

Create Manually in a Text Editor

You can also simply enter the vertex coordinates for the polylines in a text editor or the Surfer worksheet. This method is best if you have known X,Y values for your line segments. The BLN format is a one-line header followed by a list of XY coordinates for the object. The header is simply the number of data points in the object. Then save the file with a BLN extension.

Below is an example of the proper format for four separate 2-point polylines. The header consists of the number of points in the line.

```
2
-170.036813503, 41.2459673979
-157.773313416, 57.8892889456
2
-103.171471766, 40.3700031059
-81.5643300805, 51.7575389017
2
-106.091352739, 26.0625638538
-120.398791991, 44.1658483712
2
-119.230839602, 35.9901591629
-155.729396734, 19.9308138099
```

Note: You can view your BLN file in **Surfer** using the Home | New Map | Base command.

BNA Atlas Boundary File Description

Surfer imports and export Atlas Boundary files .BNA.

The Atlas Boundary File .BNA is an ASCII format file used to store spatial information including areas, curves, ellipses and points. Spatial information is only concerned with the location of objects

in space (i.e., their coordinates) and not with their attributes (such as line or fill style, marker symbol used, text labels, etc.).

File Format

The general format of the file is:

```
"Pname 1", "Sname 1", type/length
x1,y1
x2,y2
...
xn,yn
"Pname 2", "Sname 2", type/length
x1,y1
x2,y2
...
xn,yn
```

Pname

Pname is the name of the primary ID. The primary ID is used to link the object to external data.

Sname

Sname is the name of the secondary ID. The secondary ID is optional.

Type/Length

The *type/length* is an integer which identifies the object as an area, curve, ellipse or point.

X, Y

Following the *type/length* are 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.

The *type/length* field indicates the number of coordinate pairs to follow and also indicates the type of object as follows:

- Areas have a *type/length* value greater than 2. The value indicates the number of coordinate pairs to follow. Islands and lakes are concatenated to the coordinate list.
- Curves have a *type/length* value less than -1. The absolute value is the number of coordinates to follow for the curve.
- Ellipses have a *type/length* value 2. The first pair of coordinates describe the center of the ellipse. The major and minor radii are stored in the second pair of coordinates. If the minor radius is 0, the ellipse is a circle.
- If the *type/length* field is 1, the object is considered a point. One coordinate pair follows.

Attributes

The first two ID attributes for all polyline, polygon, and symbol objects are automatically exported to all .BNA files. For contour maps, the elevation is exported as the "STD_ID1" attribute for all polylines in the contour map. If other attributes are desired instead of the first two, rename those attributes to "STD_ID1" and "STD_ID2". These named attributes will be used instead of the first

and second attributes listed on the Info tab. The color, size, symbol shape, width, and other properties are not exported.

Simple and Compound Areas

Two kinds of areas exist, simple and compound. A simple area contains a starting point, a series of points specifying the area's boundary and a closing point with the same coordinate as the starting point. A compound area contains one or more subareas, such as islands or lakes. Atlas Boundary files use a special technique to specify the subareas comprising compound areas.

Example 1

A simple area with 5 points is shown in the Atlas Boundary file format:

```
"name" "attrib" 6
2.15, 3.25
3.75, 5.15
6.5, 4.3
5.5, 1.7
4.25, 3.4
2.15, 3.25
```

Example 2

A compound area consisting of a closed outer area and two islands. Here is how the coordinates should be specified in an Atlas Boundary file:

AX1,AY1	Starting point of area "A"
AX2,AY2	Points specifying boundary of area "A"
.	
.	
AXn,AYn	Ending point of area "A"
BX1,BY1	Starting point of subarea "B"
BX2,BY2	Points specifying boundary of subarea "B"
.	
.	
BXn,BYn	Ending point of subarea "B"
AX1,AY1	Starting point of area "A" (Flag Point)
CX1,CY1	Starting point of subarea "C"
CX2,CY2	Points specifying boundary of subarea "C"
.	
.	
CXn,CYn	Ending point of subarea "C"
AX1,AY1	Starting point of area "A" (Flag Point)

Each area's ending point must have the same coordinate as its starting point. The staring point of area "A" is used as a marker (called a Flag Point) to indicate the end of each subarea. This means

the first area point's coordinate must be unique and cannot appear as a coordinate within any subarea.

And, an example of what the actual file may look like:

```
""pname"" ""attrib"" 13
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
```

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

See Atlas Boundary .BNA Import Automation Options.

Export Options Dialog

See Atlas Boundary .BNA Export Options Dialog.

Export Automation Options

See Atlas Boundary .BNA Export Automation Options.

Atlas Boundary [.BNA] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

```
"Defaults=1,AreasToCurves=1"
```

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to closed curves.

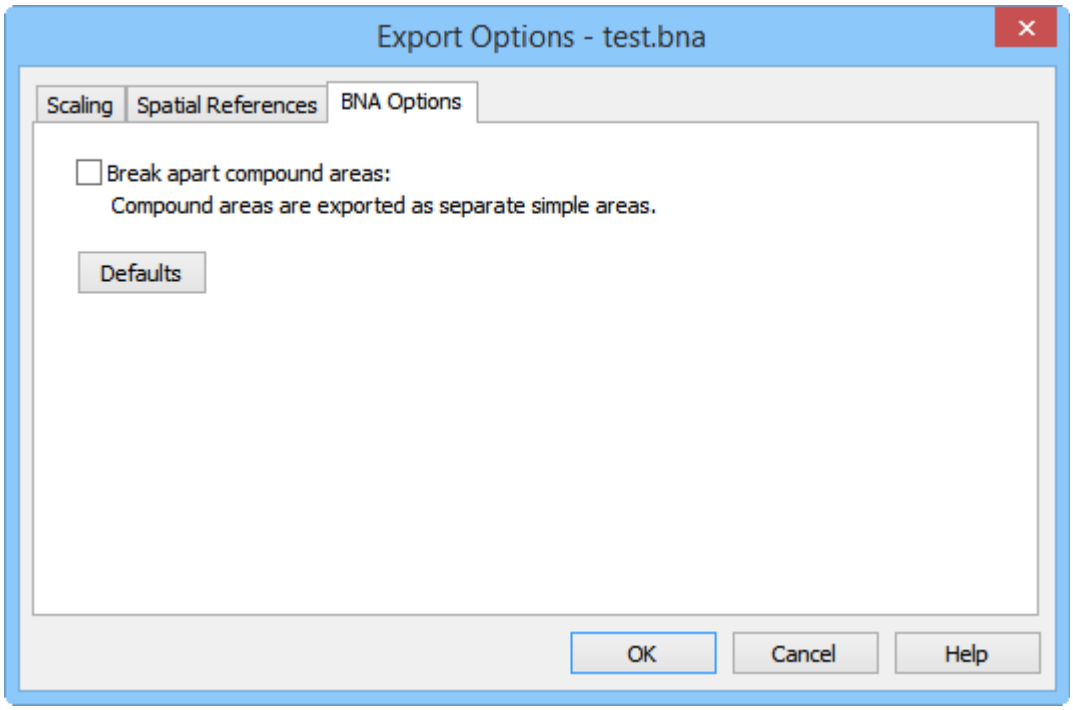
Option	Action	Default
Defaults=1	Set all options to their default values	No
ForgetOptions=1	Don't remember options for later use	No
AreasToCurves=1	Convert area objects to closed curves	No

Atlas Boundary [.BNA] Export Options Dialog

The **Export Options** dialog allows you to specify options which determine how information in the file is exported.

Attributes

The first two ID attributes for all polyline, polygon, and symbol objects are automatically exported to all .BNA files. For contour maps, the elevation is exported as the "STD_ID1" attribute for all polylines in the contour map. If other attributes are desired instead of the first two, rename those attributes to "STD_ID1" and "STD_ID2". These named attributes will be used instead of the first and second attributes listed on the Info tab. The color, size, symbol shape, width, and other properties are not exported.



*The **Export Options** dialog controls the BNA export options.*

BNA Options Page

Check the *Break apart compound areas* option to export compound areas as separate simple areas.

Scaling Page

See the Scaling Page for detailed information.

Atlas Boundary [.BNA] Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,ScalingSourceApp=0"

This would set all export options to their default values, then indicate the scaling source information should not be taken from the application, but from previously saved values.

Option	Action	Default	Description
BreakApartCompoundAreas	0 = No 1 = Yes	0	Compound areas (multi-ring polygons) will be split apart into multiple non-compound areas (simple polygons) during export.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.
FileURY	N.N		Set scaling rectangle upper right Y value.

Remarks

Scaling rectangle parameters have changed between **Surfer 8** and **Surfer 9 and newer versions**. For example:

```
Use FileURX instead of BNAUpperRightX.
Use PageLLY instead of APPLowerLeftY.
```

Older rectangle-coordinate parameters are still supported, but in new scripts use of the current parameter names is recommended.

Boolean values within options strings are not equivalent to Booleans in **Scripter** BASIC. Use "1" instead of "True" and "0" instead of "False".

Atlas Boundary (BNA) Examples

The Atlas [.BNA] file is used to store geographic information including areas, curves, and points. This format is not commonly used in **Surfer**.

Alternate BNA Format

"Pname 1", "Sname 1", type/length, x1, y1, x2, y2, x3, y3, ... xn, yn

"Pname 2", "Sname 2", type/length, x1, y1, x2, y2, x3, y3, ... xn, yn

Examples of the Format for Different Objects

Example 1: Polygon

This is an example of a **polygon**. The polygon has both a Primary and Secondary ID.

"CO","Colorado",7
-102.073,37.0033
-103.015,37.01
-103.106,37.01
-109.064,37.0033
-109.08,41.0091
-102.081,41.0157
-102.073,37.0033

Example 2: Curve

This is an example of a **curve**. In this example, this curve does not have a Primary or Secondary ID, although they can be assigned for curves by typing the information in the BNA file.

"","",-10
535.954,1681.83
541.209,1241.31
1166.1,1466.83
672.581,1723.87
604.268,1351.66
1260.25,1089.36
1260.25,1571.92
535.954,1792.18
132.202,1262.33
1260.25,837.565

Example 3: Point

This is an example of a **point**. The point has both a Primary and Secondary ID.

"Well HA-11"," Hansford Field",1

-104.351,37.847

CLR Color Spectrum File Format

Color spectrum files .CLR are used to define a spectrum, or continuous gradation of colors. This is specified by a series of anchor points with associated colors. The colors between anchor points are interpolated from the nearest anchor points.

The basic format consists of an ASCII file with header information on the first line. Subsequent lines specify anchors points, one anchor point per line.

The header consists of the following space-delimited fields:

Element	Description
Id	case-sensitive string "ColorMap" without the quotes
Version	format version number, this should be set to 1 or 2
InterpMethod	interpolation method between anchors, this should be set to 1

Subsequent lines define the anchor points, one per line. Each line has the following space-delimited fields:

Element	Description
Position	The position is the floating point percentage value (from 0.0 to 100.0). Positions must be specified in increasing order from 0 to 100 percent, and the 0 and 100 percent positions must be specified in the file.
Red	red color component (0 to 255)
Green	green color component (0 to 255)
Blue	blue color component (0 to 255)
Alpha	opacity component (0 to 255). 0 is fully transparent. 255 is fully opaque.

Alpha is not included in Version 1 files.

Examples

In the following example, the anchor points are at 0, 50, and 100. The zero position is a slightly transparent blue, the 50 percent anchor is mostly transparent green, and the 100 percent position is fully opaque yellow.

```
ColorMap 1 1
  0   0   0 255
```

```
 50   0 255   0
100 255 255   0

ColorMap 2 1
    0.000000   0   0 255 200
   50.000000   0 255   0  20
  100.000000 255 255   0 255
```

It is also possible to have coincident anchor points in a color file. Anchors and colors are interpreted in order from 0 percent to 100 percent. In the case of coincident points, you can create maps with distinct boundaries, similar to the example shown here.

```
ColorMap 2 1
    0.000000 255   0   0 255
   50.000000 255 255   0 255
   50.000000   0   0 255 255
  100.000000 255 255 255 255
```

There are many sample .CLR files located in the samples folder. A popular .CLR file is RAINBOW.CLR. The following example shows the format of RAINBOW.CLR:

```
ColorMap 1 1
  0 153 102 255
 20   0   0 255
 40   0 255   0
 60 255 255   0
 80 255 102   0
100 255   0   0
```

CLS Classed Post File Format

Classed post maps .CLS file formats are used to define a class and the symbol associated with the class. The basic format consists of an ASCII file with header information on the first line. Subsequent lines specify each class, with one class on each row. 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.

The header consists of the following space-delimited fields:

Element	Description
Id	case-sensitive string "ClassData" without the quotes
Version	format version number, this should be set to 1 or 2. 1 does not specify the line and fill color for the symbol separately.
BinningMethod	binning method, this should be set to 0, 1, or 2. 0 = Equal Number, 1 = Equal Interval, and 2 = User Defined.

Subsequent lines define the class, one per line. Each line has the following space-delimited fields:

Element	Description
Minimum	The minimum value is the lower bound for this class. The minimum value is included in the class.
Maximum	The maximum value is the upper bound for this class. The maximum value is not included in the class.
Symbol Set	The symbol set is the name of the font to use for the symbol. For example "GSI Default Symbols". The name is enclosed in double quote marks.
Symbol Index	The symbol index is the number of the symbol to use.
Color	In Format 1 CLS files, the color is for both the line and fill symbol color, specified as RGBA values, such as 81 186 198 255, with the values for R G B A separated by spaces. RGBA values range from 0 to 255 with 0 = none of the specified color or transparency and 255 = maximum of the specified color or transparency.
Fill Color	In Format 2 CLS files, the symbol fill color, specified with a color name, such as "Blue" or with RGBA values, such as "R81 G186 B198 A255". Color names and RGBA values are enclosed with double quotes. R is the red component, G is the green component, B is the blue component, and A is the alpha (opacity). RGBA values range from 0 to 255 with 0 = none of the specified color or transparency and 255 = maximum of the specified color or transparency.
Line Color	In Format 2 CLS files, the symbol line color, specified with a color name, such as "Blue" or with RGBA values, such as "R81 G186 B198 A255". Color names and RGBA values are enclosed with double quotes. R is the red component, G is the green component, B is the blue component, and A is the alpha (opacity). RGBA values range from 0 to 255 with 0 = none of the specified color or transparency and 255 = maximum of the specified color or transparency.
bitflag	An indicator that tells the program to use the line color for the fill color. As this is no longer supported and the colors are listed separately, this value must be 1.
Size	The symbol size in inches.

Examples

In the following example, 3 classes are defined. The first class has min and max values at 25 and 60, second at 60 and 75, and third at 75 and 105.1 The first class has a slightly transparent blue 0.01 inch circle (with a black outline in the type 2 CLS). The second class has a solid opaque green 0.20 inch diamond. The third class has a black 0.01 inch cross hair.

Format 1:

min max "symbol set" symbol_index color size

```
ClassData 1 0
25 60 "Default Symbols"   12   0   0 255 255    0.100000
60 75 "Default Symbols"    6   0 255   0 255    0.200000
75 105.1 "Default Symbols"    0   0   0   0 255    0.100000
```

Format 2:

min max "symbol set" "fill color" "line color" bitflag symbol_index size

```
ClassData 2 0
25 60 "GSI Default Symbols" "R0 G0 B255 A150" "Black" 1 12 0.1
60 75 "GSI Default Symbols" "Green" "Green" 1 6 0.2
75 105.1 "GSI Default Symbols" "Black" "Black" 1 0 0.1
```

CPS CPS-3 Grid File Format

Surfer imports and exports CPS-3 ASCII grid files. This import filter only supports the GRID type of the CPS formats.

File Description

The format has several file extensions: .ASC, .DAT, .GRD, .CPS, and .CPS3.

The data are written a column at a time from the left side to the right side of the grid.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

No export dialog is displayed.

Export Automation Options

No export options are available.

DBF ASCII Database [.DBF] File Description

Surfer imports data from dBase/xBase database .DBF files.

Xbase is a complex of data files .DBF, indexes .NDX, .MDX, .CDX, etc. and eventually note files .DBT for storing large amounts of formatted data in a structured form.

DBase's database system was one of the first to provide a header section for describing the structure of the data in the file. This meant that the program no longer required advance knowledge of the data structure, but rather could ask the data file how it was structured. Note that there are several variations on the .DBF file structure, and not all dBase-related products and .DBF file structures are necessarily compatible.

A second file type is the .DBT file format for memo fields. While character fields are limited to 254 characters each, a memo field is a 10-byte pointer into a .DBT file which can include a much larger text field. DBase was very limited in its ability to process memo fields, but some other xBase languages treat memo fields as strings just like character fields for all purposes except permanent storage.

DBase uses .NDX files for indexes. Some xBase languages include compatibility with .NDX files while others use different file formats.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

See DBF ASCII Database Import Automation Options

ASCII Database .DBF Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

```
"ImportCodePage=1252"
```

This would set the Unicode Language ID value to code page 1252, or the Western European (Windows) code page.

Option	Action	Default
ImportCodePage	The name of the ANSI code page to use when importing Unicode data. Valid code page number in the range 0 through 65535.	""

Note: If a companion .SHP file with the same name exists in the folder containing the .DBF file, the .DBF file is assumed to be part of an ESRI-compatible Shapefile set and the defaults as described for the *ImportCodePage* option in the Esri Shapefile .SHP Import Automation Options section apply.

Otherwise, the Language ID value stored in the .DBF file's header determines the code page used. However, if the Language ID value is zero or invalid, the OEM DOS code page determined by the machine's Control Panel locale setting will be used.

DDF SDTS TVP Topological Vector Profile and Raster Profile File Description

Surfer can import USGS SDTS Topological Vector Profile .TVP or .DDF data sets.

SDTS File Information

The Spatial Data Transfer Standard, or SDTS, is a robust way of transferring earth-referenced spatial data between dissimilar computer systems with the potential for no information loss. It is a transfer standard that embraces the philosophy of self-contained transfers, i.e. spatial data, attribute, georeferencing, data quality report, data dictionary, and other supporting metadata all included in the transfer.

The purpose of the SDTS is to promote and facilitate the transfer of digital spatial data between dissimilar computer systems, while preserving information meaning and minimizing the need for information external to the transfer. Implementation of SDTS is of significant interest to users and producers of digital spatial data because of the potential for increased access to and sharing of spatial data, the reduction of information loss in data exchange, the elimination of the duplication of data acquisition, and the increase in the quality and integrity of spatial data. SDTS is neutral, modular, growth-oriented, extensible, and flexible--all characteristics of an "open systems" standard.

The SDTS provides a solution to the problem of spatial data transfer from the conceptual level to the details of physical file encoding. Transfer of spatial data involves modeling spatial data concepts, data structures, and logical and physical file structures. To be useful, the data to be transferred must also be meaningful in terms of data content and data quality. SDTS addresses all of these aspects for both vector and raster data structures.

There are two separate types of SDTS DEM files: topological vector profile SDTS and raster profile SDTS files.

TVP

The topological vector profile SDTS files contain boundary line information and can be used as a base map (use with **File | Import** or Home | New Map | Base).

DDF

The raster profile SDTS files contain gridded elevation information and can be used to directly create a grid-based map (i.e. **Home | New Map | Contour**) or to perform grid based functions (i.e. **Grids | Edit | Assign NoData**). An uncompressed SDTS contains several files with the .DDF extension. All of the .DDF files are necessary to produce a map (i.e. you cannot copy just one .DDF file and create a map from it).

Remarks

It is not necessary to unzip the .TAR.GZ, .TAR, .ZIP, or .TGZ file. **Surfer** can read the information in the various files directly from the compressed file.

If you do unzip the .TAR.GZ, .TAR, .ZIP, or .TGZ file containing the DDF files, there is an option in WinZip (or other unzipping software) that needs to be disabled. Use the WinZip settings to disable this option. All .DDF files must be extracted into the same directory.

1. In WinZip, use the **Options | Configuration** command to open the **Configuration** dialog.
2. Click on the *Miscellaneous* tab.
3. In the Other category, un-check TAR file smart CR/LF conversion.
4. Click the *OK* button.
5. The Tar.GZ file will now properly unzip the files.

Import Options Dialog

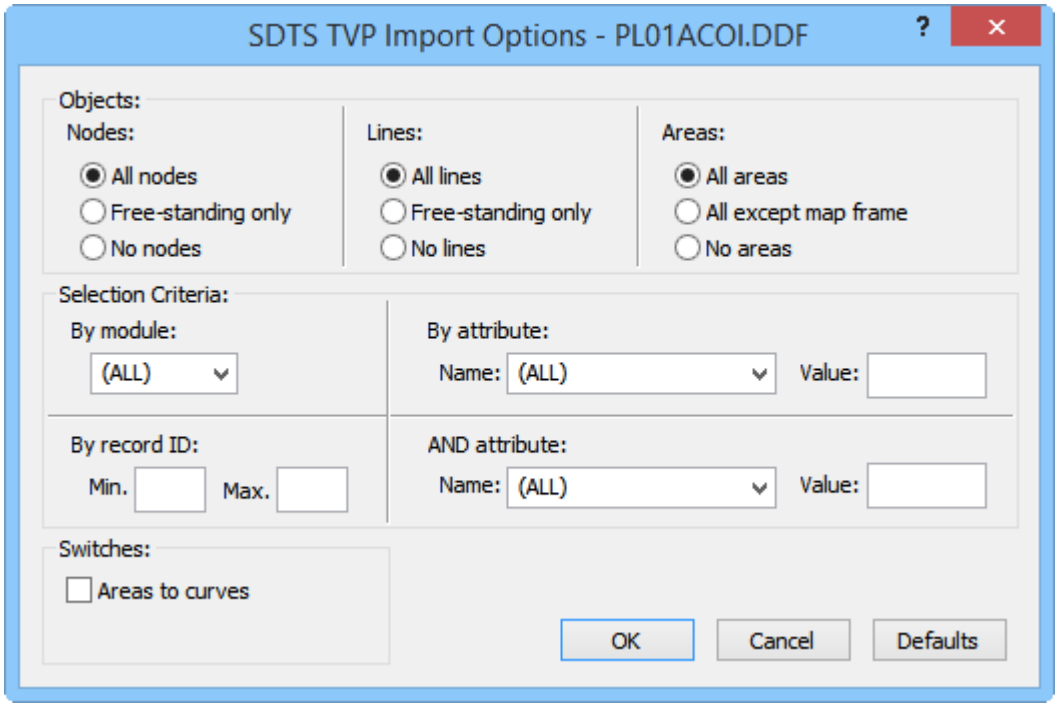
SDTS TVP Import Options Dialog

Import Automation Options

SDTS TVP Import Automation Options

SDTS Topological Vector Profile .TVP Import Options Dialog

Select any .DDF file to open the **Import Options** dialog.



*Customize the import options in the **SDTS TVP Import Options** dialog.*

Nodes Options

These options determine which Node items are imported. If *All nodes* is selected, all nodes in the SDTS data set are imported. If *Free-standing only* is selected, only those nodes that are not associated with an area or line are imported. If *No nodes* is selected, none of the nodes are imported.

Lines Options

These options determine which Line items are imported. If *All lines* is selected, all lines in the SDTS data set are imported. If *Free-standing only* is selected, only those lines that are not associated with an area are imported. If *No lines* is selected, none of the lines are imported.

Areas Options

These options determine which Area items are imported. If *All areas* is selected, all areas in the SDTS data set are imported. If *All except map frame* is selected, the areas that make up the outline of the map will not be imported. If *No areas* is selected, none of the areas are imported.

Some SDTS data sets supplied by USGS contain a map frame that is expressed in the data set as a normal polygon instead of an "invisible" polygon (i.e. a "PC" entity instead of a "PW" or "PX" entity in SDTS terminology). In such cases, the *All Except Map Frame* control will have no effect.

By Module List Box

If the name of a specific module is selected, imported items will be limited to those that reside in the specified module. If *(ALL)* is selected, items will be imported from all modules in the SDTS data set.

By Record ID Edit Boxes

Each item in the SDTS data set has a unique record ID number. To import only those items within a specific range of record IDs, enter the lowest desired record ID number in the *Min* edit box, and the highest desired record ID number in the *Max* edit box.

By Attribute, AND Attribute Controls

Application-specific attributes are associated with some items in an SDTS data set. To import only those items that have a specific attribute, select the name of the desired attribute in the *Name* list box and enter the value of the desired attribute in the *Value* edit box. If *(ALL)* is selected in the list box, items will be imported without regard to attributes. If both *By Attribute* and *AND Attribute* are specified, only those items that have both of the specified attributes will be imported.

Areas to Curves

If the *Areas to curves* box is checked, any areas in the data set will instead be imported as lines (curves) instead of polygonal areas.

Defaults Button

The *Defaults* button resets the Import Options to default values. The default options direct the Import Filter as follows: Import free-standing nodes, import free-standing lines, import all areas, don't limit by module, record ID, or attribute, don't import areas as curves, and don't synthesize Ids.

SDTS Topological Vector Profile Import Filter Messages

The following messages may appear while importing SDTS Topological Vector Profile files.

- Expected SDTS Topological Vector Profile (TVP) format. Dataset appears to contain SDTS Raster (grid) Profile.

This error appears if the selected .DDF file is in the incorrect format. If you receive this message after choosing **Home | New Map | Base**, you likely have a SDTS DEM file. Try using the **Home | New Map | Contour** command instead.

SDTS Topological Vector Profile [.TVP] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,AreasToCurves=1"

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to curves (lines).

Option	Action	Default	Description
AreasToCurves	0 = No 1 = Yes	0	Convert all polygons to polylines.
WhichLines	0 = Import all lines 1 = Import only freestanding lines 2 = Import no lines	0	Specifies which lines from the file are to be imported.
WhichNodes	0 = Import all nodes 1 = Import no nodes	0	Specifies whether nodes (points) from the file are to be imported.
WhichAreas	0 = Import all areas 1 = Import no areas	0	Specifies whether ares from the file are to be imported.
WhichText	-2 = Import all text -1 = Import no text N = any other value indicates which specific text group number to import	-2	Specifies which text group from the file is to be imported. For example, use WhichText=3 to import only the text from group 3.

DEM SDTS DEM File Description

Surfer imports elevation data in the form of a uniform lattice from SDTS Digital Elevation Model .DEM data sets.

There are two separate types of SDTS DEM files. The Topological Vector Profile SDTS files contain boundary line information and can be used as a base map (use with **File | Import** or Home | New Map | Base).

The Raster Profile SDTS files contain gridded elevation information and can be used to directly create a grid-based map such as a contour map.

Remarks

It is not necessary to unzip the .TAR.GZ, .TAR, .ZIP, or .TGZ file. **Surfer** can read the information in the various files directly from the compressed file.

If you do unzip the .TAR.GZ, .TAR, .ZIP, or .TGZ file containing the DDF files, there is an option in WinZip (or other unzipping software) that needs to be disabled. Use the WinZip settings to disable this option. All .DDF files must be extracted into the same directory.

1. In WinZip, use the **Options | Configuration** command to open the **Configuration** dialog.
2. Click on the *Miscellaneous* tab.
3. In the Other category, un-check TAR file smart CR/LF conversion.
4. Click the *OK* button.
5. The Tar.GZ file will now properly unzip the files.

Format(s) Supported for Import

2D uniform lattice; 8-, 16-, 32-bit integer, float, double

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Notes

SDTS data sets can contain elevation data or several other types of data that are not applicable to **Surfer**. If you attempt to import an SDTS data set that does not contain elevation data, an error message displays. If the error message displays after trying to create a grid based map, try clicking the **Home | New Map | Base** command instead.

DEM USGS Digital Elevation Model File Description

Surfer can import and export USGS Digital Elevation Model .DEM files.

The .DEM is a grid file and can be used to create grid based maps such as contour maps, 3D surface maps, etc.

The USGS DEM files have been replaced with the USGS SDTS files. **Surfer** reads this older file type. DEM files containing arc-second XY coordinates are automatically converted to latitude/longitude decimal degrees.

The USGS produced two DEM file types: the 7.5 minute DEM data and the 1:250,000-scale DEM data.

- The 7.5 minute DEMs correspond to the standard 1:24,000 scale quadrangles. Elevation points are provided along north-south profile lines, and use a 30 meter spacing between adjacent data points and adjacent profile lines. Point locations are referenced to the UTM (Universal Transverse Mercator) coordinate system. The 7.5 minute quadrangles are usually not oriented exactly north-south. 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. This results in the potential for a regular, stair-stepped arrangement of the data points along the edges of the map which can translate to NoData grid nodes along the edges of grid files produced in

Surfer. This can lead to discontinuous contour lines along the edges of contour maps, or NoData edges on surface plots.

- The 1:250,000 DEMs cover a 1 degree by 1 degree block representing one-half of a 1 degree by 2 degree 1:250,000-scale map. Point locations are based on the latitude/longitude coordinate system. Elevation points are provided along north-south profile lines, and use a 3 arc-second spacing between each data point and between profile lines. Three arc seconds correspond to approximately 90 meters in the north-south direction and variable spacing in the east-west direction due to convergence of meridians as latitude increases (approximately 90 meters at the equator and approximately 60 meters at 50 degrees latitude).

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options

See USGS Digital Elevation Model .DEM Export Options Dialog.

Export Automation Options

See USGS Digital Elevation Model .DEM Export Automation Options.

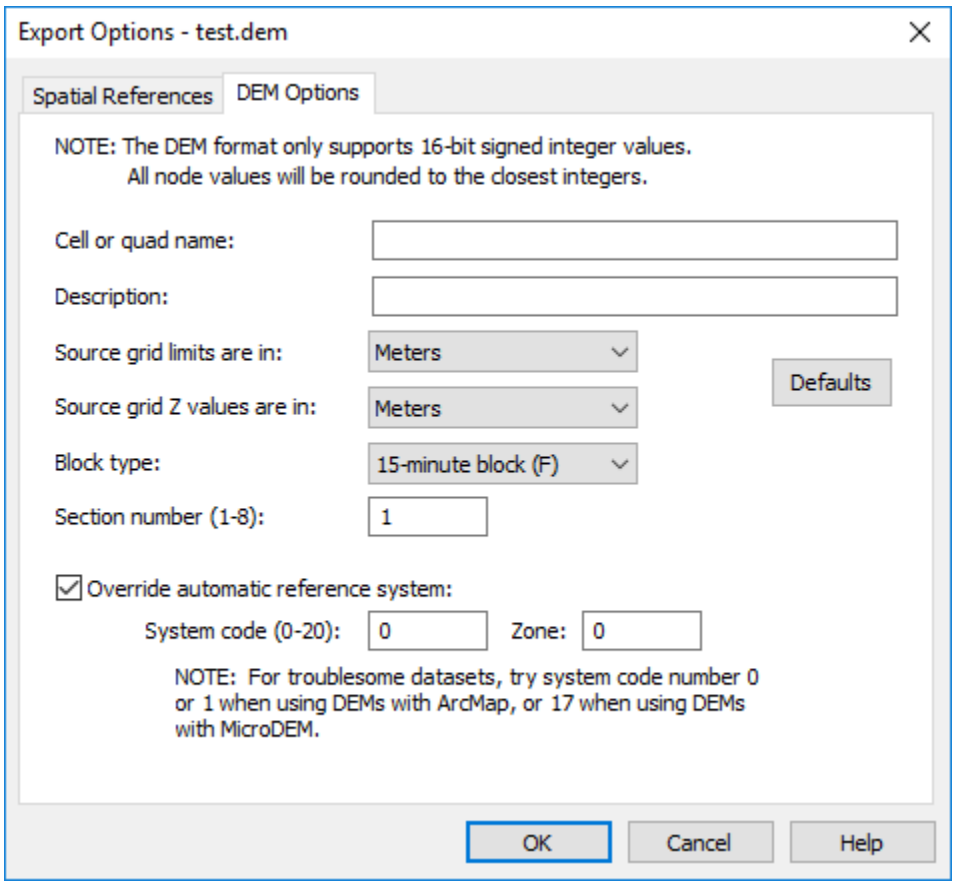
Remarks

The USGS header may contain some fields that **Surfer** does not support.

The elevation data contained in USGS DEM files are regularly spaced, but the direction of the grid lines may not exactly coincide with the grid edges. This is due to the variable angle between the UTM coordinate system (used in some DEM files) and true north. This results in the potential for a regular, stair-stepped arrangement of the grid nodes along the edges of the map which can translate to NoData grid nodes along the edges of maps produced in **Surfer**. If the DEM file contains three-second arc coordinates, the coordinates are automatically converted to latitude/longitude coordinates in **Surfer**.

USGS Digital Elevation Model .DEM Export Options Dialog

Use the **Grids | Edit | Convert** command to save a USGS .DEM file in the plot document. Use the **File | Save As** command to save a USGS .DEM file in the grid editor.



*Customize the DEM options in the **DEM Export Options** dialog.*

Cell or quad name

Type a name for the DEM image in the *Cell or quad name* box.

Description

Type a description for the DEM image in the *Description* box.

Source Grid Limits

Specify the type of units for the XY coordinates in the source grid from the *Source grid limits are in* list. Click the arrow to display the list. Possible selections are *Radians, Feet, Meters, Arcseconds,* and *Degrees*. The export units need to be the same as the map units. For example, if the map units are lat/long, and you choose Meters as the X unit code, then there will be no conversion. If the map is in lat/long, then you have to choose to export as lat/long. You put in what the units of the map are, not what you want the export file to be in.

Source Grid Z Values

Specify the type of units for the Z (vertical) coordinates in the source grid from the *Source grid Z values are in* list. Click the arrow to display the list. Possible selections are *Feet* or *Meters*. The export units need to be the same as the map units. For example, if the map units are lat/long, and you choose Meters as the X unit code, then there will be no conversion. If the map is in lat/long, then you have to choose to export as lat/long. You put in what the units of the map are, not what you want the export file to be in.

Block type

Specify the block type for the DEM image from the *Block type* list. Click the arrow to display the list.. Possible selections are *15-minute*, *7-minute*, and *Other*.

Section Number

Specify the section number by entering a number from one to eight in the *Section number (1-8)* box. Specify the section number by entering a number from 1 to 8 for a 15-minute block. Specify the section number by entering a number from 1 to 32 for a 7.5-minute block. Ignore the section number for “Other” block types.

Override Automatic Reference System

Select the *Override automatic reference system* option if you wish to override the automatic reference system code and zone in the DEM file. Clear the *Override automatic reference system* check box to automatically set the reference system code and zone.

The *System code* value indicates the projection. The *Zone* indicates the State Plane or UTM coordinate zone, if applicable. If the DEM exported from **Surfer** is not read properly by another application, you may wish to override the automatic code. For example, Esri's ArcMap 10 will not read DEMs with *System code*s greater than 3.

When the *Override automatic reference system* option is selected, the *System code* and *Zone* fields are available. Set the *System code* to a value between 0 and 20. If the *System code* is 1 (UTM) or 2 (State Plane), set the *Zone* to the appropriate coordinate zone number. See the USGS *Standards for Digital Elevation Models Part 2*, Appendices 2-E, 2-F, and 2-G for more information and a full list of the *System code* and *Zone* values.

Standards for Digital Elevation Models

Part 2: Specifications

National Mapping Program

Technical Instructions

U.S. Department of the Interior

U.S. Geological Survey

National Mapping Division

Appendix 2-I

Sectional indicator

The 30-minute DEM's are distributed in groups of files that make up a 30- by 30-minute area of coverage representing the DEM for the east or west half of a 1:100,000-scale source map. The normal distribution group is four 15-minute files per 30-minute area. The quadrangle name field in the header record contains the name of the 1:100,000-scale source map. However, the pieces or sections into which each is divided are identified within the header type A record to the size and placement of each. In byte 138-140 each section is identified by a 3 character code

XNN where:

X is a single letter indicating size

F = 15-minute block

S = 7.5-minute block

NN is a two-digit number indicating the specific quad. Figure 2-I-1 and 2-I-2 illustrate this division with the sections labeled with the code that appears in bytes 138-140 of the header record.

F01 F02 F03 F04

F05 F06 F07 F08

Figure 2-I-1

A 1:100,000-scale quad divided into eight 15-minute quads, 4 per 30-minute area.

S01 S02 S03 S04 S05 S06 S07 S08

S09 S10 S11 S12 S13 S14 S15 S16

S17 S18 S19 S20 S21 S22 S23 S24

S25 S26 S27 S28 S29 S30 S31 S32

Figure 2-I-2

A 1:100,000-scale quad divided into 32 7.5-minute quads, 16 per 30-minute area.'

USGS Digital Elevation Model .DEM Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" BlockType=1,RefOverride=1,RefSys=2,RefZone=12"

This would indicate that the DEM is a 7.5-minute block type and the reference system has been manually defined.

Option	Action	Default
BlockType	Data block type:. 0=15-minute block 1=7.5-minute block 2=Other block type.	0
RefOverride	0 = Use automatic reference system code and zone 1 = Override automatic system code and zone. Specify system code and zone values with RefSys and RefZone	0
RefSys	N = Set reference system code to an integer value between 0 and 20. Use when RefOverride = 1.	
RefZone	N = Reference Zone. Use when RefOverride = 1 and RefSys = 1 or 2.	

SectNum	Section indicator. For 15-min block it must be greater than or equal to 1 and less than or equal to 8. For 7.5-min block it must be greater than or equal to 1 and less than or equal to 32.	1
XYUnitCode	X and Y coordinate units. 0=radians 1=feet 2=meters 3=arc-seconds 4=degrees	2
ZUnitCode	Z coordinate units 1=feet 2=meters	2

Remarks

The YXUnitCode and ZUnitCode refer to the source lattice limits and the Z value units, not those in the resulting export file. If the data are in units other than the source units, the program will not convert the data units.

DGN Microstation Design v7 File Description

Surfer imports Microstation Design v7 .DGN files. DGN files are created by MicroStation and Intergraphs's Interactive Graphics Design System (IGDS). DGN files are binary CAD data files that include point, polyline, polygon, and text features with attributes. Surfer can also import v6 and older version DGN files. DGN files include file set-up information, graphic elements, and non-graphic data. Some feature style and color information is maintained when importing DGN files.

Surfer does not import DGN v8 files at this time.

Loading Microstation DGN v7 Files

Microstation DGN v7 files can be loaded to create a base map (Home | New Map | Base) with the **Import** dialog.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

References:

MicroStation 95 Reference Guide. Ch 18, pg 18-1. Bentley Systems.

DGNLib: a Microstation DGN (ISFF) Reader. Warmerdam, Frank. dgnlib.maptools.org

Microstation DGN. Geospatial Data Abstraction Library. www.gdal.org

DIC DICOm3 Medical Image File Description

Digital Imaging and Communications in Medicine (DICOM) is a standard for handling, storing, printing, and transmitting information in medical imaging. It includes a file format definition and a network communications protocol. The communication protocol is an application protocol that uses TCP/IP to communicate between systems. DICOM files can be exchanged between two entities that are capable of receiving image and patient data in DICOM format. The National Electrical Manufacturers Association (NEMA) holds the copyright to this standard. It was developed by the *DICOM Standards Committee*, whose members are also partly members of NEMA.

DICOM enables the integration of scanners, servers, workstations, printers, and network hardware from multiple manufacturers into a picture archiving and communication system (PACS). The different devices come with DICOM conformance statements which clearly state the DICOM classes they support. DICOM has been widely adopted by hospitals and is making inroads in smaller applications like dentists' and doctors' offices.

Surfer imports images and lattices from DICOM 3 medical image data sets. This filter is also able to read some files written in the obsolete ACR-NEMA format (from which the DICOM format was derived); however, Golden Software does not officially support the ACR-NEMA format.

DICOM File Format

DICOM differs from other data formats in that it groups information into data sets. That means that a file of a chest X-Ray image, for example, actually contains the patient ID within the file, so that the image can never be separated from this information by mistake.

A DICOM data object consists of a number of attributes, including items such as name, ID, etc., and also one special attribute containing the image pixel data (i.e. logically, the main object has no "header" as such - merely a list of attributes, including the pixel data). A single DICOM object can only contain one attribute containing pixel data. For many modalities, this corresponds to a single image. But note that the attribute may contain multiple "frames", allowing storage of cine loops or other multi-frame data.

DICOM uses three different Data Element encoding schemes. With Explicit VR Data Elements, for VRs that are not OB, OW, OF, SQ, UT, or UN, the format for each Data Element is: GROUP (2 bytes) ELEMENT (2 bytes) VR (2 bytes) LengthInByte (2 bytes) Data (variable length). For the other Explicit Data Elements or Implicit Data Elements, see section 7.1 of Part 5 of the DICOM Standard.

The same basic format is used for all applications, including network and file usage, but when written to a file, usually a true "header" (containing copies of a few key attributes and details of the application which wrote it) is added.

File name extensions: DICOM .DIC, .DCM and ACR-NEMA .AN1, AN2.

Format(s) Supported for Import

- device-independent bitmap; 8, 24, 32 bit per pixel
- uniform lattice; 8-, 16-, 32-bit integer, float, double

Import Options

See DICOM Import Options Dialog.

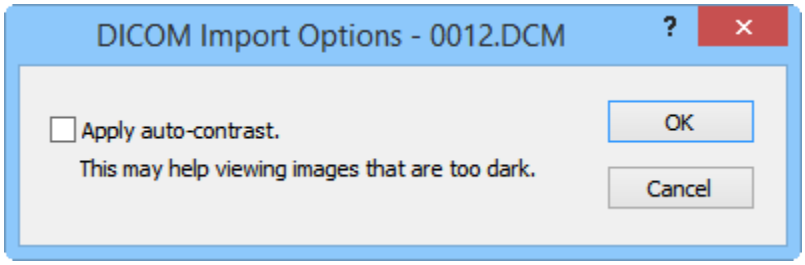
Import Restrictions/Limitations

The DICOM specification allows an unusually wide variety of different formats and encodings within the same file format. While this software can read most of the common variants of DICOM, it would not be practical to develop software to read every possible variant. Some of the known deficiencies in this implementation include:

- DICOM images that contain bit per pixel counts other than 8, 12, 16, 24 or 32 may not be readable depending on the encoding and alignment of the data.
- DICOM images that are encoded with photometric interpretation models other than RGB, grayscale, or monochrome may not be readable. In particular, some YUV encodings cannot be imported.
- Some lossless JPEG images embedded in DICOM data sets do not import. In particular, images encoded with the "Cornell" JPEG codec are not always readable.
- Some of the obscure compression algorithms allowable under the DICOM specification are not supported by this software.
- Some ACR-NEMA files do not import. Golden Software does not officially support the obsolete ACR-NEMA file formats; however, Voxler does import many ACR-NEMA files successfully.

DICOM Import Options Dialog

Import a DICOM [.AN?], [.DCM], file into **Surfer** and the **DICOM Import Options** dialog opens.



*Customize import options in the **DICOM Import Options** dialog.*

Apply Auto-Contrast

Check the *Apply auto-contrast* box to expand the dynamic range of the import data to fit the maximum extents of the import data type. This can improve visibility (contrast) on images recorded with low dynamic range.

DICOM Medical Image File .DIC, .DCM, AN? Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,AutoContrast=1"

This would first set all import options to their default values, then set the AutoContrast option value to one, which would specify that the dynamic range of the imported data will be expanded to improve the contrast of the image.

Option	Action	Default	Description
AutoContrast	0 = No 1 = Yes	0	When set to 0, the data will be imported as-is, without adjustment. When set to 1, the dynamic range of the imported data will be expanded to improve the contrast of the image.

DLG USGS Digital Line Graph File Description

The United States Geological Survey (USGS) provides digitized base map data in "line graph" form. It is available in two formats ("standard" and "optional") on either 9-track magnetic tape or CD-ROM. The CD-ROM with 1:2,000,000-scale DLG files contains data in both standard and optional formats, as well as a "graphic" format. The definitive guide to these file formats is the USGS document: "Digital Line Graphs from 1:24,000-Scale Maps: Data Users Guide 1", "Digital Line Graphs from 1:100,000-Scale Maps: Data Users Guide 2" and "Digital Line Graphs from 1:2,000,000-Scale Maps: Data Users Guide 3".

File Information

Imported DLG files are read in the "standard" and "optional" formats, and provides point, area, and curve objects.

For the 1:2,000,000-scale files, the USGS has divided the U.S. into 21 sections. On the CD-ROM, the files associated with each section are stored in a separate subdirectory.

The 21 subdirectories use the following naming convention:

SECT00

Each subdirectory starts with the letters "SECT" followed by the section number (01 to 21). The sections correspond to the following regions:

SECT01	Northeastern States
SECT02	Middle Atlantic States
SECT03	Southeastern States
SECT04	Florida
SECT05	Southern Mississippi Valley States
SECT06	Central Mississippi Valley States
SECT07	Northern Great Lakes States

SECT08	Southern Texas
SECT09	Southern Plains States
SECT10	Central Plains States
SECT11	Northern Plains States
SECT12	Arizona and New Mexico
SECT13	Southern California
SECT14	Central Pacific States
SECT15	Northwestern States
SECT16	Southeastern Alaska
SECT17	Central Alaska
SECT18	Northern Alaska
SECT19	Southwestern Alaska
SECT20	Aleutian Islands
SECT21	Hawaiian Islands

Each section has one or more data files associated with it. The file naming convention used is as follows:

S00_XX.YYY

Each file starts with the letter "S", followed by the section number (01 to 21) and an underscore (_). The XX stands for the feature code (sometimes referred to as Overlay or Base Category). Feature codes are:

PB = Political Boundaries CF = Cultural Features

AB = Administrative Boundaries ST = Streams and Rivers

RD = Roads and Trails WB = Water Bodies

RR = Railroads HP = Hypsography (Continental Divide Only)

The YYY indicates the data format:

LGS = Line Graph Standard Format

LGO = Line Graph Optional Format

GRF = Graphic Format

If a section has more than one file for a feature, an underscore (_) and a letter are used to uniquely name the files. Examples:

S01_WB_A.LGO

S01_WB_B.LGO

LGO File

An .LGO file contains 15 records of general "header" information, followed by a series of Node, Area and Line entries, in that order. Each Node is assigned a unique ID number (1,2,...). A Node entry contains the node's ID, its coordinate (all coordinates in "optional" format files are in UTM or Albers Equal Area Ellipsoid projection) and the IDs of each line segment that begins at, ends at, or passes

through the node. A "free-standing" node is one that has no line segments associated with it (i.e., the node is an isolated point).

Area ID Number

Each Area is assigned a unique ID number (1,2,...) called an area ID. An Area entry contains the area's ID, the coordinate of its "reference point", a list of the IDs of each line segment that forms the area's boundary (including islands and lakes) and a list of attribute codes assigned to the area. An area's reference point is that point on a map where a textual identifier for the area was placed (such as the name of a county).

Line ID Number

Each Line is also assigned a unique ID number (1,2,...). A Line entry contains the line's ID, the Node ID of the node it starts at, the Node ID of the node it ends at, the Area ID of the area to the left of the line, the Area ID of the area to the right of the line, a list of coordinates of the line vertices and a list of attribute codes assigned to the line segment. (Left and right are relative to the line's direction. The line was digitized from the start point to the end point) . A "free-standing" line is one that is not part of an area boundary.

Attribute Codes

Attribute codes are assigned to areas and line segments for the purpose of identifying and/or grouping them. An attribute code consists of two positive integers, a Major code value and a Minor code value. For example, USGS Section 15 contains data for Washington, Oregon, Idaho and part of Montana. In the Political Boundaries file (S15_PB.LGO), each county boundary area contains two attribute codes. One indicates which state the county was associated with (Major code = 91, Minor code = two-digit FIPS code for the state) and the other indicates which county it is (Major code = 92, Minor code = three-digit FIPS code for the county). For a list of attribute codes, see Major and Minor Attributes.

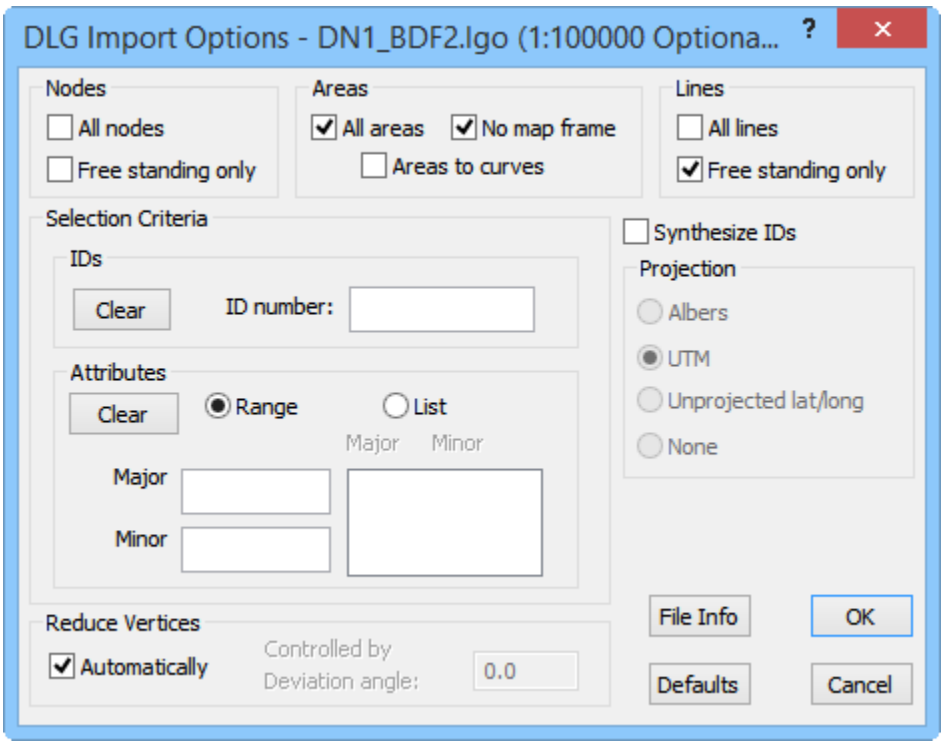
Import Options Dialog

See USGS Digital Line Graph Boundary Import Options Dialog.

Import Automation Options

See USGS Digital Line Graph Boundary Import Automation Options.

USGS Digital Line Graph .DLG Import Options Dialog



The **Import Options** dialog controls what is imported from the USGS DLG files.

Nodes, Areas, Lines

The *Nodes* , *Areas*, and *Lines* groups of check boxes control how the Import Filter handles the Node, Area and Line entries in the DLG (Digital Line Graph) file.

Nodes

The following options may be selected for the nodes:

- *All nodes* - If checked, consider all nodes. If they meet the selection criteria, they will be passed to the application as points.
- *Free standing only* - If checked, consider only free-standing nodes. If they meet the selection criteria, they will be passed to the application as points. The only free-standing nodes are the map reference points.

Areas

The following options may be selected for the areas:

- *All areas* - If checked, consider all area items. If they meet the selection criteria, they will be passed to the application.
- *No map frame* - If checked, the area that makes up the map frame won't be imported.
- *Areas to curves* - If checked, any selected areas passed to the application will be passed as a series of "curve" objects (lines), as opposed to the usual "area" objects (polygons).

Lines

The following options may be selected for the lines:

- *All lines* - If checked, consider all line items. If they meet the selection criteria, they will be passed to the application as "curve" objects.

- *Free standing only* - If checked, consider only free-standing line segments. This is the normal case, since one usually wants line segments that are area boundaries to be passed to the application as part of one or more "area" objects.

Selection Criteria

The *Selection Criteria* edit boxes allow you to specify a value (like 140) or range of values (like 6001-6009 inclusive) which limit the items considered. Leave an edit box empty to place no limit.

ID Number

Only items with the specified ID or IDs within the specified range will be passed back to the application. This is useful for importing a single item (or group of items) of one type. For example, to import just Area 100, make sure no Nodes or Lines boxes are checked, the All Areas box is checked and 100 is entered in the ID Number edit box. The Import Filter considers no nodes or lines and, within areas, only those with ID 100, and passes back one area, Area 100.

Major and Minor Attribute

Objects can be filtered by a range of major and minor attributes or by a list of major and minor attribute pairs.

Range

If the *Range* radio button in the *Attributes* group is turned on, the edit controls for the Major and Minor attributes are enabled. Only items that have at least one attribute code whose Major attribute falls within the specified range and whose corresponding Minor attribute falls within its specified range will be passed to the application. For example, to import reservoirs only, open the appropriate water bodies file (S??_WB.LGO), check only All Areas, enter nothing in the ID Number edit box (means any ID), turn on the Range radio button and enter 40 in the Major Attribute edit box (Water Bodies) and 106 (Reservoir) in the **Minor Attribute** edit box.

List

If the *List* radio button in the *Attributes* group is turned on, the list box is enabled. Only items that have at least one attribute code whose *Major* and *Minor* attributes match one of the attribute pairs in the list box will be passed to the application. To add attribute pairs into the list box, click *File Info* button to bring up the **File Info** dialog, and double-click the attribute pairs in the Attributes group in the **File Info** dialog. To remove an item from the list box select that item and then click the *Clear* button. You may remove multiple items at one time by selecting multiple items in the list box.

Projections

The Projection radio buttons control how coordinates are returned to the application. DLG files have coordinates calculated using a Universal Transverse Mercator (UTM) or Albers Equal Area Ellipsoid projection. UTM is used in 1:24,000-scale maps and 1:100,000-scale maps, while Albers is used in 1:2,000.000-scale maps.

UTM

The native file coordinates of 1:24,000-scale maps or 1:100,000-scale maps are returned to the application, but the application is also given the parameters used in the UTM projection. Use of this option makes sense only if the application understands how to handle a UTM projection.

Albers

The native file coordinates of 1:2,000,000-scale maps are returned to the application, but the application is also given the parameters used in the Albers projection. Use of this option makes sense only if the application understands how to handle an Albers projection.

Unprojected Lat/Long

The file coordinates are converted from their native form to Lat/Long and the application is informed that it is receiving Lat/Long coordinates. This can lengthen the import time considerably, since substantial computation is involved.

None

The native file coordinates are returned to the application, but the application is told that the projection is unknown.

Reduce Vertices

DLG files typically have many vertices in each line segment, often more than are needed for many tasks. The DLG Import Filter provides two methods for reducing the number of vertices.

Automatically

When you check *Automatically*, the import filter applies an algorithm which requires no further input. This algorithm reduces the number of vertices on most DLG line segments by about 50-80%.

Use Deviation Angle

Achieve finer control over vertex reduction by unchecking the *Automatically* box and entering a *Deviation Angle* (in degrees). Use small angles (5-10 degrees) to eliminate a few points, somewhat larger angles (15-25 degrees) to eliminate more points and use larger angles (30-60 degrees) to eliminate the greatest possible number of points.

USGS DLG files have no text ID items associated with Nodes, Areas or Lines. The line segments that make up the Pecos River, for example, are stored as free-standing line segments with river attribute codes, but there is no way to distinguish those line segments from any other line segments with river attributes. Sometimes, it is useful for investigation purposes to know the numeric ID of imported items. When the Synthesize IDs box is checked, the Import Filter synthesizes a Primary ID for each item using the item's type and numeric ID value. (Examples are "N14" for Node 14, "A237" for Area 237 and "L1067" for Line 1067.)

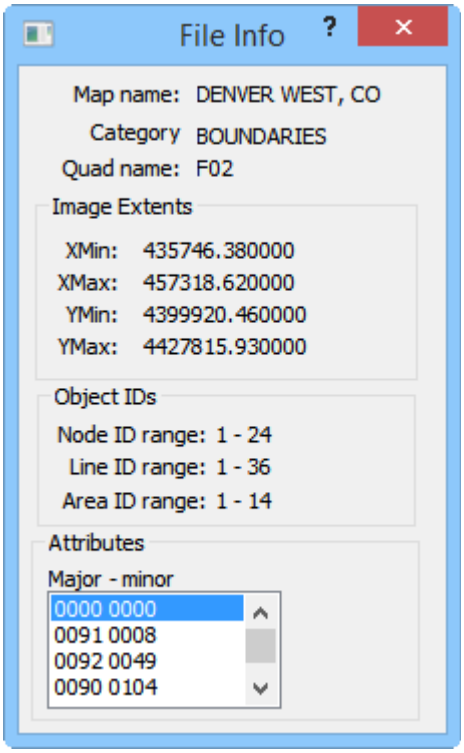
Defaults

The *Defaults* button resets the Import Options to default values. The default options direct the DLG Import Filter as follows: ignore all Nodes, consider all Areas (selected areas are returned as polygons), consider free-standing lines, place no limiting selection criteria, use UTM or Albers projection and do not synthesize text IDs.

File Info

When the *File Info* is clicked, the File Info dialog appears. The base file information is displayed, such as *Image Extents*, *Object IDs*, and *Major/Minor Attributes*.

USGS Digital Line Graph [.DLG] File Info Dialog



*The **File Info** box displays additional information about the DLG file.*

When the *File Info* button is clicked in the DLG Import Options dialog, the **File Info** dialog opens. The base file information is displayed, such as *Image Extents*, *Object IDs*, and *Major/Minor Attributes*.

USGS Digital Line Graph Boundary [.DLG] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. Parameters for the *Attributes* list box, *Range* and *List* radio buttons are not available. If the program is driven from a script, use *MajorSel* and *MinorSel* parameters instead. A typical example would be:

```
"Defaults=1,AreasToCurves=1"
```

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to closed curves.

Option	Action	Default	Description

AllNodes	0 = No 1 = Yes	0	Import all nodes (point entities).
FreeNodes	0 = No 1 = Yes	0	Import only free-standing nodes (points that aren't part of another entity).
AllAreas	0 = No 1 = Yes	1	Import all areas (polygons).
NoMapFrame	0 = No 1 = Yes	1	Don't import the entities that make up the map frame.
AreasToCurves	0 = No 1 = Yes	0	Convert areas (polygons) to curves (polygons).
AllLines	0 = No 1 = Yes	0	Import all lines.
FreeLines	0 = No 1 = Yes	1	Import only free-standing lines (lines that aren't part of another entity).
Synthesize	0 = No 1 = Yes	0	If 1, a unique object ID is synthesized for each object form the object's type, record number, an d name (if applicable).
Projection	0 = Coordinates in the file are projected (UTM or Albers Equal Area projection, depending on the file) 1 = Coordinates in the file are projected, but are to be converted to latitude/longitude during import 2 = Coordinates in the file are unknown or not projected	Varies according to the DLG header information	Specify what type of coordiantes are contained in the import file.
IDSel	M-N		Only consider entities with IDs matching M, or in the range of M-N. For example, "IDSel=1-5".
MajorSel	M-N		Only consider entities whose Major Attribute is M, or in the range of M-N. For example, "MajorSel=1-5".
MinorSel	M-N		Only consider entities whose Minor Attribute is M, or in the range of M-N. For example, "MinorSel=1-5".
ThinAuto	0 = No	1	Automatically thin vertices.

	1 = Yes		
Deviation	N.N	0.0	If ThinAuto (above) is 0, you can manually specify the amount of thinning. The Deviation value in degrees determines how aggressively vertices are thinned during import. For N.N, use small angles (5 to 10 degrees) to eliminate just a few points, larger angles (15 to 25 degrees) to eliminate more points, and even larger angles (30 to 60 degrees) to thin points very aggressively.

DOS USGS ETOPO5 File Description

Surfer can import United States Geological Survey (USGS) ETOPO5 files.

ETOPO5 is a product of a digital data base of land and sea- floor elevations on a 5-minute latitude/longitude grid.

The ETOPO5 data files are commonly available in two different formats.

- An ETOPO5 file with the .DAT file name extension is typically stored in native or " big endian" byte order, as used by Macintosh, Sun, and some other workstations.
- An ETOPO5 file with the .DOS file name extension is typically stored in converted IBM-PC/DEC-VAX (or " little endian") byte order.

Data Sources

Data sources are as follows: Ocean Areas: US Naval Oceanographic Office; USA., W. Europe, Japan/Korea: US Defense Mapping Agency; Australia: Bureau of Mineral Resources, Australia; New Zealand: Department of Industrial and Scientific Research, New Zealand; balance of world land masses: US Navy Fleet Numerical Oceanographic Center. These various data bases were originally assembled in 1988 into the worldwide 5-minute grid by Margo Edwards, then at Washington University, St. Louis, MO.

Data Formats

The data file is formatted as 16-bit BINARY INTEGERS in two byte orders: ETOPO5.DOS is in IBM-PC/DEC-VAX "swapped," lo-byte-first order. The file ETOPO5.DAT is in "normal," or hi-byte-first order, as used by Macintosh, Sun, and some other workstations. In both files, there are 2160x4320 data values, one for each five minutes of latitude and longitude, for a total of 9,331,200 points or 18,662,400 bytes.

Resolution and Accuracy

The resolution of the gridded data varies from true 5-minute for the ocean floors, the USA., Europe, Japan,and Australia to 1 degree in data-deficient parts of Asia, South America, northern Canada, and Africa.

Data values are in whole meters, representing the elevation of the CENTER of each cell.

Accuracy of the data set is hard to define, due to the disparate sources of the data. In general, the data sets for the USA, Western Europe, Korea/Japan, Australia and New Zealand are the most precise, having a horizontal resolution of five minutes of latitude and longitude, and vertical resolution of 1 meter. Data for Africa, Asia, and South America vary in resolution from +/- a few meters to only representing every 150m (500 fet), depending on the available source data. Very little detail is contained in the oceanic data shallower than 200m; the interpolation algorithm used by the US Navy to create the oceanic grid from contour charts was set to an arbitrary cutoff of -10m wherever the algorithm would have "overshot" and marked points as above sea level. An example of such an area is off Argentina, near 45S, 60W. All oceanic data are coded at least -1 m; land data are at 0 or greater, except where lake bottoms or other landlocked features go below sea level (Dead Sea, Death Valley, and in central Australia).

Data Order in the Files

The file may be thought of as having a logical record size of 8640 bytes. The data start at the North Pole (90 deg N, 0 deg 0' E) and are arranged in bands of 360 degrees x 12 points/degree = 4320 values (8640 bytes) ranging eastward from 0 deg 0' East longitude to 359 deg 55' East longitude (since it represents the North Pole, all possible longitudes still refer to a single point, thus the first band has 4320 identical values of -4290 m). The 8641st starts the latitude band for 89 deg 55' N, and so on. There is NO record for the South Pole (elevation 2810 m.)

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

DTED Digital Terrain Elevation Data [.DTED] File Description

Surfer imports digital terrain elevation data or DTED data sets.

Digital Terrain Elevation Data .DTED are grids originally produced by the United States National Imaging and Mapping Agency (NIMA). The files contain latitude/longitude coordinates with the Z units in meters.

There are multiple "levels" of DTED files. Level 0 files .DT0 are 1-degree square blocks with 30 arc-second spacing, similar to GTOPO 30 files. Level 1 files .DT1 use three arc-second spacing. Level 2 files .DT2 use one arc-second spacing.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

DXF AutoCAD DXF File Description

AutoCAD .DXF files are ASCII files (i.e., they can be edited with a text editor, word processor, or worksheet) or binary files (cannot be edited) containing records indicating graphical entities and their attributes. They provide a medium of exchange with AutoDesk's **AutoCAD** program. The format of .DXF files is complex and a detailed discussion is beyond the scope of this Help file. Many books describing the .DXF file format are widely available.

Graphics

Graphical information may be stored in the **AutoCAD** Drawing Exchange Format (.DXF). Many programs, including AutoDesk Inc.'s AutoCAD (Computer Aided Design) program can import .DXF files, allowing one to display and/or further manipulate the images. **Surfer** supports MTEXT (multi-line text block) background color in .DXF import filter.

3DSOLID is a proprietary format, and 3DSOLID entities are not supported in **Surfer.** If the .DXF file contains 3DSOLID objects, the 3DSOLID objects will be omitted from the import.

Data

AutoCAD .DXF files can contain point data that includes X, Y, and Z data. DXF files can be opened in a worksheet, used for gridding, or used for post map or classed post map display. When the DXF file is opened in the worksheet, the X and Y coordinates are displayed in columns A and B. If the DXF file contains Z values, the Z values are displayed in column C. If no Z value is included, column C contains a zero (0). When used for post map, classed post map, and gridding, the X, Y, and Z (if any) are automatically read into the appropriate locations.

ASCII or Binary Format

.DXF files can be stored in either ASCII or Binary format. ASCII .DXF files are the most versatile, since they can be displayed, edited, printed and transported to non-IBM machines (such as mainframes, minicomputers or Macintosh). However, they are somewhat bigger and take longer to read back into another application.

Table and Entities Sections

.DXF files have two important sections.

- The *Tables* section contains definitions of the various line styles and other attributes.
- The *Entities* section contains specific information about each graphical entity (line, polygon, etc.) including coordinates and references to the attributes in the defined in the Tables section. All exported graphical entities are assigned to a layer named GSLAYER.

Text

Text can be imported and exported in .DXF files.

Text can also be exported as **AutoCAD** text entities (*All text as areas* unchecked). No matter what typeface is specified in the application document, all text entities are assigned **AutoCAD's** STANDARD font. Once inside **AutoCAD**, the text entities can be edited in the normal **AutoCAD** fashion. As long as there is no shear or perspective, .DXF text entities will be sized and oriented similar to the text objects in the application document. Shear occurs when the character glyphs are not perpendicular to the text baseline. Perspective occurs when the height of glyphs in the text string are not all the same, as in a 3D view where the glyphs are smaller the farther they are from the observer.

Line Styles

Lines styles are exported with equivalent **AutoCAD**-compatible line types. The document's internal line styles are assigned the following **AutoCAD** line type names:

Document	AutoCAD
Solid	CONTINUOUS
Internal Dash	GSDASHED
Internal Dot	GSDOTTED
Internal Dash-Dot	GSDASHDOT
Internal Dash-Dot-Dot	GSDASHDOTDOT

Custom line styles in the document are assigned AutoCAD line type names of the form GSCUSTOM0, GSCUSTOM1, GSCUSTOM2, etc.

Color Numbers

Indexed .DXF color numbers are assigned to each entity. Color numbers (1,2,3,…,255) are indices into **AutoCAD's** internal color table. By convention, the first 7 color numbers are guaranteed to have known colors assigned to them by **AutoCAD**. They are:

1	Red
2	Yellow
3	Green
4	Cyan
5	Magenta
7	Black (Default for GSLAYER)

AutoCAD has a default association of colors to color numbers, but the **AutoCAD** user is free to change the colors associated with color numbers 8 through 255. When exporting to .DXF format, the color number of the color from the default **AutoCAD** color table closest to the actual color of the object is assigned to the entity in the .DXF file. Unless you use only the seven colors listed above, the color of objects inside **AutoCAD** may be different than those in the application document.

AutoCAD 2004 and later versions support true colors. When user chooses *AutoCAD 2004 or later* in the **Export Options** dialog, true colors are written to the export file.

Import Options

See AutoCAD .DXF Import Options Dialog.

Import Automation Options

See AutoCAD .DXF Import Automation Options.

Open Options

DXF files can be opened in the worksheet using the File | Open command, used for gridding, or post map or classed post map display.

Export Options

See AutoCAD .DXF Export Options Dialog.

Export Automation Options

See AutoCAD .DXF Export Automation Options.

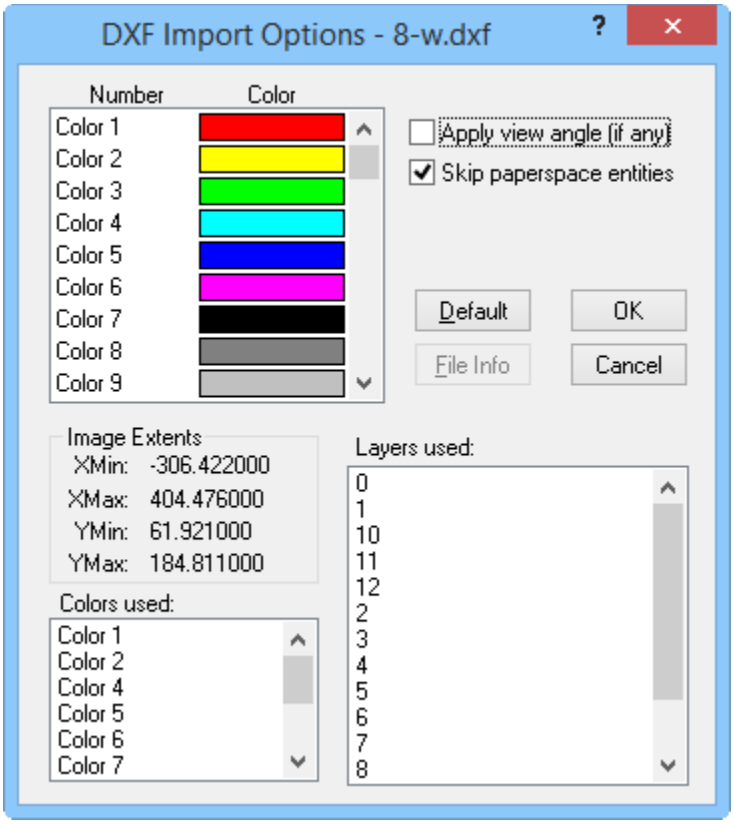
Export Contours

When a .DXF file is exported using the Export Contours command, the export options dialog is not displayed. Only spatial information is exported, and the contours are exported as POLYLINE entities defined by XY coordinates for each vertex and a Z level attribute.

AutoCAD DXF Import Options Dialog

The **Import Options** dialog allows you to specify options which determine how information in the file is imported.

AutoCAD-compatible Drawing Exchange Format .DXF files contain information describing graphical objects, such as areas, curves, points and text. The DXF Import Filter reads DXF files and structures the information in a form usable by the application.



*Customize import options in the **DXF Import Options** dialog.*

Color Number

DXF files contain no direct color information, but use color numbers (1-255) instead. There is an adhoc standard association of colors with the first 7 color numbers: Red, Yellow, Green, Cyan, Blue, Magenta and Black. By double-clicking on items in the COLOR list box, you can change the color associated with a specific color number.

Default

Pressing the *Default* button will assign a default set of colors to each color number.

Apply View Angle

If any viewing angles have been applied in AutoCAD, check *Apply view angle* to preserve these settings. The unrotated coordinates will not be preserved if this box is checked.

Skip Paperspace Entities

To import only graphical entities from AutoCAD's 'modelspace' and skip importing entities from 'paperspace', check the *Skip paperspace entities* option. If this option is not selected, entities from both 'paperspace' and 'modelspace' are imported. This option is checked, by default.

File Info

Click the *File Info* button to display the file information concerning the image extents, color numbers used and layers used in the .DXF file. The dialog extends to show this information, as shown above.

Colors Used

Selecting a color number displayed in the *Colors used* list box automatically selects that color number in the *Color* list box.

Layers Used

Double-clicking on a layer displayed in the *Layers used* list box displays the Layer Name dialog, showing the graphical entities present in the layer and a check box showing whether the layer is marked frozen (invisible) or not.

AutoCAD Entities

The point, line, and polygon AutoCAD entities are currently supported.

Import Messages

The following messages may appear while attempting to import [.DXF] files.

Incomplete Entity

Some vital groups are missing from an entity in the DXF file. Make sure the layer and other required groups are present.

Couldn't find a block to insert

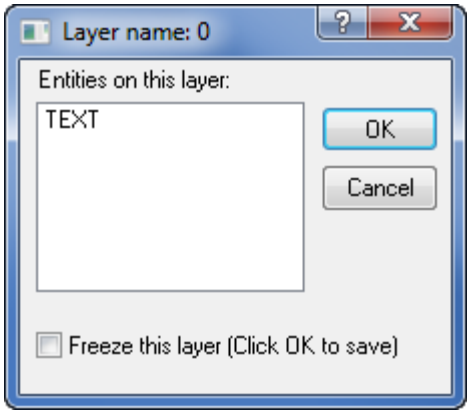
The program attempted to insert a block while the latter couldn't be found. Make sure the format of the BLOCKS section is correct, and the appropriate block is available.

Warning: 3-D extrusion not supported

The imported drawing contains one or more extruded objects which may not display properly after import. The DXF import filter software does not support the extrusion of two-dimensional objects to three dimensions within a DXF file. Click *Yes* to import the objects. Objects may appear incorrectly. Click *No* to import the .DXF file without any objects that contain an extrusions.

AutoCAD DXF Import Options - Layer Name Dialog

Double-clicking on a layer displayed in the *Layers Used* list box of the DXF Import Options dialog displays the **Layer Name** dialog, showing the graphical entities present in the layer and a check box showing whether the layer is marked frozen (invisible) or not.



*Specify the layer name for the DXF import in the **Layer Name** dialog..*

Entities on this Layer

Select a option from the *Entities on this layer* box. Left-click to select an option. The selected option will be highlighted. Only one entity can be selected at once.

Freeze this Layer

Check the *Freeze this layer (Click OK to save)* option to freeze/unfreeze layers prior to importing. Objects in layers that are marked frozen will not be imported into the application.

AutoCAD DXF Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,Color3=0;0;255"

This would first set all import options to their default values, then set the color associated with DXF Color Table entry #3 to Blue. All entities associated with Color Table entry #3 will be Blue. The DXF Color Table contains 255 entries (1-255).

Option	Action	Default	Description
ApplyViewAngle	0 = No 1 = Yes	0	Apply the view angle from the DXF file (if any) to the coordinates during import. Only applies if the DXF file being imported has a defined view angle.
SkipPaperspace	0 = No, import "paper space" entities 1 = Yes, skip "paper space" entities	0	Skip importing any "paper space" entities in the DXF drawing. "Model space" entities are always imported.
Color	N=R;G;B	See below	Any color in the DXF color table may be altered by using a ColorN

	for default color table	option where N is the color number of the color to be changed. The R, G, and B values specify the color channel saturations for each of the red, green, and blue channels respectively, and each must be between 0 and 255. For example: To set color number 2 to white, the option would be Color2=255;255;255. Multiple ColorN options may be given, one for each color.

The first seven colors in the default Color Table are:

Color #	R;G;B Values	Color
1	255;0;0	Red
2	255;255;0	Yellow
3	0;255;0	Green
4	0;255;255	Cyan
5	0;0;255	Blue
6	255;0;255	Magenta
7	0;0;0	Black

Import Messages

The following messages may appear while attempting to import [.DXF] files.

Incomplete Entity

Some vital groups are missing from an entity in the DXF file. Make sure the layer and other required groups are present.

Couldn't find a block to insert

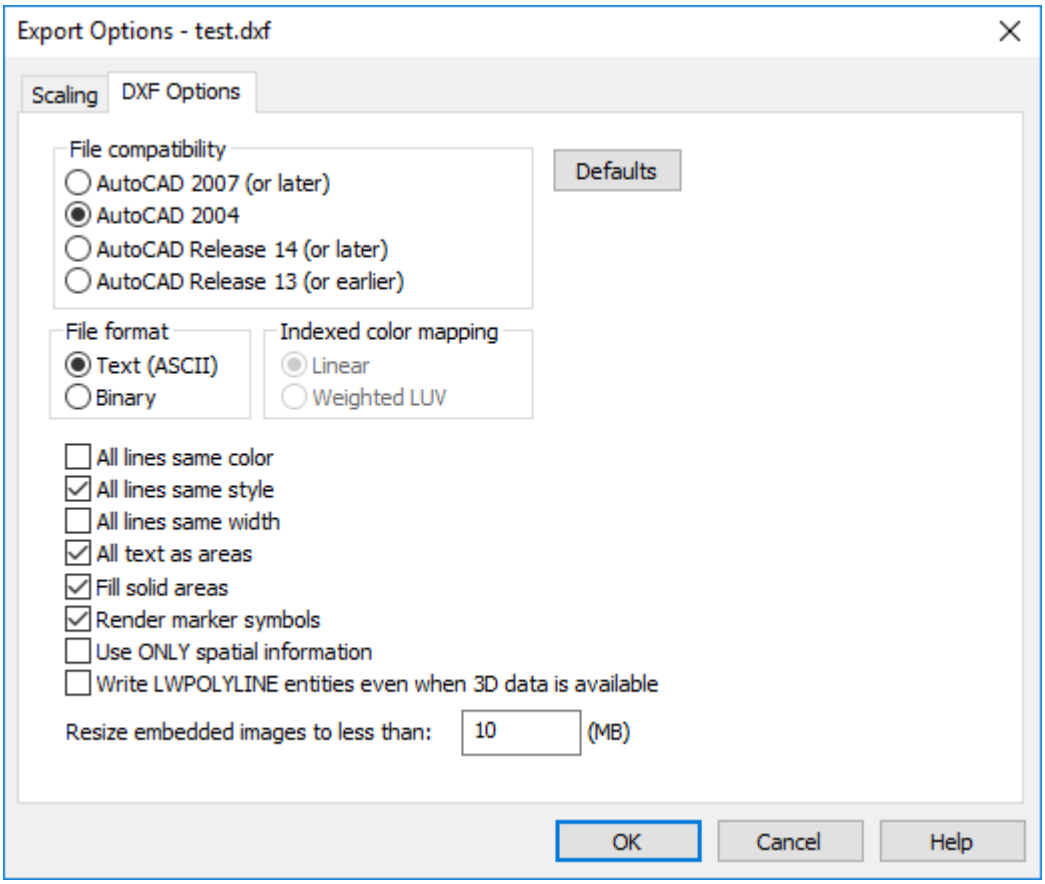
The program attempted to insert a block while the latter couldn't be found. Make sure the format of the BLOCKS section is correct, and the appropriate block is available.

Warning: 3-D extrusion not supported

The imported drawing contains one or more extruded objects which may not display properly after import. The DXF import filter software does not support the extrusion of two-dimensional objects to three dimensions within a DXF file. Click *Yes* to import the objects. Objects may appear incorrectly. Click *No* to import the .DXF file without any objects that contain an extrusions.

AutoCAD DXF Export Options Dialog

The **Export Options** dialog allows you to specify options which determine how information in the file is exported.



*Customize DXF export options in the **Export Options** dialog, **DXF Options** page.*

File Compatibility

Select the appropriate file compatibility. Available options are AutoCAD 2007 (or later), AutoCAD 2004, AutoCAD Release 14 (or later), and AutoCAD Release 13 (or earlier).

AutoCAD 2007 (or later)

Choose *File compatibility* of *AutoCAD 2007 (or later)* if the DXF file needs to be imported into AutoCAD 2007 or later. The 2007 version supports Unicode character encoding in the UTF-8 format.

AutoCAD 2004

Choose *File compatibility* of *AutoCAD 2004 (or later)* if the DXF file needs to be imported into AutoCAD 2004 or later. The 2004 version supports Unicode character encoding in the \U+XX format. The 2004 version (and all newer versions) also support RGB color support.

AutoCAD Release 14 (or later)

Choose *File compatibility* of *AutoCAD Release 14 (or later)* if the DXF file needs to be imported into AutoCAD Release 14 or later. The version 14 and previous versions support indexed color mapping.

AutoCAD Release 13 (or earlier)

Choose *File compatibility* of *AutoCAD Release 13 (or earlier)* if the DXF file needs to be imported into an earlier release of AutoCAD.

File Format

Choose *Text (ASCII)* or *Binary* to specify the format of the exported DXF file. See the AutoCAD DXF File Description for information on the organization of DXF files.

Indexed Color Mapping

Select the *Linear* or *Weighted LUV* option when either *AutoCAD Release 14 (or later)* or *AutoCAD Release 13 (or earlier)* is selected as the *File compatibility.* Note: Even though the older file formats can be imported into AutoCAD Release 14 and AutoCAD 2004 or later, the AutoCAD Release 14 or later file format uses several features that result in smaller, faster loading DXF files, and AutoCAD 2004 or later supports true color in addition to the indexed color.

General Export Options

The following export options pertain to the objects within the DXF file.

All Lines Same Color

Choose *All lines same color* if you don't want an AutoCAD color number (1-255) assigned to each of your lines. The default color for the layer will be used instead. All exported graphical entities are assigned to a layer named GSLAYER.

All Lines Same Style

Choose *All lines same style* if you want exported lines to be assigned the default style (for the GSLAYER) when imported into **AutoCAD**. Otherwise, exported lines will retain their style (solid, dashed, etc.).

All Lines Same Width

Choose *All lines same width* if you want exported lines to be assigned the default width (for the GSLAYER) when imported into **AutoCAD**. Otherwise, exported lines will be the width assigned in the application document.

All Text As Areas

Text can be exported as DXF solid polygons (*All text as areas* checked). These polygons will always be oriented properly. Whether or not these solid polygons (like all solid polygons) will be filled or not is controlled by the *Fill solid areas* option (see below).

Text can also be exported as **AutoCAD** text entities (*All text as areas* unchecked). No matter what typeface is specified in the application document, all text entities are assigned **AutoCAD's** STANDARD font. Once inside **AutoCAD**, the text entities can be edited in the normal **AutoCAD** fashion. As long as there is no shear, perspective, or clipping, DXF text entities will be exported as text. This means that the DXF text entities will be sized and oriented similar to the text objects in the application document. When shear, perspective, or clipping occur, the text is exported as solid polygons. Shear occurs when the character glyphs are not perpendicular to the text baseline. Perspective occurs when the height of glyphs in the text string are not all the same, as in a 3-D

view where the glyphs are smaller the farther they are from the observer. Clipping occurs when part of the text object is partially inside and partially outside the map limits.

Fill Solid Areas

Choose *Fill solid area*s if you want the interior of solid areas (polygons) to be filled. Otherwise, the areas will be exported as AutoCAD CLOSED POLYLINE entities.

Render Marker Symbols

Check the *Render marker symbols* check box if the marker symbols should be exported to the DXF file. The marker symbols are exported as lines and polygons. When the *Render marker symbols* option is not checked, marker symbols are exported to the DXF file as points.

Use ONLY Spatial Information

Choose *Use ONLY spatial information* if you want to export only spatial information and not object attributes or text labels. Spatial information is only concerned with the location of objects in space (i.e., their coordinates) and not with their attributes (such as line or fill style, marker symbol used, text labels, etc.) For example, if this option is chosen, all text will be ignored, markers will be exported as point entities instead of polygonal glyphs and coordinates output to the DXF file will be stored in map units instead of inches. This is useful when exporting base maps when only the spatial information is desired.

The AutoCAD program's behavior when importing DXF files (via the DXFIN command) is different depending on whether the **AutoCAD** drawing file [.DRW] is brand new or already contains drawing entities. If the file is brand new, attributes (such as line style) are loaded from the Tables section, so lines encountered in the Entities section will have the proper line style (solid, dash, dash-dot, etc.). However, if an old drawing file is already open, **AutoCAD** will ignore the Tables section and only read the Entities section. If the DXF file contains lines with styles not already defined, **AutoCAD** will issue an error message and abort the DXF import. It is recommended you choose the All lines same style option when exporting DXF files that will be imported into existing **AutoCAD** drawings. **AutoCAD** will then assign the default style to all lines in the imported layer (named GSLAYER).

Write LWPOLYLINE Entities

Check the *Write LWPOLYLINE entities even when 3D data is available* check box to export all lines as 2D LWPOLYLINE entities. Z values will be written as attributes when available. Clear the *Write LWPOLYLINE entities even when 3D data is available* check box to export 3D objects as 3D POLYLINE entities and 2D objects as LWPOLYLINE entities. Z values will be written as attributes when available. By default the *Write LWPOLYLINE entities even when 3D data is available* check box is not checked.

Resize Embedded Images to Less Than

The *Resize embedded images to less than* option specifies the maximum size (in megabytes) an embedded image is allowed to be. If an exported image exceeds this size, its resolution will be reduced so it doesn't exceed the designated maximum size. Increase this value to get better looking images at the expense of larger export files.

Defaults

The *Defaults* button sets all buttons and check boxes to default conditions.

The Scaling Page

See the Scaling Page for detailed information.

AutoCAD DXF Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,FormatASCII=0,AllColorsSame=1"

This would set all export options to their default values, then indicate the DXF file will be written in binary format and that all colors will be the mapped to the default AutoCAD color.

Option	Action	Default	Description
FileCompatibility	13 = AutoCAD Release 13 (or earlier) 14 = AutoCAD Release 14 (or later) 18 = AutoCAD 2004 21 = AutoCAD 2007 (or later)	14	Specifies which version of the DXF format is to be used for export.
FormatASCII	0 = Binary 1 = Text (ASCII)	1	File format of exported DXF file. The ASCII file format is larger than the DXF binary format, but is compatible with a wider variety of software programs.
MaxBitmapSizeInMB	N	10	For AutoCAD 2004 format, this option specifies the largest size allowed for an individual bitmap in the DXF file, in Megabytes. Any exported bitmap larger than this size, will have its resolution reduced so it does not exceed the maximum size.
AllColorsSame	0 = No 1 = Yes	0	Convert all colors to the default color.
AllStylesSame	0 = No 1 = Yes	0	Convert all line styles to the default line style.
AllWidthsSame	0 = No 1 = Yes	0	Convert all line widths to the default line width.
AllTextToAreas	0 = No 1 = Yes	0	Convert all text entities in the exported document to solid areas
FillSolidAreas	0 = No	0	Fill solid area interiors.

	1 = Yes		
UseSpatialInfo	0 = No 1 = Yes	0	Use ONLY spatial information. Only the basic geometry (lines, areas, text) will be written to the DXF file, and all line styles, colors, fills, and other attributes will be discarded.
ColorMapping	0 = Linear 1 = Weighted LUV	0	Choose the indexed color mapping as either weighted LUV export color mapping, or linear export color mapping.
RenderMarkers	0 = No 1 = Yes	1	Render marker symbols as lines and polygons. When set to 0, marker symbols are exported to the DXF file as points.
WriteLWPOLYLINE	0 = No 1 = Yes	0	Render both 2D and 3D objects as 2D LWPOLYLINE entities in the output DXF. When set to 0, 3D objects will be exported as 3D POLYLINE entities, and 2D objects will be exported as LWPOLYLINE entities.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.
FileURY	N.N		Set scaling rectangle upper right Y value.

Remarks

Scaling rectangle parameters have changed between **Surfer 8** and **Surfer 9 and newer versions**. For example:

```
Use FileURX instead of CGMUpperRightX.
Use PageLLY instead of APPLowerLeftY.
```

Surfer 8 and older rectangle-coordinate parameters are still supported, but in new scripts use of the current parameter names is recommended.

Boolean values within options strings are not equivalent to Booleans in **Scripter** BASIC. Use "1" instead of "True" and "0" instead of "False".

E00 Esri ArcInfo Export Format (E00) File Description

Esri ArcInfo Export Format .E00 files are ASCII files containing topological entities and their attributes. They provide a medium of exchange between Esri application programs on different hardware and operating system platforms (Windows, UNIX, etc.).

Loading Files

Esri ArcInfo Export Format .E00 files can be imported with the **File | Import** command or the **Home | New Map | Base** command.

Import Options

See Esri ArcInfo Export Format .E00 Import Options Dialog

Import Automation Options

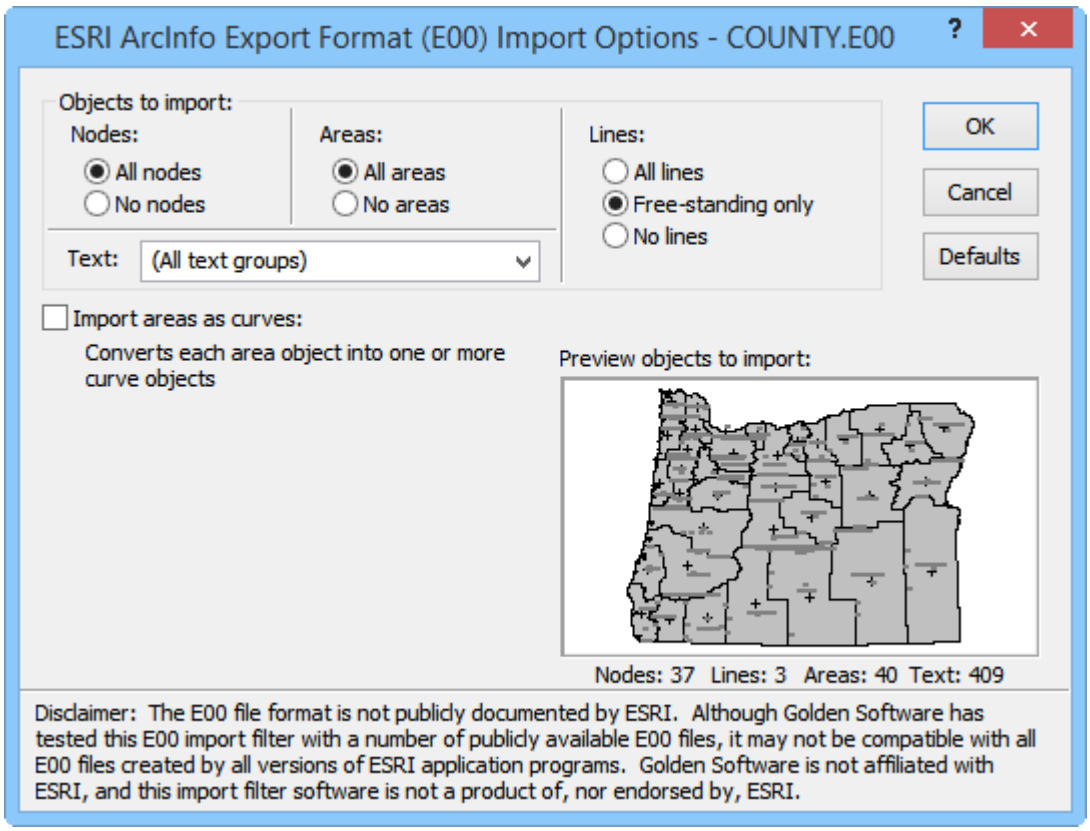
See Esri ArcInfo Export Format .E00 Import Automation Options

Disclaimer

The E00 file format is not publicly documented by Esri. Although Golden Software has tested this E00 import filter software with a number of publicly available E00 files, it may not be compatible with all E00 files created by all versions of Esri application programs. Golden Software is not affiliated with Esri, and this import filter software is not a product of, nor endorsed by, Esri.

Esri ArcInfo Export Format [.E00] Import Options Dialog

The **Import Options** dialog allows you to specify options which determine how information in the file is imported. Each of the controls in this dialog is described below.



*Select the E00 import options in the **Import Options** dialog.*

Nodes

These options determine which *Node* items are imported.

- If *All nodes* is selected, all nodes in the drawing are imported.
- If *No nodes* is selected, none of the nodes are imported.

Areas

These options determine which *Area* items are imported.

- If *All areas* is selected, all areas in the file are imported.
- If *No areas* is selected, none of the areas are imported.

Lines

These options determine which *Line* items are imported.

- If *All lines* is selected, all lines in the file are imported.
- If *Free-standing only* is selected, only those lines that are not associated with an area are imported.
- If *No lines* is selected, none of the lines are imported.

Text

The selection in this list determines which groups of text items are imported.

- If *All text groups* is selected, all text items from the import file are imported.
- If *No Text* is selected, no text items from the import file are imported.

- If the name of a specific group is selected in the list, only the text items from that group are imported. Some import files don't contain any named groups of text items, in which case the only selections possible will be *All Text Groups* or *No Text*.

Import Areas as Curves

Check the *Import areas as curves* box to convert each area object into one or more curve (line) objects.

Preview Objects To Import Display

This area of the dialog displays a rough preview of the items that are selected for import from the import file. Any lines selected for import are displayed in black. Any areas selected for import are displayed in light gray with a black border. Any nodes selected for import are displayed as black crosses. Any text items selected for import are displayed as dark gray rectangles. Any changes to the dialog controls that effect which objects are selected for import will be reflected in the preview display.

Esri ArcInfo Export Format [.E00] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,AreasToCurves=1"

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to closed curves.

Option	Action	Default	Description
AreasToCurves	0 = No 1 = Yes	0	Convert all polygons to polylines during import.
AllLines	0 = No 1 = Yes	0	Consider all lines.
WhichLines	0 = Import all lines 1 = Import only freestanding lines 2 = Import no lines	0	Specifies which lines from the file are to be imported
WhichNodes	0 = Import all nodes 1 = Import no nodes	0	Specifies whether nodes (points) from the file are to be imported.
WhichAreas	0 = Import all areas 1 = Import no areas	0	Specifies whether ares from the file are to be imported.
WhichText	-2 = Import all text -1 = Import no text N = any other value indicates which specific	-2	Specifies which text group from the file is to be imported. For example, use WhichText=3 to import only the text from group 3.

	text group number to import

E00 DEM Grid File Description

Esri Format .E00 grid files are ASCII files containing elevation data. They provide a medium of exchange between Esri application programs on different hardware and operating system platforms (Windows, UNIX, etc.).

Loading Files

Esri Format .E00 grid files can be loaded in the Open Grid dialog.

Import Options

No import options dialog is displayed.

Import Automation Options

No import options are available.

Disclaimer

The E00 file format is not publicly documented by Esri. Although Golden Software has tested this E00 import filter software with a number of publicly available E00 grid files, it may not be compatible with all E00 files created by all versions of Esri application programs. Golden Software is not affiliated with Esri, and this import filter software is not a product of, nor endorsed by, Esri.

ECW ER Mapper [.ECW] File Description

Surfer imports ERMapper .ECW files.

Enhanced Compression Wavelet .ECW is an open standard wavelet compression image format developed by Earth Resource Mapping. The file format is optimized for aerial and satellite imagery, and efficiently compresses very large images with fine, alternating contrast. This is a lossy compression format. A region can be specified for import in the .ECW Image Import Options dialog, or the image can be imported as a read-only image. Each of these options decreases the import time for large ECW files.

The .ECW file format has the following properties:

- Embeds map projection information
- Fast compression (about 1.5 MB of compressed file per second on 1 GHz processor)
- Typical compression ratios between 1:10 and 1:100
- Possible decompression of selected regions without the need to decompress the whole file
- Data flow compression allows for compression of big files with small RAM requirements

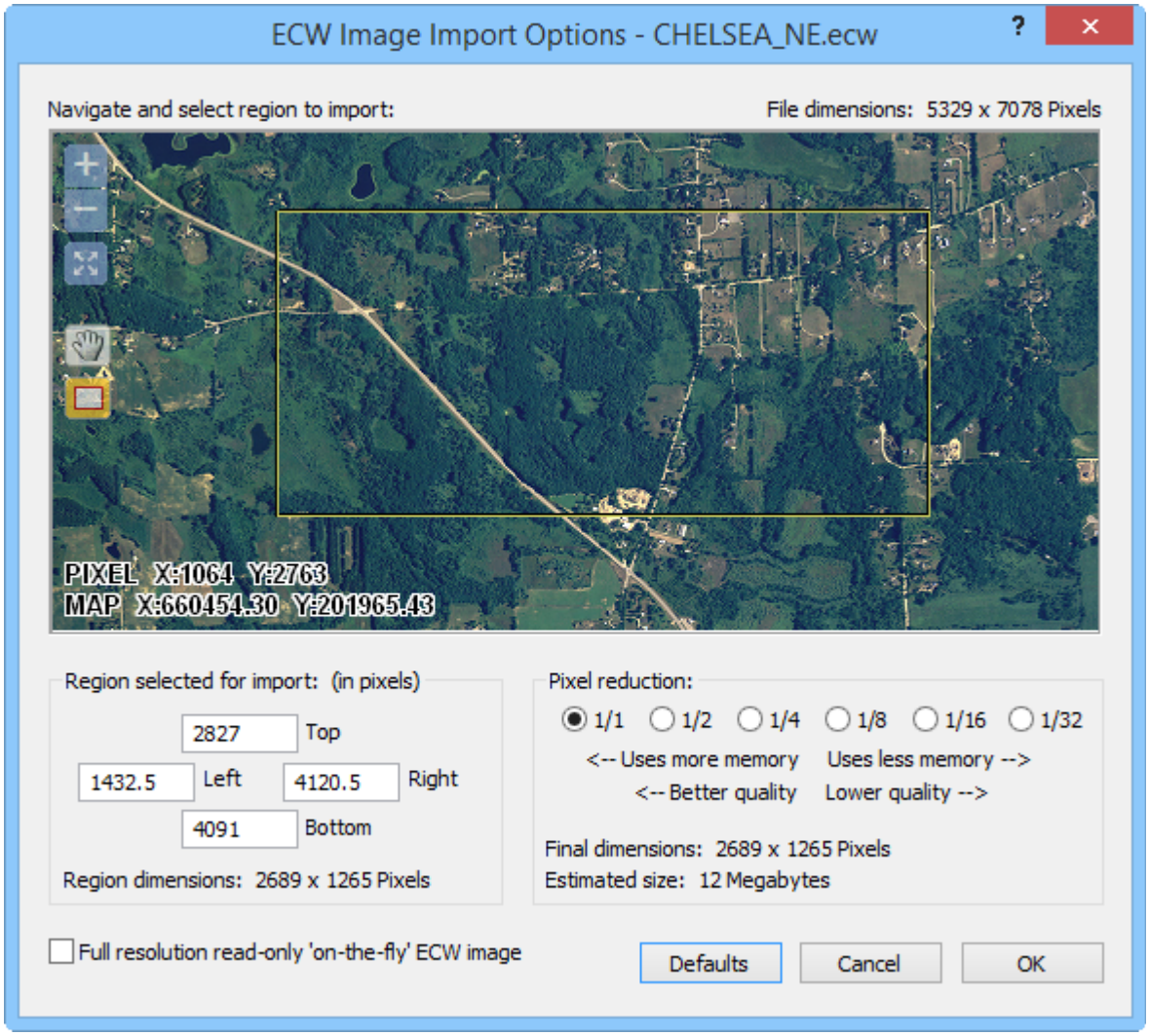
Import Options

See .ECW Image Import Options Dialog.

Import Automation Options

See ER Mapper .ECW Import Automation Options.

ER Mapper .ECW Import Options Dialog



*Customize the ECW pixel reduction, import region, or read-only import in the **ECW Image Import Options** dialog.*

Image Preview

The preview section displays a picture of the area to be imported. The preview section contains a low resolution preview of the area.

- 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 import. The area to import is indicated in the preview section by a yellow outline.
- Click the button to fit the entire image in the preview window.
- Click the button on the left side of the image preview to quickly move the image in the preview section. Click and hold the left mouse button down and drag the image to change the view. Changing the view does not affect the area to import. The area to import 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 imported. The initial region includes the entire image extents. Click and hold the left mouse button down and drag the mouse over the area to import. The zoom extents update and the yellow box coincides with the area drawn. The *Region selected for import* section also updates. Only the portion of the image highlighted by the yellow box will be imported.

Import Region

The region for import can be selected in the image preview or in the *Region selected for import* section. The initial region includes the entire image extents. Type a value in pixels into the *Top*, *Left*, *Right*, and *Bottom* fields to specify the imported region. The yellow outline in the image preview is updated as the values are changed.

Pixel Reduction

Since some ECW images can be extremely large once expanded into memory, the import filter allows the image to be reduced in dimensions by 1/2 to 1/32 of the original size via the options in the **ECW Image Import Options** dialog. Choose from *1/1* (Uses the most memory, best quality), *1/2, 1/4, 1/8, 1/16,* to *1/32* (Uses the least memory, lowest quality). The *Final dimensions* field displays the final dimensions of the imported region. The *Estimated size* field displays the estimated image size in Megabytes (MB).

Read-only Import

The *Full resolution read-only 'on-the-fly' ECW image* option imports the full ECW file in its native form as a highly-compressed, read-only image. If a portion of the image is needed for display or output, the necessary portion is extracted and decompressed "on the fly." This option provides quick import and interactive panning and zooming without using large amounts of RAM or disk space. However once an image is imported with *Full resolution read-only 'on-the-fly' ECW image* checked, the image cannot be modified.

ER Mapper [.ECW] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Scale=16"

This would reduce the scale of the map to 1/16th .

Option	Action	Default	Description
Scale	1 = 1/1 (uses most memory, best quality) 2 = 1/2 4 = 1/4 8 = 1/8 16 = 1/16 32 = 1/32 (uses least memory, lowest quality)	1	Since some images can be extremely large once expanded into memory, the import filter allows the image to be reduced in dimensions by 1/2 to 1/32 of the original size.
LRX	integer	maximum X	Use LRX to limit the import to a subregion of the file. LRX is the X pixel coordinate for the lower right corner of the desired subregion.
LRY	integer	maximum Y	Use LRY to limit the import to a subregion of the file. LRY is the Y pixel coordinate for the lower right corner of the desired subregion.
ULX	integer	0	Use ULX to limit the import to a subregion of the file. ULX is the X pixel coordinate for the upper left corner of the desired subregion.
ULY	integer	0	Use ULY to limit the import to a subregion of the file. ULY is the Y pixel coordinate for the upper left corner of the desired subregion.

Remarks

The pixel coordinates increase in the X direction from left to right and increase in the Y direction from top to bottom. The pixel coordinate origin is the upper left corner. For example an image with a 2000 pixel width and 3000 pixel height has the following coordinates: the upper left corner is coordinate (0,0) and the lower right corner is the x and y maximums (2000,3000).

However, the image still will import correctly even if the ULY is used for the maximum Y and LRY is used for the minimum Y.

EMF Windows Enhanced Metafile File Description

Windows can store graphical objects (i.e. pictures) in a special form called an Windows Enhanced Metafile .EMF file. Such pictures can be stored on disk or in the Windows Clipboard. When these pictures are imported (or pasted from the Clipboard), the objects can be optionally separated and stored individually.

Windows Metafiles are intended to be portable between applications and may contain both vector and image components. In contrast to raster formats such as .JPEG and .GIF which are used to store image (bitmap) graphics such as photographs, scans and graphics, Windows Metafiles generally are used to store line-art, illustrations and content created in drawing or presentation applications. Most Windows clipart is in the .EMF or .WMF format.

Windows Metafile .WMF is a 16-bit format introduced in Windows 3.0. It is the native vector format for Microsoft Office applications such as Word, PowerPoint, and Publisher. A newer 32-bit version with additional commands is called Enhanced Metafile .EMF. .EMF is also used as a graphics language for printer drivers.

Import Options

See Windows Enhanced Metafile Import Options Dialog.

Import Automation Options

See Windows Enhanced Metafile Import Automation Options.

Export Options Dialog

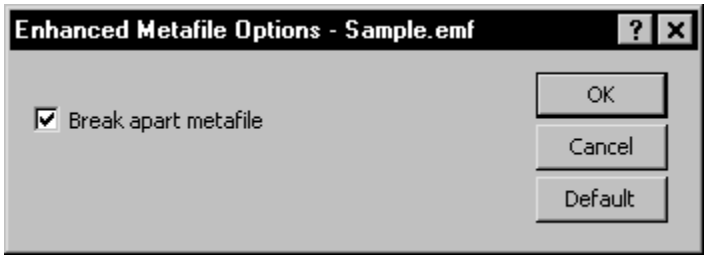
See Windows Enhanced Metafile Export Options Dialog.

Export Automation Options

See Windows Enhanced Metafile Export Automation Options.

Windows Enhanced Metafile .EMF Import Options Dialog

The **Enhanced Metafile Options** dialog allows you to specify options which determine how information in the file is imported. A Windows Enhanced Metafile .EMF is a collection of objects combined together to produce an image.



*Specify the EMF import options in the **Enhanced Metafile Options** dialog.*

Break Apart Metafile

Check the *Break apart metafile* option to break the metafile into its constituent graphical objects during import. Uncheck the option to leave the metafile as a single group object.

Windows Enhanced Metafile .EMF Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

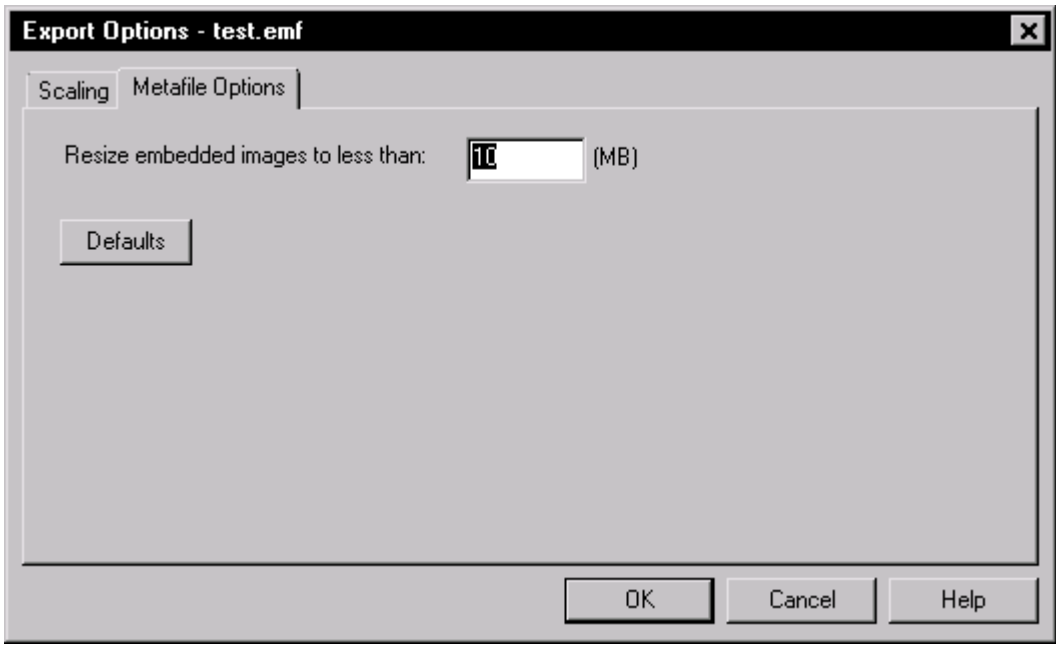
```
"Defaults=1,BreakApart=0"
```

This would first set all import options to their default values, then set the BreakApart option value to zero, which specifies the metafile contents are to remain together as a single unit.

Option	Action	Default	Description
BreakApart	0 = No 1 = Yes	1	Break metafile apart into individual graphical objects during import.

Windows Enhanced Metafile .EMF Export Options Dialog

The **Export Options** dialog allows you to specify options which determine how information in the file is imported. A Windows Enhanced Metafile [.EMF] is a collection of objects combined together to produce an image.



*Specify the EMF file export options in the **Export Options** dialog.*

Resize Embedded Images

The *Resize embedded images to less than* option specifies the maximum size (in megabytes) an embedded image is allowed to be. If an exported image exceeds this size, its resolution will be reduced so it doesn't exceed the designated maximum size. Increase this value to get better looking images at the expense of larger export files.

Defaults

The *Defaults* button sets all options to default conditions.

Scaling Page

See the Scaling Page for detailed information.

Windows Enhanced Metafile [.EMF] Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,AllTextToPolygons=0"

This would set all export options to their default values, then indicate that all text is not to be exported as polygons (i.e., it is to remain as text).

Option	Action	Default	Description
MaxBitmapSizeInMB	N	10	This option specifies the largest size allowed for an individual bitmap in the EMF file, in Megabytes. Any exported bitmap larger than this size will be downscaled to this size during export.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.
FileURY	N.N		Set scaling rectangle upper right Y value.

Remarks

Boolean values within options strings are not equivalent to Booleans in **Scripter** BASIC. Use "1" instead of "True" and "0" instead of "False".

EPS Encapsulated PostScript File Description

Surfer exports Encapsulated PostScript files .EPS files.

EPS File Format

At a minimum, an EPS file contains a *BoundingBox DSC comment*, describing the rectangle containing the image described by the EPS file.

Identifying EPS files

Because of the different ways in which EPS previews are handled, there is no one way to identify an EPS file.

- A Windows-format EPS file containing a TIFF or WMF preview must start with the four bytes containing, in hexadecimal, C5 D0 D3 C6. Bear in mind these files are widespread on all platforms.
- In all other cases an EPS file must start with a line %!PS-Adobe-a.b EPSF-c.d where a, b, c and d are all single digit numbers.
- A Mac-format EPS file is accompanied by a resource fork. The preview is a PICT resource with ID 256. An EPS file on the Mac is expected to have a file type code of "EPSF", whether or not it has a preview.
- An EPSI file will contain a line starting %%BeginPreview: in the DSC prolog.
- In many cases no preview is present at all.

Export Options Dialog

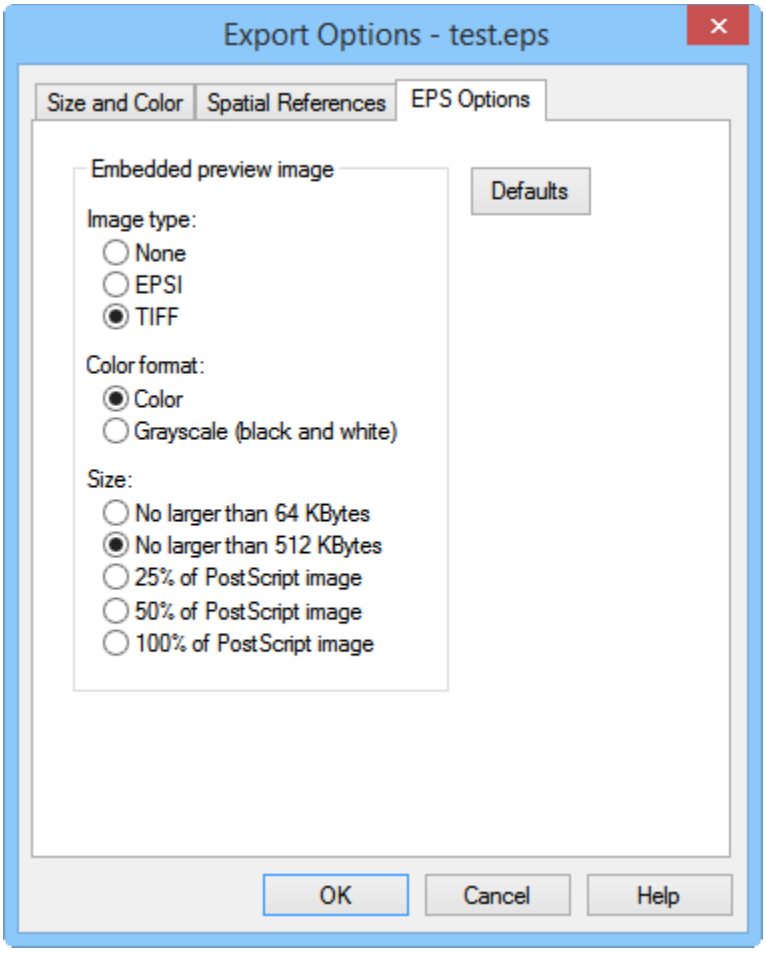
See Encapsulated PostScript .EPS Export Options Dialog, Size and Color, and Spatial References

Export Automation Options

See Encapsulated PostScript Automation Export Options

Encapsulated PostScript [.EPS] Export Options Dialog

The **Export Options** dialog allows you to specify options which determine how information in the file is exported.



*Specify the EPS options in the **EPS Options** page of the **Export Options** dialog.*

Image Type

The *Image type* controls determine how the EPS preview image is encoded in the export file. Possible selections are *EPSI*, *TIFF*, or *None*. Choosing *EPSI* or *TIFF* inserts an image into the beginning of the EPS file. Selecting *None* inserts no preview image. Some programs need the preview image in the file so that the file imports correctly.

Color Format

The *Color format* controls determine whether the preview image is exported in *Color* or *Grayscale (black and white)*. Setting the *Color Format* to *Grayscale (black and white)* does not affect the colors in the EPS file. On the image preview colors are changed.

Size

The Size controls determine what size of EPS preview image is encoded in the export file. The selections are No larger than 64KBytes, No larger than 512Kbytes, 25% of PostScript Image, 50% of PostScript Image, or 100% of PostScript Image. Selecting a smaller size will give less precision on the image preview, but will not affect the EPS file.

Default

Click the *Default* button to return the EPS options to the default settings.

Encapsulated PostScript .EPS Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,ScalingSourceApp=0"

This would set all export options to their default values, then indicate the scaling source information should not be taken from the application, but from previously saved values.

Option	Action	Default	Description
Width	N		Sets the width of the exported image in pixels. Either a Width or an HDPI option may be specified, but not both.
Height	N		Sets the height of the exported image in pixels. Either a Height or a VDPI option may be specified, but not both.
HDPI	N	96	Sets the number of horizontal pixels in the exported image that corresponds to a distance one horizontal inch on the Surfer plot. Either a Width or an HDPI option may be specified, but not both. The default HDPI is the resolution of the display device driver on your computer (this is 96 dots per inch on many Windows display devices, but number can very from device to device).
VDPI	N	96	Sets the number of vertical pixels in the exported image that corresponds to a distance of one vertical inch on the Surfer plot. Either a Height or a VDPI option may be specified, but not both. The default VDPI is the resolution of the display device driver on your computer (this is 96 dots per inch on many Windows display devices, but number

			can very from device to device).
KeepAspect	0 = No 1 = Yes	1	If set to 1, KeepAspect ajusts the aspect ratio (the width versus height) of the exported image to match the aspect ratio of the Surfer plot that is being exported. If an option is given to set the Width or HDPI of an image, KeepAspect automatically assigns the appropriate Height or VDPI to maintain the same aspect ratio as the Surfer plot. Likewise, if an option is given to set the Height or VDPI of an image, KeepAspect automatically assigns the appropriate Width or HDPI to maintain the same aspect ratio of the Surfer plot. Note that the KeepAspect option is ignored if both the Width and Height options or the HDPI and VDPI options are specified. If set to 0, the aspect ratio of the exported image is determined solely by the Width and Height or HDPI and VDPI options. The size is not adjusted to match the aspect ratio of the Surfer plot.
IgnoreRefInfo	0 = Save spatial references 1 = Ignore spatial references	1	Ignore any spatial reference information.
SaveRefInfoAsInternal	0 = No 1 = Yes	0	Save spatial references to internal format (if possible).
SaveRefInfoAsBlueMarbleRSF	0 = No 1 = Yes	0	Save spatial references as Blue Marble .RSF file.
SaveRefInfoAsESRIWorld	0 = No 1 = Yes	0	Save spatial references as Esri World file.
SaveRefInfoAsGSIREF	0 = No 1 = Yes	0	Save spatial references as Golden Software Reference (version 1) file.
SaveRefInfoAsGSIREF2	0 = No 1 = Yes	0	Save spatial references as Golden Software Reference (version 2) file.
PreviewType	0 = None 1 = EPSI	2	Specifies whether an embedded preview image is to be included in the EPS file,

	2 = TIFF		and in what format the embedded preview image is to be stored.
PreviewColor	0 = Grayscale 1 = Color	1	Choose whether the preview image is exported in Color or Grayscale.
PreviewSize	0 = < 64 KB 1 = < 512 KB 2 = 25% of PostScript Image 3 = 50% of PostScript Image 4 = 100% of PostScript Image	1	Choose what size of EPS preview image is encoded in the export file.

Creating .EPS Files

Encapsulated PostScript .EPS files can be created in **Surfer** by exporting or by printing to a file. The print driver must be a PostScript driver.

To create an .EPS file using the File | Export command. Use these steps to export an .EPS file:

1. In Surfer, click **File | Export**.
2. Type a *File name*, set the *Save as type* to EPS Encapsulated PostScript (*.eps), and click *Save*.
3. Set any options in the **Export Options** dialog and click *OK*.

The EPS file produced in this manner does contain the image preview.

You can also create an EPS file using **File | Print**. However, the EPS file produced in this manner does not contain an image preview. **Surfer** and other programs need an image preview to import EPS files.

1. Click the Windows **Start** button and open the **Printer** settings. This is different for each Windows operating system.
2. Click the **Add Printer** button. Follow the directions and add any PostScript printer (usually PS appears in the printer driver's title).
3. Select FILE for the printer port.
4. Open the printer properties by right-clicking on the printer and selecting **Properties**. PostScript printer drivers usually contain a **PostScript** page. Under *PostScript output format* on the **PostScript** page choose *Encapsulated PostScript (EPS)*. There may not be a **PostScript** page for some printer drivers.
5. To create an EPS file from **Surfer** choose File | Print and select the PostScript printer from the *Name* list.
6. The **Print to File** dialog is displayed. Type in the path and file name to use for the EPS file. Make sure to use the EPS extension on the file. Click *OK* and the map is written to the EPS file.

ERS ER Mapper Grid [.ERS] File Description

Surfer can import and export ER Mapper grid format .ERS files.

The ER Mapper grid format consists of two files: a binary data file (no file suffix) and an ASCII header file .ERS.

The ER mapper grid file does not have a file extension. It should have the same format as a .BIL file, and it must have an associated ASCII header file with the same file name but with .ERS file extension. This header file contains data about the image such as the number of rows and columns in the image and latitude and longitude.

These files can be read directly by the ER Mapper software or by other packages such as ERDAS IMAGINE

Loading .ERS Files

ER Mapper .ERS files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .ERS files. Alternatively, the ER Mapper .ERS file format is available when prompted to save a grid file.

Export Options Dialog

No export options dialog is displayed.

Export Automation Options

No export options are available.

FLD AVS Field [.FLD] File Descriptions

The AVS Field data set file format originated with the AVS5 and AVS/Express applications. The data set consists of a file with the file name extension .FLD, which describes the data set; plus possibly one or more additional data files referenced if the data are not contained within the .FLD file itself.

Surfer imports and exports several different types of data from AVS Field data sets.

Loading .FLD Files

AVS Field .FLD files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed. During import, any external data files referenced from within the .FLD file are assumed to reside in the same directory where the .FLD file resides.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .FLD files. Alternatively, the AVS Field .FLD file format is available when prompted to save a grid file.

Export Options

See AVS .FLD Export Options Dialog.

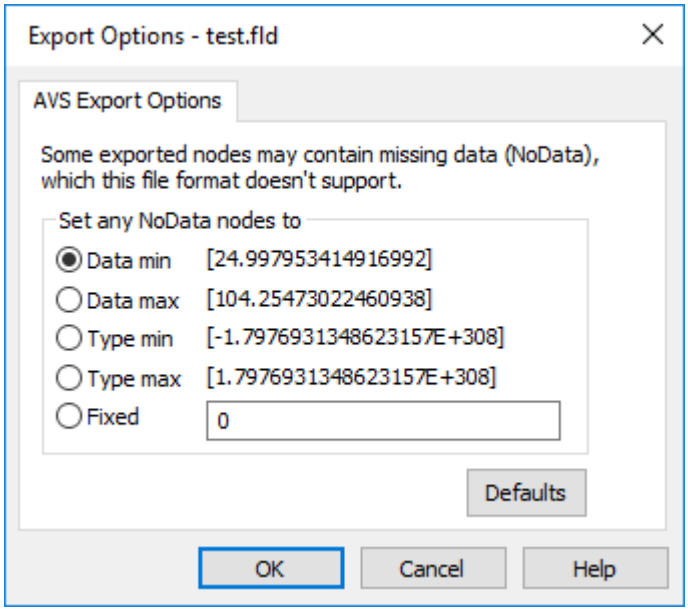
Exported AVS data sets always place the data values inside the .FLD file, not in external data files.

Export Automation Options

See AVS .FLD Export Automation Options

AVS Field .FLD Export Options Dialog

The **Export Options** dialog is only displayed if the lattice contains one or more nodes with the NoData value. The dialog controls determine how NoData values are treated in the export file.



*Control how the NoData nodes are exported in the **Export Options** dialog.*

Data Min

Choose *Data min* to export NoData nodes with the indicated minimum value (varies depending on contents of lattice).

Data Max

Choose *Data max* to export NoData nodes with the indicated maximum value (varies depending on contents of lattice).

Type Min

Choose *Type min* to export NoData nodes with the indicated minimum value for the lattice's data type (usually zero).

Type Max

Choose *Type max* to export NoData nodes with the indicated maximum value of the lattice's data type.

Fixed

Choose *Fixed* to export NoData nodes with some other value, and enter the desired value in the adjacent edit control.

AVS Field .FLD Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" BlankMethod=BM_FIXED, FixedBlankValue=1.70141E+038"

This would indicate that the NoData value for a lattice or grid node is a custom value, and that value is equal to **Surfer's** NoData value.

Option	Action	Default
BlankMethod	Method for assigning values to NoData nodes: BM_DATA_MIN = Minimum value of the lattice BM_DATA_MAX = Maximum value of the lattice. BM_TYPE_MIN = Minimum value of the inherent data type. BM_TYPE_MAX = Maximum value of the inherent data type. BM_FIXED = A user specified value	BM_TYPE_MAX
FixedBlankValue	Used if BlankMethod is BM_FIXED	No

FLT, HDR Esri Binary Float Grid [.FLT, .HDR] File Description

Surfer can import and export Esri floating point grid files in Esri Spatial Analyst .FLT, .HDR format.

The Esri Binary Float Grid format consists of 2 files: a binary .FLT image file and ASCII .HDR header file with the same file name but different file extension. For example, Test.FLT and Test.HDR.

Loading .FLT Files

Esri Float Grid .FLT files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .FLT files. Alternatively, the Esri .FLT file format is available when prompted to save a grid file.

Export Options Dialog

When the input grid is georeferenced, see Spatial References. By default, an Esri .PRJ file is created automatically when exporting this Esri file type.

Export Automation Options

N/A

GEOJSON GeoJSON Data Interchange Format File Description

Surfer imports GeoJSON Data Interchange Format .GEOJSON files. GeoJSON is based on the JavaScript Object Notation and encodes geographic structures. GeoJSON files may include points, polylines, and polygons. GeoJSON uses the World Geodetic System 1984 coordinate system.

For detailed file structure information, see RFC 7946 - The GeoJSON Specification

Loading .GeoJSON Files

GeoJSON files can be loaded to create a base map (Home | New Map | Base) with the **Import** dialog.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

References: *The GeoJSON Specification*. Butler, et al. Published by GeoJSON WG. 2016. www.geojson.org

GEOREF Golden Software Georeference File Description

The Golden Software Georeference GEOREF file stores control point information for an image. The GEOREF file is a text file that defines the warp method and control point source and target coordinates.

Format

The GEOREF file includes a header that defines the file version, warp method, and columns:

```
Surfer Georeference Control Point File
Version=1
Method=<WarpMethod>
Name,Raster X,Raster Y,Longitude,Latitude,Active
```

The WarpMethod value indicates the warp method:

Warp Method	Value
Affine Polynomial	3
First Order Polynomial	4
Second Order Polynomial	5
Third Order Polynomial	6
Thin Plate Spline	8
Natural Cubic Spline	9
Marcov Spline	10
Exponential Spline	11
Rational Quadratic Spline	12
Inverse Distance Squared	13

Following the header, each line in the GEOREF file contains a control point with comma-separated values for the control point Name, Raster X, Raster Y, Longitude, Latitude, and Active state.

GIF Graphics Interchange Format

The Graphics Interchange Format .GIF is an image format that was introduced by CompuServe in 1987 and has since come into widespread usage on the World Wide Web due to its wide support and portability.

File Format

The format supports up to 8 bits per pixel, allowing a single image to reference a palette of up to 256 distinct colors chosen from the 24-bit RGB color space. It also supports animations and allows a separate palette of 256 colors for each frame. The color limitation makes the GIF format unsuitable for reproducing color photographs and other images with continuous color, but it is well-suited for simpler images such as graphics or logos with solid areas of color.

GIF images are compressed using the Lempel-Ziv-Welch (LZW) lossless data compression technique to reduce the file size without degrading the visual quality. This compression technique was patented in 1985. Controversy over the licensing agreement between the patent holder, Unisys, and CompuServe in 1994 inspired the development of the Portable Network Graphics (PNG) standard; since then all the relevant patents have expired.

Usage

- Sharp-edged line art (such as logos) with a limited number of colors. This takes advantage of the format's lossless compression, which favors flat areas of uniform color with well defined edges (in contrast to JPEG, which favors smooth gradients and softer images).
- Used to store low-color sprite data for games.
- Used for small animations and low-resolution film clips.
- In view of the general limitation on the GIF image palette to 256 colors, it is not usually used as a format for digital photography. Digital photographers use image file formats capable of reproducing a greater range of colors, such as TIFF, RAW or the lossy JPEG, which is more suitable for compressing photographs.
- The PNG format is a popular alternative to GIF images since it uses better compression techniques and does not have a limit of 256 colors, but PNGs do not support animations.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

See Size and Color, Spatial References, and GIF Export Options

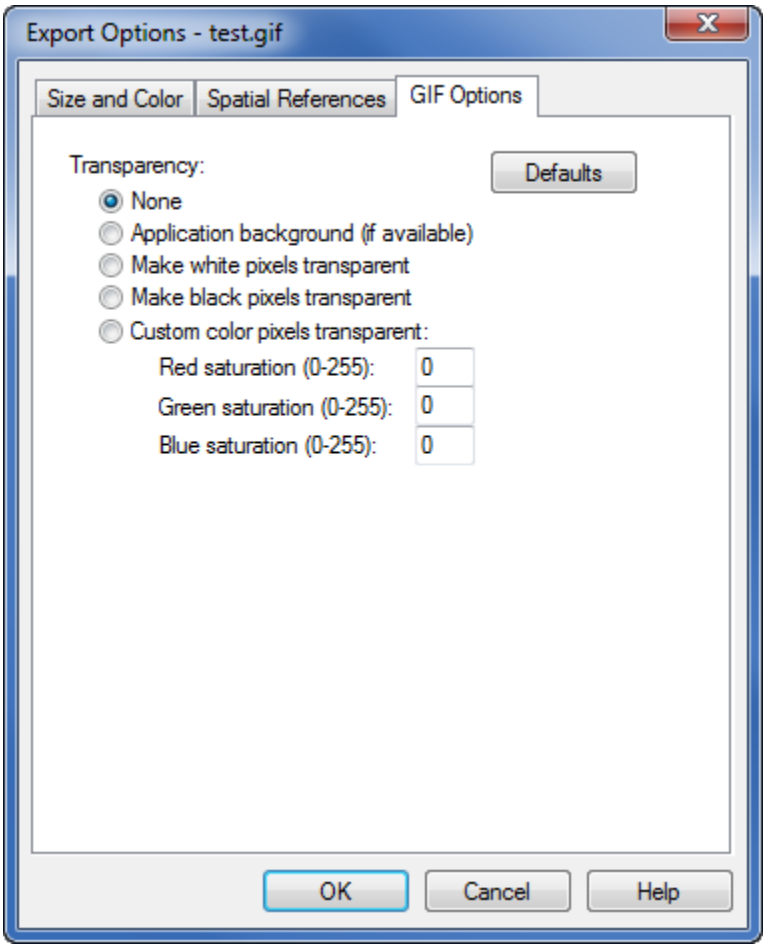
Export Automation Options

See Image (Bitmap) Export Automation Options

.GIF Export Options Dialog

The Graphics Interchange Format .GIF is an image format that was introduced by CompuServe in 1987 and has since come into widespread usage on the World Wide Web due to its wide support and portability. GIF files support transparency.

Use the **Export Options** dialog to specify the Size and Color, Spatial References options and the transparency options for the .GIF file. To export the .GIF file with transparency, click on the **GIF Options** tab in the **Export Options** dialog.



*Specify the transparency setting on the **GIF Options** page in the **Export Options** dialog.*

Select *None* to export the image with no transparency.

Select *Application background (if available)* option to export any background areas as transparent. The .GIF file will have transparent background areas. All drawn parts in the plot window are exported solid.

Select *Make white pixels transparent* to make all white areas of the image transparent. Select *Make black pixels transparent* to make all black areas of the image transparent. Select *Custom color pixels transparent* to select a specific color by *Red saturation (0-255), Green saturation (0-255),*

and *Blue saturation (0-255)* in the image to be transparent. When any of these options are selected, pixels with this color in the foreground of the image can be transparent.

Resolution

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.

GML Geography Markup Language File Description

Surfer imports GML files with the File | Import command, or GML files can be used to create base maps with the Home | New Map | Base command. GML files are also server by WFS servers when using the Download Online Maps dialog.

File Information

GML files are XML format files defined by the Open Geospatial Consortium. The standards, schemas, and documentation for the GML file format are available from the Open Geospatial Consortium here.

Import Options Dialog

No **Import Options** dialog is displayed.

Import Automation Options

There are no import automation options for this format.

GPX GPS Exchange Format File Description

GPS exchange files, .GPX files, are created from GPS devices.

File Structure

GPX files are XML format text files. The .GPX files can contain way points, routes, tracks, and various attribute information. GPX files use the World Geodetic System 1984 coordinate system.

- Way points describe an unrelated group of points. For instance, this can be a collection of place names. Each way point is imported and displayed as the default symbol.
- Routes are a series of connected way points. Typically a route is a plan on where to go. Routes are imported and displayed as polylines. One point routes are imported and displayed as the default symbol.
- Tracks are a series of connected way points. A track is where the device has actually been. Tracks are imported and displayed as polylines. One point tracks are imported and displayed as the default symbol.

Way point, route, and track names are imported for each object. These are listed as attributes for each object on the Info tab in the Properties window.

Loading a GPX

Use the File | Import command to import the GPX file as drawn objects in the plot window. The way points, routes, and tracks are imported into a single group when the **Import** command is

used. Use the Home | New Map | Base or **Home | Add to Map | Layer | Base** command to add the GPX information as a base map.

Import Options

No import options dialog is displayed.

Import Automation Options

No import automation options exist.

GRD Geosoft Grid File Description

Surfer can import and export Geosoft .GRD grid version 2 format in compressed and uncompressed format and export .GGF format. Sometimes there is an associated .GI projection file that comes with the binary .GRD file.

File Description

Geosoft grid is a binary format for storing raster data typically used for geophysical and elevation data. Two files describe each grid (.GRD and .GI). Technical information on this file format is reportedly available in the manual for Geosoft's Montaj software application.

Loading .GRD Files

Geosoft .GRD files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .GRD or .GGF files. Alternatively, the Geosoft .GRD or .GGF file format is available when prompted to save a grid file.

Export Options Dialog

When the input grid is georeferenced, see Spatial References

Export Automation Options

No export options are available.

GRD Golden Software Grid file format

Surfer imports and exports two-dimensional uniform lattices (elevation grids) in the **Surfer** Grid file format. Both ASCII and binary variants are supported.

Files with the .GRD extension are **Surfer** grid files. To preserve faulting information and to use double precision values in **Surfer**, be sure to save the grid in the Surfer 7 format. If you need to use a grid in **Surfer 5** or **Surfer 6**, save the grid as Surfer 6 Text .GRD or Surfer 6 Binary .GRD. Keep in mind that these two formats do not retain fault information or double precision values.

Surfer 7 Grid File Format

Surfer 7 grid files .GRD use a tag-based binary format to allow for future enhancements. Each section is preceded by a tag structure, which indicates the type and size of the following data. If a program does not understand or want a particular type of data, it can read the associated tag and quickly skip to the next section. In general, sections can appear in any order except for the first, which must be a Header section.

Data types used in **Surfer 7** grid files:

Type	Description
long	32 bit signed integer
double	64 bit double precision floating point value

Each section is preceded by a tag structure with the following format:

Element	Type	Description
Id	long	The type of data in the following section. See the next table for a list of valid values.
Size	long	The number of bytes in the section (not including this tag). Skipping this many bytes after reading the tag will align the file pointer on the next tag.

Tag ID Values

Tag Id values. The 0x prefix indicates a hexadecimal value:

Id	Description
0x42525344	*Header* section – must be the first section within the file.
0x44495247	*Grid* section – describes a 2D matrix of Z values.
0x41544144	*Data* section – contains a variable amount of data. The size of the data section is given by the *Size* field in the tag structure.
0x49544c46	*Fault Info* section – describes the fault traces used when creating the grid.

Header Section

The *Header* section must be the first section in the file and has the following format:

Element	Type	Description
Version	long	Version number of the file format. Can be set to 1 or 2

If the version field is 1, then any value >= BlankValue will be assigned the NoData value using Surfer's NoData value, 1.70141e+038.

If the version field is 2, then any value == BlankValue will be assigned the NoData value using Surfer's NoData value, 1.70141e+038.

Grid Section

The *Grid* section consists of a header that describes a 2D matrix of values, followed by the matrix itself. This section encapsulates all of the data that was traditionally referred to as a grid:

Element	Type	Description
nRow	long	number of rows in the grid
nCol	long	number of columns in the grid
xLL	double	X coordinate of the lower left corner of the grid
yLL	double	Y coordinate of the lower left corner of the grid
xSize	double	spacing between adjacent nodes in the X direction (between columns)
ySize	double	spacing between adjacent nodes in the Y direction (between rows)
zlo	double	minimum Z value of the grid. NoData nodes are not included in the minimum.
zhi	double	maximum Z value of the grid. NoData nodes are not included in the maximum. Note that if all nodes are NoData nodes, the zlo=zhi= 1.71041e38.
BlankValue	double	nodes are assigned NoData if greater than or equal to this value. NoData values appear as 1.71041e38.

Data Section

A *Data* section containing the 2D matrix of values (doubles) must immediately follow a *Grid* section. Within the *Data* section, the grid is stored in row-major order, with the lowest row (minimum Y) first

Fault Info Section

A *Fault Info* section describes the fault geometry used to create the grid. *Fault Info* sections have the following format:

Element	Type	Description
nTraces	long	number of fault traces (polylines)
nVertices	long	total number of vertices in all the traces

data section	variable-sized data block consisting of an array of Trace structures immediately followed by the array of vertices

Data Section

A *Data* section containing an array of *Trace* structures and an array of *Vertex* structures must immediately follow a *Fault Info* section. The number of *Trace* structures in the array is nTraces, and the number of *Vertex* structures is nVertices.

Trace Structure

Element	Type	Description
iFirst	long	0-based index into the vertex array for the first vertex of this trace
nPts	long	number of vertices in this trace

Vertex Structure

Element	Type	Description
x	double	X coordinate of the vertex
y	double	Y coordinate of the vertex

Example

The following example illustrates the layout for a 5 row by 10 column grid:

Element	Type	Description
0x42525344	long	Tag: Id for Header section
4	long	Tag: Size of Header section
1	long	Header Section: Version
0x44495247	long	Tag: ID indicating a grid section
72	long	Tag: Length in bytes of the grid section
5	long	Grid Section: nRow
10	long	Grid Section: nCol
0.0	double	Grid Section: xLL
0.0	double	Grid Section: yLL
1.0	double	Grid Section: xSize
1.75	double	Grid Section: ySize
25.0	double	Grid Section: zMin
101.6	double	Grid Section: zMax
0.0	double	Grid Section: Rotation

1.70141e38	double	Grid Section: BlankValue
0x41544144	long	Tag: ID indicating a data section
400	long	Tag: Length in bytes of the data section (5 rows x 10 columns x 8 bytes per double)
Z11, Z12, …	double	Data Section: First (lowest) row of matrix. 10 doubles
Z21, Z22, …	double	Data Section: Second row of matrix. 10 doubles
Z31, Z32, …	double	Data Section: Third row of matrix. 10 doubles
Z41, Z42, …	double	Data Section: Fourth row of matrix. 10 doubles
Z51, Z52, …	double	Data Section: Fifth row of matrix. 10 doubles

Surfer 6 Binary Grid File Format

Surfer 6 grid files .GRD use a layout similar to the Surfer 6 Text grid format. The only difference is in the identification string and that **Surfer 6** grid files are binary. Data types used in **Surfer 6** grid files include:

Type	Description
char	single byte
short	16 bit signed integer
float	32 bit single precision floating point value
double	64 bit double precision floating point value

The **Surfer 6** format has the following layout:

Element	Type	Description
id	char	4 byte identification string ‘ DSBB’ which identifies the file as a **Surfer 6** binary grid file.
nx	short	number of grid lines along the X axis (columns)
ny	short	number of grid lines along the Y axis (rows)
xlo	double	minimum X value of the grid
xhi	double	maximum X value of the grid
ylo	double	minimum Y value of the grid
yhi	double	maximum Y value of the grid
zlo	double	minimum Z value of the grid. NoData nodes are not included in the minimum.
zhi	double	maximum Z value of the grid. NoData nodes are not included in the maximum. Note that if all nodes are assigned NoData, the zlo=zhi= 1.71041e38.
z11, z12, …	float	first row of the grid. Each row has a constant Y coordinate. The first row corresponds to ylo, and the last row

		corresponds to yhi. Within each row, the Z values are ordered from xlo to xhi. NoData values appear as 1.71041e38.
z21, z22, ...	float	second row of the grid
z31, z32, ...	float	third row of the grid
...	float	all other rows of the grid up to yhi

Surfer 6 Text Grid Format

Surfer 6 text grid files .GRD contain five header lines that provide information about the size and limits of the grid, followed by a list of Z values. The fields within ASCII grid files must be space or tab delimited.

The listing of Z values follows the header information in the file. The Z values are stored in row-major order starting with the minimum Y coordinate. The first Z value in the grid file corresponds to the lower left corner of the map. This can also be thought of as the southwest corner of the map, or, more specifically, the grid node of minimum X and minimum Y. The second Z value is the next adjacent grid node in the same row (the same Y coordinate but the next higher X coordinate). When the maximum X value is reached in the row, the list of Z values continues with the next higher row, until all the rows of Z values have been included.

The general format of an ASCII grid file is:

id	The identification string DSAA that identifies the file as an ASCII grid file.
nx ny	nx is the integer number of grid lines along the X axis (columns) ny is the integer number of grid lines along the Y axis (rows)
xlo xhi	xlo is the minimum X value of the grid xhi is the maximum X value of the grid
ylo yhi	ylo is the minimum Y value of the grid yhi is the maximum Y value of the grid
zlo zhi	zlo is the minimum Z value of the grid. NoData nodes are not included in the minimum. zhi is the maximum Z value of the grid. NoData nodes are not included in the maximum. Note that if all nodes have the NoData value, the zlo=zhi= 1.71041e38.
grid row 1	
grid row 2	
grid row 3	

…	These are the rows of Z values of the grid, organized in row order. Each row has a constant Y coordinate. Grid row 1 corresponds to ylo and the last grid row corresponds to yhi. Within each row, the Z values are arranged from xlo to xhi. NoData values appear as 1.71041e38.

Example

The following example grid file is ten rows high by ten columns wide. The first five lines of the file contain header information. X ranges from 0 to 9, Y ranges from 0 to 7, and Z ranges from 25 to 97.19. The first Z value shown corresponds to the lower left corner of the map and the following values correspond to the increasing X positions along the bottom row of the grid file. This file has a total of 100 Z values.

```
DSAA
10 10
0.0 9.0
0.0 7.0
25.00 97.19
91.03 77.21 60.55 46.67 52.73 64.05 41.19 54.99 44.30 25.00
96.04 81.10 62.38 48.74 57.50 63.27 48.67 60.81 51.78 33.63
92.10 85.05 65.09 53.01 64.44 65.64 52.53 66.54 59.29 41.33
94.04 85.63 65.56 55.32 73.18 70.88 55.35 76.27 67.20 45.78
97.19 82.00 64.21 61.97 82.99 80.34 58.55 86.28 75.02 48.75
91.36 78.73 64.05 65.60 82.58 81.37 61.16 89.09 81.36 54.87
86.31 77.58 67.71 68.50 73.37 74.84 65.35 95.55 85.92 55.76
80.88 75.56 74.35 72.47 66.93 75.49 86.39 92.10 84.41 55.00
74.77 66.02 70.29 75.16 60.56 65.56 85.07 89.81 74.53 51.69
70.00 54.19 62.27 74.51 55.95 55.42 71.21 74.63 63.14 44.99
```

Import Options Dialog

N/A

Import Automation Options

N/A

Export Options Dialog

See GRD Surfer 6 Text Grid Export Options Dialog

Export Automation Options

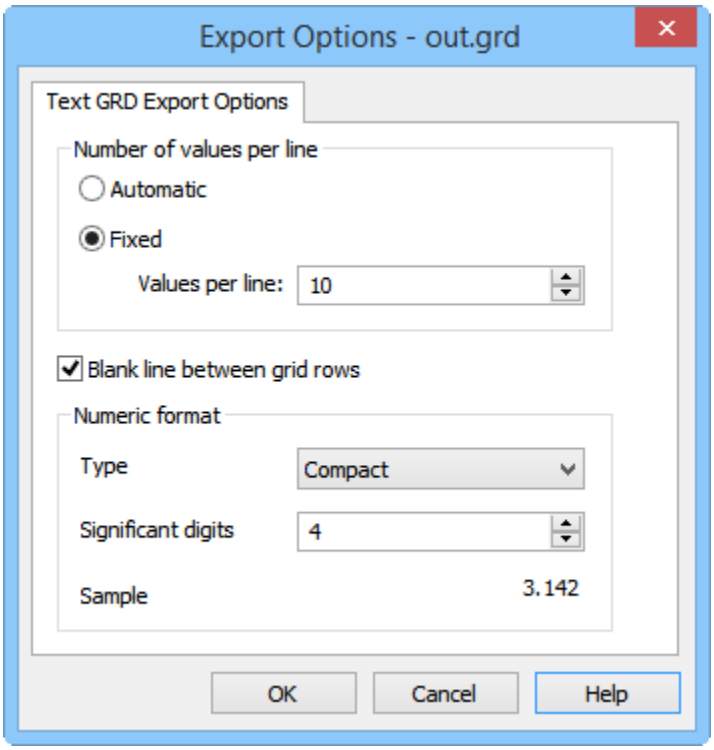
See GRD Surfer 6 Text Grid Export Automation Options

GRD Surfer 6 Text Grid Export Options Dialog

Surfer can export grids in the Surfer 6 Text Grid (*.grd) format.

Text GRD Export Options

The **Text GRD Export Options** page of the **Export Options** dialog contains Text GRD specific export options.



*Select options for Surfer 6 Text Grid export in the **Export Options** dialog.*

Number of Values Per Line

The *Number of values per line* options determine the number of x values per line for each grid row. Select *Automatic* to have each grid row use one line in the text file. Select *Fixed* to specify the number of values on each line in the text file.

When *Fixed* is selected, the *Values per line* option is enabled. Type the desired number of values per line in the *Values per line* field or click the button to change the value.

Blank Line Between Grid Rows

Check the *Blank line between grid rows* to include a blank line between grid rows in the text file. When *Blank line between grid rows* is not checked the Text GRD file will not have any blank lines.

Numeric Format

The *Numeric format* section determines how the grid z values are represented in the grid file. To change the format type click the current selection in the *Type* field and select *Exponential*, *Fixed*, or *Compact* from the list.

- *Exponential* writes the grid z values in scientific notation. Specify the number of decimal digits by typing a value into the *Decimal digits* field. Alternatively click the button to change the *Decimal digits* value.

- *Fixed* writes the grid z values with a fixed number of decimal digits. Specify the number decimal digits by typing a value into the *Decimal digits* field. Alternatively click the button to change the *Decimal digits* value.
- *Compact* writes the grid z values with the fewest possible number of characters, subject to the *Significant digits* value. Once the *Significant digits* limit is met, decimal values are rounded up. For example with 4 *Significant digits*, the value 12.345 is written 12.35 in the text file. When the *Significant digits* limit is met by the integer part of a number, scientific notation is used for the value. For example with 4 significant digits, the value 12345.67 is written 1.235e+04 in the text file.

The *Sample* displays a preview of the selected number format options.

GRD Surfer 6 Text Grid (*.grd) Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of a series of comma-separated keywords that specify the behavior of the import. A typical example would be:

"UserselectValuesPerLine = 1, ValuesPerLine=5, BlankLinePerGridRow=1, NumericFormatType=1, NumericFormatDigits=5"

This would set the number of values on a line to 5, adds a blank line between rows of grid nodes, and sets the numeric format to fixed with 5 significant digits.

Option	Action	Default	Description
ValuesPerLine	Integer	""	Optional. Sets the number of grid nodes on a line in the grid file. Set to 0 to save grids with the same number of values per line as the number of grid columns.
BlankLinePerGridRow	0 = No blank lines per row. 1 = Append a blank line after every row of output.	0	Optional. Sets whether a blank line should appear in the grid after every row of grid nodes.
NumericFormatType	0 = Exponential 1 = Fixed 2 = General	2	Sets the numeric format for the grid node values.
NumericFormatDigits	Integer from 1 to 17		Sets the number of significant digits for the grid node values.

GRIB Global Weather Data Grid File Description

Surfer imports and exports GRB weather data grid files.

A GRIB grid is a binary file format developed for use in the weather and meteorological industries to transport and manipulate weather data and is the foundation of weather forecasts.

File Format

The file format is maintained by the World Meteorological Organization. The format is described in detail at http://www.wmo.int/pages/prog/www/WDM/Guides/Guide-binary-2.html.

File name extensions

.GRB

Loading Files

World Meteorological Organization .GRB files are similar to grid files and can be loaded to create a grid based map (e.g. **Home | New Map | Contour**), or when using the functions in the Grid tab.

Import Options Dialog

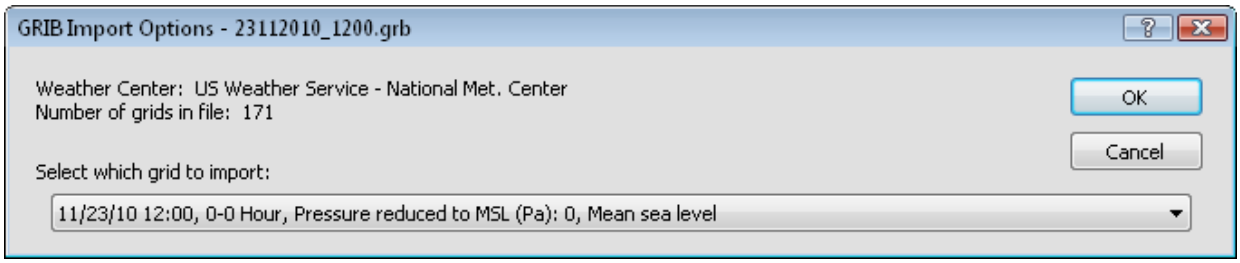
See GRIB Import Options

Import Automation Options

See GRIB Import Automation Options.

GRIB Import Options Dialog

Import a World Meteorological Organization .GRB file into **Surfer** and the **GRIB Import Options** dialog opens.



*Select which grid to import in the **GRIB Import Options** dialog.*

Select Which Grid to Import

Click on the existing grid name in the *Select which grid to import*. In the list, click on the desired grid and click *OK*. The grid is imported.

GRIB Global Weather Data Grid Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,WhichGrid=2"

This would first set all import options to their default values, then set the grid number 2 for import.

Option	Action	Default	Description
WhichGrid	-1 = Import all grids Integer = grid number	""	Specifies which grid to import. This is the desired grid's 0-based sequence number within the file. 0 = first grid, 1 = second grid, etc.

GSB Golden Software Boundary File Description

The .GSB format is a proprietary Golden Software file format. There are several different versions of GSB files, so older Golden Software applications may not be able to read .GSB files exported from newer applications.

Golden Software Boundary files contain boundary objects including areas, curves and points. Primary and Secondary IDs are usually associated with each object. The objects have no attributes (such as color or line style) associated with them.

GS Boundary files are binary files (i.e., they can't be created or modified with a text editor or word processor) that are usually used as base maps. Information indicating the type of projection used (if any) is also stored in the file.

Attributes

All ID attributes for polylines, polygons, and symbol objects are automatically exported to all .GSB files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map. The color, size, symbol shape, width, and other properties are not exported.

Loading GSB Files

Golden Software Boundary .GSB files can be imported with the **File | Import** command, or more commonly the **Home | New Map | Base** command.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

See Golden Software GSB Import Automation Options

Export Options

See Golden Software .GSB Export Options

Export Automation Options

See Golden Software .GSB Export Automation Options

Golden Software Boundary .GSB Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

```
"Defaults=1,AreasToCurves=1"
```

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to closed curves.

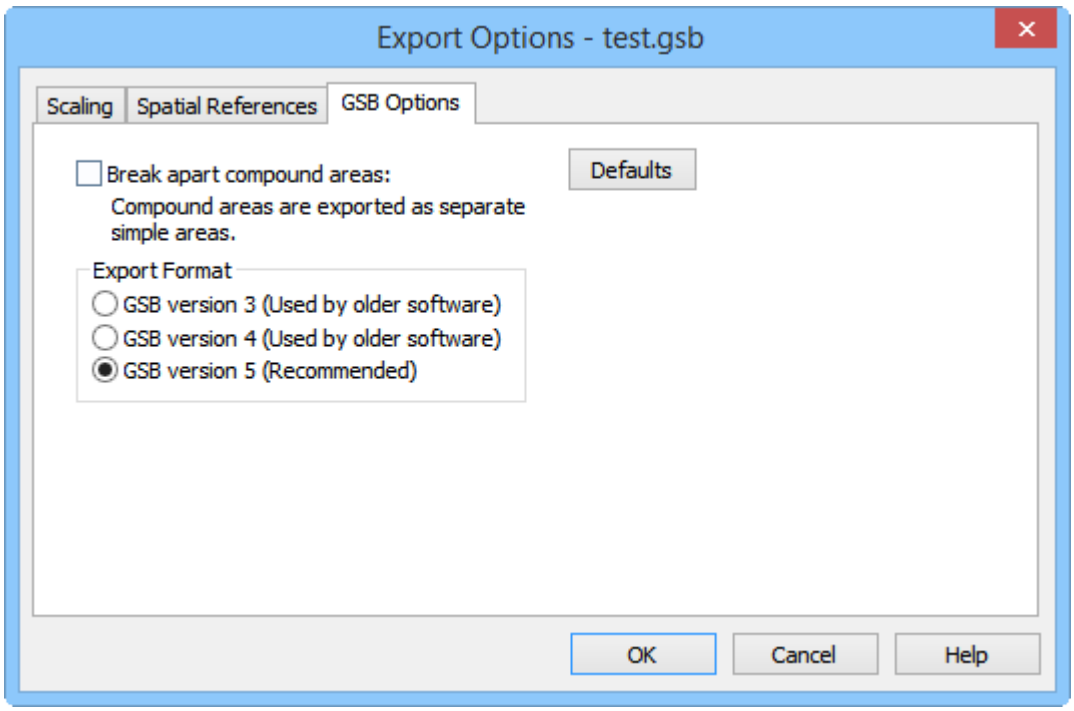
Option	Action	Default
`Defaults=1`	Set all options to their default values	No
`ForgetOptions=1`	Don't remember options for later use	No
`AreasToCurves=1`	Convert area objects to closed curves	No

Golden Software Boundary .GSB Export Options Dialog

The **Export Options** dialog allows you to specify options which determine how information in the file is exported.

Attributes

All ID attributes for polylines, polygons, and symbol objects are automatically exported to all .GSB files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map. The color, size, symbol shape, width, and other properties are not exported.



*Select the GSB export options in the **Export Options** dialog.*

Break Apart Compound Areas

Check the *Break apart compound areas* option to export compound areas as separate simple areas.

Export Format

- Choose *GSB version 3* or *GSB version 4* if you require an export file that can be imported by certain older versions of Golden Software application software.
- Otherwise, choose *GSB version 5*.

Defaults

The *Defaults* button sets all controls to default conditions.

Golden Software Boundary [.GSB] Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,AreasToCurves=1"

This would first set all export options to their default values, then specify that all areas be output as lines.

Option	Action	Default	Description

BreakApartCompoundAreas	0 = No 1 = Yes	0	Compound areas (multi-ring polygons) will be split apart into multiple non-compound areas (simple polygons) during export.
GsbVersion	3 = GSB version 3 4 = GSB version 4 5 = GSB version 5	5	Specify the version of the exported GSB file.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.
FileURY	N.N		Set scaling rectangle upper right Y value.

GSI Golden Software Interchange File Description

Golden Software Interchange .GSI files are binary files containing records indicating graphical entities and their attributes. They provide a medium of exchange between Golden Software programs.

Attributes

All ID attributes for all polyline, polygon, and symbol objects are automatically exported to all .GSI files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map.

Transparency and Fill

All lines, fills, text, images, and vector objects with transparency are exported and imported with the transparency. Objects with bitmap fills are imported and exported with the fill and transparency. Objects with gradient fill are exported and imported with the gradient fill.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

See Golden Software .GSI Import Automation Options

Export Options Dialog

See Golden Software .GSI Export Options

Export Automation Options

See Golden Software .GSI Export Automation Options

Golden Software Interchange .GSI Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,PrimaryIDField=1"

This would first set all import options to their default values, then set the *PrimaryIDField* option value to 1, which would specify that the field number one for each attribute ID record will be assigned to imported objects' IDs.

Option	Action	Default
`Defaults=1`	Set all options to their default values	No
`ForgetOptions=1`	Don't remember options for later use	No
`PrimaryIDField=n`	Specify that field number n of each attribute ID record will be assigned to the primary ID string of each graphical object that is imported from the interchange file.	0
`SecondaryIDField=`*n*	Same as above, except applies to the secondary ID.	0

Golden Software Interchange .GSI Export Options Dialog

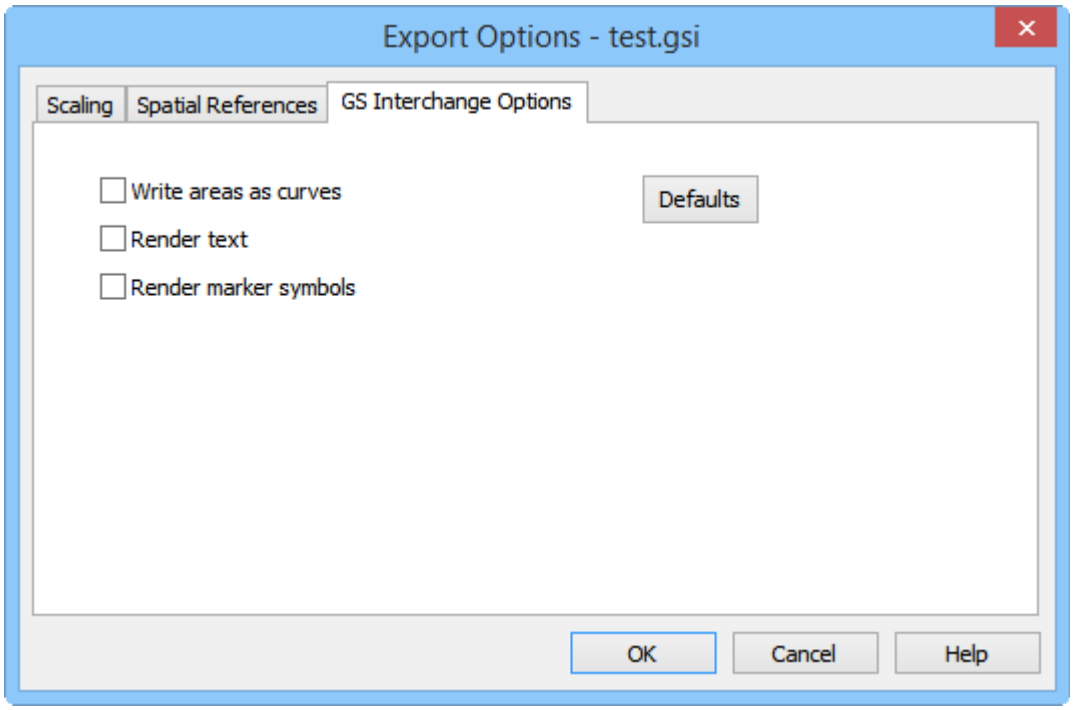
The **Export Options** dialog allows you to specify options which determine how information in the file is exported.

Transparency and Fill

All lines, fills, text, images, and vector objects with transparency are exported and imported with the transparency. Objects with bitmap fills are imported and exported with the fill and transparency. Objects with gradient fill are exported and imported with the gradient fill.

Attributes

All ID attributes for all polyline, polygon, and symbol objects are automatically exported to all .GSI files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map.



*Select the GSI export options in the **Export Options** dialog.*

Write Areas to Curves

Check *Write areas as curves* to cause all area (polygonal) objects to be exported as curve (polyline) objects.

Render Text

Check *Render text* to cause all text objects to be exported as areas and lines. When *Render Text* is unchecked, text is exported as text.

As long as there is no clipping, GSI text entities will be exported as text. This means that the GSI text entities will be sized and oriented similar to the text objects in the application document. When clipping occurs, the text is exported as solid polygons. Clipping occurs when part of the text object is partially inside and partially outside the map limits.

Render Marker Symbols

Check *Render marker symbols* to cause markers to be exported as areas and lines.

Defaults

The *Defaults* button sets all controls to default conditions.

Golden Software Interchange .GSI Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,AreasToLines=1"

This would first set all export options to their default values, then specify that all areas be output as lines.

Option	Action	Default	Description
AreasToLines	0 = No 1 = Yes	0	Output polygons to exported GSI file as polylines.
RenderMarkers	0 = No 1 = Yes	0	If set to 0, marker symbols will be output to the export file as marker objects. If set to 1, marker symbols will be output to the export file as lines and areas to retain the shape and look of the symbol.
RenderText	0 = No 1 = Yes	0	If set to 0, text will be output to the export file as text objects. If set to 1, text will be output to the export file as lines and areas to retain the shape and look of the font face.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.

FileLLY	N.N	Set scaling rectangle lower left Y value.
FileURX	N.N	Set scaling rectangle upper right X value.
FileURY	N.N	Set scaling rectangle upper right Y value.

GXF Grid Exchange [.GXF] File Description

Surfer imports and exports elevation data in Grid Exchange .GXF file format.

The Grid eXchange File .GXF is an open standard ASCII file format for exchanging gridded data among different software systems. Software that supports the .GXF standard will be able to import properly formatted .GXF files and export grids in .GXF format.

File Format

A GXF file is an ASCII file made up of a number of labeled data objects and comments. Each labeled data object has a label line followed by one or more data lines. A label line is identified by a '#' character in the first column followed immediately by an upper-case label. The data associated with that label are found on one or more lines that follow that label.

Any lines that are not part of a labeled data object are ignored and can be used to place comments with a .GXF file. Programs that read .GXF files ignore such comment lines while searching for the next GXF data object.

All lines in a .GXF file must be less than or equal to 80 characters in length. If the last non-white space character on a line is a '\', the next line is assumed to be a continuation of the current line (except for #GRID data lines). Spaces and tab characters are white space characters. Note that continuation lines are a feature of GXF revision 3, and they should only be required for defining projection parameters that require more than 80 characters. They are not required and cannot be used for writing grid data to the GXF (in the #GRID object). This allows GSF readers written for GXF revision 2 and earlier to read GXF revision 3 files.

Any name or string parameters that are part of a data object and which themselves contain spaces, must be enclosed in double quotes. Parameters on data lines may be separated by a space or comma.

For example, the following file satisfies this format:

```
This is a very small 4 x 5 point grid.

#POINTS
5
#ROWS
4

These lines are skipped because they are not part of a data object.
```

```
#GRID
135.28    122.21    119.64    163.25    199.15
145.38    132.45    120.32    121.42    205.18
140.13    151.48    132.91    119.12    219.67
132.67    150.56    140.45    102.89    218.41
```

This file contains two comment lines at the beginning, followed by two labeled data objects, another comment, and finally a #GRID data object. The #GRID object is always the last object and indicates the beginning of the gridded data. This example contains the minimum information required to define a grid, and other grid information, such as location, storage, sense, sampling separations, etc, assume default values. The next section documents all data objects defined in this revision. The only required data objects are #POINTS, #ROWS, and #GRID. All other GXF objects will assume default value. However, a recommended minimum set of defined data objects would also include #PTSEPARATION, #RWSEPARATION, #XORIGIN, and #YORIGIN, since these objects serve to define the minimum amount of information needed to locate the grid in an assumed coordinate system.

Loading .GXF Files

Grid Exchange .GXF files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

See Grid Exchange .GXF Export Options Dialog.

Export Automation Options

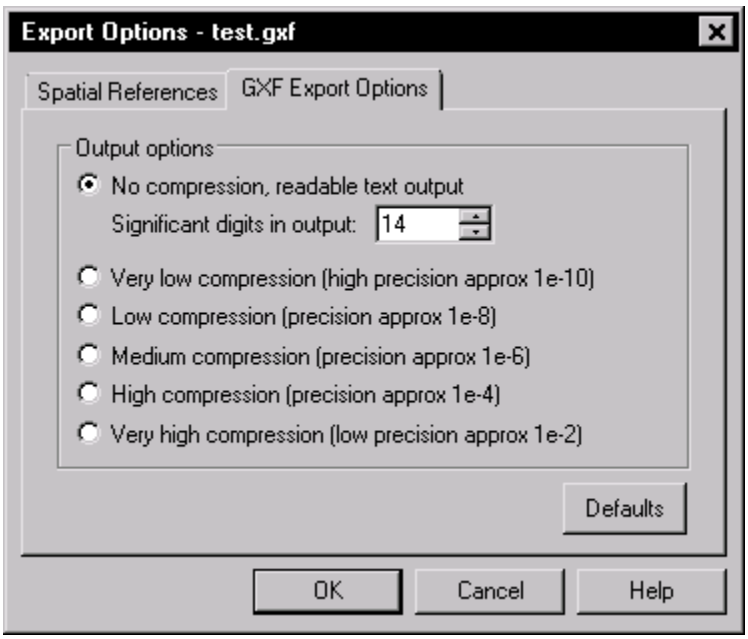
See Grid Exchange .GXF Export Automation Options

References

SEG Gravity and Magnetic Committee, GXF Grid eXchange File, May 2, 2001.
http://www.geosoft.com/resources/technotes/pdfs/gxfr3d9_1.pdf

Grid Exchange .GXF Export Options Dialog

Use the **Grids | Edit | Convert** command in the plot window, or the **File | Save As** command in the Grid Editor to save a .GXF file and open the **GXF Export Options** dialog.



*Specify the GXF output options in the **Export Options** dialog.*

Output Options

The controls in the output options dialog determine which compression format is used when exporting the data. Compression methods that yield higher compression yield lower data precision, and vice versa.

No Compression, Readable Text Output

The *No compression, readable text output* selection exports the data in a human-readable ASCII text format with the specified number of significant digits in each number.

Very Low, Low, Medium, High, Very High Compression

The *Very low compression, Low compression, Medium compression, High compression,* and *Very high compression* selections export the data in a binary floating-point format with the data precision indicated in parenthesis ( ) next to the corresponding selection.

Defaults

Click the *Defaults* button to return the options to the default settings.

Grid Exchange GXF Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" Compression=5,SigFigs=1"

This would indicate that the file should be exported with maximum compression and Z values should include one significant digit.

Option	Action	Default
Compression	Specifies the number of base-90 digits to use for compression: 0 = no compression, max = 5	0
SigFigs	Number of significant figures used when outputting ASCII floating point numerals.	14

HDF Hierarchical Data Format [.HDF] File Description

Surfer imports and exports data sets in Hierarchical Data Format (HDF) Version 4.

Hierarchical Data Format, commonly abbreviated HDF, HDF4, or HDF5 is a library and multi-object file format for the transfer of graphical and numerical data between computers. It was created by the NCSA, but is currently maintained by The HDF Group. The freely available HDF distribution consists of the library, command-line utilities, test suite source, Java interface, and the Java-based HDF Viewer (HDFView).

HDF supports several different data models, including multidimensional arrays, raster images, and tables. Each defines a specific aggregate data type and provides an API for reading, writing, and organizing the data and metadata. New data models can be added by the HDF developers or users.

HDF is self-describing, allowing an application to interpret the structure and contents of a file without any outside information. One HDF file can hold a mixture of related objects which can be accessed as a group or as individual objects. Users can create their own grouping structures called "vgroups."

Loading HDF Files

HDF files are similar to grid files and can be loaded to create a grid based map (i.e. **Home | New Map | Contour**), or when using the functions in the **Grids** tab.

Import Options Dialog

If the HDF data set contains multiple entities, you will be presented with a list of entities from which to choose.

Import Automation Options

N/A

Saving HDF Files

Use the **Grids | Edit | Convert** command in the plot window, or the **File | Save As** command in the grid node editor to save a .HDF file and open the HDF Export Options dialog. Alternatively, the .HDF format is available when prompted to save a grid file. **Surfer** saves in the HDF version 4 format.

Export Options Dialog

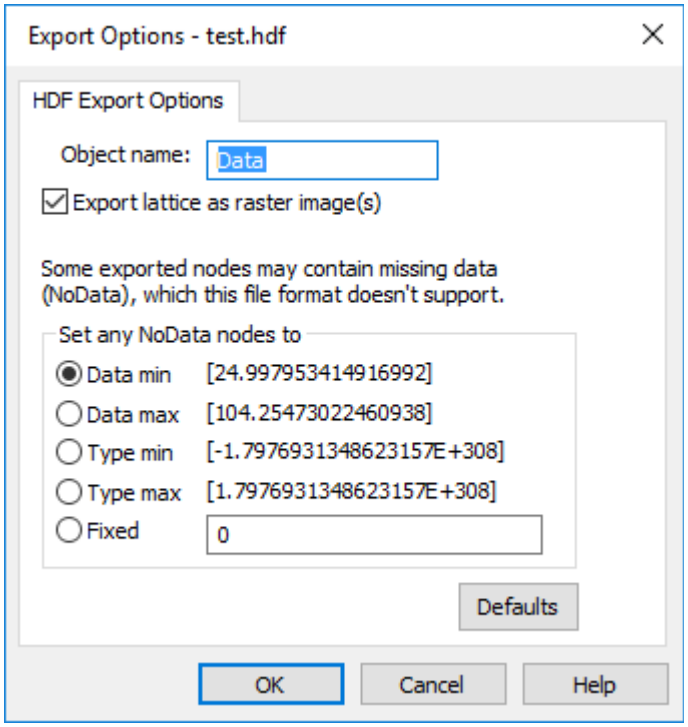
See Hierarchical Data Format .HDF Export Options Dialog.

Export Automation Opitons

See Hierarchical Data Format .HDF Export Automation Options.

Hierarchical Data Format .HDF Export Options Dialog

The export options dialog is only displayed if the lattice contains one or more nodes assigned the NoData value. The dialog controls determine how NoData values are treated in the export file. Use the **Grids | Edit | Convert** command in the plot window, or the **File | Save As** command in the grid node editor to save a .HDF file and open the **HDF Export Options** dialog.



*Control how the NoData nodes are exported in the **Export Options** dialog.*

Object name

The name of the HDF root object may be entered into the space next to *Object name*.

Export Lattice as Raster Image(s)

If the object being exported is a lattice, it can be exported in raster image format instead of lattice format by checking the *Export lattice as raster image(s)* box. If this box is not checked, lattices are exported as lattice data.

Set Any NoData Nodes To

The controls in the *Set any NoData nodes to* section determine how NoData values are treated in the export file.

Data Min

Choose *Data min* to export NoData nodes with the indicated minimum value (varies depending on contents of lattice).

Data Max

Choose *Data max* to export NoData nodes with the indicated maximum value (varies depending on contents of lattice).

Type Min

Choose *Type min* to export NoData nodes with the indicated minimum value for the lattice's data type (usually zero).

Type Max

Choose *Type max* to export NoData nodes with the indicated maximum value of the lattice's data type.

Fixed

Choose *Fixed* to export NoData nodes with some other value. Enter the desired value in the adjacent edit control.

Defaults

Click the *Defaults* button to return the settings on the **HDF Export Options** tab to the default values.

Hierarchical Data Format .HDF Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" BlankMethod=BM_FIXED, FixedBlankValue=1.70141E+038"

This would indicate that the NoData value for a lattice or grid node is a custom value, and that value is equal to **Surfer's** NoData value.

Option	Action	Default
BlankMethod	Method for assigning values to NoData nodes: BM_DATA_MIN = Minimum value of the lattice BM_DATA_MAX = Maximum value of the lattice.	BM_TYPE_MAX

		BM_TYPE_MIN = Minimum value of the inherent data type. BM_TYPE_MAX = Maximum value of the inherent data type. BM_FIXED = A user specified value	
	FixedBlankValue	Used if BlankMethod is BM_FIXED.	No
	ExpDSRootName	Object name.	Empty string
	ExpDSRootName	A non-zero value means export HDR Raster Image whenever possible.	0

HDR GTopo30 [.HDR, .STX, .DEM] File Description

Surfer imports elevation data from GTopo30 data sets.

A GTopo30 data set consists of multiple files. The following files must exist in the same directory:

- .HDR contains header information
- .STX the statistics file, used to obtain zmin, zmax
- .DEM contains the actual grid nodes

GTopo30 files are extremely large. **Surfer** can read the files, though approximately 220 MB of RAM is needed for this operation due to the grid size. You can use Grids | Resize | Extract to make the file smaller. You must have the associated .DEM, .HDR, and .STX files to use GTopo30 files in **Surfer**. These files are necessary for computing the X, Y, and Z limits. Some GTopo30 files contain a "no data" value of -9999 in the file. The no data value is set to zero elevation in **Surfer**.

Loading .HDR Files

GTopo30 .HDR files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu..

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

HGT NASA SRTM Data File Description

Surfer can import HGT binary grid files. **Surfer** can import HGT binary grid files directly from a compressed (zipped) file, if desired. HGT Files are produced by the Shuttle Radar Topography Mission, of NASA.

The SRTM data sets result from a collaboration between NASA, National Geospatial-Intelligence Agency, German space agency, and the Italian space agency. The files contain near-global DEM files of the Earth using radar interferometry. The SRTM instrument consisted of the Spaceborne Imaging Radar-C (SIR-C) hardware set modified with a Space Station-derived mast and additional antennae to form an interferometer with a 60 meter long baseline.

The SRTM data have undergone a sequence of processing steps resulting in several data versions having slightly different characteristics. In addition, the different naming conventions used by the NGA and NASA can lead to some confusion. In the first step raw SRTM radar echo data were processed in a systematic fashion using the SRTM Ground Data Processing System (GDPS) supercomputer system at the Jet Propulsion Laboratory. This processor transformed the radar echoes into strips of digital elevation data, one strip for each of the 1000 or so data swaths. These strips were then mosaicked into just less than 15,000 one degree by one degree cells and formatted according to the Digital Terrain Elevation Data (DTED) specification for delivery to NGA, who are using it to update and extend their DTED products. The DTED specification can be found in MIL-PDF-89020b.pdf on this server.

The data were processed on a continent-by-continent basis beginning with North America and proceeding through South America, Eurasia, Africa, Australia and Islands, with data from each continent undergoing a “block adjustment” to reduce residual errors. These data were also reformatted into the SRTM format, detailed in Section 3 below, and placed on this server as Version 1.0.

In the next step NGA applied several post-processing procedures to these data including editing, spike and well removal, water body leveling and coastline definition as described in the document SRTM_Edit_Rules.doc on this server. Following these "finishing" steps data were returned to NASA for distribution to the scientific and civil user communities as well as the public. These data were also reformatted into the SRTM format and are referred to as Version 2. The figure below shows a portion of cell N34W119.hgt, demonstrating the difference between the edited and unedited data.

Data Formats

The names of individual data tiles refer to the longitude and latitude of the lower-left (southwest) corner of the tile (this follows the DTED convention). For example, the coordinates of the lower-left corner of tile N40W118 are 40 degrees north latitude and 118 degrees west longitude. To be more exact, these coordinates refer to the geometric center of the lower left sample, which in the case of SRTM3 data will be about 90 meters in extent.

SRTM1 data are sampled at one arc-second of latitude and longitude and each file contains 3601 lines and 3601 samples. The rows at the north and south edges as well as the columns at the east and west edges of each cell overlap and are identical to the edge rows and columns in the adjacent cell.

SRTM3 data are sampled at three arc-seconds and contain 1201 lines and 1201 samples with similar overlapping rows and columns. This organization also follows the DTED convention. Unlike DTED, however, 3 arc-second data are generated in each case by 3x3 averaging of the 1 arc-second data - thus 9 samples are combined in each 3 arc-second data point. Since the primary error source in the elevation data has the characteristics of random noise this reduces that error by roughly a factor of three.

This sampling scheme is sometimes called a "geographic projection", but of course it is not actually a projection in the mapping sense. It does not possess any of the characteristics usually present in true map projections, for example it is not conformal, so that if it is displayed as an image geographic features will be distorted. However it is quite easy to handle mathematically, can be easily imported into most image processing and GIS software packages, and multiple cells can be assembled easily into a larger mosaic (unlike the pesky UTM projection, for example.)

Refer to NASA/JPL SRTM web site for additional information.

Import Options Dialog

No import options dialog is displayed.

HTM HTML Image Map File Description

Surfer exports HTML Image Map files. An HTML Image Map is a PNG image with click-able, hyperlinked areas.

A **Surfer** plot is exported to a PNG image, and the polygons, rectangles, and rounded rectangles are also exported to an HTML document overlaid on the PNG image. When the HTML file is opened in a web browser, the polygons retain the hyperlinks and link descriptions assigned in **Surfer**. Users can manually edit the HTML file after exporting to add or edit the hyperlink URLs.

Export Options Dialog

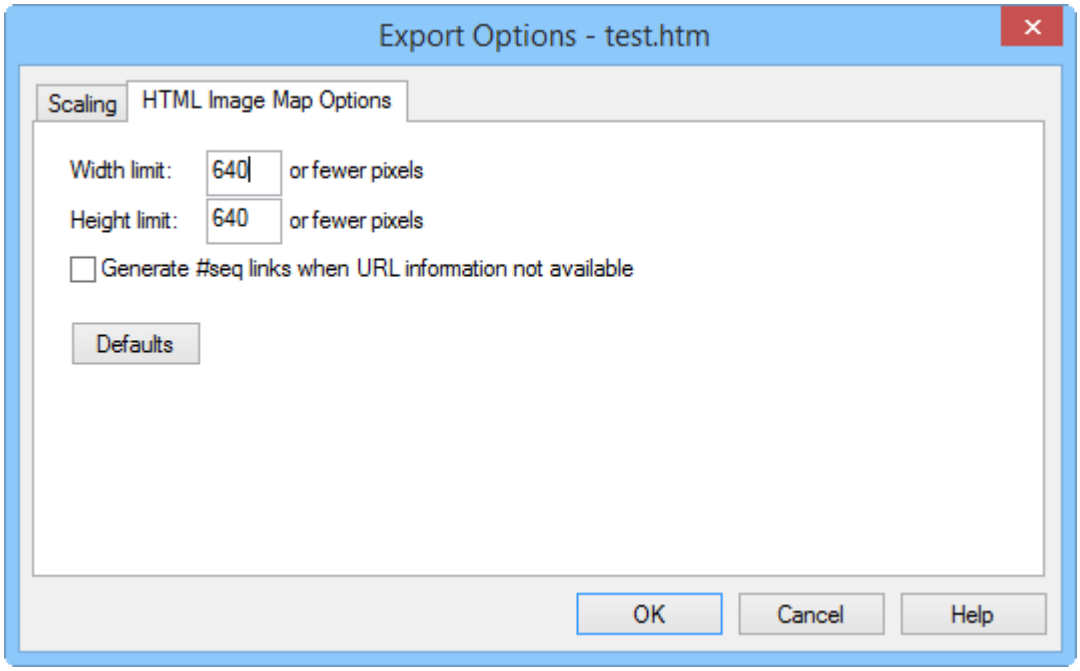
See Scaling and HTML Image Map Export Options Dialog

Export Automation Options

See HTML Image Map Export Automation Options

Export Options Dialog - HTML Image Map Page

The **HTML Image Map Options** page is located in the **Export Options** dialog.



*Set image height and width limits and generate #seq links for polygons without URL information in the **HTML Image Map Options** page.*

Size Limits

Enter a value in pixels into the *Width limit* and *Height limit* fields to specify the maximum size of the PNG image and image map.

Links

Click the *Generate #seq links when URL information not available* check box to manually edit links in the HTM or HTML file after the export is completed. Links are generated, #seq1, #seq2, etc., for any polygons without a hyperlink specified in **Surfer**.

Default Settings

Click the Defaults button to return the options to their default values: Width limit 640 or fewer pixels, Height limit 640 or fewer pixels, and Generate #seq links when URL information not available is not selected.

HTML Image Map .HTM .HTML Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,AllAreas=1"

This would first set all export options to their default values, then specify that all polygons without URL information have #seq links in the HTM file.

Option	Action	Default	Description
WidthLimit	N	640	A value in pixels to specify the maximum image width.
HeightLimit	N	640	A value in pixels to specify the maximum image height.
AllAreas	0 = No 1 = Yes	0	When the value is 0, links are not generated for polygons without a specified hyperlink. When the value is 1, links are generated in the form of #seq1, #seq2, etc., for polygons without hyperlinks.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.

PageLLY	N.N	Set application page rectangle lower left Y value.
PageURX	N.N	Set application page rectangle upper right X value.
PageURY	N.N	Set application page rectangle upper right Y value.
FileLLX	N.N	Set scaling rectangle lower left X value.
FileLLY	N.N	Set scaling rectangle lower left Y value.
FileURX	N.N	Set scaling rectangle upper right X value.
FileURY	N.N	Set scaling rectangle upper right Y value.

Add Hyperlinks and Link Descriptions to Polygons

Surfer can export plots to HTML Image Map files. When opened in a web browser, polygons, rectangles, and rounded rectangles with hyperlinks can be clicked to guide users to a specific file or website. When a user hovers the cursor over the polygon, the URL and link description is displayed in the browser.

To Add a Hyperlink and Link Description to objects in a base layer:

1. Select the layer containing the objects in the plot window or Contents window.
2. Click the **Map Tools | Layer Tools | Open Attribute Table** command to open the Attribute Table.
3. Click the button to add the hyperlink attribute with the New Attribute Name dialog. In the *Attribute name* field, enter one of the following valid names and press enter:
    1. URL
    2. Hyperlink
    3. hyperlink
4. If desired, click the button to add the link description with the New Attribute Name dialog. In the *Attribute name* field, type "LinkDescription" and press enter. The attribute is case sensitive and must match exactly to export link descriptions to HTML Image Map files.
5. In the *URL*, *Hyperlink*, or *hyperlink* attribute column, type the hyperlink file location or web address for each desired object.
6. Type a link description into the *LinkDescription* attribute column for the desired object.

To Add a Hyperlink and Link Description to an object:

1. Select the object in the plot window or Contents window.
2. Click the Info page of the **Properties** window.
3. Click the *Add* button next to *Add new attribute* to add the hyperlink attribute. The New Attribute Name dialog opens. In the *Attribute name* field, enter one of the following valid names and press enter:
    1. URL
    2. Hyperlink
    3. hyperlink
4. If desired, click the *Add* button next to *Add new attribute* to add the link description. The New Attribute Name dialog opens. In the *Attribute name* field, type "LinkDescription" and

press enter. The attribute is case sensitive and must match exactly to export link descriptions to HTML Image Map files.

5. In the **Properties** window, type the hyperlink file location or web address into the hyperlink attribute field for each desired object.
6. In the **Properties** window, type a link description into the *LinkDescription* attribute field for the desired object.

IMG Analyze 7.5 Medical Image Files

Surfer imports and exports Analyze 7.5 medical image data sets as uniform lattices. Analyze 7.5 format is a file format for storing MRI data. An Analyze 7.5 data set consists of two files, the .HDR file and the .IMG file.

File name extensions:

- The .HDR file contains information about the format of the data set.
- The .IMG file contains the image data.

Analyze 7.5 format can contain multiple volumes (i.e., time points), each volume with multiple slices (i.e., Z dimensions), each slice with X and Y dimensions and each pixel (i.e., node) with possible multiple components. Each volume should have the same X, Y and Z dimension and component info.

Format(s) Supported for Import

3D uniform lattice; 8-, 16-, 32-bit integer, float, double

Format(s) Supported for Export

3D uniform lattice; 8-, 16-, 32-bit integer, float, double

Loading .IMG Files

Analyze 7.5 IMG files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Node Editor to export .IMG files. Alternatively, the Analyze 7.5 .IMG file format is available when prompted to save a grid file.

Export Options

See Analyze 7.5 Medical Image .IMG Export Options Dialog

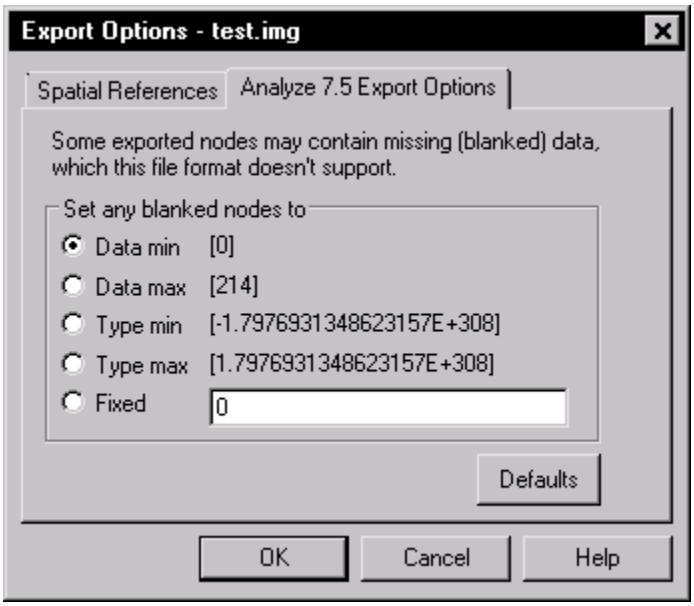
Export Automation Options

See Analyze 7.5 Medical Image .IMG Export Automation Options

Analyze 7.5 Medical Image [.HDR, .IMG] Export Options Dialog

Surfer imports and exports Analyze 7.5 medical image data sets as uniform lattices. Analyze 7.5 format is a file format for storing MRI data.

The **Export Options** dialog is only displayed if the lattice contains one or more nodes assigned the NoData value. The dialog controls determine how NoData values are treated in the export file.



Control how the NoData nodes are exported in the ***Export Options*** *dialog.*

Data Min

Choose *Data min* to export NoData nodes with the indicated minimum value (varies depending on contents of lattice).

Data Max

Choose *Data max* to export NoData nodes with the indicated maximum value (varies depending on contents of lattice).

Type Min

Choose *Type min* to export NoData nodes with the indicated minimum value for the lattice's data type (usually zero).

Type Max

Choose *Type max* to export NoData nodes with the indicated maximum value of the lattice's data type.

Fixed

Choose *Fixed* to export NoData nodes with some other value. Enter the desired value in the adjacent edit control.

Defaults

Click the *Defaults* button to return the options to the default values.

Analyze 7.5 Medical Image .HDR, .IMG Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" BlankMethod=BM_FIXED, FixedBlankValue=1.70141E+038"

This would indicate that the NoData value for a lattice or grid node is a custom value, and that value is equal to **Surfer's** NoData value.

Option	Action	Default
BlankMethod	Method for assigning values to NoData nodes: BM_DATA_MIN = Minimum value of the lattice BM_DATA_MAX = Maximum value of the lattice. BM_TYPE_MIN = Minimum value of the inherent data type. BM_TYPE_MAX = Maximum value of the inherent data type. BM_FIXED = A user specified value	BM_TYPE_MAX
FixedBlankValue	Used if BlankMethod is BM_FIXED	No

IMG ERDAS Imagine Image File Description

Surfer imports ERDAS Imagine IMG files. ERDAS Imagine uses .img files to store raster data with the ERDAS Imagine Hierarchal File Format (HFA) structure. The contents of an ERDAS Imagine .IMG file is not fixed and can be an image or a grid.

If the ERDAS Imagine IMG file is an image file, the file can be imported in four ways:

- As an image with the File | Import command.
- As a base layer with the Home | New Map | Base or **Home | Add to Map | Layer | Base** command.
- As a grid-based layer (e.g. Contour layer with **Home | New Map | Contour** or **Home | Add to Map | Layer | Contour**). If the image is loaded as a grid-based layer, the Z value will be in the range from 0 to 1.
- With the Open Grid dialog via one of the applicable Grid Menu commands.

If the ERDAS Imagine IMG file is a grid file, the file must be imported as a grid-based layer or with the **Open Grid** dialog.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

INFO Leica Confocal Scanning File Description

Surfer imports image data from Leica Confocal Scanning (LEICA) or LCS Microscope data sets. Each data set consists of one or more image slices.

There are several different data set formats generated by the various Leica Laser Confocal Scanning .LCS Microscopes. The format supported by the **Surfer** import filter is the microscope's NATIVE (or "raw") storage format. This format uses an .INFO file to describe a series of images slices stored in a raw pixel array format. The base name of the .INFO file is also used for each of the images, e.g., if "mydata.info" is the header, then the images might appear in "mydata.12" or "mydata.13." The numbering of the image files is specified in the range parameters in the .INFO file. The first image is not always image number zero.

The microscope's software also has the option of storing image slices as TIFF files, which are not supported by this particular filter; however, Golden Software does provide a TIFF import filter for reading those.

Loading .INFO Files

Leica Confocal Raw Slice .INFO files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

JPG File Interchange Format File Description

Surfer imports and exports JPEG JFIF raster image files.

File Format

The JFIF meta data resides in the JPEG Application Segment APP0, having the zero-terminated ASCII string "JFIF" as segment header.

The JPEG File Interchange Format, .JFIF, is a image file format standard. It is a format for exchanging JPEG encoded files compliant with the JPEG Interchange Format .JIF standard. It solves some of JIF's limitations in regard to simple JPEG encoded file interchange. As with all JIF compliant files, image data in JFIF files is compressed using the techniques in the JPEG standard, hence JFIF is sometimes referred to as "JPEG/JFIF".

JFIF defines a number of details that are left unspecified by the JPEG standard (ISO/IEC IS 10918-1, ITU-T Recommendation T.81):

Resolution and aspect ratio

The JPEG standard does not include any method of coding the resolution or aspect ratio of an image. JFIF provides resolution or aspect ratio information using an application segment extension to JPEG. It uses Application Segment #0, with a segment header of 'JFIF\x00', and specifies that this must be the first segment in the file, hence making it simple to recognise a JFIF file. Exif images recorded by digital cameras generally do not include this segment, but typically comply in all other respects with the JFIF standard.

Color Space

JPEG does not define which color encoding is to be used for images. JFIF defines the color model to be used: either Y for greyscale, or YCbCr as defined by CCIR 601. Since this is not an absolute color space — unless an ICC profile, colorspace metadata, or an sRGB tag is provided and interpreted – a decoded JFIF image will be in a device-dependent RGB colorspace. Hence, JFIF does not by itself provide a mechanism for accurately transporting color-managed images across the Internet.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

See Bitmap Export Options Dialog for additional information.

Export Automation Options

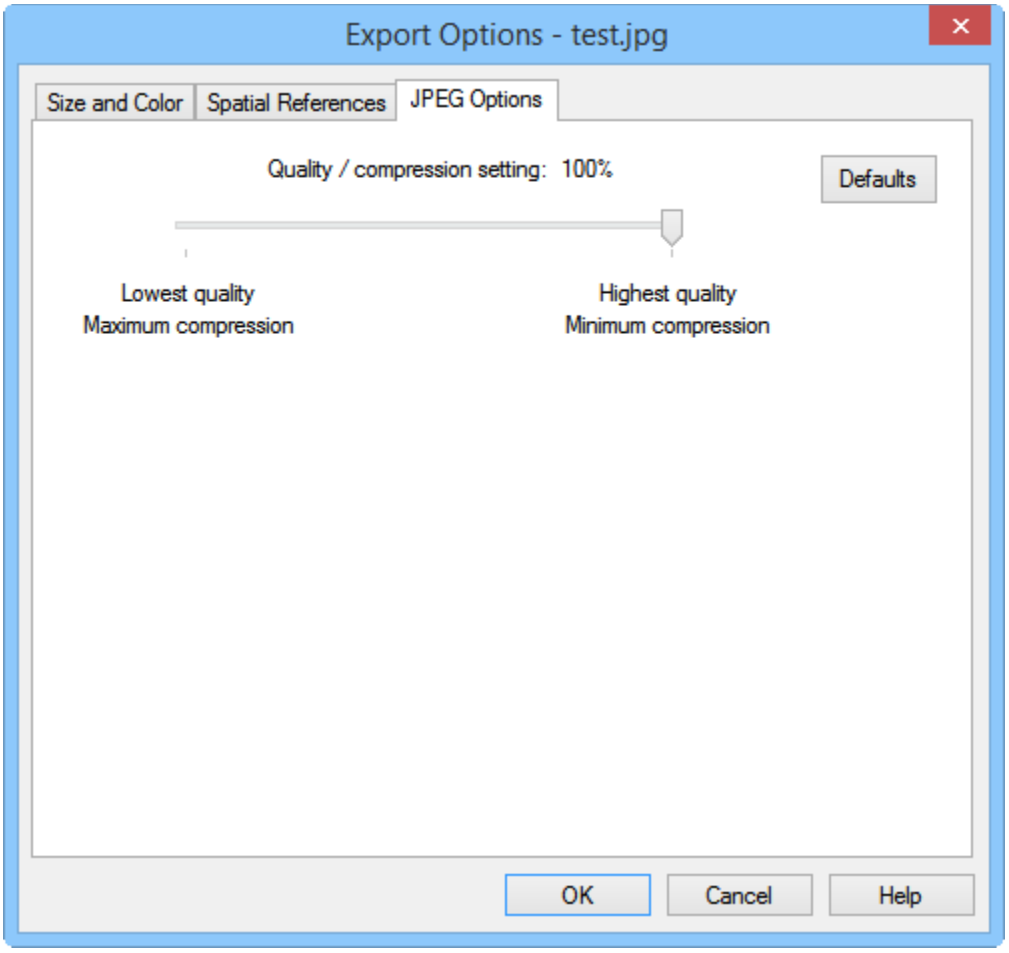
See Bitmap Export Automation Options for additional information.

Legal Notice

This software is based in part on the work of the Independent JPEG Group.

Export Options Dialog - JPEG Options Page

The **JPEG Options** page is located in the **Export Options** dialog.



*Specify the Quality/Compression Setting on the **JPEG Options** page of the **Export Options** dialog.*

Quality/Compression Settings

Adjust the slider for the desired compromise between image compression and image quality. Move the slider to the left for *Lowest quality/Maximum compression.* Move the slider to the right for *Highest quality/Minimum compression.*

Defaults

The *Defaults* button sets all controls to their default values.

JP2 JPEG2000 Interchange Format File Description

Surfer imports and exports JP2 JPEG 2000 raster image files. The JPEG 2000 File Interchange Format, .JP2, is a image file format standard with additional wavelet compression techniques. It is a format for exchanging JP2 encoded files compliant with the JPEG Interchange Format standard. This format is based on the ISO standard (ISO 15444/6). JPEG2000 offers both lossless and lossy compression and creates smaller file sizes than JPG exports. The image is also better quality than the traditional JPG format.

Spatial Reference Information

Some applications associate spatial reference information (such as projection, datum, and georeference parameters) with bitmap images representing a region of the Earth's surface. If the program you're running has spatial reference information to export, the option to save the information will appear on the **Spatial References** tab when exporting. The GeoJP2 format allows the spatial reference information to be stored in one file, along with the image.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

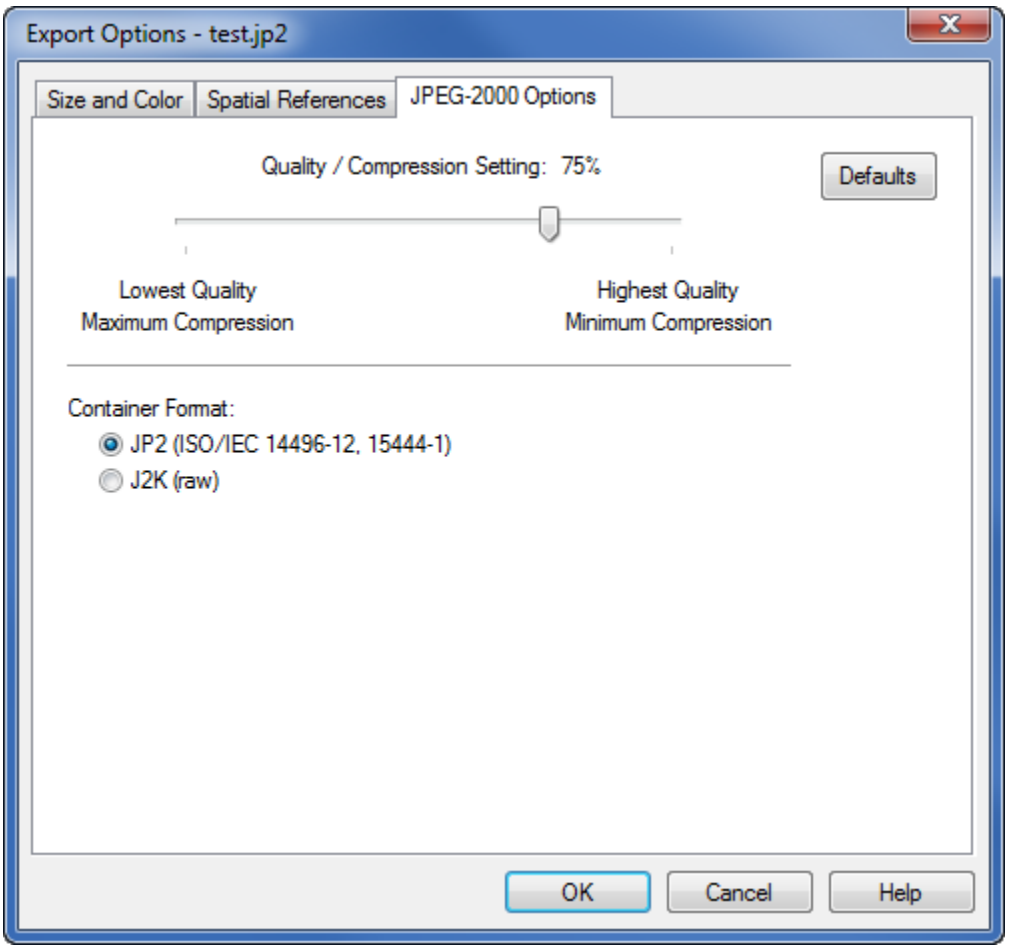
See Size and Color, Spatial References, and JPEG-2000 Options

Export Automation Options

See Image Bitmap Export Automation Options for additional information.

Export Options Dialog - JPEG-2000 Options Page

The JPEG-2000 Options page is located in the Export Options dialog.



*Specify the Quality/Compression Setting on the **JPEG Options** page of the **Export Options** dialog.*

Quality/Compression Settings

Adjust the slider for the desired compromise between image compression and image quality. Move the slider to the left for *Lowest Quality/Maximum Compression.* Move the slider to the right for *Highest Quality/Minimum Compression.*

Container Format

The *Container Format* controls the export format. Select *JP2 (ISO/IEC 14496-12, 15444-1)* to export the image in the ISO JP2 compliant format. This format exports the image data, and organization and contents of the file. Select *J2K (raw)* to export only the image data.

Defaults

The *Defaults* button sets all controls to their default values.

KML KMZ Google Earth Keyhole Markup File Description

Surfer can import and export .KML and .KMZ files.

.KML and .KMZ files are imported into Google Earth to display information, such as contour maps or point locations, on Google Earth maps. Both .KML and .KMZ files contain the same information. The difference is that .KMZ files are compressed versions of .KML files. When exporting to .KML or .KMZ files, images are exported to the same directory specified by the Export dialog.

Coordinate System

.KML and .KMZ files require that the coordinates be in latitude and longitude degrees. This means that the *Map* object in the Contents window is converted to lat/long (WGS84) when exporting. The program does this conversion internally. No change is needed to the *Map*. **Note:** It is important to the *Scaling source* is *Map* in the **Export Options** Scaling page. If the *Scaling source* is *Paper space (page units)* the map will end up in Africa.

Attributes

The first and second attributes for polyline, polygon, and symbol objects are automatically exported to all .KML and .KMZ files. The first attribute is used for object names. The second attribute is used for an object's description. For contour maps, the elevation is exported to the polyline object name for all polylines in the contour map.

Import Options Dialog

See KML Google Earth Import Options Dialog

Import Automation Options

See KML Google Earth Import Automation Options

Export Options Dialog

See KML Export Options Dialog, Scaling, Spatial References

Export Automation Options

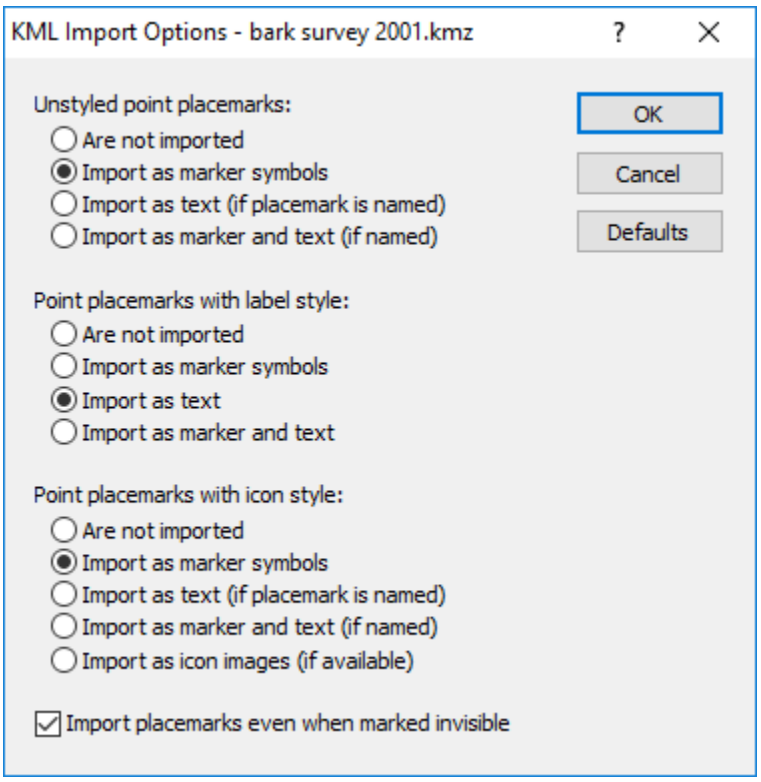
See KML Export Automation Options

Google Earth .KML and .KMZ Import Options Dialog

When using **File | Import** command to import a Google Earth KML or KMZ file, the **KML Import Options** dialog specifies how point placemarks are imported.

KML Import Options Dialog

The **KML Import Options** dialog contains three options for Google Earth KML/KMZ file import.



*Specifiy point placemark options in the **KML Import Options** dialog.*

Unstyled Point Placemarks

Select an option for unstyled point placemarks in the *Unstyled point placemarks* section. The *Are not imported* option does not import the placemarks. The *Import as marker symbols* option imports the placemarks as marker symbols. The *Import as text* option imports the placemark as text. The placemark must be named to be imported as text. The *Import as marker and text* option imports the placemark as both a marker symbol and text. Again, the text is only imported if the placemark is named.

Point Placemarks with Label Style

Select an option for label styled point placemarks in the *Point placemarks with label style* section. The *Are not imported* option does not import the placemarks. The *Import as marker symbols* option imports the placemarks as marker symbols. The *Import as text* option imports the placemark as text. The *Import as marker and text* option imports the placemark as both a marker symbol and text.

Point Placemarks with Icon Style

Select an option for icon styled point placemarks in the *Point placemarks with icon style* section. The *Are not imported* option does not import the placemarks. The *Import as marker symbols* option imports the placemarks as marker symbols. The *Import as text* option imports the placemarks as text if the placemarks are named. The *Import as marker and text* imports the placemarks as marker symbols and text if the placemarks are named. The *Import as icon images* option imports the placemark as icon images if the raster images are available.

Import Placemarks Marked Invisible

Check the *Import placemarks even when marked invisible* check box to include polyline, polygon, and/or point placemarks when they are flagged as insvisible, i.e. visibility = 0. Only visible placemarks will be imported when the *Import placemarks even when marked invisible* check box is not checked.

Google Key Markup .KML and .KMZ Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults = 1, LabelPointsMode=3"

This would first set all import options to their default values, then imports point placemarks with label style as marker and text symbols.

Option	Action	Default	Description
UnstyledPointsMode	0 = Not imported 1 = Import as marker symbols 2 = Import as text (if placemark is named) 3 = Import as marker and text (if named)	1	Specifies how unstyled point placemarks are imported. Placemarks must be named to import as text with values 2 and 3.
LabelPointsMode	0 = Not imported 1 = Import as marker symbols 2 = Import as text 3 = Import as marker and text	2	Specifies how placemarks with label style are imported. Placemarks must be named to import as text with values 2 and 3.
IconPointsMode	0 = Not imported 1 = Import as marker symbols 2 = Import as text 3 = Import as marker and text 4 = Import as icon images (if available)	1	Specifies how placemarks with icon style are imported. Placemarks must be named to import as text with values 2 and 3.
ImportInvisible	0 = False 1 = True	0	Specifies whether placemarks are imported even when marked invisible. 1 to import all placemarks. 0 to import only visible placemarks.

Google Earth .KML and .KMZ Export Options Dialog

When using the **File | Export** command to export to a .KML or .KMZ file, the **Export Options** dialog specifies the Scaling options and Spatial Reference options. In addition, the **KML Options** page is available to control how exported objects are created in the .KML or .KMZ file.

Coordinate System

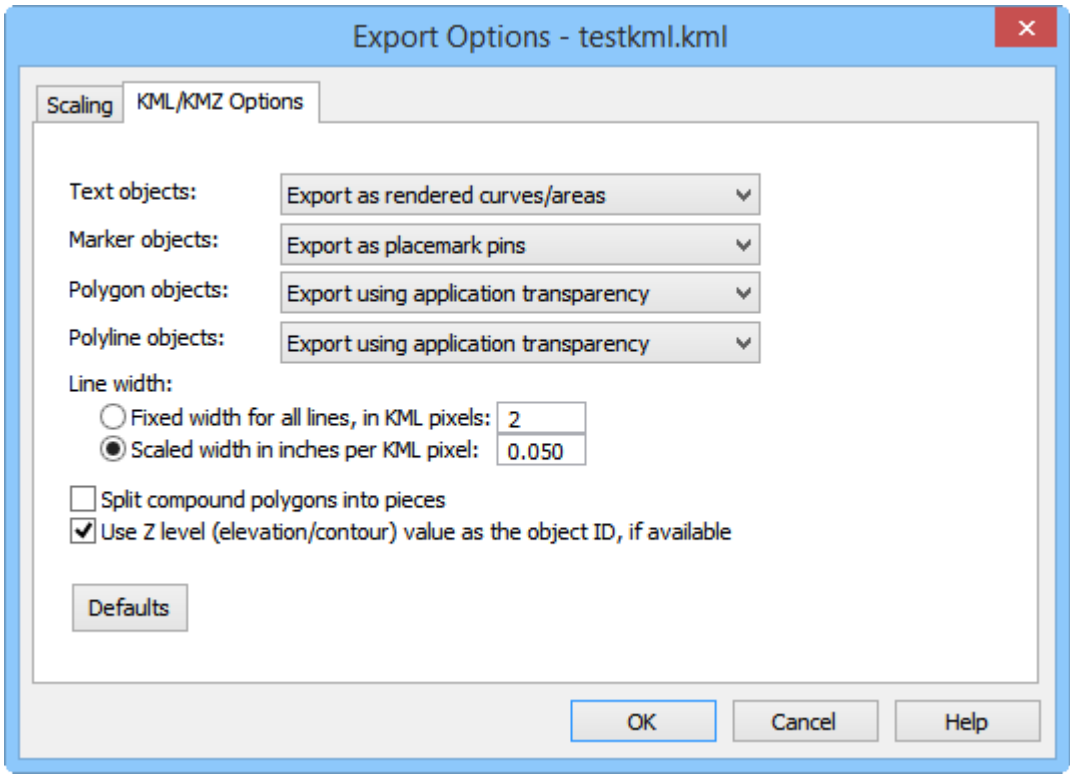
.KML and .KMZ files require that the coordinates be in latitude and longitude degrees. This means that the *Map* object in the Contents window is converted to lat/long (WGS84) when exporting. The program does this conversion internally. No change is needed to the *Map*.

Attributes

The first and second attributes for polyline, polygon, and symbol objects are automatically exported to all .KML and .KMZ files. The first attribute is used for object names. The second attribute is used for an object's description. For contour maps, the elevation is exported to the polyline object name for all polylines in the contour map.

Export Options Dialog

The **Export Options** dialog is displayed when exporting a Google Earth .KML or .KMZ file. The dialog controls determine how text, symbols, polygons, and polylines are treated when exporting these files.



*Set the KML options in the **Export Options** dialog.*

Text Objects

When the *Text objects* is set to *Export as placemark pins with text descriptions*, text is output as point placemarks with the text as the object name. These appear in Google Earth as yellow pushpins. This is the default *Text objects* setting.

Changing the *Text objects* option to *Export as rendered curves/areas* converts all text to polylines and/or polygons, as appropriate. Changing the *Text objects* to *Export as labels* converts the text to placemarks where the placemarks visible markers is the actual text. Unlike rendered text, the label stays the same size on the screen as the Google Earth view is zoomed in and out. Changing the *Text objects* to *Don't export text objects* removes all text from the .KML or .KMZ file.

Marker Objects

When the *Marker objects* is set to *Export as icon pictographs (GIF format)*, symbols are saved to .GIF files. A .GIF file is created for each unique symbol in the **Surfer** file. When sending the .KML or .KMZ file to other users, be sure to include all .GIF files, as well. Otherwise, symbols will not appear. Small red boxes will appear at the locations in Google Earth, indicating the symbols are not found. This is the default *Marker objects* setting.

Changing the *Marker objects* to *Export as placemark pins*, converts all symbols to Google Earth yellow pushpins. Changing the *Marker objects* to *Export as rendered curves/areas,* converts all symbols to polylines and polygons, as appropriate. Changing the *Marker objects* to *Don't export marker symbols* removes all symbols from the .KML or .KMZ file.

Polygon Objects

When the *Polygon objects* is set to *Export using application transparency*, the polygons are exported with the transparency option set in the **Properties** window. This is the default option.

Changing the *Polygon objects* to *Export all areas as opaque*, makes all polygons in the .KML or .KMZ file 100% opaque. No transparency is shown for any objects. Changing the *Polygon objects* to *Export all areas with 50% transparency* makes all polygons 50% transparent, regardless of the transparency level in the **Surfer** file. Changing the *Polygon objects* to *Don't export areas* removes all polygons from the .KML or .KMZ file.

Polyline Objects

When the *Polyline objects* is set to *Export using application transparency,* the polylines are exported with the transparency option set in the **Properties** window. This is the default option.

Changing the *Polyline objects* to *Export all curves as opaque* makes all polylines in the .KML or .KMZ file 100% opaque. No transparency is shown for any objects. Changing the *Polyline objects* to *Don't export curves* removes all polylines from the .KML or .KMZ file.

Line Width

When the *Line width* is set to *Fixed width for all lines, in KML pixels*, the line width for all lines in the .KML or .KMZ file are the same width. Set the width by highlighting the existing value and typing a new value. The smaller the number, the thinner the line. The default value is 2, which makes the lines 2 pixels wide in the .KML or .KMZ file.

When the *Line width* is set to *Scaled width in inches per KML pixel*, the line width for each line is individually determined. The line width in **Surfer** is divided by the number in the box to determine how many pixels are used for each line. The default value is 0.050 which means that a line in

Surfer that is 0.050 inches is equivalent to 1 pixel in the .KML or .KMZ file. The *Scaled width in inches per KML pixel* is the default setting for *Line Width*.

Split Compound Polygons

Check the box next to *Split compound polygons into pieces* to break complex polygons into smaller polygons. Uncheck this box to export the complex polygons as a single object. The default is unchecked. Checking this box does make the .KML or .KMZ file size larger, due to the large number of small polygons being created. When unchecked, some very complex polygons may appear as unfilled in Google Earth.

Z Level as Object ID

Check the box next to Use Z level (elevation/contour) value as the object ID, if available to export the objects with the Z level as the object ID. When the Use Z level (elevation/contour) value as the object ID, if available option is not checked, the first attribute is used for the object name.

Defaults

The *Defaults* button sets all options to default conditions.

KML Layers

When exporting KML files, each map layer is exported to a separate folder in the KML. In Google Earth, this shows as separate folders under the KML file name when imported and displayed on the map. To turn on or off each map layer separately, click on the check box next to the map layer name.

Scaling Page

Scaling information can be set on the Scaling page.

Spatial References

The spatial reference information can be set on the Spatial References page. Check the desired file formats. It is recommend that *GS Reference (Version 2) file* and/or the *Esri .PRJ file* option be checked. Click *OK* and the file is saved.

Limitations

- All line styles are exported as solid lines.
- All fill styles are exported as solid fill.
- Transparency of all objects is controlled on the **KML/KMZ Options** page in the **Export Options** dialog.
- Image transforms are limited to 2D displacement, scaling, and Z rotation.

Google Key Markup .KML and .KMZ Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

```
"Defaults=1,RenderText=1"
```

This would first set all export options to their default values, then convert the text to lines.

Option	Action	Default	Description
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.
FileURY	N.N		Set scaling rectangle upper right Y value.
TextMode	0 = Pins 1 = Polygons/Polylines 2 = Disable text	0	If set to 0, text is exported as placemarks (pins) with text in the description. If set to 1, the text is rendered as polygons and/or polylines. If set to 2, text export is disabled.
MarkerMode	0 = Pins 1 = Polygons/Polylines 2 = Icon pictographs (GIF) 3 = Disable Markers	2	If set to 0, markers are exported as pins. If set to 1, markers are exported as polygons and/or polylines. If set to 2, markers are exported as icon pictographs in GIF format. If set to 3, marker export is disabled.

AreaMode	0 = Application-supplied alpha transparency 1 = Opaque 2 = 50% transparent 3 = Disable areas	0	If set to 0, polygons are exported with the application-supplied transparency. If set to 1, all polygons are exported opaque. If set to 2, all polygons are exported 50% transparent. If set to 3, polygon export is disabled.
LineMode	0 = Application-supplied alpha transparency 1 = Opaque 2 = Disable lines	0	If set to 0, lines are exported with the application-supplied transparency. If set to 1, all lines are exported opaque. If set to 2, line export is disabled.
LineWidthMode	0 = Fixed width 1 = Scaled	1	If set to 0, lines are exported with the same fixed width. If set to 1, lines are exported with scaling.
LineWidthPels	N	2	Line width in Google Earth pixels, if LineWidthMode = 0
LineScaleInchesPerPel	N	0.050	Line scale in inches per Google Earth pixel, if LineWidthMode = 1
DecomposeCompoundPolygons	0 = No 1 = Yes	0	If set to 0, complex polygons are created as complex polygons. Some large polygons may appear unfilled in Google Earth with this setting. If set to 1, complex polygons are split into multiple smaller polygons. This increases the .KML or .KMZ file size.
ZLevelAsName	0 = No 1 = Yes	0	If set to 1, the object Z level will be used for the exported object name. If set to 0, the first attribute is used for the object name.

LAS LiDAR Binary File Description

The ASPRS LiDAR LAS binary file format stores 3D point data. LiDAR stands for Light Detection and Ranging data. This data is generally created by software which combines GPS, IMU, and laser pulse range data to produce X, Y, and Z point data.

Click Home | New Map | Specialty | Point Cloud to create a point cloud map with one or more LAS or LAZ files. Click File | Open or Data | Edit | Merge and select the LAS or LAZ file to view the LAS/LAZ data in the worksheet window.

Surfer imports compressed (LAZ) and uncompressed (LAS) LiDAR Binary files.

File Format

The format contains binary data consisting of a header block, variable length records, and point data. All data is in little-endian format. The header block consists of a public block followed by variable length records. The public block contains generic data such as point numbers and coordinate bounds. The variable length records contain variable types of data including projection information, metadata, and user application data.

Currently, **Surfer** imports 1.0, 1.1, 1.2, 1.3, and 1.4 format LAS files.

Classification Number	Meaning	Point Cloud Classification Colormap Default Color
0	Created, never classified	Black
1	Unclassified	20% Black
2	Ground	Dark Brown
3	Low Vegetation	Forest Green
4	Medium Vegetation	Dull Green
5	High Vegetation	Spring Green
6	Building	Banana Yellow
7	Low Point (noise)	Light Orange
8	Model Key (Mass) point	Blue Purple
9	Water	Blue
10	Railroad	Purple
11	Road Surface	70% Black
12	Overlap	10% Black
13	Wire - Guard (Shield)	Yellow
14	Wire - Conductor (Phase)	Pale Yellow
15	Transmission Tower	Moon Green
16	Wire - Structure Connector (e.g. Insulator)	Magenta
17	Bridge Deck	Powder Blue
18	High Noise	Red
19-39	Reserved	20% Black
40	Bathymetric Point	Twilight Blue
41	Water Surface	Deep Blue

42	Derived Water Surface	Blue Purple
43	Submerged Oject	Pale Purple
44	IHO S57 Object	Light Violet
45	No Bottom Found Bathymetric Point	Deep Navy Blue
46-255	Reserved	20% Black

Import Options Dialog

See LiDAR Import Filtering Dialog for opening the LAS/LAZ file in the worksheet or with the Grid Data command.

See Import Points Dialog for creating a point cloud layer.

Import Automation Options

See LiDAR Import Automation Options

Export Options Dialog

See Export LAS/LAZ

Export Automation Options

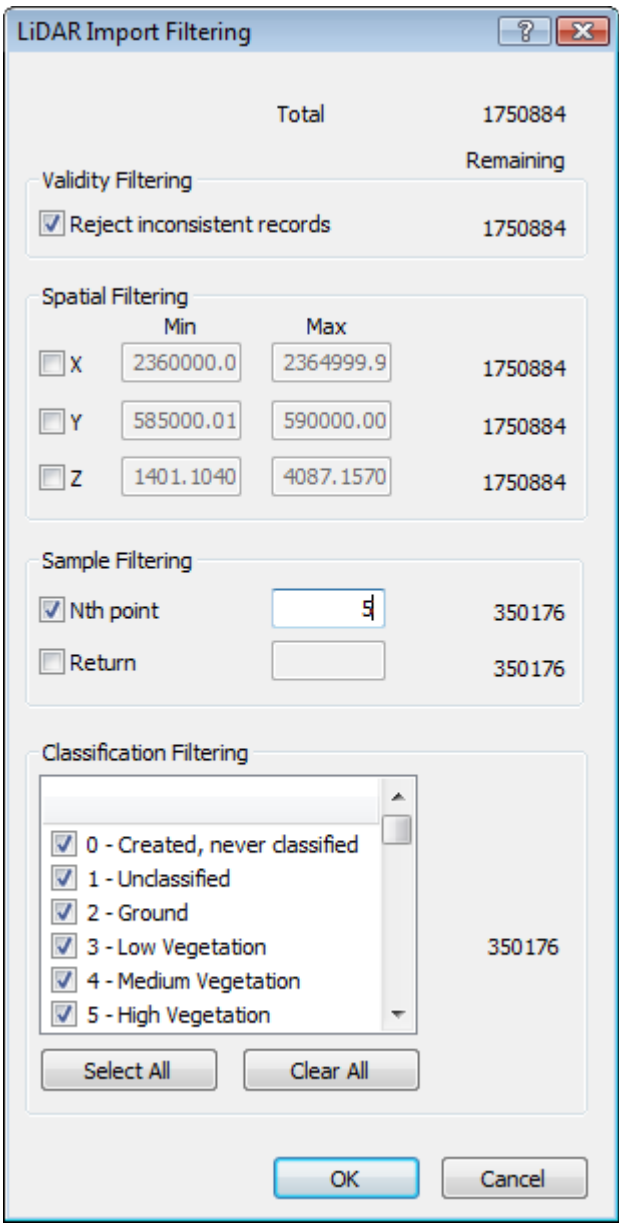
Surfer does not currently export LiDAR LAS data via automation

References:

LAS Specification v1.4 - R13. American Society for Photogrammetry & Remote Sensing. 15 July 2013. http://www.asprs.org/

LiDAR Import Filtering Dialog

Click the Data | Edit | Merge command in a worksheet or the File | Open command to load a .LAS or .LAZ file in the worksheet window. The **LiDAR Import Filtering** dialog allows you to specify options which determine how information in the file is imported. The **LiDAR Import Filtering** dialog is also displayed when gridding LAS or LAZ data.



*Customize import options in the **LiDAR Import Filtering** dialog.*

Total Points

The total number of points in the data file are listed at the top right of the dialog. As LiDAR data is typically extremely large, it is recommended that either *Spatial Filtering* or *Sample Filtering* be applied to the data.

Validity Filtering

Check the box next to the *Reject inconsistent records* option to remove all invalid or inconsistent data from import. When this box is unchecked, all data are passed to the import filter when importing a .LAS file.

Spatial Filtering

Check the boxes next to *X, Y,* or *Z* to filter the data based on the X, Y, or Z data limits. This is useful for selecting only a portion of the data based on geographic extents. After checking the box, highlight the value next to *X, Y,* or *Z* in either the *Min* or *Max* column. Type a new value. The number of points in the *Remaining* column will update to show how many points will exist in the data table after the filtering has been done on the data.

Sample Filtering

Check the box next to *Nth point* or *Return* to filter the data further. The *Nth point* removes every nth point, as set by the value *N*. If you want every other point, type 2 in the box next to *Nth point*. If you want every third point, type 3 in the box.

The *Return* value filters the data based on the laser return. Type the value of the laser return in the box next to *Return*. The return is the filter applied directly to the LAS data detecting waveform peaks. The timing of the peaks is given in the LAS file as a distinct *Return*. Most LiDAR systems designed for topographic mapping are optimized to record 3 return pulses. First returns can be used to create digital surface models that include features above the ground surface, such as buildings, bridges, and tree canopy. Intermediate returns are helpful in separating vegetation from solid objects among the above ground features. Final returns are normally the first approximation of the bare ground surface. If you only want the data from the ground surface, you might type in 3 for the *Return* value.

Classification Filtering

To import only data with a certain classification, check the box next to the classification type. For instance, if you only want to import data that is classified as LiDAR class *2-Ground*, uncheck all other boxes in the *Classification Filtering* section and check the box next to *Ground*.

Select All or Clear All

Click the *Select All* button to check all of the classes in the *Classification Filtering* section. Click the *Clear All* button to uncheck all of the classes in the *Classification Filtering* section. After clicking the *Clear All* button, one or more individual classes should be checked. Otherwise, no information will be imported.

OK or Cancel

Click *OK* to close the **LiDAR Import Filtering** dialog and import the .LAS file. Click *Cancel* to close the **LiDAR Import Filtering** dialog and not import the .LAS file.

LAS LiDAR Binary File Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

Example 1

This would first set all options to the default values, specifies a range of x values, clears the import flag from all classes, then specifies which classes (2 and 3) to import.

```
"Defaults=1,AcceptXMin=2776000,AcceptXMax=2777000,AcceptNoClasses=1,AcceptClass[2]=1,AcceptClass[3]=1"
```

Example 2

This would first set all options to the default values, specifies a range of x values, applies the import flag to all classes, specifies which classes (0 and 4) to exclude from import, then includes only every 5th point.

"Defaults=1,AcceptXMin=2776000,AcceptXMax=2777000,AcceptAllClasses=1,AcceptClass[0]=0,AcceptClass[4]=0,AcceptNthPoint=5"

Option	Action	Default	Description
RejectInconsistent	0 = No 1 = Yes	0	1 to reject inconsistent records, 0 to keep inconsistent records.
AcceptXMin	Value		Minimum X value, greater than or equal to
AcceptXMax	Value		Maximum X value, less than or equal to
AcceptYMin	Value		Minimum Y value, greater than or equal to
AcceptYMax	Value		Maximum Y value, less than or equal to
AcceptZMin	Value		Minimum Z value, greater than or equal to
AcceptZMax	Value		Maximum Z value, less than or equal to
AcceptNthPoint	Value		Filters out every Nth point.
AcceptReturn	Value		Filters by laser return value
AcceptClass	0 = No 1 = Yes		0 rejects the specified class, and 1 accepts the specified class. Takes the form "AcceptClass[x]=n" where x is class number (0-31) and n = 0 or 1 To specify individual classes for import you must first clear the class import flags by including AcceptNoClasses=1 in the options string. To exclude individual classes for import you should first set all class import flags by including AcceptAllClasses=1 or Defaults=1 in the options string.
AcceptAllClasses	0 = No 1 = Yes		1 to select all classes for import. Individual classes can be overridden by AcceptClass[x]=0 specifications.

AcceptNoClasses	0 = No 1 = Yes	1 to clear all class selections. Must be used if specifying classes with AcceptClass[x]=1 specifications.

LAT Iris Explorer [.LAT] File Description

Surfer imports and exports Iris Explorer .LAT files. IRIS Explorer is an Open Inventor based product. Open Inventor provides a de facto standard for 3D scene definition and was the basis of the VRML (Virtual Reality Modeling Language) extension to HTML and the World Wide Web.

The IRIS Explorer *Lattice* data type is a generalized multi-dimensional array. It differs from arrays in other languages by being able to store both node-based values and coordinate information for each of the nodes. Being extremely flexible, it is the most commonly used data type in IRIS Explorer, used almost exclusively to store data and pass it between modules.

The *Data* part of a lattice can be multi-dimensional. If required, multiple data values can be stored at each node.

The *Coordinate* part of a lattice can have a varying number of dimensions, dependent on the number of dimensions that the data space occupies. There are three different types of coordinate information: uniform, perimeter, curvilinear.

Format supported for Import

3D uniform lattice

Format supported for Export

3D uniform lattice

Loading .LAT Files

Iris Explorer .LAT files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

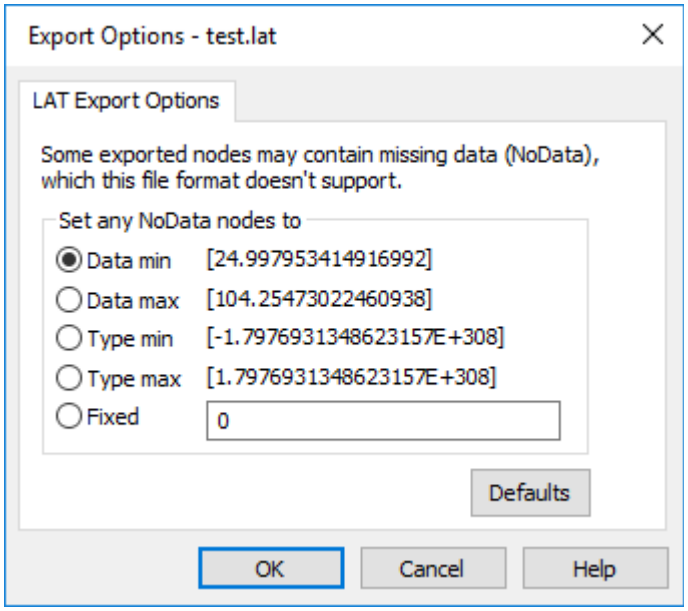
See Iris Explorer .LAT Export Options Dialog.

Export Automation Options

See Iris Explorer .LAT Export Automation Options.

Iris Explorer .LAT Export Options Dialog

The export options dialog is only displayed if the lattice contains one or more nodes assigned the NoData value. The dialog controls determine how NoData values are treated in the export file. Use the **Grids | Edit | Convert** command in the plot window, or the **File | Save As** command in the grid node editor to save a .LAT file and open the **LAT Export Options** dialog.



*Control how the NoData nodes are exported in the **Export Options** dialog.*

Set Any NoData Nodes To

The controls in the *Set any NoData nodes to* section determine how NoData values are treated in the export file.

Data Min

Choose *Data min* to export NoData nodes with the indicated minimum value (varies depending on contents of lattice).

Data Max

Choose *Data max* to export NoData nodes with the indicated maximum value (varies depending on contents of lattice).

Type Min

Choose *Type min* to export NoData nodes with the indicated minimum value for the lattice's data type (usually zero).

Type Max

Choose *Type max* to export NoData nodes with the indicated maximum value of the lattice's data type.

Fixed

Choose *Fixed* to export NoData nodes with some other value, and enter the desired value in the adjacent edit control.

Defaults

Click the *Defaults* button to return the settings on the page to the default options.

Iris Explorer .LAT Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" BlankMethod=BM_FIXED, FixedBlankValue=1.70141E+038"

This would indicate that the NoData value for a lattice or grid node is a custom value, and that value is equal to **Surfer's** NoData value.

Option	Action	Default
BlankMethod	Method for assigning values to NoData nodes: BM_DATA_MIN = Minimum value of the lattice BM_DATA_MAX = Maximum value of the lattice. BM_TYPE_MIN = Minimum value of the inherent data type. BM_TYPE_MAX = Maximum value of the inherent data type. BM_FIXED = A user specified value	BM_TYPE_MAX
FixedBlankValue	Used if BlankMethod is BM_FIXED.	No

LVL Level File Format

Level .LVL files are used to specify level and property data for various map types within **Surfer**. Level files are ASCII files with one record per line. Fields within the line are separated by commas or white space. Comments are delimited by a single quote character, and may be placed at the beginning of a line or the end of a record.

LVL Format 1

There are three different forms of the level file format. The first and simplest form consists of a single elevation value per record. For example, the following file can be used to specify logarithmic intervals for a contour map:

```
0.1
1
```

```
10
100
1000
```

LVL Format 2

The second form of the level file format is indicated by specifying the LVL2 keyword on the first line. In this form of level file, each subsequent record has the following general syntax:

```
Level Flags LColor LStyle LWidth FFGColor FBGColor FPattern FMode FScale FType
```

The fields are space delimited and defined as follows:

Element	Description
Level	Level is the Z value of the level.
Flags	Flags represents labels and hachures where 0=normal (no labels, no hachures), 1=labeled, 2=hached, and 3=labeled and hached.
LColor	LColor is the color of lines used to represent the level. The color must be enclosed in double quotes.
LStyle	LStyle is the style of lines used to represent the level, and it is one of the line style names as listed in the Line Styles topic. The name must be enclosed in double quotes.
LWidth	LWidth is the line width in mils (thousandths of an inch).
FFGColor	FFGColor is the fill foreground color. The color must be enclosed in double quotes.
FBGColor	FBGColor is the fill background color. The color must be enclosed in double quotes.
FPattern	FPattern is one of the fill style names as appears in the Fill Pattern palette within the dialogs. The name must be enclosed in double quotes.
FMode	FMode is the background mode for patterns where 1=transparent and 2=opaque.

All colors are specified by name as listed in the Colors topic or by the special syntax "Rxxx Gyyy Bzzz." In this form, the red, green, and blue components are substituted for the xxx, yyy, and zzz fields, respectively. Each component can range from 0 to 255.

Here is an example of a version 2 level file that defines 4 levels and their properties:

```
LVL2
20 1 "Black" "Solid" 0 "Black" "Black" "Solid" 2
25 0 "Black" "Solid" 0 "R15 G15 B15" "R15 G15 B15" "Solid" 2
30 0 "Black" "Solid" 0 "R31 G31 B31" "R31 G31 B31" "Solid" 2
35 0 "Black" "Solid" 0 "R47 G47 B47" "R47 G47 B47" "Solid" 2
```

LVL Format 3

The third form of the level file format is indicated by specifying the LVL3 keyword on the first line. In this form of level file, each subsequent record has the following general syntax:

```
Level Flags LColor LStyle LWidth FVersion FFGColor FBGColor FPattern OffsetX
OffsetY ScaleX ScaleY Angle Coverage
```

The fields are space delimited and defined as follows:

Element	Description
Level	Level is the Z value of the level.
Flags	Flags represents labels and hachures where 0=normal (no labels, no hachures), 1=labeled, 2=hached, and 3=labeled and hached.
LColor	LColor is the color of lines used to represent the level. The color must be enclosed in double quotes.
LStyle	LStyle is the style of lines used to represent the level, and it is one of the line style names as listed in the Line Styles topic. The name must be enclosed in double quotes.
LWidth	LWidth is the line width in mils (thousandths of an inch).
FVersion	Unused, set to 1
FFGColor	FFGColor is the fill foreground color. The color must be enclosed in double quotes.
FBGColor	FBGColor is the fill background color. The color must be enclosed in double quotes.
FPattern	FPattern is one of the fill style names as appears in the Fill Pattern palette within the dialogs. The name must be enclosed in double quotes.
OffsetX	OffsetX controls the X location of the image pattern within the geometry.
OffsetY	OffsetY controls the Y location of the image pattern within the geometry.
ScaleX	ScaleX controls the density in the X direction of an image pattern.
ScaleY	ScaleY controls the density in the Y direction of an image pattern.
Angle	Angle controls the rotation of the pattern. This should always be set to zero.
Coverage	Coverage determines whether an image is stretched or tiled. 0 = tiled, 1 = stretched.

All colors are specified by name as listed in the Colors topic or by the special syntax "Rxxx Gyyy Bzzz Aaaa." In this form, the red, green, and blue components are substituted for the xxx, yyy, and zzz fields, respectively. The opacity value is substituted for the aaa value. Each component can range from 0 to 255.

Here is an example of a version 3 level file that defines 5 levels and their properties:

```
LVL3
'Level Flags LColor LStyle LWidth FVersion FFGColor FBGColor FPattern OffsetX
OffsetY ScaleX ScaleY Angle Coverage
2100 1 "R216 G0 B216 A255" ".2 in. Dash" 0 1 "Powder Blue" "R255 G255 B255 A156"
"Crosshatch" 0 0 1 1 0 0
2300 0 "R216 G0 B216 A255" ".2 in. Dash" 12 1 "R255 G0 B255 A207" "R255 G255 B255
A0" "Dashed Horizontal" 0 0 1 1 0 0
```

```
2500 0 "R216 G0 B216 A255" ".2 in. Dash" 25 1 "R102 G153 B102 A130" "R255 G255
B255 A0" "Solid" 0 0 1 1 0 0
2700 0 "R216 G0 B216 A255" ".2 in. Dash" 37 1 "Powder Blue" "R255 G255 B255 A0"
"Andesite 1" 0 0 1 1 0 0
2900 0 "R216 G0 B216 A255" ".2 in. Dash" 50 1 "Red" "R255 G255 B255 A0" "Coarse,
medium-graded sand" 28 0 1.55 1.55 0 0
```

MIF MapInfo Interchange Format File Description

MapInfo Interchange Format .MIF files contain boundary objects including areas, curves and points. The objects optionally have attributes (such as color or line style) associated with them.

.MIF files are text files that are usually used as base maps.

MID File

Each. MIF file is usually accompanied by a file with the same name, but with the .MID file name extension. The .MID file contains attribute information about the objects in the map. This information is imported in the objects' attributes. The .MID file can be opened in the worksheet with the File | Open command.

Limitations

The MIF file format limits attribute values to 254 characters. If an attribute value has more than 254 characters, a warning message will be issued. MapInfo may be unable to read the exported MIF file. It is suggested that the attribute values be limited to only 254 characters.

The MIF file format limits all attributes for a single object to a total of 4000 characters. This includes all attribute names and attribute values. If an object has more than 4000 characters, a warning message will be issued. MapInfo may be unable to read the exported MIF file. It is suggested that the number of characters be reduced so that all attribute names and values are limited to less than 4000 characters.

The MIF file format requires that all attribute names start with a character other than a number. When an attribute name starts with a number in **Surfer**, the attribute name will automatically be renamed to start with an underscore. For example, the attribute name *1table* will be renamed to *_1table*.

Attributes

All attributes for all polyline and polygon objects are automatically exported to all .MIF files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map.

Loading .MIF Files

MapInfo .MIF files can be loaded with the File | Import and Home | New Map | Base commands.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

See MapInfo Interchange Format (MIF) Import Automation Options

Export Options Dialog

See MapInfo Interchange Format (MIF) Export Options Dialog

Export Automation Options

See MapInfo Interchange Format (MIF) Export Automation Options

MapInfo Interchange Format .MIF Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

```
"Defaults=1,AreasToCurves=1"
```

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to closed curves.

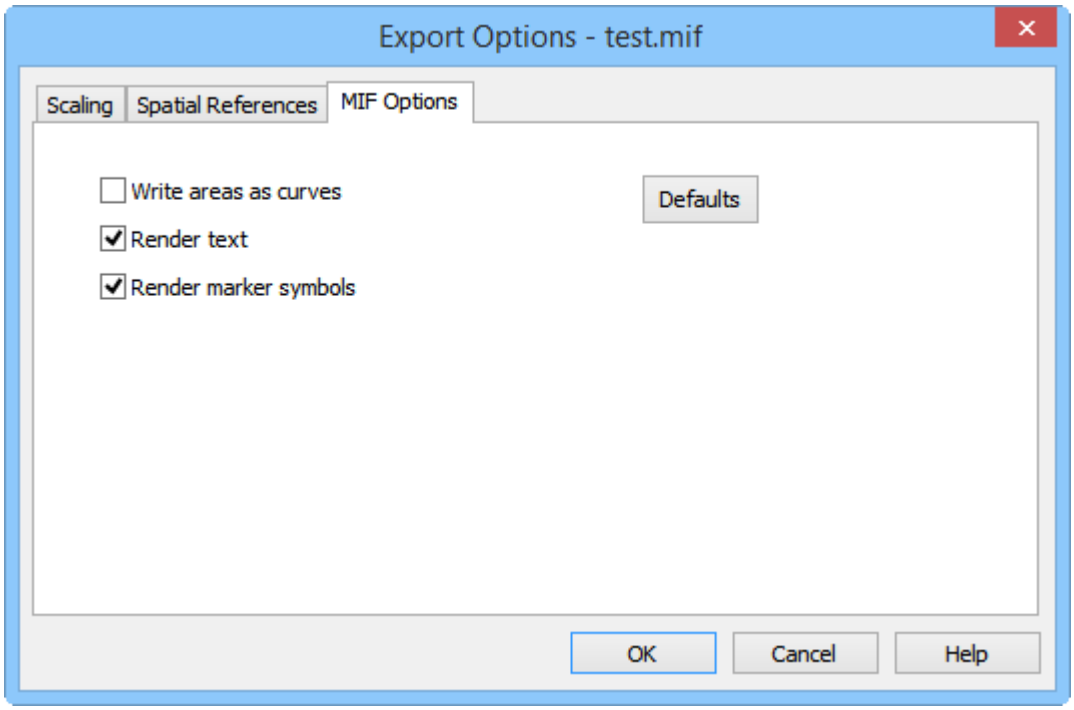
Option	Action	Default
`Defaults=1`	Set all options to their default values	No
`ForgetOptions=1`	Don't remember options for later use	No
`AreasToCurves=1`	Convert area objects to closed curves	No
`IgnoreStyles=1`	Import as base map (ignore styles, colors and fill)	No
`PrimaryIDField=`*n*	Specify that field number n of each record in the optional MID database will be assigned to the primary ID string of each object that is imported from the MIF input file. If the MIF file is not accompanied by a MID file, this option has no effect.	0
`SecondaryIDField=`*n*	Same as above, except applies to the secondary ID.	0

MapInfo Interchange Format .MIF Export Options Dialog

The **Export Options** dialog allows you to specify options which determine how information in the file is exported.

Attributes

All attributes for all polyline and polygon objects are automatically exported to all .MIF files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map.



*Specify the MIF export options in the **Export Options** dialog, **MIF Options** page.*

Write Areas to Curves

Check *Write areas as curves* to cause all area (polygonal) objects to be exported as curve (polyline) objects.

Render Text

Check *Render text* to cause all text objects to be exported as areas and lines. When *Render Text* is unchecked, text is exported as text.

As long as there is no shear, perspective, or clipping, MIF text entities will be exported as text. This means that the MIF text entities will be sized and oriented similar to the text objects in the application document. When shear, perspective, or clipping occur, the text is exported as solid polygons. Shear occurs when the character glyphs are not perpendicular to the text baseline. Perspective occurs when the height of glyphs in the text string are not all the same, as in a 3-D view where the glyphs are smaller the farther they are from the observer. Clipping occurs when part of the text object is partially inside and partially outside the map limits.

Render Marker Symbols

Check *Render marker symbols* to cause markers to be exported as areas and lines.

MapInfo Interchange Format .MIF Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,AreasToLines=1"

This would first set all export options to their default values, then specify that all areas be output as lines.

Option	Action	Default	Description
AreasToCurves	0 = No 1 = Yes	0	Output polygons to exported MIF file as polylines.
RenderMarkers	0 = No 1 = Yes	0	If set to 0, marker symbols will be output to the export file as marker objects. If set to 1, marker symbols will be output to the export file as polylines and polygons to retain the shape and look of the symbol.
RenderText	0 = No 1 = Yes	0	If set to 0, text will be output to the export file as text objects. If set to 1, text will be output to the export file as polylines and polygons to retain the shape and look of the font face.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.

FileURY	N.N	Set scaling rectangle upper right Y value.

NC NetCDF File Description

Surfer can import NetCDF .NC and .GRD lattice file formats. **Surfer** can export NetCDF .NC lattice file formats. These files are uniform grids.

NetCDF is a set of software libraries and self-describing, machine-independent data formats that support the creation, access, and sharing of array-oriented scientific data. It is also a community standard for sharing scientific data. NetCDF was developed and is maintained at Unidata, part of the University Corporation for Atmospheric Research (UCAR) Community Programs (UCP). Unidata is funded primarily by the National Science Foundation.

A NetCDF file may contain multiple variables. An import options dialog shows all importable variables available in the file. Only one variable may be imported at a time, and only the selected variable will be imported. The **Import Options** dialog will not be displayed if only one variable is present in the NetCDF file.

Loading Files

NetCDF NC grid files are similar to **Surfer** grid files and can be loaded to create a grid based map (i.e. Home | New Map | Contour), or when using the functions in the Grids tab.

Import Options Dialog

NetCDF Import Options Dialog

Import Automation Options

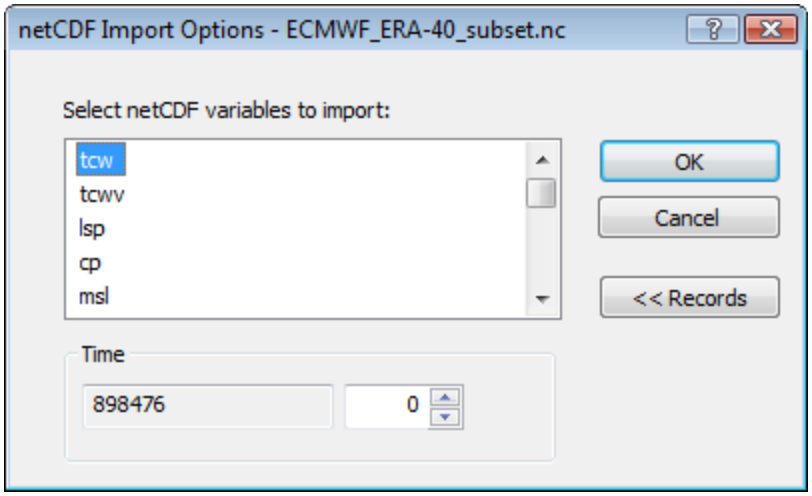
See NetCDF Import Automation Options

Export Options

See Export Options - Spatial References. Use the Grids | Edit | Convert command in the plot document, or the File | Save As command in the Grid Editor to export an NetCDF Network Common Data Form .NC file.

NetCDF Import Options Dialog

Surfer imports NetCDF .NC and .GRD lattice files. These files are uniform grids.



Select the netCDF variable to import and set the Time record.

Variable

A NetCDF file may contain multiple variables. In the *Select netCDF variables to import* section, click on the desired variable to import. Only one variable may be imported at a time, and only the selected variable will be imported.

Records

The *Records* button is available if the selected variable is a record variable. If the variable is not a record variable, the *Records* button is unavailable.

Time

The *Time* option lets the user select which record to load. The coordinate value for the unlimited dimension at that record index is shown in the text control on the left. Note that if a variable has an unlimited dimension other than the first dimension, it will not be considered importable by the filter.

The filter treats record variables specially. Record variables have one or more unlimited dimensions. The n-th record contains all the variable data for the n-th dimension index.

NetCDF .NC File Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,VariableName=toc"

This would first set all import options to their default values, then set the VariableName option value to *toc*, which would specify the variable to import.

Option	Default	Description

VariableName	""	Set to the variable name to import. The variable name must match exactly with the variable name listed in the file.
RecordIndex	""	Set to the record index to import for variable names with records.

NetCDF .NC Grid File Format Export Options

Surfer exports NetCDF NC grid files. These files are uniform grids.

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .NC files. The Spatial References page is displayed when exporting a .NC file when the grid file contains a coordinate system. The dialog controls how spatial reference information is stored.

PDF Adobe Acrobat Portable Document Format [.PDF] File Description

Portable Document Format, .PDF, is a file format used for document exchange. PDF is used for representing two-dimensional documents. PDF was created by Adobe Systems.

File Structure

A .PDF file consists primarily of eight object types.

- *Boolean* values, representing true or false
- Numbers
- Strings
- Names
- *Arrays*, ordered collections of objects
- *Dictionaries*, collections of objects indexed by *Names*
- *Streams*, usually containing large amounts of data
- The *Null* object

Import

Click the Home | New Map | Base command to create a base map from a PDF file. Click the File | Import command to import the PDF file as a group of objects or images. See PDF Import Options Dialog

Export

Click the **File | Export** command to export as a PDF (Raster) (*.PDF) file or a PDF (Vector) (*.PDF) file. Alternately, you can use the **File | Print** command to print to a PDF driver if you have one installed.

GeoPDF

When a map with a defined coordinate system is exported to a Vector PDF file format, a GeoPDF file is created automatically. The scaling source for the GeoPDF is changed on the Scaling page in the **Export Options** dialog. When a map with a defined coordinate system is exported to a Raster PDF file format, the georeference options for the GeoPDF are changed on the Scaling and Spatial

References pages in the **Export Options** dialog. The *Internal file format* option must be checked on the **Spatial References** page to create the GeoPDF.

An add-on, such as TerraGo toolbar, may be needed to view the GeoPDF options in Adobe's Acrobat or PDF Reader programs. The map layer and map must have an assigned coordinate system in **Surfer** prior to the export.

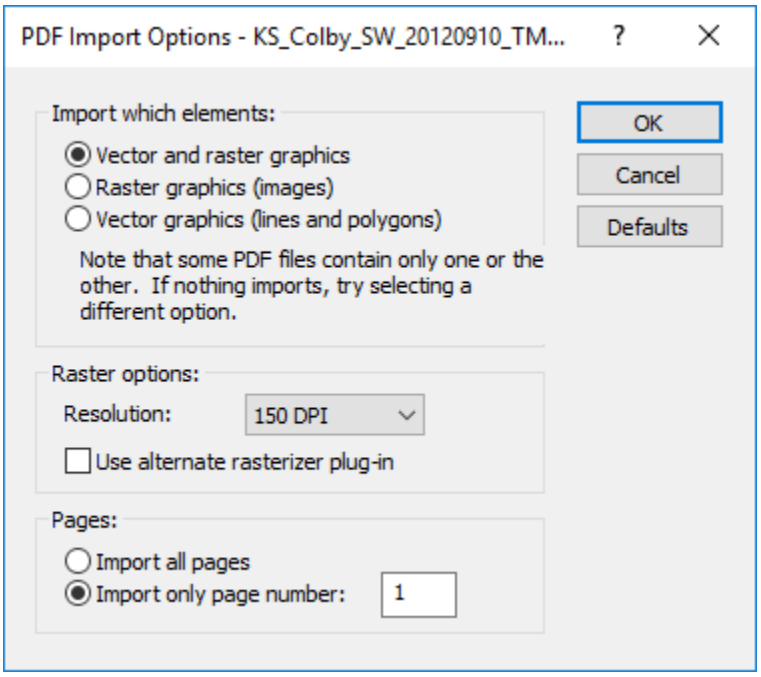
GeoPDF files can be imported as a base map in **Surfer**.

PDF Import Options Dialog

The **PDF Import Options** dialog is displayed when importing PDF files. Vector graphics from the PDF file are imported as lines and polygons. Raster graphics from the PDF file are imported as images.

GeoPDF

When the PDF file is a GeoPDF file and contains coordinates, the PDF is imported with the georeferenced coordinate system.



*The **Import Options** dialog controls what is imported from the PDF file.*

Import Which Elements

Select which elements you wish to import from the PDF in the *Import which elements* section.

- Select *Vector and raster graphics* to import both vector and raster graphics from the file.
- Select *Raster graphics (images)* to import only images from the file.
- Select *Vector graphics (lines and polygons)* to import only polylines and polygons from the file.

A PDF file may not contain both graphic types. Selecting *Vector graphics (lines and polygons)* when the PDF file contains only raster graphics will result in an empty import. If you are unsure which graphics types are present in the PDf file, select *Vector and raster graphics*.

Raster Options

The *Raster options* section includes the import options for raster graphics.

Resolution

The *Resolution* option controls the resolution of the imported images. Available options are 50, 75, 100, 125, 150, 200, 250, 300, 600, or *Custom* DPI. The higher the DPI, the clearer the image is when imported, but the larger the file size. The default is 150 DPI. When *Custom DPI* is selected, type a value into the *Render resolution* box to specify the resolution of the imported image. The *Render resolution Custom DPI* value must be between 50 and 1200.

Alternate Rasterizer

The *Use alternate rasterizer plug-in* option uses a different import filter to read in raster graphics. If the raster graphics appear incorrect after import when the *Use alternate rasterizer plug-in* option is cleared, import the PDF with the *Use alternate rasterizer plug-in* option selected.

Pages

When a PDF contains more than one page, the *Pages* options control which page is imported into the program. *Import all pages* imports each page in the PDF. *Import only page number* imports only the specified page number into the program. To change the page number to import, select the *Import only page number* option and type the page number to import.

PDF Adobe Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Resolution=300, WhichPage=7"

This would set the resolution of the images in the PDF file to 300 DPI and imports only page 7.

Option	Action	Default	Description
WhichPage	0 = all pages *n* = specific page number	0	The page number of the page to import. 0 imports all pages. Type the specific page number to import only that page.
Resolution	Value from 50 to 1200	150	The resolution of the imported PDF in DPI.
DoVector	0 = False 1 = True	1	Set to 1 to import vector graphics. Set to 0 to ignore vector graphics.

DoRaster	0 = False 1 = True	1	Set to 1 to import raster graphics. Set to 0 to ignore raster graphics.
UseAltPdfPlugIn	0 = False 1 = True	0	Set to 1 to use the alternate rasterizer for raster import. Set to 0 to use the default rasterizer for raster import.

Adobe Acrobat .PDF Raster Export Options Dialog

Adobe Acrobat Portable Document Format .PDF is a file format used for document exchange. PDF is used for representing two-dimensional documents. PDF was created by Adobe Systems.

Use a raster .PDF to export the entire **Surfer** project as an image in .PDF format. Use the **Export Options** dialog to specify the *Image size in pixels, Dots per inch, Color format*, and to *Maintain aspect ratio*. The higher resolution image you select, the larger the .PDF file will be. All images, lines, text, maps, etc. will be exported into the single image object in the raster .PDF file. Raster objects lose quality at low resolutions or when you zoom in. The raster .PDF output will look exactly like your original project.

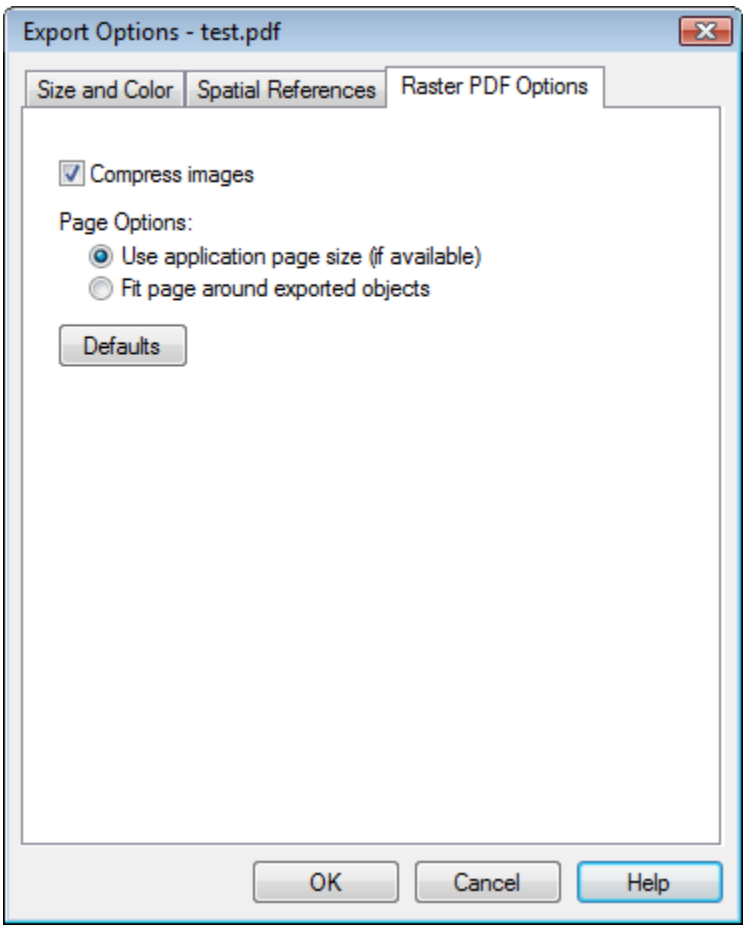
GeoPDF

When a map with a defined coordinate system is exported to a Raster PDF file format, the georeference options for the GeoPDF are changed on the Scaling and Spatial References pages in the **Export Options** dialog. The *Internal file format* option must be checked on the **Spatial References** page to create the GeoPDF.

An add-on, such as TerraGo toolbar, may be needed to view the GeoPDF options in Adobe's Acrobat or PDF Reader programs. The map layer and map must have an assigned coordinate system in **Surfer** prior to the export.

Export Options Dialog (for Raster PDF)

When using the **File | Export** command to export to a PDF (Raster) file, the **Export Options** dialog allows you to specify the Size and Color options and Spatial Reference options. In addition, the **Raster PDF Options** page is available to compress the image in the .PDF file.



*Check the Compress images option on the **Raster PDF Options** page to make the file size for raster .PDF files smaller.*

Compress Images

Check the box next to *Compress images* to apply .PDF compression to the images in the .PDF file. This produces smaller raster .PDF files with no loss in quality. When this option is unchecked, .PDF files will be larger.

Page Options

The *Page Options* section controls the page size in the exported .PDF file. Select the *Use application page size (if available)* to use the paper size defined in the Page Setup dialog. If *Fit page around exported objects* is selected, the page size in the .PDF file will be only as large as the objects being exported.

When the *Use application page size (if available)* is selected and objects extend beyond the page defined in the **Page Setup** dialog, the objects are clipped and not displayed in the PDF file.

Adobe Acrobat .PDF Vector Export Options Dialog

Adobe Acrobat Portable Document Format .PDF is a file format used for document exchange. PDF is used for representing two-dimensional documents. PDF was created by Adobe Systems. The vector .PDF export supports solid pattern, image, and gradient patterns.

Use a vector PDF to export the layers in the project as layers in the PDF. The layer visibility can be toggled on or off when viewing the PDF.

Use a vector .PDF to export the vector objects in the project (lines and text) as vector objects in the PDF. This makes the objects clear and sharp. Vector objects do not lose quality when you zoom in. It doesn't matter how much you zoom in, the line and text will always look crisp.

The images in your project will also export as images in the vector .PDF. Vector .PDF files are smaller than Raster .PDF files and are usually higher quality. The drawback is that some fill patterns are not supported, so the output may not look exactly the same as your original project.

Objects that are transparent or partially transparent in **Surfer** are exported with the transparency enabled in the .PDF.

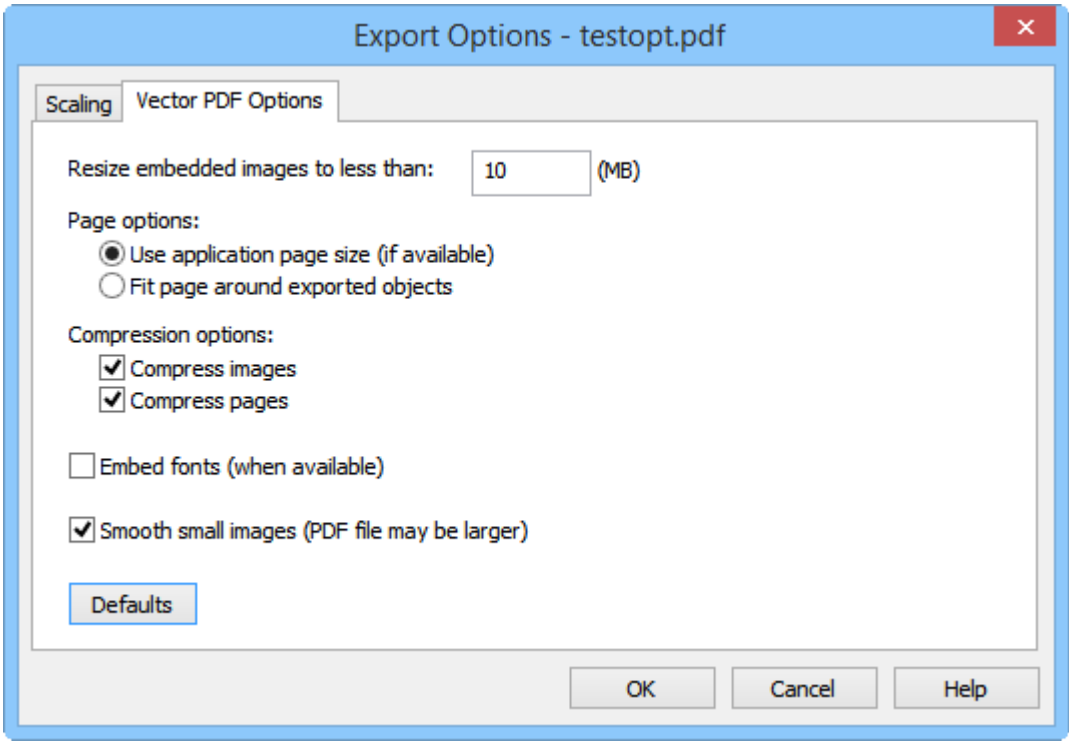
GeoPDF

When a map with a defined coordinate system is exported to a Vector PDF file format, a GeoPDF file is created automatically. The scaling source for the GeoPDF is changed on the Scaling page in the **Export Options** dialog.

An add-on, such as TerraGo toolbar, may be needed to view the GeoPDF options in Adobe's Acrobat or PDF Reader programs. The map layer and map must have an assigned coordinate system in **Surfer** prior to the export.

Vector PDF Options Page

In addition, the **Vector PDF Options** page is available to resize and compress image and to include the page size in the .PDF file.



*Set image compression and page size options on the **Vector PDF Options** page in the **Export Options** dialog.*

Resize Embedded Images

The *Resize embedded images to less than* option specifies the size in megabytes that an embedded image is resized to during export. Enter a value in the *MB* box. If an exported image exceeds this size, its resolution will be reduced so it doesn't exceed the designated maximum size. Increase this value to get better looking images at the expense of larger export files. The default value of *Resize embedded images to less than* option is 10MB.

As long as there is no shear, perspective, or clipping, PDF text entities will be exported as text. This means that the PDF text entities will be sized and oriented similar to the text objects in the application document. When shear, perspective, or clipping occur, the text is exported as solid polygons. Shear occurs when the character glyphs are not perpendicular to the text baseline. Perspective occurs when the height of glyphs in the text string are not all the same, as in a 3-D view where the glyphs are smaller the farther they are from the observer. Clipping occurs when part of the text object is partially inside and partially outside the map limits.

Page Options

The *Page Options* section controls the page size in the exported .PDF file. Select the *Use application page size (if available)* to use the paper size defined in the Page Setup dialog. If *Fit page around exported objects* is selected, the page size in the .PDF file will be only as large as the objects being exported.

When the *Use application page size (if available)* is selected and objects extend beyond the page defined in the **Page Setup** dialog, the objects are clipped and not displayed in the PDF file.

Compression Options

The *Compression Options* section contains options that can make the PDF file size smaller.

Compress Images

Check the box next to *Compress images* to apply .PDF compression to the images in the .PDF file. This produces smaller vector .PDF files with no loss in quality. When this option is unchecked, .PDF files will be larger.

Compress Pages

Check the box next to *Compress pages* to apply .PDF compression to the entire page stream. This produces significantly smaller vector .PDF files with no loss in quality. When this option is unchecked, .PDF files will be larger.

Embed Fonts

Check the box next to *Embed fonts (when available)* to store the fonts inside the .PDF file. This produces a larger vector .PDF file, but the font in the .PDF file will exactly match the font in the **Surfer** file. When this option is unchecked, the .PDF file will be smaller, but the text may not appear the same in the .PDF file as it does in the **Surfer** file. Some fonts cannot be embedded. A different font is substituted when this is the case.

Smooth Small Images

Surfer uses bilinear interpolation to smooth the display of reduced large images and enlarged small images. Adobe Reader uses bilinear interpolation to smooth the display of a reduced large image, however it does not smooth the display when enlarging small images. Therefore some small images may appear "blocky" in a PDF output when compared to the display in the **Surfer** program.

Click the *Smooth small images (PDF file may be larger)* check box to smooth small images in the output PDF file. The *Smooth small images* option uses bilinear interpolation to upscale images less than 1MB by 2x, 4x, or 8x its original size up to, but not greater than, 1MB.

Defaults

Click the *Defaults* button to return the options to the default values.

Scaling Page

See Scaling

Adobe Acrobat Portable Document Format .PDF (Vector) Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options.

Option	Action	Default	Description
MaxBitmapSizeInMB	N	10	This option specifies the largest size allowed for an individual image in the PDF file, in Megabytes. Any exported bitmap larger than this size, will

			have its resolution reduced so it does not exceed the maximum size.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.
FileURY	N.N		Set scaling rectangle upper right Y value.
CompressImages	0 = False 1 = True	1	Compresses images in the PDF to reduce file size.
CompressPages	0 = False 1 = True	1	Compress page streams in the PDF to reduce file size.
EmbedFonts	0 = False 1 = True	0	Embed fonts in the PDF file. When set to 1, the file size increases, but the text in the PDF looks identical to the **Surfer** file.
FitPage	0 = Use application page size 1 = Fit page around exported objects	0	Sets the size of the PDF to the application page size, if true. Otherwise, PDF page size is set to the size of the exported objects.
SmoothSmallImages	0 = False 1 = True	1	Upscales the image resolution, and therefore file size, for small images less than 1MB. This smooths the display of small images in the output PDF.

PLT Golden Software PlotCall File Description

Surfer can import Golden Software PlotCall .PLT files.

PlotCall files .PLT contain line graphics designed to be output on pen plotters. The curve, point and text objects of a PlotCall file are imported into **Surfer** with the **File | Import** command. There is no capability to export PlotCall files.

In the PlotCall file, each pen used is assigned a number. There can be up to 16 pens used in a PlotCall file.

PlotCall files can be either ASCII files (i.e., they can be edited with a text editor or word processor) or binary files (can't be edited) containing commands. Each command occupies one record and begins with a two-letter operation code (op-code) to determine its function. The currently supported op-codes are:

Op-code	Function
MA	Move Absolute
PA	Plot Absolute
TR	Translate
SC	Scale
PS	Plot String
SS	Set Symbol Set
RO	Rotate
PI	Pivot
SP	Select Pen

The general format of most commands is "op-code X,Y" where X and Y are coordinate values measured in inches. See Golden Software's PlotCall manual for a detailed description of each command. PlotCall files are usually produced by Golden Software's DOS applications, or by special user-written programs on PCs or mainframes.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

See PLT Golden Software PlotCall Import Automation Options

Golden Software PlotCall .PLT Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,Pen3=0;0;255;1;0.05"

This would first set all import options to their default values, then set the parameters for lines which are drawn with Pen #3. The lines will be Blue, dashed and 0.05 inches wide.

Option	Action	Default
Defaults=1	Set all options to their default values	No
ForgetOptions=1	Don't remember options for later use	No
PenN=R;G;B;S;W;N	Lines drawn with Pen #N are: Red intensity "R" (0-255) Green intensity "G" (0-255) Blue intensity "B" (0-255) Style "S" (See below) Width "W" inches Contain "N" custom segments	See below 0

PlotCall files can specify up to 16 pens (1-16). The default width is 0.0 (thinest line possible but still visible). The default line attributes associated with each pen are shown in the following table:

Pen #	R;G;B Values	Color	Style
1	0;0;0	Black	0 (Internal Solid)
2	0;0;255	Blue	0 (Internal Solid)
3	0;255;0	Green	0 (Internal Solid)
4	0;255;255	Cyan	0 (Internal Solid)
5	255;0;0	Red	0 (Internal Solid)
6	255;0;255	Magenta	0 (Internal Solid)
7	255;255;0	Yellow	0 (Internal Solid)
8	255;255;255	White	0 (Internal Solid)
9	0;0;0	Black	1 (Internal Dashed)
10	0;0;255	Blue	1 (Internal Dashed)
11	0;255;0	Green	1 (Internal Dashed)
12	0;255;255	Cyan	1 (Internal Dashed)
13	255;0;0	Red	1 (Internal Dashed)
14	255;0;255	Magenta	1 (Internal Dashed)
15	255;255;0	Yellow	1 (Internal Dashed)
16	255;255;255	White	1 (Internal Dashed)

The different line styles are:

Style	Line
-1	Custom (See below)
0	Internal Solid
1	Internal Dashed
2	Internal Dotted
3	Internal Dash-Dot
4	Internal Dash-Dot-Dot

Internal Style

The five internal styles indicate a generic style and leave it to the display device to decide how best to render the line.

Custom Style

A custom style allows (and requires) you to specify the length of the strokes and gaps used when rendering the line.

PLY Stanford Polygon File Description

PLY is a computer file format known as the *Polygon File Format* or the *Stanford Triangle Format*. The format was designed to store three dimensional data from 3D scanners. It supports a relatively simple description of a single object as a list of nominally flat polygons. A variety of properties can be stored including: color and transparency, surface normals, texture coordinates and data confidence values. The format permits one to have different properties for the front and back of a polygon.

File Format

Files are organized as a header, that specifies the elements of a mesh and their types, followed by the list of elements itself, usually vertices and faces eventually other entities such as edges, samples of range maps, and triangle strips can be encountered.

The header of both ASCII and binary files is ASCII text. Only the numerical data that follows the header is different between the two versions. The header always starts with the line 'ply'. The header helps to identify this as a genuine 'PLY' file.

```
ply
```

The second line indicates which variation of the PLY format this is. It should be one of:

```
format ascii 1.0
format binary_little_endian 1.0
format binary_big_endian 1.0
```

Future versions of the standard will change the revision number at the end - but 1.0 is the only version currently in use.

Comments may be placed in the header by using the word 'comment' at the start of the line. Everything from there until the end of the line should then be ignored. eg:

```
comment This is a comment!
```

The 'element' keyword introduces a description of how some particular data element is stored and how many of them there are. Hence, in a file where there are 12 vertices, each represented as a floating point (X,Y,Z) triple, one would expect to see:

```
element vertex 12
property float x
property float y
property float z
```

Other 'property' lines might indicate that colours or other data items are stored at each vertex and indicate the data type of that information. Regarding the data type there are two variants, depending on the source of the ply file, the type can be specified with one of *char uchar short ushort int uint float double*, or one of *int8 uint8 int16 uint16 int32 uint32 float32 float64*. For an object with ten polygonal faces, one might see:

```
element face 10
property list uchar int vertex_index
```

The word 'list' indicates that the data are a list of values - the first of which is the number of entries in the list (represented as a 'uchar' in this case) and each list entry is (in this case) represented as an 'int'.

At the end of the header, there must always be the line:

```
end_header
```

In the ASCII version of the format, the vertices and faces are each described one to a line with the numbers separated by white space. In the binary version, the data are simply packed closely together at the 'endianness' specified in the header and with the data types given in the 'property' records. For the common "property list..." representation for polygons, the first number for that element is the number of vertices that the polygon has and the remaining numbers are the indices of those vertices in the preceding vertex list.

Import

Stanford Polygon (.PLY) files can be imported into **Surfer**. The geometry (shape) information from PLY models is the only item imported. Any surface/material properties of the PLY model are ignored. No import options dialog is displayed.

Import Automation Options

No import options are available.

Reference

This implementation is based on the file format described in Greg Turk's 1998 "The PLY Polygon File Format" document, which can be found on the Internet.

PNG Portable Network Graphic File Description

Surfer imports and exports Portable Network Graphic .PNG files.

Portable Network Graphics .PNG is an image format that employs lossless data compression. PNG was created to improve upon and replace GIF (Graphics Interchange Format) as an image-file format not requiring a patent license.

PNG supports palette-based, grayscale, or RGB images. PNG does not support CMYK color spaces.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import automation options are available.

Export Options Dialog

See Size and Color, Spatial References, and PNG Options

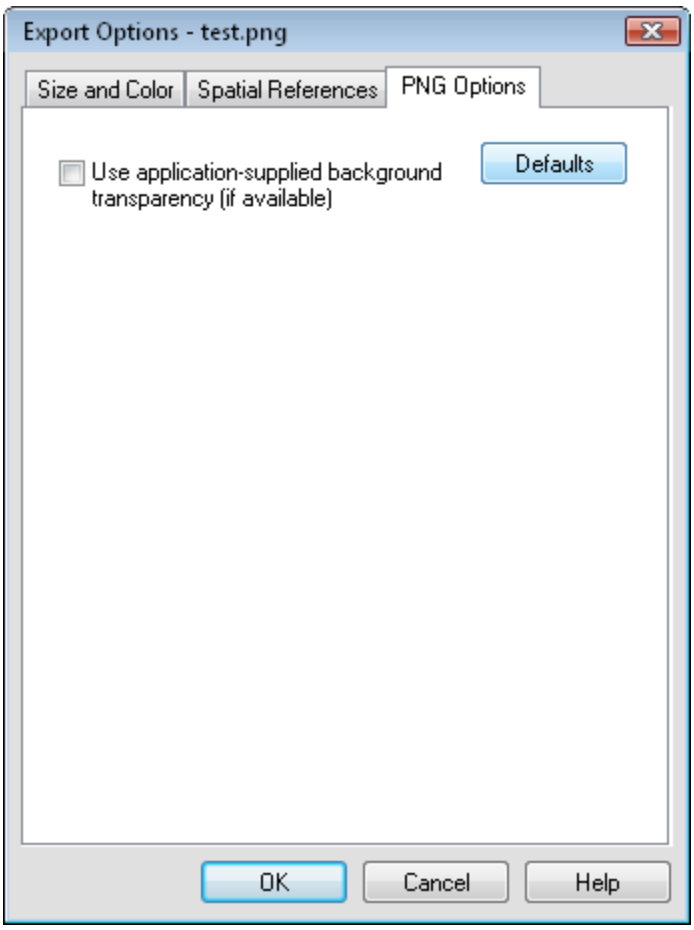
Export Automation Options

See Bitmap Export Automation Options

.PNG Export Options Dialog

Portable Network Graphics .PNG is an image format that employs lossless data compression. PNG files support transparency.

Use the **Export Options** dialog to specify the Size and Color and Spatial References options. Set the transparency options on the **PNG Options** page. To export the .PNG file with all background areas transparent, click on the **PNG Options** tab in the **Export Options** dialog. Check the *Use application-supplied background transparency (if available)* option. The .PNG file will have transparent background areas.



*Specify the transparency setting on the **PNG Options** page in the **Export Options** dialog.*

PNM Portable Any Map File Description

Surfer imports Portable Any Map .PNM, Portable Pixel Map .PPM, Portable Gray Map .PGM, and Portable Bitmap .PBM files. **Surfer** exports .PNM files.

Although there are both binary and ASCII variants of this file format, the current version of the filter always writes binary files when exporting.

The portable pixmap file format .PPM, the portable graymap file format .PGM and the portable bitmap file format .PBM specify rules for exchanging graphics files. They provide very basic functionality and serve as a least-common-denominator for converting pixmap, graymap, or image files between different platforms. Several applications refer to them collectively as the PNM format (portable anymap).

The .PGM and .PPM formats (both ASCII and binary versions) have an additional parameter for the maximum value in a line between the X and Y dimensions and the actual pixel data.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import automation options are available.

Export Options Dialog

See Size and Color and Spatial References

Export Automation Options

See Bitmap Export Automation Options

RAS, SUN Sun Raster Image (SUN, RAS) File Description

Surfer imports and exports Sun Raster .RAS and .SUN image files.

The Sun Raster .RAS and .SUN file format originated at Sun Microsystems and is a common file format for storing bitmap images on UNIX and Solaris workstations.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import automation options are available.

Export Options Dialog

See Size and Color and Spatial References

Export Automation Options

See Bitmap Export Automation Options

RAW Binary Lattice File Description

Surfer imports and exports uniform lattices from raw (untyped) binary files. The lattice dimensions and several other aspects of the file format can be controlled.

Since this filter works with the headerless/untyped files, it is important to knows what options are appropriate for the data in question.

The RAW file is a simple headerless binary file consisting of lattice nodes arranged according to the export options that were selected when the file was created. No specific file name extension is required, but .RAW and .BIN are commonly used for these files.

Loading .ERS Files

.RAW files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

See RAW Binary Lattice Files Import Options Dialog.

Import Automation Options

See RAW Binary Lattice Files - Import Automation Options.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Editor to export .ERS files. Alternatively, the ER Mapper .ERS file format is available when prompted to save a grid file.

Export Options Dialog

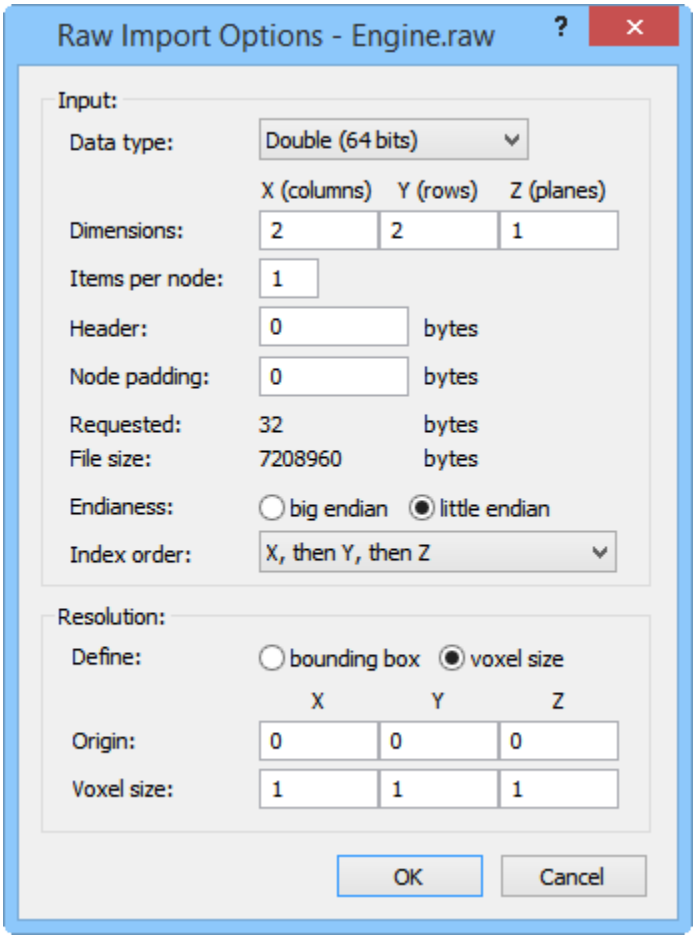
See RAW Binary Lattice Files - Export Options Dialog.

Export Automation Options

See RAW Binary Lattice Files - Export Automation Options.

RAW Binary Lattice File - Import Options Dialog

The **Raw Import Options** dialog is displayed when importing a RAW Binary Lattice file.



Specify Input and Resolution options in the ***Raw Import Options*** *dialog.*

Data Type

The *Data type* option allows you to select the type of data to be read from the import file and how many data components of this type should be read for each lattice node.

Dimensions

The *Dimensions* option allows you to specify the three dimensions of the lattice in the *X (columns)*, *Y (rows)*, and *Z (planes)* boxes. The values must be positive.

Items per Node

The *Items per node* value displays the number of components that are associated with each point in the .RAW file.

Header

The *Header* option allows you to specify how many bytes of non-data should be skipped at the beginning of the file before reading the first item of data. Enter a value in the *Header* box.

Node Padding

The *Node padding* option allows you to specify how many bytes of non-data should be skipped between each data item. This value is zero for most files. Enter a value in the *Node padding* box.

Requested Bytes (Static control)

The *Requested bytes* option displays how many bytes would be read from the file based on your selection in the above controls.

File Size (Static control)

The *File size* option displays the actual size of the file in bytes.

Endianess

The *Endianess* option allows you to specify whether the import data are in *little-endian* or *big-endian* byte order. Click the button to left of *little-endian* or *big-endian* to select a byte order. Integers are usually stored as a sequence of bytes, so that the encoded value can be obtained by simple concentration. The two most common are litte-endian and big-endian. Little-endian refers to increasing numeric significance with increasing memory addresses. Big-endian is the opposite and refers to "big end first".

Index Order

The *Index order* option allows you to specify whether the precedence of the lattice data in the file. The first option is indexed fastest and the last option is indexed slowest. So, *X, then Y, then Z* would have X indexed fastest and Z slowest. *Z, then Y, then X* would have Z indexed fastest and X slowest.

Resolution

The *Resolution* controls allows you to specify the size (in application display units) of the imported lattice. Two methods are available:

- *bounding box*, which allows you to specify the extents of the lattice's bounding box (from which the voxel size is calculated); and
- *voxel size*, which allows you to specify the extents of each voxel (from which the bounding box is calculated).

Min and Max Coord

When *bounding box* is set for the *Resolution*, the *Min coord* and *Max coord* boxes become available. Type in the values at the bottom left corner (*Min coord)* and upper right corner (*Max coord)* for the bounding box.

Origin and Voxel Size

When *voxel size* is set for the *Resolution*, the *Origin* and *Voxel size* options become available. The origin is the location from which the first voxel is set. The *Voxel size* is the size of the voxels in map units.

RAW Binary Lattice Import Automation Options

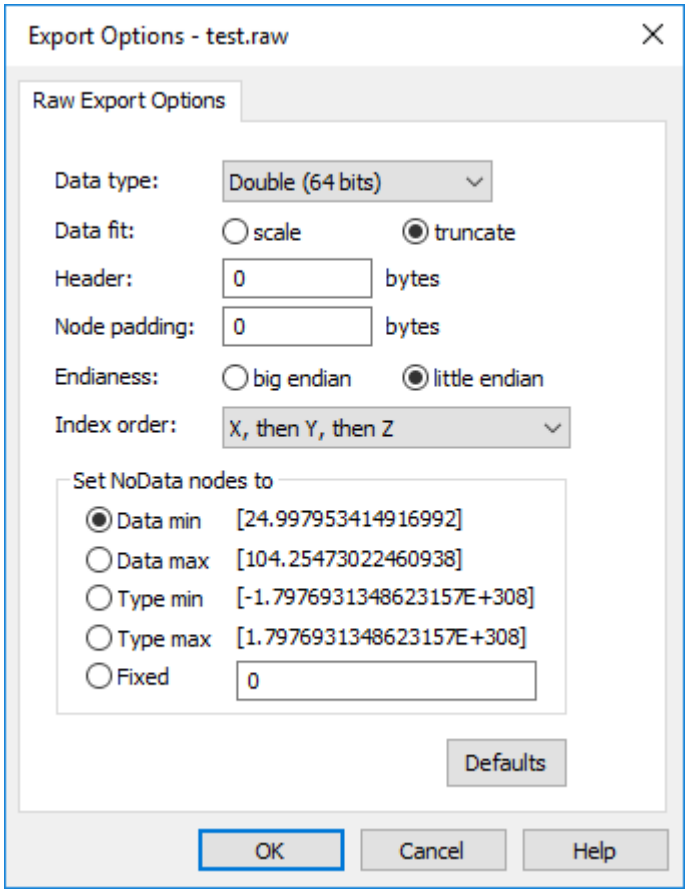
Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options.

Option	Action	Default
DataType	Data type for data nodes in file: 0=Signed 8 bits (character) 1=Unsigned 8 bits (BYTE), 2=Signed 16 Bits (short) 3=Unsigned 16 Bits (short) 4=Signed 32 Bits(integer) 5=Unsigned 32 Bits (integer) 6=Signed 64 Bits 7=32-Bit float 8=64-Bit Double	1
SkipAfter	Number of data elements to read for each node.	No
ItemsPerNode	Convert area objects to closed curves	No
DimX	Number of x columns in lattice.	2
DimY	Number of y rows in lattice.	2
DimZ	Number of z columns (planes) in lattice.	1
StartOffset	Offset of first data node in file.	No
BigEndian	True if data are in Motorola MSB (big endian)format. False if data are in Intel LSB (little endian) format.	False
IndexOrder	Index order: 0="X, then Y, then Z" 1="Y, then X, then Z" 2="Z, then X, then Y" 4="X, then Z, then Y" 5="Y, then Z, then X" 6="Z, then Y, then X". 0	0
XMin	Smallest X value in grid.	No
YMin	Smallest Y value in grid.	No
ZMin	Smallest Z value in grid.	No
XMax	Largest X value in grid.	No
YMax	Largest Y value in grid.	No
ZMax	Largest Z value in grid.	NoNo
XSpacing	Spacing between x nodes in grid.	No

YSpacing	Spacing between Y nodes in grid.	No
ZSpacing	Spacing between Z nodes in grid.	No
HasSpacing	True if RawMin and RawSpacing are valid. False if Raw Min and RawMax are valid.	False

RAW Binary Lattice File - Export Options Dialog

The **Raw Export Options** dialog is displayed when exporting a RAW Binary Lattice file.



*Specify the export options in the **Raw Export Options** dialog.*

Data Type

The *Data Type* option allows you to select the type of data that is written to the export file.

Data Fit

The *Data fit* option allows you to specify whether the export source data are scaled to truncated to fit the export data type chosen above. This is particularly important when the source data are larger (i.e. doubles, long integers) than the export data type (i.e. floats, short integers).

Header Bytes

The *Header* option allows you to specify how many bytes of non-data should be skipped at the beginning of the file before writing the first item of data. Enter a value in the *Header* box.

Node Padding

The *Node padding* option allows the user to specify how many bytes of zeros should be written between each data item. This value is zero for most files. Enter a value in the *Node padding* box.

Endianess

The *Endianess* option allows you to specify whether the export data are in *little-endian* or *big-endian* byte order. Click the button to left of *little-endian* or *big-endian* to select a byte order. Integers are usually stored as a sequence of bytes, so that the encoded value can be obtained by simple concentration. The two most common are litte-endian and big-endian. Little-endian refers to increasing numeric significance with increasing memory addresses. Big-endian is the opposite and refers to "big end first".

Index Order

The *Index order* option allows the user to specify whether the precedence of the lattice data in the file is in x,y,z (x fastest) or z,y,x (z fastest) order.

Set NoData Nodes To

The controls in the *Set NoData nodes to* section determine how NoData values are treated in the export file.

Data Min

Choose *Data min* to export NoData nodes with the indicated minimum value (varies depending on contents of lattice).

Data Max

Choose *Data max* to export NoData nodes with the indicated maximum value (varies depending on contents of lattice).

Type Min

Choose *Type min* to export NoData nodes with the indicated minimum value for the lattice's data type (usually zero).

Type Max

Choose *Type max* to export NoData nodes with the indicated maximum value of the lattice's data type.

Fixed

Choose *Fixed* to export NoData nodes with some other value, and enter the desired value in the adjacent edit control.

Defaults

Click the *Defaults* button to return all settings on the page to the default values.

RAW Binary Lattice File .RAW, .BIN Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" BlankMethod=BM_FIXED, FixedBlankValue=1.70141E+038"

This would indicate that the NoData value for a lattice or grid node is a custom value, and that value is equal to **Surfer's** NoData value.

Option	Action	Default
DataType	Data type for data nodes in file: 0=Signed 8 bits (character) 1=Unsigned 8 bits (BYTE), 2=Signed 16 Bits (short) 3=Unsigned 16 Bits (short) 4=Signed 32 Bits(integer) 5=Unsigned 32 Bits (integer) 6=Signed 64 Bits 7=32-Bit float 8=64-Bit Double	1
SkipAfter	Use application-supplied scaling parameters	No
StartOffset	Offset of first data node in file.	0
BigEndian	True if data are in Motorola MSB (big endian)format. False if data are in Intel LSB (little endian) format.	False
IndexOrder	Index order: 0="X, then Y, then Z" 1="Y, then X, then Z" 2="Z, then X, then Y" 4="X, then Z, then Y" 5="Y, then Z, then X" 6="Z, then Y, then X".	0
BlankMethod	Method for assigning values to NoData nodes: BM_DATA_MIN = Minimum value of the lattice BM_DATA_MAX = Maximum value of the lattice. BM_TYPE_MIN = Minimum value of the inherent data type. BM_TYPE_MAX = Maximum value of the inherent data type.	BM_TYPE_MAX

	BM_FIXED = A user specified value	
FixedBlankValue	Used if BlankMethod is BM_FIXED	No

RGB Silicon Graphics (RGB) Image File Description

Surfer imports and exports Silicon Graphics Image .RGB, .RGBA, and .BW files.

This file format .RGB, .RGBA, .BW originated on Silicon Graphics workstations and is/was used in a variety of high-end imaging applications, both Unix- and Windows-based. The file name extension is sometimes used to indicate the format of the image contained in the file, but is not required to do so. The extensions are typically .BW for black and white images, .RGB for 24-bit color images, and .RGBA for 32-bit color images with an alpha channel.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import automation options are available.

Export Options Dialog

See Size and Color and Spatial References

Export Automation Options

See Bitmap Export Automation Options

RST Idrisi Raster Image [.RST, .IMG] File Description

Surfer imports the Idrisi Raster Image .RST format. It requires the associated .RCS file to be present in the same directory. Sometimes there is also an associated .REF file, depending on the refsystem member of the IDR_RDC struct inside the file.

File Format

IDRISI starts in the upper-left corner (row 0/column 0), then advances column by column and row by row.

Idrisi grids have either a .RST or .IMG extension. Every data set come with a set of files including a .DOC or .RDC file. **Surfer** supports all Idrisi grid types except the RGB type.

The associated .RDC file must be present in the same directory as the .RST. Sometimes there is also an associated .REF file, depending on the refsystem member of the IDR_RDC struct.

- The old IDrisi data sets consist of paired .IMG/.DOC.
- The New IDrisi data sets consist of paired .RST/.RDC.

Allowable .RST, .IMG formats are ASCII, Binary and Packed Binary forms. However, most routines in the IDRISI application expect the binary format only.

Loading .IMG and .RST Files

Idrisi Raster Image .IMG and .RST files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

RT* TIGER/Line File Description

Surfer can import TIGER/Line .RT* files. A TIGER/Line file dataset usually includes many .RT* files in a single directory. TIGER/Line files are extracts of selected geographic and cartographic information from the Census TIGER® database. TIGER/Line vector files include point, polyline, and polygon features that represent land attributes and political or statistical areas, such as boundaries, census tracts, roads, cities, rivers, etc. TIGER/Line files contain only geographic information, but can be linked to United States Census Bureau demographic data for analysis.

The TIGER/Line file format is a text-based format. The TIGER/Line file format was replaced by a shapefile specification in 2008.

Importing TIGER/Line Files

Create a map from TIGER/Line files with the Home | New Map | Base command. Select any TIGER/Line .RT* file in the **Import** dialog to load the dataset.

Import Options Dialog

There are no TIGER/Line .RT* import options.

Import Automation Options

There are no TIGER/Line .RT* import automation options.

References: *2002 TIGER/Line Technical Documentation*, prepared by the U.S. Census Bureau, 2003. www.census.gov.

SEG SEG-SP1 Data File Format

The .SP1 SEG standard data exchange file format is a format widely used in the geophysical industry to exchange data for shotpoint locations for seismic surveying. **Surfer** currently imports and exports version SP1 file formats. Shotpoint locations are usually those computed locations which are the best estimates of where actual data points are located in the field. The locations are derived from a variety of complex field data. This format is applicable for both land and marine

locations and has sufficient flexibility for use in 3D seismic surveys. This can include gravity, magnetic data, or other data about each shotpoint. For additional information on specifics about each file format, refer to the Society for Exploration Geophysicists.

SP1 files can be opened by clicking the File | Open command or by clicking the File | Import command in a worksheet window. Alternatively, the SP1 format can be used to create a base map or post map in the plot window. When importing into **Surfer**, the line name, shotpoint number, and non-zero Z value is imported and displayed in attribute columns for each point.

The SEG and SP1 file formats are fairly flexible and can include data in latitude and longitude or in easting, northing, and depth/elevation formats. When loading latitude/longitude data, the WGS84 datum is assumed.

Import Options Dialog

See SEG SP1 Import Options

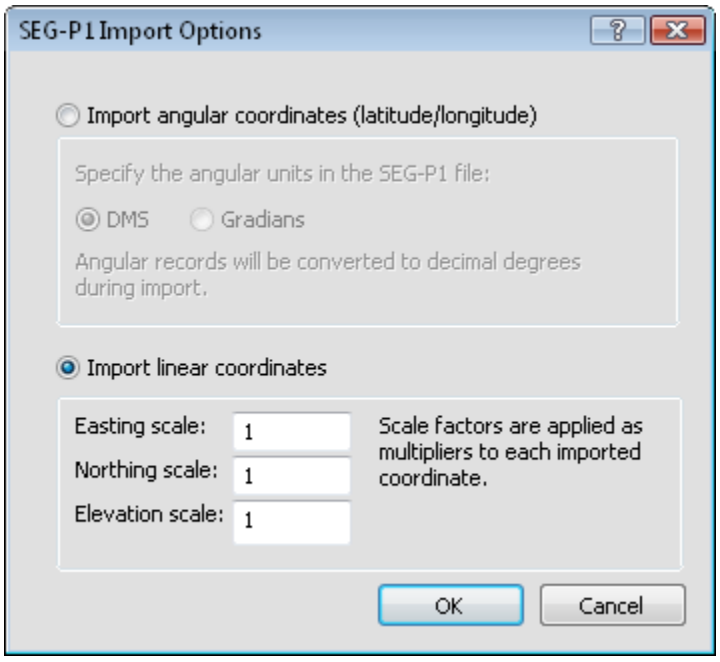
Export Options Dialog

See Scaling page

SEG P1 Import Options Dialog

The **SEG-P1 Import Options** dialog allows you to specify options which determine how information in the file is imported.

A SEG P1 file is a collection of point locations, usually used in geophysical shotpoints.



*Specify the SP1 import options in the **SEG-P1 Import Options** dialog.*

Coordinate Format

Select *Import angular coordinates (latitude/longitude)* if the data in the SP1 or SEG file is in latitude and longitude. Select *Import linear coordinates* if the data in the SP1 or SEG file is in a Cartesian coordinate system or an unknown linear coordinate system. Units can be in degrees, gradians, meters, feet, or any unknown units.

DMS or Gradians

When the *Import angular coordinates (latitude/longitude)* is selected, the *Specify the angular units in the SEG-P1 file* option becomes available. If the file contains degrees, minutes, and seconds, select *DMS*. If the file contains gradians, select *Gradians*.

Scale Factors

When the *Import linear coordinates* is selected, the *Easting scale, Northing scale,* and *Elevation scale* options are available. Highlight the existing value and type the new scale factor value in the box. Scale factors are multiplied by the values in the import file to get appropriate coordinates. This might be necessary if the *Easting* and *Northing* are in one set of units and the *Elevation* is in another set of units.

OK or Cancel

Click *OK* to continue importing the tabular SP1 data into a new table or the vector format SP1 file into a new base map. Click *Cancel* to quit importing the file.

SHP Esri Shapefile File Description

Esri Shapefiles are typically usually used in **Surfer** as base maps. Compressed (ZIP, TAR, TAR.GZ, TGZ) Shapefiles can be imported directly into **Surfer**. However if the compressed folder contains more than one Shapefile, only the first Shapefile is imported.

Esri Shapefiles are in a binary file format (i.e., they can't be created or modified with a text editor or word processor) that is compatible with Arc/Info, Arc/View, and other Esri application programs. This format is used to store spatial information including boundary objects such as areas, curves, and points. Spatial information is only concerned with the location of objects in space (i.e., their coordinates) and not with their attributes (such as line or fill style, marker symbol used, text labels, etc.).

Four types of files are produced with each export:

Filename Extension	Description
.SHP	Contains the coordinates of each object in the drawing.
.SHX	Contains the file offset of each object in the .SHP file.
.DBF	Contains the attribute text associated with each object in the .SHP file.
.CPG	Contains the Unicode code page number.

In each of the .SHP, .SHX, and .DBF files, the shapes in each file correspond to each other in sequence. That is, the first record in the .SHP file corresponds to the first record in the .SHX and .DBF files, and so on. The .SHP and .SHX files have various fields with different endianness, so as an implementor of the file formats you must be very careful to respect the endianness of each field and treat it properly.

Overview

A shapefile is a digital vector storage format for storing geometric location and associated attribute information. This format lacks the capacity to store topological information. The shapefile format was introduced with ArcView GIS version 2 in the beginning of the 1990s. It is now possible to read and write shapefiles using a variety of free and non-free programs.

Shapefiles are simple because they store primitive geometrical data types of points, lines, and polygons. These primitives are of limited use without any attributes to specify what they represent. Therefore, a table of records will store properties/attributes for each primitive shape in the shapefile. Shapes (points/lines/polygons) together with data attributes can create infinitely many representations about geographical data. Representation provides the ability for powerful and accurate computations.

While the term "shapefile" is quite common, a "shapefile" is actually a set of several files. Three individual files are normally mandatory to store the core data that comprises a shapefile. There are a further eight optional files which store primarily index data to improve performance. Each individual file should conform to the MS DOS 8.3 file nameing convention (8 character file name prefix, fullstop, 3 character file name suffix such as shapefil.shp) in order to be compatible with past applications that handle shapefiles. For this same reason, all files should be located in the same folder.

Shapefiles deal with coordinates in terms of X and Y, although they are often storing longitude and latitude, respectively. While working with the X and Y terms, be sure to respect the order of the terms (longitude is stored in X, latitude in Y).

Mandatory files

.SHP - shape format; the feature geometry itself

.SHX - shape index format; a positional index of the feature geometry to allow seeking forwards and backwards quickly

.DBF - attribute format; columnar attributes for each shape, in dBase III format

Optional files

.PRJ - projection format; the coordinate system and projection information, a plain text file describing the projection using well-known text format

.SBN and .SBX - a spatial index of the features

.FBN and .FBX - a spatial index of the features for shapefiles that are read-only

.AIN and .AIH - an attribute index of the active fields in a table or a theme's attribute table

.IXS - a geocoding index for read-write shapefiles

.MXS - a geocoding index for read-write shapefiles (ODB format)

.ATX - an attribute index for the .dbf file in the form of shapefile.columnname.atx (ArcGIS 8 and later)

.SHP.XML- metadata in XML format

.CPG - file containing the single value code page to be used for ANSI to Unicode translation of attribute text in associated .DBF files.

Attributes

All attributes for all polyline, polygon, and symbol objects are automatically exported to all .SHP files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map. All attributes are automatically imported.

Shapefile shape format .SHP

The main file [.SHP] contains the primary geographic reference data in the shapefile. The file consists of a single fixed length header followed by one or more variable length records. Each of the variable length records includes a record header component and a record contents component. A detailed description of the file format is given in the Esri Shapefile Technical Description.[1] This format should not be confused with the AutoCAD shape font source format, which shares the .shp extension.

The main file header is fixed at 100 bytes in length and contains 17 fields; nine 4-byte (32-bit signed integer or int32) integer fields followed by eight 8-byte (double) signed floating point fields:

Bytes	Type	Endianness	Usage
0-3	int32	big	File code (always hex value 0x0000270a)
4-23	int32	big	Unused; five uint32
24-27	int32	big	File length (in 16-bit words, including the header)
28-31	int32	little	Version
32-35	int32	little	Shape type (see reference below)
36-67	double	little	Minimum bounding rectangle (MBR) of all shapes contained within the shapefile; four doubles in the following order: min X, min Y, max X, max Y
68-83	double	little	Range of Z; two doubles in the following order: min Z, max Z
84-99	double	little	Range of M; two doubles in the following order: min M, max M

The file then contains any number of variable-length records. Each record is prefixed with a record-header of 8 bytes:

Bytes	Type	Endianness	Usage
0-3	int32	big	Record number
4-7	int32	big	Record length (in 16-bit words)

Following the record header is the actual record:

Bytes	Type	Endianness	Usage
0-3	int32	big	Shape type (see reference below)
4-	-	-	Shape content

The variable length record contents depend on the shape type. The following are the possible shape types:

Value	Shape Type	Fields
0	Null shape	None
1	Point	X, Y
3	Polyline	MBR, Number of parts, Number of points, Parts, Points
5	Polygon	MBR, Number of parts, Number of points, Parts, Points
8	MultiPoint	MBR, Number of points, Points
11	PointZ	X, Y, Z, M
13	PolylineZ	*Mandatory*: MBR, Number of parts, Number of points, Parts, Points, Z range, Z array *Optional*: M range, M array
15	PolygonZ	*Mandatory*: MBR, Number of parts, Number of points, Parts, Points, Z range, Z array *Optional*: M range, M array
18	MultiPointZ	*Mandatory*: MBR, Number of points, Points, Z range, Z array *Optional*: M range, M array
21	PointM	X, Y, M
23	PolylineM	*Mandatory*: MBR, Number of parts, Number of points, Parts, Points *Optional*: M range, M array
25	PolygonM	*Mandatory*: MBR, Number of parts, Number of points, Parts, Points *Optional*: M range, M array
28	MultiPointM	*Mandatory*: MBR, Number of points, Points *Optional Fields*: M range, M array
31	MultiPatch	*Mandatory*: MBR, Number of parts, Number of points, Parts, Part types, Points, Z range, Z array *Optional*: M range, M array

In common use, shapefiles containing Point, Polyline, and Polygon are extremely popular. The "Z" types are three-dimensional. The "M" types contain a user-defined measurement which coincides with the point being referenced. Three-dimensional shapefiles are rather uncommon, and the measurement functionality has been largely superseded by more robust databases used in conjunction with the shapefile data.

Shapefile shape index format (.shx)

The shapefile index contains the same 100-byte header as the [.SHP] file, followed by any number of 8-byte fixed-length records which consist of the following two fields:

Bytes	Type	Endianness	Usage
0-3	int32	big	Record offset (in 16-bit words)
4-7	int32	big	Record offset (in 16-bit words)

Using this index, it is possible to seek backwards in the shapefile by seeking backwards first in the shape index (which is possible because it uses fixed-length records), reading the record offset, and using that to seek to the correct position in the [.SHP] file. It is also possible to seek forwards an arbitrary number of records by using the same method.

Shapefile attribute format .DBF

Attributes for each shape are stored in the xBase (dBase) format, which has an open specification.

Shapefile attribute format .CPG

file containing the single value code page to be used for ANSI to Unicode translation of attribute text in associated .DBF files.

Shapefile projection format .PRJ

The projection information contained in the [.PRJ] file is critical in order to understand the data contained in the [.SHP] file correctly. Although it is technically optional, it is most often provided, as it is not necessarily possible to guess the projection of any given points. The file is stored in well-known text (WKT) format.

Some typical information contained in the [.PRJ] file is:

- Geographic coordinate system
- Datum (geodesy)
- Spheroid
- Prime meridian
- Map projection
- Units used
- Parameters necessary to use the map projection, for example:
- Latitude of origin
- Scale factor
- Central meridian
- False northing
- False easting
- Standard parallels

Shapefile spatial index format (.sbn)

This is a binary spatial index file, which is used only by Esri software. The format is not documented, and is not implemented by other vendors. The [.SBN] file is not strictly necessary, since the [.SHP] file contains all of the information necessary to successfully parse the spatial data.

Limitations

Topology and shapefiles

Shapefiles do not have the ability to store topological information. ArcInfo coverages and Personal/File/Enterprise Geodatabases do have the ability to store feature topology.

Spatial representation

The edges of a polyline or polygon are defined using points, which can give it a jagged edge at higher resolutions. Additional points are required to give smooth shapes, which requires storing quite a lot of data compared to, for example, bézier curves, which can capture complexity using smooth curves, without using as many points. Currently, none of the shapefile types support bézier curves.

Data storage

Unlike most databases, the database format is based on older xBASE standard, incapable of storing null values in its fields. This limitation can make the storage of data in the attributes less flexible. In ArcGIS products, values that should be null are instead replaced with a 0 (without warning), which can make the data misleading. This problem is addressed in ArcGIS products by using Esri's Personal Geodatabase offerings, one of which is based on Microsoft Access.

Mixing shape types

Each shapefile can technically store a mix of different shape types, as the shape type precedes each record, but common use of the specification dictates that only shapes of a single type can be in a single file. For example, a shapefile cannot contain both Polyline and Polygon data. Thus, well (point), river (polyline) and lake (polygon) data must be kept in three separate files.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

See Esri Shapefile Import Automation Options

Export Options Dialog

See Esri Shapefile Export Options Dialog

Export Automation Options

See Esri Shapefile Export Automation Options and Scaling

Esri Shapefile .SHP Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

```
"Defaults=1,AreasToCurves=1"
```

This would first set all import options to their default values, then set the *AreasToCurves* option value to one, which would specify that any areas imported be converted to closed curves.

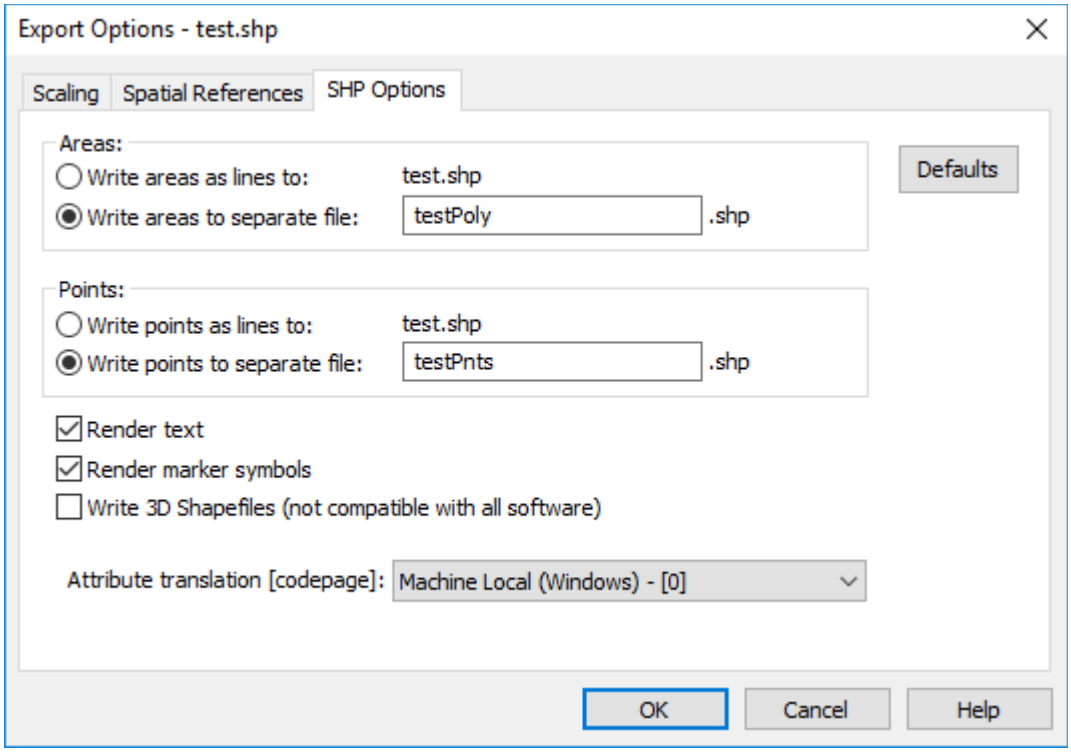
Option	Action	Default
Defaults=1	Set all options to their default values	No
ForgetOptions=1	Don't remember options for later use	No
AreasToCurves=1	Convert area objects to closed curves	No
PrimaryIDField=*n*	Specify that field number *n* of each record in the optional DBF database will be assigned to the primary ID string of each object that is imported from the SHP input file. If the SHP file is not accompanied by a DBF file, this option has no effect.	0
SecondaryIDField=*n*	Same as above, except applies to the secondary ID.	0
ImportCodePage	The number of the ANSI code page to use when importing Unicode data. Valid code page number in the range 0 through 65535. If a .CPG file exists in the directory with the same name as the .SHP file, the code page stored in the .CPG file will be used. If a .DBF file exists in the directory with the same name as the .SHP file, the .DBF file attribute data must be translated from ANSI text to Unicode representation. If no .CPG file exists, the Language ID value stored in the .DBF file's header determines the code page used. However, if the Language ID value is zero or invalid, the "Western European (Windows)" code page (1252) will be used.	""

Esri Shapefile .SHP Export Options Dialog

The **Export Options** dialog, **SHP Options** page allows you to specify options which determine how information in the file is exported.

Attributes

All attributes for all polyline, polygon, and symbol objects are automatically exported to all .SHP files. For contour maps, the Z value is exported as the "ZLEVEL" attribute for all polylines in the contour map.



*Specify the SHP file export options in the **Export Options** dialog.*

SHP Options Page

Note that an Esri Shapefile (.SHP) file may contain either line objects, area objects, or point objects, but not a combination of different object types.

Write Objects as Lines

All objects can be written as line objects. The *Write areas as lines* and *Write points as lines* options may be both selected, or either of the aforementioned controls may be selected individually, to write only the corresponding objects as lines. Alternatively, areas and/or points can be written to separate files by selecting the *Write areas to separate file* and/or *Write points to separate file* controls, in which case valid file name(s) must be typed into the adjacent edit control(s).

Render Text

By default, text is not output. Selecting the *Render text* control will instead cause text to be rendered as lines and/or areas as appropriate. If this box is not checked, text will not be output as the .SHP format does not support text.

Render Marker Symbols

By default, marker symbols in the drawing are output as points. Selecting the *Render marker symbols* control will instead cause marker symbols to be rendered as lines and/or areas as appropriate.

Write 3D Shapefiles

Check the *Write 3D Shapefiles (not compatible with all software)* check box to export contour lines and point, polyline, and polygon objects in a base layer as 3-dimensional objects, where each vertex for the point, polyline or polygon comprises an X, Y and Z coordinate.

- When exporting a contour layer, the Z value of the contour line is used as the Z coordinate in the shapefile. The Z coordinate for each contour line is also stored as the ZLEVEL attribute in the DBF file.
- When exporting objects in a base layer, the objects must have an attribute called ZLEVEL with the Z coordinate information. The ZLEVEL attribute is used as the Z coordinate in the shapefile, and all attributes (including ZLEVEL) are stored in the DBF file.

When *Write 3D Shapefiles (not compatible with all software)* is not checked, contour lines and objects in a base layer are exported as 2-dimensional objects, i.e. with X and Y coordinates only. When exporting a contour layer, the Z coordinate for each contour line is stored as the ZLEVEL attribute in the DBF file. For base layers, all attributes are exported to the DBF file.

You would export a contour layer as a 3D shapefile to import it into another software that can visualize this type of data, e.g. ArcScene. Not all software can read 3D shapefiles. Uncheck the *Write 3D Shapefiles* checkbox for maximum compatibility. In most cases, exporting the contours as a 2D shapefile with the Z coordinates in the DBF is sufficient.

Attribute Translation

By default, the "Machine Local (Windows)" code page (0) will be used to translate 16-bit Unicode attribute text to 8-bit ANSI text stored in the companion .DBF file. This can be changed by selecting a different code page from the drop down list. This code page will be stored in a companion .CPG file and also in the .DBF file header. If characters from multiple languages are present, consider selecting “UTF-8 - [65001]”.

Defaults

The *Defaults* button sets all options to default conditions.

Scaling Page

Scaling information can be set on the Scaling page.

Spatial References

The spatial reference information can be set on the Spatial References page. By default, if the map is in a defined coordinate system, the Esri .PRJ file is automatiocally created for this Esri file type.

Esri Shapefile .SHP Export Automation Options

Since the Export Options dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options. A typical example would be:

"Defaults=1,AreasToFile=0,PointsToFile=0"

This would first set all export options to their default values, then specify that all points and areas be output to as lines within the single shapefile.

Option	Action	Default	Description

AreasToFile	0 = No, write areas as lines 1 = Yes, write areas as separate file	1	Write areas to separate file.
PointsToFile	0 = No, write areas as lines 1 = Yes, write areas as separate file	1	Write points to separate file.
RenderMarkers	0 = No, export as points 1 = Yes, render markers	1	Any marker objects will be converted to areas for export; otherwise markers are exported as points.
RenderText	0 = No, discard text 1 = Yes, render text	1	Any text objects will be converted to areas for export; otherwise text is discarded.
Write3DObjects	0 = No, write 2D SHP 1 = Yes, write 3D SHP	0	Points, lines, and areas will be written with X, Y, and Z coordinates in the shapefile
ExportCodePage	Numeric value	Western European (Windows) = 1252	The number of the ANSI code page to use when exporting Unicode data. Valid code page number in the range 0 through 65535. The *ExportCodePage* value is stored in the .CPG file and .DBF file in the same directory with the same name as the .SHP file. If characters from multiple languages are present, consider using code page 65001 (UTF-8).
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.

FileLLX	N.N	Set scaling rectangle lower left X value.
FileLLY	N.N	Set scaling rectangle lower left Y value.
FileURX	N.N	Set scaling rectangle upper right X value.
FileURY	N.N	Set scaling rectangle upper right Y value.

Remarks

Note that if *Shp_AreasToFile* is set to 1, the file name used for the areas file will be the same as the specified export file name with the suffix *Poly* added to the basename. For example, if the specified export file name is *ColoradoHydrography.SHP*, the areas file will be *ColoradoHydrographyPoly.SHP*.

Likewise, if *Shp_PointsToFile* is set to 1, the file name used for the points file will be the same as the specified export file name with the suffix *Pnts* added to the basename.

SID LizardTech MrSID File Description

The MrSID (pronounced Mister Sid) is an acronym that stands for *multiresolution seamless image database*. It is a file format developed and patented by LizardTech for encoding of georeferenced raster graphics. Unlike many other file formats, any rectangular region of the image can be quickly extracted from the file without decompressing the entire file. A region can be specified for import in the MrSID Import Options dialog, or the image can be imported as a read-only image. Each of these options decreases the import time for large MrSID files.

Import Options Dialog

See MrSID Import Options Dialog.

Import Automation Options

See MrSID Import Automation Options.

Third Party Notice

Portions of this computer program are copyright (C) 1995-2008 Celartem, Inc., doing business as LizardTech. All rights reserved. MrSID is protected by U.S. Patent No. 5,710,835. Foreign Patents Pending.

LizardTech MrSID .SID Import Options Dialog

*Customize the SID pixel reduction, import region, or read-only import in the **SID Image Import Options** dialog.*

Image Preview

The preview section displays a picture of the area to be imported. The preview section contains a low resolution preview of the area.

- 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 import. The area to import is indicated in the preview section by a yellow outline.
- Click the button to fit the entire image in the preview window.
- Click the button on the left side of the image preview to quickly move the image in the preview section. Click and hold the left mouse button down and drag the image to change the view. Changing the view does not affect the area to import. The area to import 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 imported. The initial region includes the entire image extents. Click and hold

the left mouse button down and drag the mouse over the area to import. The zoom extents update and the yellow box coincides with the area drawn. The *Region selected for import* section also updates. Only the portion of the image highlighted by the yellow box will be imported.

Import Region

The region for import can be selected in the image preview or in the *Region selected for import* section. The initial region includes the entire image extents. Type a value in pixels into the *Top*, *Left*, *Right*, and *Bottom* fields to specify the imported region. The yellow outline in the image preview is updated as the values are changed.

Pixel Reduction

Since some SID images can be extremely large once expanded into memory, the import filter allows the image to be reduced in dimensions by 1/2 to 1/32 of the original size via the options in the **SID Image Import Options** dialog. Choose from *1/1* (Uses the most memory, best quality)*, 1/2, 1/4, 1/8, 1/16,* to *1/32* (Uses the least memory, lowest quality). The *Final dimensions* field displays the final dimensions of the imported region. The *Estimated size* field displays the estimated image size in Megabytes (MB).

Read-only Import

The *Full resolution read-only 'on-the-fly' MrSID image* option imports the full MrSID file in its native form as a highly-compressed, read-only image. If a portion of the image is needed for display or output, the necessary portion is extracted and decompressed "on the fly." This option provides quick import and interactive panning and zooming without using large amounts of RAM or disk space. However once an image is imported with *Full resolution read-only 'on-the-fly' MrSID image* checked, the image cannot be modified.

LizardTech MrSID .SID Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Scale=16"

This would reduce the scale of the image to 1/16th .

Option	Action	Default	Description
Scale	1 = 1/1 (uses most memory, best quality) 2 = 1/2 4 = 1/4 8 = 1/8 16 = 1/16 32 = 1/32 (uses least memory, lowest quality	1	Since some images can be extremely large once expanded into memory, the import filter allows the image to be reduced in dimensions by 1/2 to 1/32 of the original size.

LRX	integer	maximum X	Use LRX to limit the import to a subregion of the file. LRX is the X pixel coordinate for the lower right corner of the desired subregion.
LRY	integer	maximum Y	Use LRY to limit the import to a subregion of the file. LRY is the Y pixel coordinate for the lower right corner of the desired subregion.
ULX	integer	0	Use ULX to limit the import to a subregion of the file. ULX is the X pixel coordinate for the upper left corner of the desired subregion.
ULY	integer	0	Use ULY to limit the import to a subregion of the file. ULY is the Y pixel coordinate for the upper left corner of the desired subregion.

Remarks

The pixel coordinates increase in the X direction from left to right and increase in the Y direction from top to bottom. The pixel coordinate origin is the upper left corner. For example an image with a 2000 pixel width and 3000 pixel height has the following coordinates: the upper left corner is coordinate (0,0) and the lower right corner is the x and y maximums (2000,3000).

However, the image still will import correctly even if the ULY is used for the maximum Y and LRY is used for the minimum Y.

STK Metamorph File Description

Surfer imports lattice data from Metamorph .STK data sets.

A Metamorph .STK data set contains one or more stacked bitmap images (slices). This file format has historically been used to record data from some makes of confocal scanning microscopes.

File name extension: .STK

Format(s) Supported for Import

uniform lattice; 8-, 16-bit integer.

Loading .STK Files

Metamorph .STK files can be loaded to create a grid based map (i.e. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options

No import options dialog is displayed.

Import Automation Options

No import options are available.

STL STereoLithography 3D Mesh File Description

Surfer exports grids in the STL file format in the Save Grid As dialog. Convert an existing grid to STL with the Grids | Edit | Convert command, or save a grid in the grid editor with File | Save As.

The Stereolithography 3D Mesh STL file format is used primarily for 3D printing, prototyping, and computer-aided manufacturing. An STL file is a triangular representation of a 3-dimensional surface geometry. The surface is broken down into small triangles or facets. Each facet is described by the triangle's vertices and unit normal vector.

The STL file format does not contain any scale or unit information. STL files can be saved in ASCII or binary format.

STL ASCII Format

The ASCII format can be viewed in any text editor software. However, ASCII format STL files are considerably larger than binary STL files. The following block is an example of the format structure:

```
solid gridmesh
facet normal i j k
   outer loop
      vertex x1 y1 z1
      vertex x2 y2 z2
      vertex x3 y3 z3
   end loop
end facet
endsolid gridmesh
```

Each facet is described by the unit normal vector (i,j,k) and each vertex is described by its coordinates (x,y,z) between the solid and endsolid lines. The numerical data (i,j,k and x,y,z) are single precision floating point values.

STL Binary Format

The binary format is significantly more compact than the ASCII format. The binary format includes an 80-byte ASCII header, an integer value indicating the number of facets, and then a series values describing the unit normal vector, facet vertices, and attribute value for each facet.

STL References

See *StereoLithography Interface Specification,* 3D Systems, Inc., October 1989 for more information on the STL file format specifications.

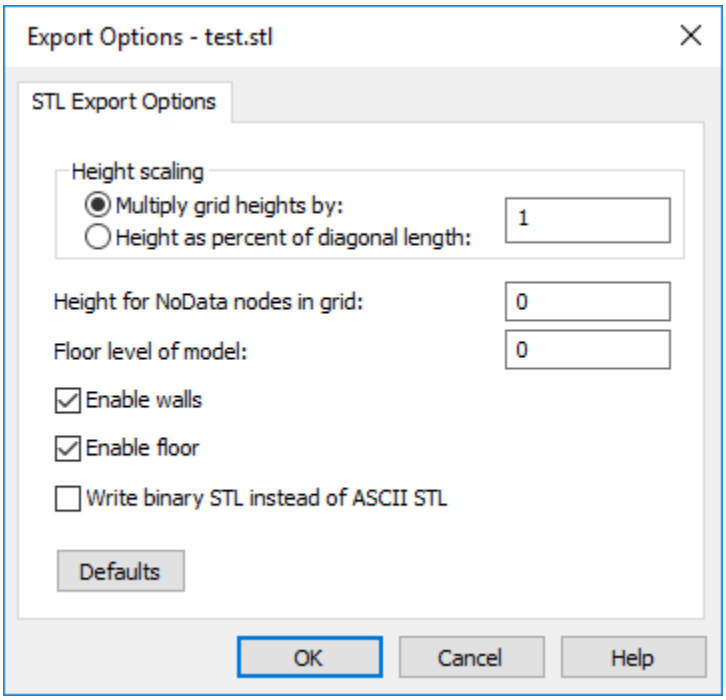
Export Options

See Spatial References and STL Export Options Dialog

See STL Export Automation Options

STL Export Options Dialog

The **STL Export Options** dialog is displayed when saving a grid in the STL Stereolithography 3D Mesh file format.



*Control model scaling and features in the **STL Export Options** dialog.*

Grid Height Scaling

The grid Z values can be scaled in one of two methods. Select *Multiply grid heights by* to apply a scale factor to the grid Z values. Select *Height as a percent of diagonal length* to scale the maximum height of the model as a percentage of the grid's diagonal length.

Scale the grid Z values by selecting *Multiply grid heights by* and typing a scaling factor in the field. Values less than 1 decrease the grid Z values. Values greater than 1 increase the grid Z values. The default value is 1, where grid Z values are not scaled when saving the grid in the STL format.

Scale the grid Z values by selecting *Height as a percent of diagonal length* and typing a percentage in the field. Values less than 100 make the model height less than the model's diagonal length. Values greater than 100 make the model height greater than the model's diagonal length. A value of 100 makes the model height equal to the model's diagonal length.

The Height for NoData nodes in grid value is scaled by the Height scaling. The Floor level of model value is not affected by the Height scaling.

NoData Node Value

Specify the value for NoData grid nodes in the *Height for NoData nodes in grid* field. Nodes assigned the NoData value will be mapped to the *Height for NoData nodes in grid* value in the

output STL file. The default value is 0. However, 0 may not be appropriate for your grid range. A value equal to or near the grid Z minimum is recommended.

The *Height for NoData nodes in grid* value is transformed by the specified *Height scaling*. For example, if you remap NoData grid nodes to 100 and set *Multiply grid heights by* to 0.25, the resulting height value in the STL file is 25 for NoData nodes.

Floor Level

Specify the floor level for the STL model in the *Floor level of model* field. The *Floor level of model* value determines the surface's height above the floor and the wall height. The default value is 0. However, 0 may not be appropriate for your grid range.

The *Floor level of model* value is not affected by the *Height scaling*.

Walls and Floors

Check the *Enable walls* check box to include walls in the output STL file. Check the *Enable floor* check box to include a floor on the bottom of the output STL file. The *Enable walls* and *Enable floor* check boxes are checked by default.

File Format

Check the *Write binary STL instead of ASCII STL* check box to write the STL in the binary format standard. Binary format output STL files are considerably smaller than ASCII format STL files. Clear the *Write binary STL instead of ASCII STL* check box to write the STL in the ASCII format standard. See the file description for more information on the binary and ASCII formats.

Defaults

Click the *Defaults* button to reset the export options to their default values.

OK and Cancel

Click *OK* to save the grid as a model in the STL file format. Click *Cancel* to close the **STL Export Options** dialog without saving the STL file.

STL Stereolithography 3D Mesh Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1, HeightScale=1.5, BlankHeight=70,EnableFloor=0"

This would first set the options to their default values, then scale the grid Z values by 150% and remove the model floor from the output file.

Option	Action	Description	Default
HeightModePercent	0 = False	1 makes the HeightScale value a percentage of the	0

	1 = True	model's maximum horizontal extent, which is typically the longest diagonal. 0 makes the HeightScale value a scaling factor to apply to Z values.	
HeightScale	N	Value by which to multiply the grid Z values and BlankHeight before exporting to the STL 3D model when HeightModePercent = 0. Value is the maximum height of the model described as a percentage of the model's diagonal length when HeightModePercent = 1.	1
BlankHeight	N	Remap NoData grid nodes to the specified value.	0
FloorHeight	N	Specify the model floor height.	0
EnableWalls	0 = Disable walls 1 = Enable walls	Set to 1 to include walls in the STL 3D model. Set to 0 to exclude walls from the model.	1
EnableFloor	0 = Disable floor 1 = Enable floor	Set to 1 to include a floor in the STL 3D model. Set to 0 to exclude the floor from the model.	1
WriteBinary	0 = ASCII format 1 = binary format	Set to 0 to write to the ASCII STL file format. Set to 1 to write to the binary STL file format.	0

SVG Scalable Vector Graphics File Description

SVG files are an XML based vector image format designed for use with two-dimensional graphics. SVG files can contain raster graphics, vector polylines and polygons, and text. When exported from Golden Software programs, SVG files contain text that is drawn as vector polylines and polygons. Objects are typically grouped in the .SVG file.

SVG is an open standard available for import and export from many programs. SVG is intended to be portable between applications. The SVG file format is maintained by the W3C SVG Working Group.

Export Options Dialog

See Export Options - Scaling Page

SYLK Spreadsheet [.SLK] File Description

Surfer imports and exports data to SYLK spreadsheet [.SLK] files.

Symbolic Link SYLK is a Microsoft file format typically used to exchange data between applications, specifically spreadsheets. SYLK files conventionally have a [.SLK] suffix. From within a spreadsheet data can be exported in the SYLK format. Composed of only displayable ANSI characters, it can be easily created and processed by other applications, such as databases.

Microsoft does not publish a SYLK specification. Variants of the format are supported by Multiplan, Microsoft Excel, Microsoft Works, OpenOffice.org, and Gnumeric.

Note that even if a SYLK file is created by an application that supports Unicode (for example Microsoft Excel), the SYLK file will be encoded in the current system's ANSI code page, not in Unicode. If the application contained characters that were displayable in Unicode but have no codepoint in the current system's code page, they will be converted to question marks ('?') in the SYLK file.

If a SYLK file is created in another application and loaded into the worksheet, there might be special formatting information in the file that the worksheet cannot use. In these cases, the data file is loaded without the special formatting, but if the file is saved in a SYLK format in the worksheet the special formatting information is lost.

Technically, the SYLK specification has a 65,536 row limit and a 256 column limit. However, SYLK is an ASCII format so it is possible to have an unlimited size. The Golden Software SYLK Import and Export filters allow limits greater than the specifications indicate, so larger worksheets can be saved in SYLK format. However, other spreadsheet programs may not be able to load these large SYLK files.

Sample SYLK code

As an example, the following SYLK code in a text file with the [.SLK] extension:

```
ID;P
C;Y1;X1;K"Row 1"
C;Y2;X1;K"Row 2"
C;Y3;X1;K"Total"
C;Y1;X2;K11
C;Y2;X2;K22
C;Y3;X2;K0;ER1C2+R2C2
E
```

would be displayed like this when read by an appropriate spreadsheet:

Row 1	11
Row 2	22
Total	33

Loading .SLK Files

.SLK files can be loaded with the File | Open command.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Saving .SLK Files

.SLK files can be saved from a worksheet using the File | Save As command.

Export Options Dialog

No export options dialog is displayed.

Export Automation Options

No export options are available.

TAB MapInfo Table File Description

Surfer imports MapInfo Table TAB vector map files. MapInfo Table TAB files define a table structure. When combined with the appropriate .MAP, .ID, and .DAT files, MapInfo TAB files can include geographic features such as points, lines, polygons, and text with attributes. Some feature style and color information is maintained when importing TAB files. The associated .MAP, .ID, and .DAT files must be in the same directory as the .TAB file to import the vector .TAB file.

MapInfo TAB files are also used to georeference raster images. For information about a georeferencing TAB file, see *Export Options - Spatial References Page*. A georeference TAB file cannot be loaded as a vector map.

File Structure

MapInfo TAB text files are the main file for a MapInfo table and include the table structure. MapInfo DAT binary files include attribute data. MapInfo MAP binary files store the geographic objects. MapInfo ID files link the geographic objects to graphic data.

Loading MapInfo TAB files

TAB vector map files can be loaded to create a base map (Home | New Map | Base) with the **Import** dialog. Select the TAB file to create the map. The directory must also include the .MAP, .ID, and .DAT associated files.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

References:

MapInfo Dataset Format. Library of Congress. www.loc.gov

MapInfo TAB and MIF/MID. Geospatial Data Abstraction Library. www.gdal.org

TGA Truevision Targa [.TGA] File Description

TARGA is an acronym for Truevision Advanced Raster Graphics Adapter; TGA is an initialism for Truevision Graphics Adapter. Today, most people refer to the format as the "TARGA File Format".

File Format

Truevision's (now AVID) TGA file format, often referred to as TARGA file format, is a raster graphics file format. It was the native format of Truevision Inc.'s TARGA and VISTA boards, which were the first graphic cards for IBM-compatible PCs to support Highcolor/truecolor display. This family of graphic cards was intended for professional computer image synthesis and video editing with PCs; for this reason, usual resolutions of TGA image files match those of the NTSC and PAL video formats.

TGA files commonly have the extension .TGA on PC DOS/Windows systems. The format can store image data with 8, 16, 24, or 32 bits of precision per pixel, the maximum 24 bits of RGB and an extra 8-bit alpha channel. Color data can be color-mapped, or in direct color or truecolor format; optionally, a lossless PackBits RLE compression can be employed.

Uncompressed 24-bit TGA images are relatively simple compared to several other prominent 24-bit storage formats: A 24-bit TGA contains only an 18-byte header followed by the image data as packed RGB data. In contrast, BMP requires padding rows to 4-byte boundaries, TIFF and PNG are metadata containers that do not place the image data or attributes at a fixed location within the file.

Import Options

No import options dialog is displayed.

Import Automation Options

No import automation options are available.

Export Options

See Size and Color and Spatial References

Export Automation Options

See Bitmap Export Automation Options

TIF Tagged Image File Description

The tagged image file format .TIF or .TIFF is a file format for storing images, including photographs and line art. The TIFF format is widely supported by a variety of applications.

Surfer can import and export .TIF and .TIFF files.

When a .TIF or .TIFF file contains attribute information, **Surfer** imports this information and stores it on the Info tab. For a full list of the available attributes in a .TIF or .TIFF file, refer to the online reference guide.

The TIFF specification allows an unusually wide variety of different formats and encodings within the same file format. While the filter can read most of the common variants of TIFF, it would be impractical to develop software to read every possible variant. The TIFF filter supports a wide variety of TIFF files that use the PlanarConfiguration 2 encodings.

TIFF images that contain bit per pixel counts other than 1, 4, 8, 16, 24, or 32 may not be readable depending on the encoding and alignment of the data.

TIFF images that are encoded with photometric interpretation models other than RGB, YCbCr, grayscale, or monochrome may not be readable.

Some of the compression algorithms allowable under the TIFF specification are not supported by **Surfer**.

GeoTIFF

Only a single map object can be exported to a GeoTIFF. If multiple map objects exist in the **Surfer** file, all maps should be overlaid before exporting. If other drawn objects (legends, scale bars, text, etc.) exist in the **Surfer** file, these objects should be temporarily deleted before exporting.

Import Options

No import options dialog is displayed when importing the TIFF file as an image or base layer.

When opening a TIFF file as a grid with the Open Grid dialog, the TIFF Import Options dialog is displayed.

Import Automation Options

No import automation options are available when importing the TIFF file as an image or base layer.

When opening a TIFF File as a grid, see TIF Tagged Image File Import Automation Options.

Export Options

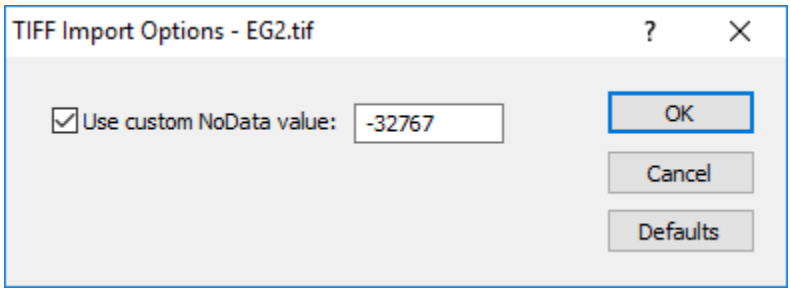
See Size and Color, Spatial References, and TIFF Export Options

Export Automation Options

See Bitmap Export Automation Options

TIF Tagged Image Import Options Dialog

Surfer can import TIF image files. TIF file can also be used as grid files. When a TIF file is used as a grid file, the **TIFF Import Options** dialog is displayed.



*Set the NoData value in the **TIFF Import Options** dialog.*

Use Custom NoData Value

Leave the *Use custom NoData value* option cleared to use the NoData value specification in the TIF file. If you wish to specify the NoData value, select the *Use custom NoData value* option, and type the desired NoData value in the field.

Some applications may export a TIF raster without correctly specifying the NoData (or blanking) value in the file header. In these cases **Surfer** will treat all the values in the grid as valid. If your grid-based map looks incorrect, use the Grid Info command and/or the Z value in the status bar cursor coordinates to determine the correct NoData value. The NoData value is likely to be the minimum or maximum which can be determined with **Grid Info**. Hover the cursor over a known NoData (blanked) region and read the value in the cursor coordinates section of the status bar. Next create a new map with the TIF grid file and specify the correct NoData value in the **TIFF Import Options** dialog.

Defaults

Click *Defaults* to return the **TIFF Import Options** dialog to the default settings.

TIF Tagged Image File [.TIF, .TIFF] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,CustomNoData=1,CustomNoDataValue=-32767"

This would first set all import options to their default values, then use a custom NoData value for the grid file.

Option	Action	Default	Description

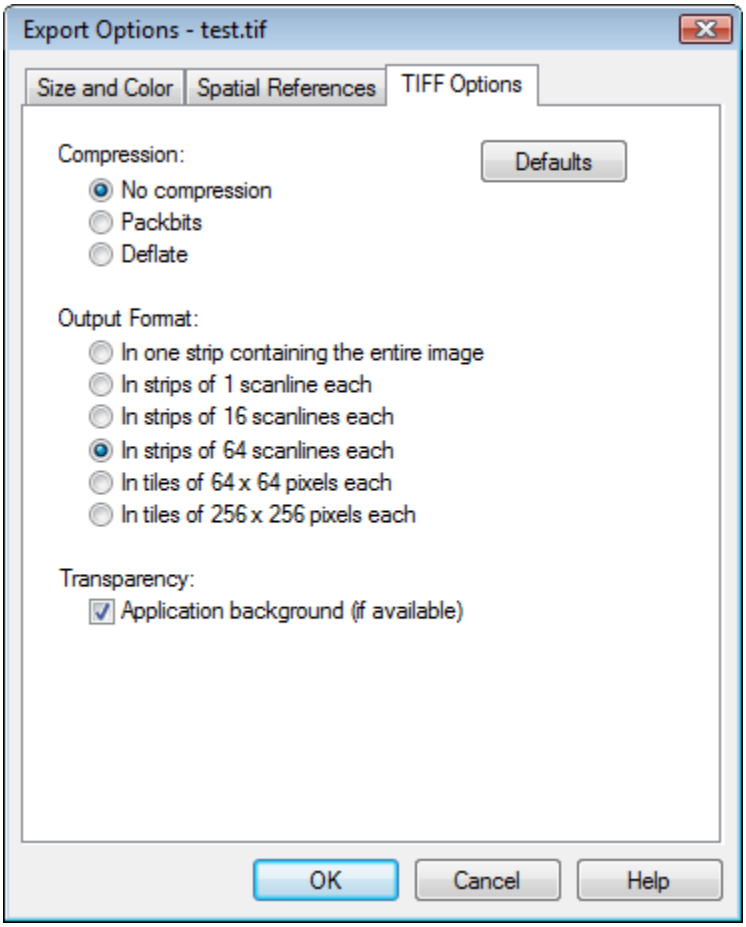
CustomNoData	0 = No 1 = Yes	0	0 to use the NoData value specified in the TIF file header. 1 to specify a custom NoData value.
CustomNoDataValue	N		Specify the NoData value for the TIF file.

Remarks

These import automation options are only used when importing the TIF file as a grid, for example with LoadFile2 [Grid]. There are no import options when importing a TIF image, e.g. with Import2, or using the TIF image to create a base map, e.g. with AddRasterBaseMap.

Tagged Image .TIF Export Options Dialog

Surfer can import and export .TIF, .TIFF files.



*Specify TIF export options in the **Export Options** dialog.*

Compression

- *No compression* results in a very large output file size.
- *Packbits* compression involves finding repeated data values; as a result, it is a good choice for images without large color ranges.

- *Deflate* is a dictionary encoding method that produces significant reduction in file size for most images without losing any image information.

Output Format

Choose from a variety of *strip* and *tile* output formats. The option *In one strip containing the entire image* is the least efficient option but has the highest degree of compatibly with other software.

Transparency

Use the **Export Options** dialog to specify the Size and Color options and the transparency options for the .TIF file. To export the .TIF file with all background areas transparent, click on the **TIFF Options** tab in the **Export Options** dialog. Check the *Application background (if available)* option. The .TIF file will have transparent background areas. In order for the TIF to be transparent, the *Color depth* must be set to 4-bit, 8-bit, or 32-bit on the **Size and Color** tab.

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.

Size and Color Page

See Size and Color

Spatial References Page

See Spatial References

TXT Formatted Text Grid [.FTG, .CSV, .DAT, .TXT] File Description

Formatted Text Grid [.FTG, .CSV, .DAT, .TXT] files are regularly spaced tables or matrices of Z values. Import a Formatted Text Grid file with the Open Grid dialog. The **Open Grid** dialog is accessed via one of the grid-based map layer commands (e.g. Home | New Map | Contour Map) or an applicable Grid tab command.

Surfer can import Formatted Text Grid files with a number of matrix formats. Below are four examples of possible formats. The first example shows tables where the first row and column contain information about the X and Y coordinates.

	A x	B y	C z	D
1		x1	x2	x3
2	y1	z11	z21	z31
3	y2	z12	z22	z32
4	y3	z13	z23	z33

	A x	B y	C z	D
1		y1	y2	y3
2	x1	z11	z12	z13
3	x2	z21	z22	z23
4	x3	z31	z32	z33

In the above images, the first row and column contain X or Y axis coordinates. In the image on the left the X

coordinate advances by column. In the image on the right the X coordinate advances by row

In the following example shows tables where all values in the table represent a Z value, and X and Y coordinate information is not included in the first row or column.

	A x	B y	C z
1	z11	z21	z31
2	z12	z22	z32
3	z13	z23	z33

	A x	B y	C z
1	z11	z12	z13
2	z21	z22	z23
3	z31	z32	z33

In the above images, X and Y coordinate information is not included in the first row and column. In the image on the left, the X coordinate advances by column. In the image on the right, the X coordinate advances by row.

When the first row and column do not contain coordinate information, it is necessary that the first data value corresponds to the x-axis and y-axis minimum coordinates. If the table is not formatted with the value corresponding to the smallest x-axis and y-axis coordinate in the first row and column, then one of two manipulations must be performed. Either add a first row and first column with coordinate information, or reformat the table where the first data value is the Z value at the x-axis and y-axis minimums.

The Formatted Text Grid file must contain a rectangular, regularly spaced matrix of values. The x-axis coordinates and y-axis coordinates must increase by an equal incremental value with each row or column. However, the incremental value does not need to be the same for the x-axis and y-axis.

	A x	B y	C z	D
1		1	2	3
2	5	12.3	17.2	15.4
3	8	11.1	18.8	17.9
4	11	8	16.4	20.1

Above is an example of a regularly spaced Formatted Text Grid

In the preceding image the Formatted Text Grid file includes x-axis coordinate information in the first row and y-axis information in the first column. The x-axis coordinate minimum is 1 (Column B) and increases by the step value of 1 in Columns C and D. The y-axis minimum is 5 (Row 2) and increases by the step value of 3 in Rows 3 and 4.

Import Options Dialog

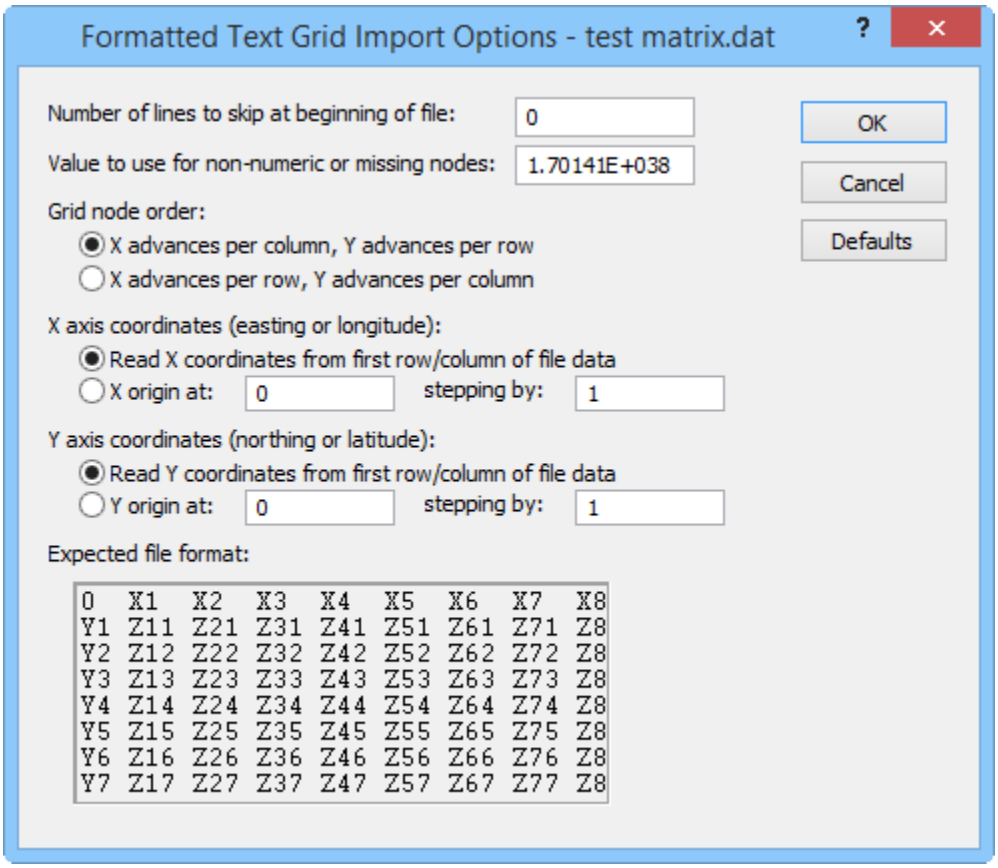
See TXT Formatted Text Grid Import Options Dialog

Import Automation Options

See TXT Formatted Text Grid Import Automation Options

TXT Formatted Text Grid Import Options Dialog

Surfer can import Formatted Text Grid files.



*Specify the Formatted Text Grid options in the **Import Options** dialog.*

Skip Lines at Beginning of File

If the coordinate information or table of Z values does not start at the first line of the file, then you must specify the *Number of lines to skip at beginning of file*. Type a value representing the number of lines to skip in the *Number of lines to skip at beginning of file* field.

Non-Numeric or Missing Values

If a non-numeric or missing node is encountered during import, **Surfer** will input a user-defined value into the resulting grid. Type a number in the *Value to use for non-numeric or missing nodes* field to specify the value. The default value is the **Surfer** NoData (blanking or null) value, 1.70141E+38.

Grid Node Order

The *Grid node order* specifies whether the x-axis coordinates and y-axis coordinates advance per column or row. Click the *X advances per column, Y advances per row* or *X advances per row, Y advances per column* radio button that describes the format of the Formatted Text Grid file.

X Axis Coordinates

If the Formatted Text Grid file contains x-axis coordinates in the first row or column, click the *Read X coordinates from first row/column of file data* radio button.

If the file does not contain x-axis coordinate information, click the radio button next to *X origin at*. Next, type the minimum x-axis coordinate value in the *X origin at* field. Finally type the value by which the x-axis coordinate increases with row/column in the *stepping by* field.

Y Axis Coordinates

If the Formatted Text Grid file contains y-axis coordinates in the first row or column, click the *Read Y coordinates from first row/column of file data* radio button.

If the file does not contain y-axis coordinate information, click the radio button next to *Y origin at*. Next, type the minimum y-axis coordinate value in the *Y origin at* field. Finally type the value by which the y-axis coordinate increases with row/column in the *stepping by* field.

Expected File Format

The *Expected file format* preview provides a visual representation of the import option selections. The *Expected file format* field updates as the import options are changed. X-axis coordinate values are represented by *X#* in the *Expected file format* field. Y-axis coordinate values are represented by *Y#* in the *Expected file format* field. Z values are represented by *Z##*, where the first number is the corresponding x-axis coordinate, and the second number is the corresponding y-axis coordinate.

Formatted Text Grid [.CSV, .DAT, .FTG, .TXT] Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

```
"Defaults=1,XFromFile=True"
```

This would first set all import options to their default values, then set the XFromFile option value to True, which would specify that the x-coordates are read from the first row of the file.

Option	Action	Default	Description
Defaults	0 = No 1 = Yes	0	When set to 1, the default import options will be applied.
XFromFile	0 = No 1 = Yes	0	When set to 0, the x-axis coordinates are specified by the XOrigin and XStep values. When set to 1, the x-axis coordinates are read from the first row/column of the file.
YFromFile	0 = No 1 = Yes	0	When set to 0, the y-axis coordinates are specified by the YOrigin and YStep values. When

			set to 1, the y-axis coordinates are read from the first row/column of the file.
XOrigin		0	This is the minimum x-axis coordinate value
YOrigin		0	This is the minimum y-axis coordinate value
XStep		1	This is the incremental amount by which the x-axis value increases with each row/column.
YStep		1	This is the incremental amount by which the y-axis value increases with each row/column.
NoData		1.40141E+38	This is the value that is inserted when a non-numeric or missing node is encountered.
SkipLines		0	This is the number of lines to skip at the beginning of the file.
AxisFlip	0 = No 1 = Yes	0	When set to 0, x-axis advances by column, and y-axis advances by row. When set to 1, x-axis advances by row, and y-axis advances by column.

VCT Idrisi Binary Vector File Description

Surfer imports the Idrisi Binary Vector .VCT format. VCT files are paired with documentation files (.VDC). Sometimes there is also an associated .REF file. Each IDRISI Binary Vector .VCT file includes only one feature type: point, polyline, polygon, or text. Each feature is described by a single numeric attribute and one or more XY coordinate pairs.

For detailed file structure information, download the TerrSet Help System from Clark Labs.

Loading .VCT Files

Idrisi Vector VCT files can be loaded to create a base map (Home | New Map | Base) with the **Import** dialog.

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

References: *TerrSet Manual*. Published by Clark Labs, Clark University. www.clarklabs.org

VTK Visual Tool Kit File Description

Surfer can import and export Visualization Tool Kit .VTK data sets. This implementation is based on version 4.2 of the VTK File Format documentation from Kitware.

VTK data sets can contain several types of lattice data and/or geometric figures. The content of VTK files can be in ASCII text format or a mixed binary/ASCII format in which headers and parameters are in ASCII format but the data values are in binary format.

File name extension: .VTK

Format(s) Supported for Import

- 3D uniform lattice; 8-, 16-, 32-bit integer, float, double
- point set; double

Format(s) Supported for Export

3D uniform lattice; 8-, 16-, 32-bit integer, float, double

Import Options

No import options dialog is displayed.

Import Automation Options

No import options are available.

Import Restrictions/Limitations

The implementation of the software does not directly support geometric figures in VTK data sets. Geometric figures are currently imported as **Surfer** point sets, with one point at each vertex in the geometric model.

Export Options

See Visual Tool Kit .VTK Export Options Dialog for more information.

Export Automation Options

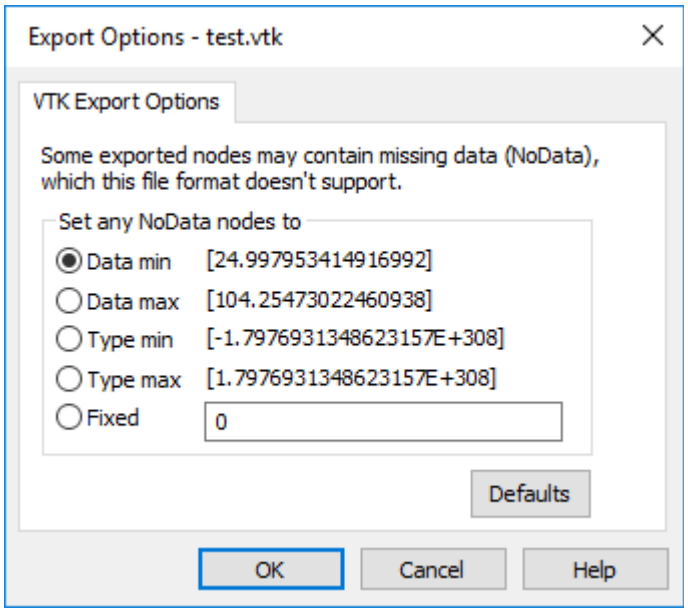
See Visual Tool Kit .VTK Export Automation Options

Export Restrictions/Limitations

This implementation of **Surfer** does not support exporting of geometric figures; only lattices can be exported as .VTK files.

Visual Tool Kit .VTK Export Options Dialog

The **Export Options** dialog is only displayed if the lattice contains one or more nodes assigned the NoData value. The dialog controls determine how NoData values are treated in the export file. Use the **Grids | Edit | Convert** command in the plot window or the **File | Save As** command in the grid node editor to open the **Export Options** dialog.



*Set NoData node values for .VTK export in the **Export Options** dialog.*

Set Any NoData Nodes To

The controls in the *Set any NoData nodes to* group determine how NoData values are treated in the export file.

Data Min

Choose *Data min* to export NoData nodes with the indicated minimum value (varies depending on contents of lattice).

Data Max

Choose *Data max* to export NoData nodes with the indicated maximum value (varies depending on contents of lattice).

Type Min

Choose *Type min* to export NoData nodes with the indicated minimum value for the lattice's data type (usually zero).

Type Max

Choose *Type max* to export NoData nodes with the indicated maximum value of the lattice's data type.

Fixed

Choose *Fixed* to export NoData nodes with some other value, and enter the desired value in the adjacent edit control.

Defaults

Click the *Defaults* button to return all settings in the dialog to the default values.

Visual Tool Kit [.VTK] Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

" BlankMethod=BM_FIXED, FixedBlankValue=1.70141E+038"

This would indicate that the NoData value for a lattice or grid node is a custom value, and that value is equal to **Surfer's** NoData value.

Option	Action	Default
BlankMethod	Method for assigning values to NoData nodes: BM_DATA_MIN = Minimum value of the lattice BM_DATA_MAX = Maximum value of the lattice. BM_TYPE_MIN = Minimum value of the inherent data type. BM_TYPE_MAX = Maximum value of the inherent data type. BM_FIXED = A user specified value	BM_TYPE_MAX
FixedBlankValue	Used if BlankMethod is BM_FIXED	No

WMF Windows Metafile Description

Windows stores graphical objects (i.e., pictures) in a special form called a metafile. Such pictures can be stored on disk or in the Windows Clipboard. When these pictures are imported (or pasted from the Clipboard), the objects can be optionally separated and stored individually.

Windows Metafiles are intended to be portable between applications and may contain both vector and image components. In contrast to raster formats such as .JPEG and .GIF which are used to store image (bitmap) graphics such as photographs, scans and graphics, Windows Metafiles generally are used to store line-art, illustrations and content created in drawing or presentation applications. Most Windows clipart is in the .WMF or .EMF format.

Windows Metafile .WMF is a 16-bit format introduced in Windows 3.0. It is the native vector format for Microsoft Office applications such as Word, PowerPoint, and Publisher. A newer 32-bit version with additional commands is called Enhanced Metafile (EMF). EMF is also used as a graphics language for printer drivers.

The Windows metafile formats are Windows Clipboard (Picture) .CLP, Windows Metafile (Enhanced) .EMF, and Windows Metafile .WMF.

Import Options

See Windows Metafile Import Options Dialog.

Import Automation Options

See Windows Enhanced Metafile Import Automation Options.

Export Options Dialog

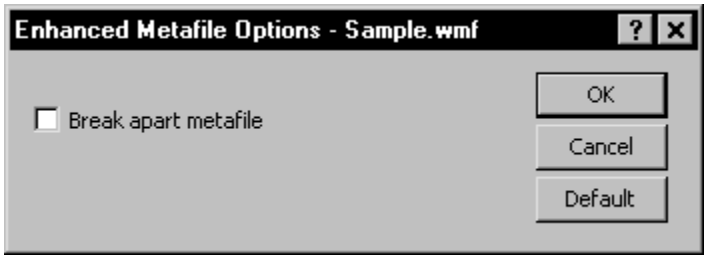
See Windows Enhanced Metafile Export Options Dialog.

Export Automation Options

See Windows Enhanced Metafile Export Automation Options.

Windows Metafile Import Options Dialog

The **Enhanced Metafile Options** dialog allows you to specify options which determine how information in the file is imported. A Windows Metafile .WMF and Enhanced Windows Metafile .EMF are collections of objects combined together to produce an image.



*Specify the import options in the **Enhanced Metafile Options** dialog.*

Break Apart Metafile

Check the *Break apart metafile* option to separate the objects into individual entities. Do not select this option if you want the objects to remain a single collection.

Import-Filter Messages

The following messages may appear while attempting to import .WMF, .EMF or .CLP files.

Clipboard file contains no picture in metafile form!

The clipboard file may contain information in a variety of forms (text, bitmap, metafile). This Graphical Import Filter component can only import pictures stored in Windows metafile format. The clipboard file you selected contains no picture in metafile format. You may still be able to get the information into the Golden Software application by launching the Windows Clipboard Viewer utility (CLIPBRD.EXE), opening the .CLP file, switching back to the Golden Software application and then selecting **Home | Clipboard | Paste | Paste Special**.

No Destination Bitmap for Bit Pattern

Attempt to create a bit pattern on a bitmap while the latter doesn't exist. Make sure the file has been created correctly.

Windows Metafile .WMF and .EMF Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

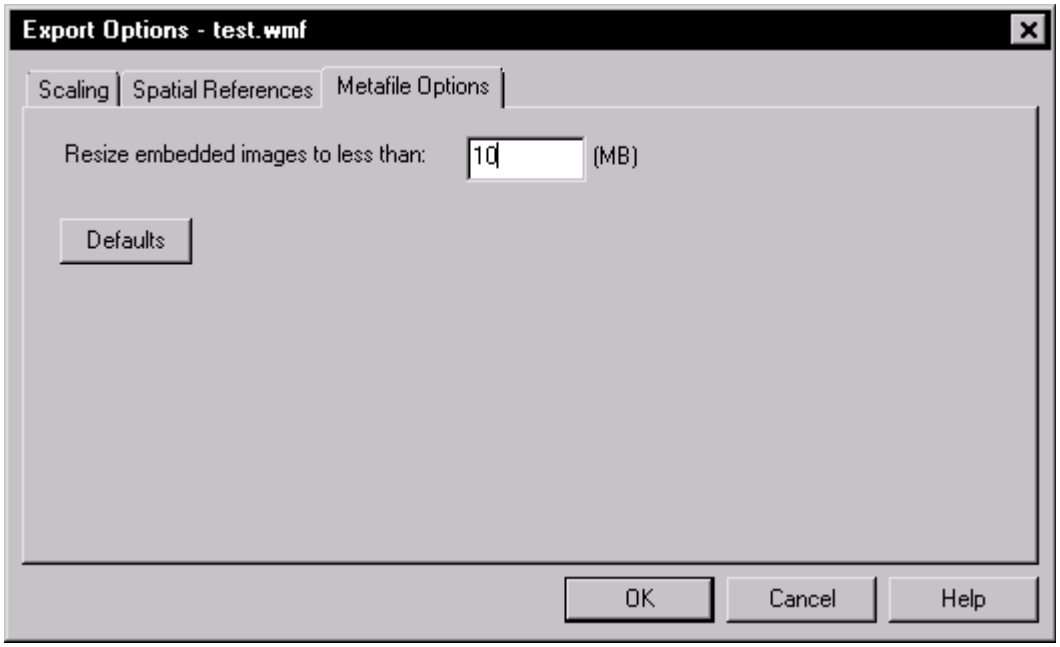
"Defaults=1,BreakApart=0"

This would first set all import options to their default values, then set the BreakApart option value to zero, which specifies the metafile contents are to remain together as a single unit.

Option	Action	Default	Description
BreakApart	0 = No 1 = Yes	1	Break metafile apart into individual graphical objects during import.

Windows Metafile .WMF Export Options Dialog

The **Export Options** dialog allows you to specify options which determine how information in the file is exported. A Windows Metafile .WMF is a collection of objects combined together to produce an image.



*Specify the WMF file export options in the **Export Options** dialog.*

Resize Embedded Images

The *Resize embedded images to less than* option specifies the maximum size (in megabytes) an embedded image is allowed to be. If an exported image exceeds this size, its resolution will be reduced so it doesn't exceed the designated maximum size. Increase this value to get better looking images at the expense of larger export files.

Defaults

The *Defaults* button sets all options to default conditions.

Scaling Page

See the Scaling Page for detailed information.

Windows Metafile .WMF and .EMF Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options.

Option	Action	Default	Description
MaxBitmapSizeInMB	N	10	This option specifies the largest size allowed for an individual bitmap in the EMF file, in Megabytes. Any exported bitmap larger than this size, will have its resolution reduced so it does not exceed the maximum size.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.

FileURY	N.N	Set scaling rectangle upper right Y value.

X AVS X-Image [.XIMG] File Description

Surfer can import and export AVS X-Image .X and .XIMG files.

The AVS X-Image format originated on UNIX workstations and is typically used to store true-color images containing an alpha channel, a feature that most other image file formats lacked at the time this file format was developed. An AVS X-Image file is a binary file containing a raster image with 8-bits each for the red, green, blue, and alpha channels (32 bits total per pixel).

Import Options Dialog

No import options dialog is displayed.

Import Automation Options

No import options are available.

Export Options Dialog

See Size and Color and Spatial References pages.

Export Automation Options

See Image Bitmap Export Automation Options

XLSX XLSM XLS Excel Workbook File

Microsoft Excel .XLS, .XLSX, and .XLSM files contain data and retain some cell formatting in **Surfer**. Some information, such as formulas, is ignored. Excel files can preserve all formatting information available in the Golden Software worksheet.

Surfer supports the import of .XLS, .XLSX, and .XLSM Excel files. **Surfer** supports the export of .XLS and .XLSX files.

Excel .XLS

Excel .XLS files are Microsoft Excel documents. Worksheet cell data and some cell formatting are retained with this format. Other types of information, such as formulas, are ignored.

Excel .XLS format files can preserve all formatting information available in the Golden Software worksheet. This format has a 65,536-row limit and a 256-column limit in Excel 97 and greater. Therefore, this format cannot be used to store very large data sets.

Excel .XLSX

Excel .XLSX files are Microsoft Excel 2007 XML spreadsheets. Worksheet cell data and some cell formatting are retained with this format. Other types of information, such as formulas, are ignored.

Excel .XLSX format files can preserve all formatting information available in the Golden Software worksheet. This format has a 1,048,576-row limit and a 16,384-column limit in Excel 2007 and greater.

Excel .XLSM

Excel .XLSM files are Microsoft Excel 2007/2010 XML spreadsheets. XLSM files can contain macros and VBA scripts. Worksheet cell data and some cell formatting are retained with this format. Other types of information, such as formulas, are ignored.

Excel .XLSM format files are not saved in **Surfer**.

Use Caution when Saving Excel Files!

A file can be saved in an Excel format from **Surfer** worksheet, ***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 file is opened and saved as an .XLS or .XLSX file from the **Surfer** worksheet, be aware that only the single worksheet is saved in the document. If the existing file is overwritten, all the unused worksheets are destroyed. In this case, a warning message is issued. The message reads: *Saving this worksheet will destroy all but one of the sheets in the existing *.xls, *.xlsx file. To overwrite the file, click OK. To choose a different file name, click Cancel.*

Special Characters Used in Excel Files

There are a number of special characters that can be contained in an Excel file that the worksheet cannot handle in the same way as Excel. For these characters, **Surfer** substitutes a reasonable representation so the value displayed in the cell will look similar to what was displayed in Excel.

Font Color in Excel Files

Font color is maintained when opening XLSX and XLSM files in the worksheet. However XLS files are opened with black font, regardless of font color in the XLS file.

Sheet Names

The sheet name of an Excel file is listed in the gridding report, cross validation report, and data statistics report.

Import Options Dialog

See Excel XLS Import Options Dialog and Excel XLSX Import Options Dialog

Import Automation Options

See Import Automation Options String.

Export Options

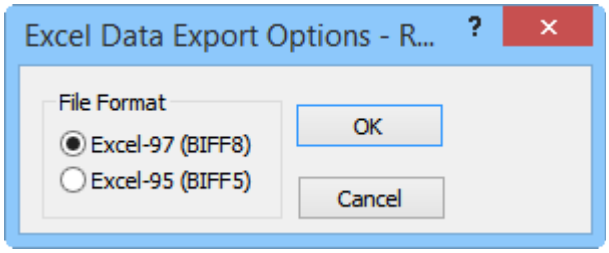
See Excel Export Options Dialog.

Export Automation Options

See Export Automation Options String

Excel .XLS Export Options Dialog

Surfer can export .XLS and .XLSX files. The **Excel Data Export Options** dialog allows you to specify if the Excel file will be written in *Excel 97* or *Excel 95* format.



*Select the File Format to use when exporting in the **Excel Data Export Options** dialog.*

File Format

XLS Files can be saved in either *Excel 97 (BIFF 8)* or *Excel 95 (BIFF 5)* formats.

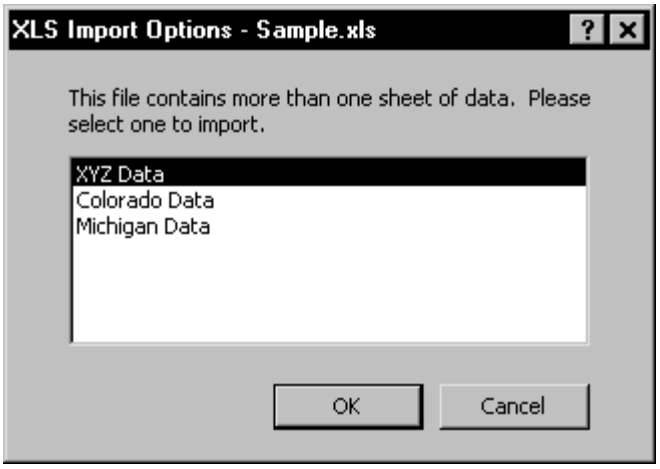
Use Caution when Saving Excel Files!

A file can be saved in an Excel format from the **Surfer** worksheet, 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 file is opened and saved as an .XLS or .XLSX file from the **Surfer** worksheet, be aware that only the single worksheet is saved in the document. If the existing file is overwritten, all the unused worksheets are destroyed. In this case, a warning message is issued. The message reads: *Saving this worksheet will destroy all but one of the sheets in the existing *.xls , *.xlsx file. To overwrite the file, click OK. To choose a different file name, click Cancel.*

The **Export Options** dialog does not appear when saving XLSX format files. **Surfer** does not save Excel .XLSM format files.

Excel .XLS Import Options Dialog

Surfer can import .XLS and .XLSX file types. **Surfer** can import one sheet of Excel spreadsheet data at a time. If you are importing an Excel file with multiple sheets, the **XLS Import Options** dialog will appear.

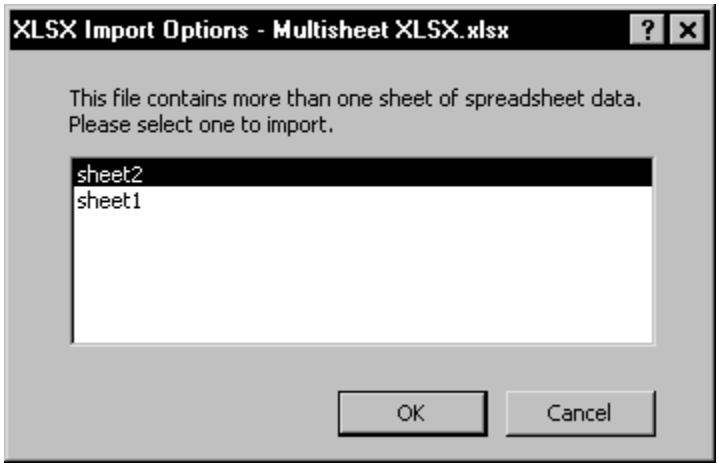


*Select the sheet to import in the **XLS Import Options** dialog.*

Select one sheet to import and click the *OK* button.

Excel .XLSX Import Options Dialog

Surfer can import .XLS and .XLSX file types. **Surfer** can import one sheet of Excel spreadsheet data at a time. If you are importing an Excel file with multiple sheets, the **XLS Import Options** dialog will appear. Select one sheet to import and click the *OK* button.



*Select the sheet to import in the **XLSX Import Options** dialog.*

Special Excel Characters

Excel files can contain numeric cell formatting (codes like "#,###.00", for example). When an Excel file is imported, the worksheet stores the Excel number formatting internally so that the same formats are preserved when the file is saved. The worksheet only understands a subset of the codes. When it finds a character that it does not understand, the worksheet prints the literal format character. The worksheet displays a reasonable representation of the unsupported Excel characters so the value displayed in the cell looks similar to what was displayed in Excel.

Here is a summary of all the Excel number format characters that the worksheet understands:

Character	Character Value	Definition
\	Escape	The character following the escape is treated as a literal character. It is not interpreted as formatting character.
"	Quote	All characters between the first quote and the next quote are printed exactly as they appear. Quote characters themselves can printed by using the escape sequence \"
#	Digit placeholder	If the number has more digits to the right of the decimal point than there are number of characters, the number is rounded to as many decimal places as there are number of characters. If the number being printed has more digits to the left of the decimal point than there are number of characters, all the digits are printed.
0	Digit placeholder (Force a digit)	This is the same as the # character except that if there are fewer digits in the number than there are 0 format characters then a 0 is printed in the same place it appears in the format string.
?	Digit or space digit place holder	This is the same as the 0 character except that this prints a space instead of 0. In Excel this is also used for formatting fractions, but this meaning is not supported by the worksheet.
.	Prints a decimal point	This determines how many digits are printed to the right and left of the decimal point depending on how many digit formatting characters, # or 0, appear on either side of the decimal point in the format string. The actual character printed may not be a period '.' depending on the number format preferences set in the control panel.
,	Comma separate	If this format character appears anywhere in a format string, except the exponent, then the number is printed with thousands separators or whatever primary digit grouping is specified in the control panel.
%	Percent	If this format character appears anywhere in a format string, except the exponent, then the number is multiplied by 100. The percent character is printed in the same place as it appears in the format string.
e+	Exponent	Exponents specify scientific notation. The e+ or e- specification must be followed by one or more of the digit formatting characters (# or 0). The number of digit formatting characters after the e+ or e- determines how many exponent digits are printed. The E+ or e+ notation prints a minus sign in front of negative exponents and a plus sign in front of positive exponents. The E- or e- notation prints a minus sign in front of negative exponents. The exponent values displayed are an even multiple of the number of digit formatting characters to the left of the decimal point. For example, in the format string "##0.0e+##" there are three digit formatting characters left of the decimal point. This means that possible exponent values are e-06, e-03, e+00, e+03, e+06, etc.

E+	Exponent	See above
e-	Exponent	See above
E-	Exponent	See above
$	Currency	This character is replaced with the local currency character.
_	Variable width	The underscore and the character that follows it are replaced with a space. In Excel the space is the same width as the character after the underscore.
*	Repeat	The character after the asterisk is printed. In Excel the character after the asterisk is printed enough times to fill the cell width.
;	Terminator	Marks the end of the format string. Nothing after the semicolon will be used for formatting. In Excel, a format string can have several parts separated by semi-colons: "positive fmt;negative fmt;zero fmt;text fmt."
[color]	Specifies the color to print the number	One of the recognized color names must appear between the square brackets. It is invalid to use square bracket anywhere else in the format string. Recognized color names are BLACK, WHITE, RED, GREEN, BLUE, YELLOW, MAGENTA, and CYAN.

Characters Not Currently Supported

All other characters are printed exactly as they appear in the format string. Excel format characters not currently supported include:

- ; Excel allows several different formats per cell depending on whether the number is positive, negative, or zero. The worksheet will only use the positive portion of the Excel format string.
- [*condition*] Excel allows numbers to be formatted according to their value (i.e. if the number is greater that 100 use one format, otherwise use a different format). The worksheet does not support these options.
- @ Text place holder (text in the cell replaces this character)
- m Month or minute
- mm Month or minute
- mmm Month
- mmmm Month
- d Day of month
- dd Day of month
- ddd Day of week
- dddd Day of week
- h Hour
- hh Hour
- AM/PM Displays times in AM PM format instead of 24 hour format
- am/ pm
- A/P
- a/p

The import filter converts most of the cells with a time and date format into appropriately formatted strings. The cell appearance is similar, although math functions are not available on the imported strings. Note: if the file is saved in Excel format, the original Excel-formatted dates will be lost.

XYZ Grid [.DAT] File Description

Surfer exports to the XYZ grid file format.

The XYZ file is a simple encoding of 3D points into an ASCII text file. Each line of the file contains three numbers. Each triplet of three numbers comprises the X, Y, and Z coordinates, respectively, of a point in 3D space.

Import Options Dialog

Surfer does not import the XYZ .DAT grid file. You can open the file using the File | Open command. The file will open in the **Surfer** worksheet window. No import option dialog is displayed.

Import Automation Options

No import options are available.

Saving Files

Use the Grids | Edit | Convert command in the plot document, or **File | Save As** in the Grid Node Editor to export .DAT files. Alternatively, the XYZ .DAT grid file format is available when prompted to save a grid file.

Export Options

No export option dialog is displayed.

Export Automation Options

No export options are available.

XYZ Points [.CSV, .DAT, .TXT, .XYZ] File Description

Surfer plots can be exported to a .DAT or .CSV data file that includes XYZ coordinates and attribute values. The first three columns are the X, Y, and Z coordinates followed by columns for each attribute. Exporting a CSV file will include column headers as the first line of the CSV file. The Z coordinate column can be left out of the DAT or CSV file in the **XYZ Options** dialog. Specify the *Scaling source* in the Scaling page of the **Export Options** dialog to export the coordinates in page or map units. The XYZ Points format is a vector format. Only vector objects will be written to the XYZ points data file.

XYZ Export Options Dialog

See XYZ Points File Export Options Dialog

See Export Options - Scaling Page

See Export Options - Spatial References

XYZ Export Automation Options

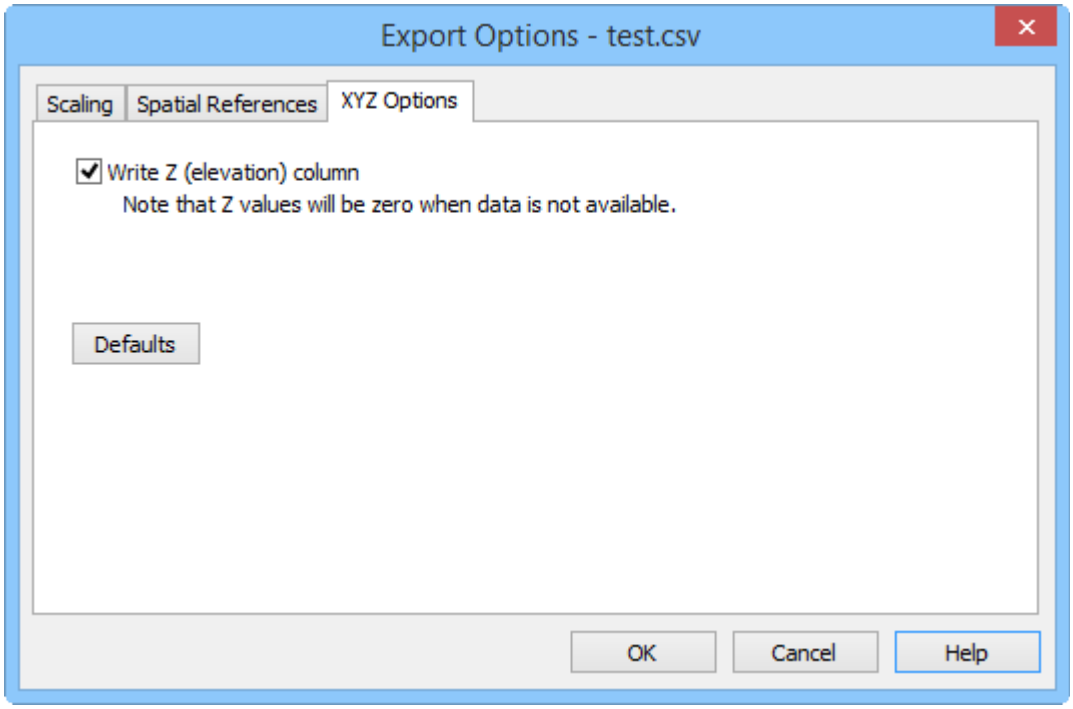
See XYZ Points File Export Automation Options

XYZ Points Export Options Dialog

Surfer can export plot documents as CSV or DAT XYZ points files.

XYZ Options Dialog

The **XYZ Options** page of the **Export Options** dialog contains the XYZ points export specific options.



*Specify export options for the DAT or CSV XYZ points file in the **XYZ Options** dialog.*

Z Column

By default the third column in the exported file contains the Z coordinate information for the points. When an object does not have Z coordinate information, the Z column value is 0. When the *Write Z (elevation) column* check box in unchecked the DAT or CSV file only includes X and Y coordinates and attribute values. The *Write Z (elevation) column* can be unchecked when no objects have elevation data or when the elevation data is undesired in the output file. Click the check box to check or uncheck the *Write Z (elevation) column* option.

Defaults

Click the *Defaults* button to restore the default XYZ export options, where *Write Z (elevation) column* is checked.

XYZ Points Export Automation Options

Since the **Export Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various export options.

Option	Action	Default	Description
WriteZ	0 = No 1 = Yes	1	Set to 1 to write the Z values in the third column of the data file. If a Z value is not available, 0 will be written in the third column. Set to 0 to only export the X and Y coordinates.
ScalingSource	0 = previously saved 1 = application-supplied	1	Use previously saved or application-supplied scaling source.
SaveScalingInfo	0 = No 1 = Yes	0	Save scaling parameters for later use.
PageLLX	N.N		Set application page rectangle lower left X value.
PageLLY	N.N		Set application page rectangle lower left Y value.
PageURX	N.N		Set application page rectangle upper right X value.
PageURY	N.N		Set application page rectangle upper right Y value.
FileLLX	N.N		Set scaling rectangle lower left X value.
FileLLY	N.N		Set scaling rectangle lower left Y value.
FileURX	N.N		Set scaling rectangle upper right X value.
FileURY	N.N		Set scaling rectangle upper right Y value.

ZMAP Zmap Grid File Format

Surfer imports and exports Z-Map Plus ASCII grid files. This import filter only supports the GRID type of the Z-Map Plus formats.

File Description

Z-Map Plus grid files are used by Geographix applications such as IsoMap to store gridded elevation data.

The format has several file extensions: .ASC, .DAT, .GRD, .XYZ, .ZMAP, .ZYC, .ZYCOR

The data are written a column at a time from the left side to the right side of the grid.

Loading Z-Map Plus Files

Z-Map Plus .ASC, .DAT, .GRD, .XYZ, .ZMAP, .ZYC, .ZYCOR files can be loaded to create a grid based map (e.g. **Home | New Map | Contour**) with the Open Grid dialog, or when using the functions in the **Grid** menu.

Import Options Dialog

See ZMap Import Options Dialog

Import Automation Options

See ZMAP Import Automation Options.

Export Options Dialog

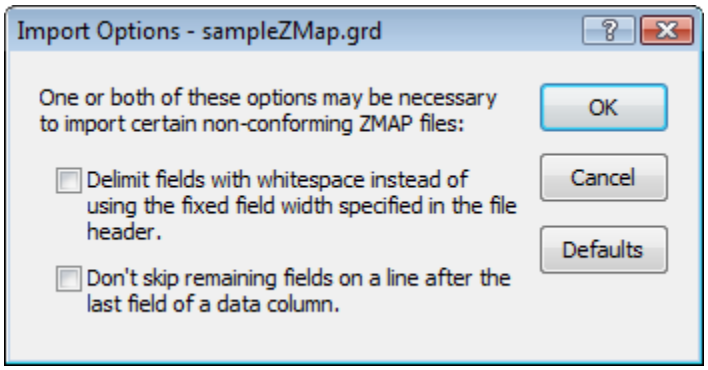
When the input grid is georeferenced, see Spatial References

Export Automation Options

No export options are available.

ZMap GRD Import Options Dialog

When selecting a ZMap grid file, the following **Import Options** dialog appears. Set the options and click *OK* to create the map from the grid.



*Customize the field delimiter in the **Import Options** dialog.*

Field Delimiter

Normally the data fields in a ZMAP grid file are of fixed size and separated by spaces or tabs. But this is not required by the ZMAP grid format. When the *Delimit fields with whitespace instead of using the fixed field width specified in the file header* box is unchecked, most ZMAP grid files will be imported into **Surfer** correctly. Checking the box allows files to be read where the fields are not separated by spaces or tabs. By default, this box is unchecked.

Skip Non-Standard Fields

Most ZMAP grid files have a set number of fields on each line. Some non-conforming ZMAP grid files contain additional placeholders after the set number of fields. These fields are ignored when the

Don't skip remaining fields on a line after the last field of a data column box is checked. When unchecked, these additional fields are imported as the first several fields of the next row. By default, this box is unchecked.

ZMAP Z-MAP Grid File Import Automation Options

Since the **Import Options** dialog is not displayed when the program is driven from an automation script, an options string can be specified in the script. The string consists of comma-separated parameters, which specify the behavior of the various import options. A typical example would be:

"Defaults=1,WhitespaceDelimiters=1"

This would first set all import options to their default values, then set the WhitespaceDelimiters option value to 1, which specifies white space as the data field delimiters.

Option	Action	Default	Description
WhitespaceDelimters	0 = Delimit data fields with fixed field width 1 = Delimit data fields with whitespace	""	Specifies how data fields are delimited upon import
DontSkipExtra	0 = skip any remaining fields on a line after the last field of a data column 1 = Don't skip remaining fields on a line after the last field of a data column	""	

Index

1-grid vector maps, 619
2-grid vector maps, 622
3D DXF, 99, 1002
3D DXF data, 80
3D label lines, 553
3D SHP, 99, 1002
3D surface, 567, 570, 572, 574, 708, 745
3D view points, 1034
3D wireframe, 586, 587, 958
95% and 99% confidence interval for the mean, 183
95%confidence interval for the mean, 183
99% confidence interval for the mean, 183
a brief introduction to the major Surfer automation objects, 1089
a gridding example, 197
a note about the documentation, 74
about Surfer, 71
absolute value, 150, 942
accessing objects, 1087
accessing Surfer, 1087
accidentally move, 978
accumulated horizontal distance, 438
across then down, 1020
active, 214
active cell, 36, 138, 140, 141
active cell edit box, 138
active cell location, 138
active cell location box, 140
ActiveX automation members, 1078, 1087
ActiveX automation objects, 1087
ActiveX object, 1077
actual size, 994
Add, 121, 767
add a color scale bar to a 3D wireframe, 713
add a color scale bar to a filled contour map, 713
add a map background color, 692
add a map layer, 767
add blank line, 1242
add component, 306
add contour labels, 98, 526
add contour levels, 503
add data source, 672
add grids, 447
add levels, 503
add map, 767
add map source, 672
add method, 1095
add scale bar, 696
add sever, 672
add text, 116, 866
add, delete, and move contour labels, 98
adding a color scale, 109
adding a color scale bar, 708, 713
adding a map layer, 107, 767
adding a map title, 112
adding a mesh, 105
adding a post map layer, 114
adding a shaded relief map layer, 110
adding a unique column identifier, 86
adding an overlay, 107, 767
adding color zones to a 3D wireframe, 588
adding labels to the post map, 100
adding new data, 86
adding transparency to map layers, 109
ADF, 390, 1153
adf export, 1153
adf export dialog, 1153
adjacent maps, 54
adjust to, 1016
adobe acrobat [.PDF] export automation options, 1303
adobe acrobat [.PDF] raster export options dialog, 1299
adobe acrobat [.PDF] vector export options dialog, 1301
advanced contour level properties, 95
advanced options, 258, 259, 260, 264, 271, 1039
advanced settings, 1039, 1044
advanced tutorial lessons, 118, 121, 124
affected levels, 510, 514, 516, 518, 522, 527, 530
albers, 193, 817, 827
albers conic, 827
albers equal area, 827
Align, 982, 985
align maps, 770, 772
align objects, 982
aligning several maps on the same page, 772
alignment, 166, 867, 919
all data search, 226
all files, 158
all recognized types, 158
all sessions, 1039
always reset, 1039
always split, 1057
AM, 390, 1154, 1156
ambient, 572, 745
Amira Mesh [.AM, .COL] Export Automation Options, 1156
amira mesh [.AM, .COL] files, 1154
Analyze 7.5 Medical Image [.HDR, .IMG] Export Automation Options, 1266
analyze 7.5 medical image [.HDR, .IMG] export options dialog, 1265
anchor color, 950
anchor nodes, 954
angle, 227, 229, 264, 299, 539, 561, 729
angle (direction) data, 626
angle column, 537, 561
angle component grid file, 621
angular divisions, 214, 291
anisotropy, 229, 284, 299
ANOVA, 214
antarctic, 826
antialias lines, 1034
antialias redraw, 1034
antialias text, 1034

apostrophe, 1068
Application Object, 1087, 1096
Application Property, 1096
apply a gradient to class symbols, 550
approximate density, 265
arctic, 826
area, 419, 423, 428, 429, 762
area reshape, 895
area to download, 663
area to line, 893
arithmetic average, 187
arrange icons, 996
arrange lock, 978
arrange lock position, 978
arrange maps, 770
arrange menu commands, 974, 976, 979, 980, 981, 982, 987, 988, 989
arrange objects, 27
array variables, 1081
arrays, 1083
artificial data, 214
ascending, 174
ASCII, 1002
ASCII DAT, TXT, and CSV data files, 1131
ASCII files, 1131, 1241
ASCII grid file format, 1241
ascii import, 153, 1133
ASCII table, 174
assign columns, 192
assign coordinate, 395, 817
assign coordinate system, 193, 395, 809, 813, 817, 826, 827
assign projection, 193, 809, 813, 817
assign xyz columns, 192
assigning color fill based on a fill spectrum, 520
assigning color fill to specific contour levels, 520
assigning colors to a color spectrum, 950
assigning coordinates to an image base map, 778
assigning fill and line properties to blanked areas, 521
assigning gradational line properties, 512
assigning line properties to specific contour levels, 515
assigning the same line properties to all contour levels, 513
asterisk, 19
Atlas BNA, 130, 1123, 1171, 1172, 1173
Atlas BNA Examples, 1173
atlas boundary [.BNA] export options dialog, 1171
atlas boundary export automation options, 1172
atlas boundary import automation options, 1170
attribute query, 795
attribute search, 795
auto hide, 27
auto redraw, 995
AutoCAD DXF export, 99, 1002
AutoCAD DXF export automation options, 1211
autoCAD DXF export options dialog, 1208
AutoCAD DXF import, 465, 999
autoCAD DXF import automation options, 1206
autoCAD DXF import options dialog, 1203
autocorrelation estimator, 295
autocovariance estimator, 295
AutoFit, 303
automatic scaling, 593, 599, 733
automatic variogram fitting dialog box, 303
automatically check for updates, 1028
automation, 1100
automation example, 1102, 1103, 1104, 1106, 1107
automation examples, 1102
automation help, 1087, 1089
available date formats, 133
available date time formats, 133
available time formats, 133
average, 187
average data value, 214
average deviation, 183
average distance, 265
average every, 898
AVS, 1229, 1230
AVS Field [.AVS] Export Automation Options, 1230
avs field [.FLD] export options dialog, 1229
AVS X-Image [.XIMG] Export Automation Options, 1230
Axes Collection, 1087, 1095, 1107
axis attributes, 729
axis date/time labels, 729
axis direction, 733
axis labels, 729
axis limits, 733
axis maximum, 733
axis minimum, 733
Axis Object, 1087, 1107
axis plane, 729
axis position, 733
axis properties, 729, 730, 733, 738
axis scaling, 733
axis tick labels, 102
azimuthal, 829
azimuthal equidistant, 829
azimuthal projection, 829
back, 980
backcolor, 1044
background, 692
background cell, 167
background color, 518, 520, 867, 923
background map color, 692
background mode, 518
backup files before saving, 1025
backward, 981
banded interleave, 1160
BAS, 1102
base, 578, 589
base map, 458, 463, 465, 663
base map from data, 458
base map from server, 663
base map general page, 465
base map properties, 463, 465, 487
base object, 1095
base parameters, 589
BASIC, 1059
basin, 613

basin color, 613
basis kernel functions, 260
bathymetric, 1148
between cols, 390
between rows, 390
bezier, 883
biharmonic operator, 412, 414
BIL, 390, 1160
bilinear, 574
bilinear interpolation, 447, 450
BIN, 390, 1318
binary grid file (Surfer 6 format), 1240
binary lattice files - export options dialog, 1316
binning method, 547
bins, 547
BIP, 390, 1160
bitmap (image), 916, 999, 1002, 1123, 1141, 1144
bitmap (image) coordinates, 778
bitmap (image) export, 1002
Bitmap (Image) Export Automation Options, 1144
bitmap (image) import, 465, 999
bitmap (image) pattern, 927
bitmap (image) properties general page, 947
black and white, 1020
blank, 362, 375, 387, 1160
blank cells, 170
blank inside, 375, 777
blank line between grid rows, 1242
blank node, 362
blank node when searching, 224
blank outside, 375, 777
blank value, 362, 431
blanked areas, 375
blanked grids and volume, 430
blanked nodes, 380, 389, 567
blanked planar area, 423
blanked regions, 493, 521, 590, 593
blanking a grid file, 124, 375
blanking file, 130, 233, 438, 775, 777, 1123
blanking file format, 1160
blanking with search ellipse, 227
blend colors, 950
blend overlay and surface colors, 574
BLN, 130, 233, 375, 438, 775, 777, 1123, 1160, 1163, 1165
block diagram, 567
block select, 973
BMP, 999, 1002, 1123
BNA, 130, 1002, 1123, 1170, 1171, 1172, 1173
BNA curve example, 1173
BNA file example, 1173
BNA point example, 1173
BNA polygon, 1173
bold, 867, 919
bonne, 830
bonne projection, 830
boolean, 209, 1080
boolean expressions, 209
border, 578, 692
boundary, 458, 999, 1002, 1123
boundary files, 42
boundary number, 438
break a line, 902, 903, 1067
break apart, 777, 989
break apart complex polygon, 912
break apart islands, 912
break apart islands/lakes, 912
break apart lakes, 912
break apart layer, 777
break apart overlay, 777
break at intersections, 903
break line, 902, 903
break polyline, 902, 903
breakline example, 234
breakline format, 234
breaklines, 233, 234
breaklines and faults, 233
breakpoint, 1064
browse, 1078
brush, 347
brush grid, 347
BSQ, 390, 1160
buffer, 906
buffer polygon, 906
buffer zone, 906
built-in functions and procedures, 1070
bulk contour levels, 95
button appearance, 1047
button persistence, 1025
byte, 1080
calculate area, 762
calculate contour levels by logarithm, 495
calculate distance, 762
calculate perimeter, 762
calculating the agreement between a grid and the original data file, 445
calculating the volume between two surfaces, 429
calculating the volume of a blanked grid, 430
calculating the volume of a lake, 431
calculating the Z value of a surface at a point, 447
calculation, 157, 170
calculus, 397, 400, 406, 407, 408, 409, 410, 411, 412, 415
cancel, 35
can't find coordinate system, 813
cartesian data, 621, 626
cascade windows, 996
cassini, 831
cassini projection, 831
cell alignment, 166
cell background, 167
cell edit box, 140, 141
cell format, 157
cell number format, 165
cell numeric format, 165
cell properties, 165, 166, 167
cells, 130
center, 867
center on page, 1018
centimeters, 1025
central difference, 599
change all symbols at once, 550
change blank value, 431
change data file, 535, 544, 561
change default columns for XYZ data, 192
change grid, 567
change grid file, 493, 593, 599
change ID, 975
change image coordinates, 778
change individual contour level, 505
change points, 894, 895
change polygon shape, 893, 895

change polyline shape, 892, 893, 895
change projection in worksheet, 193
change the page setup, 1012
change the view, 678
change units, 1025
changes, 80
changing base map properties, 463
changing colors, 106
changing properties in a base map, 463
changing the grid node editor, 25
changing the layout, 25
changing the managers, 25
changing the map limits, 631
changing the menu bar, 25
changing the object manager location, 27
changing the opacity of a color spectrum, 950
changing the plot window, 25
changing the post map properties, 99
changing the toolbars, 25
changing the worksheet, 25
changing tick label properties, 102
changing tick labels, 102
characteristics of projections, 824
check for update, 70, 1028
choosing gridding methods, 237, 238, 242
choosing methods based on the number of XYZ data points, 238
circle, 889
class list box, 547
class module, 1076; example, 1077
class scatter color file format, 541
class symbol fill color, 550
class symbol gradient, 550
class symbol properties, 550
class symbol set all symbols the same, 550
class symbol set all symbols to a color range, 550
class symbol size dialog, 552
class symbol size properties, 552
class symbol size range, 552
class symbol uniform, 550
classed post layer, 541
classed post legend, 702
classed post map, 541, 544, 547, 553, 561
classed post map changes, 80
classes page, 547
classical variogram estimator, 295
clear, 157
clear selected cells, 157
clear transform, 974
click tolerance, 1033
clip map, 688
clip symbols at map limits, 627, 631
clipping symbols on vector maps, 631
close, 67, 1103
close group, 989
close method, 1095
close polylines, 892
close script, 1059
closing the object manager, 27
CLP, 1123
CLP export, 1002
CLP import, 465, 999
CLR, 106, 627, 950, 954
CLS, 547
Code, Class, and Object Modules, 1075
coefficient of kurtosis, 184
coefficient of skewness, 185, 214
coefficient of variation, 186, 214, 265
coefficient of variation filter, 385
COL, 390, 1154, 1156
collate, 1022
collection objects, 1087, 1095
collections, 1087
color, 106, 867, 950, 959, 960, 962, 1043, 1045
color column, 537
color fill, 520, 587, 950
color filled contours, 520, 950
color filled wireframe, 587
color list, 962
color method, 537
color modulation, 574
color names, 962
color on wireframe maps, 588
color palette, 959
color relief map, 593
color scale, 80, 109, 567, 593, 627, 708, 713
color scale bar, 708, 713
color scale bar visibility, 708, 713
color scaling, 627
color selection, 959
color spectrum, 950, 954
color spectrums, 949, 955
color syntax, 1043
color zones, 588
color zones page, 583, 589
colormap, 80
colormap dialog, 950
colors, 593, 599, 960, 1043
colors dialog, 960
column, 147, 149, 150, 168
column formats, 157, 168
column labels, 139
column letters, 138
column width, 139, 147, 168
columns to rows, 191
combine, 767, 988
combine boundaries, 902, 903, 913
combine curves, 902
combine grids, 447
combine islands, 913
combine islands/lakes, 913
combine lakes, 913
combine lines, 902
combine maps, 54, 91, 767, 971
combine multiple polygons, 913
combine multiple polylines, 902
combine polygons, 903, 913
combine polylines, 902
combining maps, 54, 767, 768
combining maps when exporting, 1006
comma delimiters, 1131
command line, 1063
comments, 1068
common graphic files, 999

comparison of gridding methods, 237
compass gradient filters, 385
compassed-based grid notation, 397
compatibility, 42, 68
compatibility with Visual BASIC, 1059
complex lines, 80
complex polygon, 913
complex polygon to multiple polygons, 912
composite object, 988, 989
compute transform, 170
concatenate, 170
conformal, 843
conjoin polygons, 913
connect curves, 902
connect lines, 902
connect multiple polylines, 902
connect points, 535
connect points in a post map, 535
connect points in post map, 80
connect polylines, 902
connection line, 535
contact us, 71, 79
contour export, 1002
contour fill, 518, 520, 522
contour interval, 93, 495, 503
contour labels, 98, 526, 527
contour level, 505
contour levels, 93, 344, 495, 503, 513, 515
contour levels dialog, 503
contour line properties, 94, 512, 513, 514, 515, 516, 521
contour lines, 99, 503, 510, 1002
contour map, 89, 492, 493, 713, 1102
contour map properties, 493
contour map properties dialog, 493, 695
contour smoothing, 453, 532
convergence, 254
convert, 390
convert boundary to polyline, 893
convert polygon to polyline, 893
convert polygons, 893
convert polyline to polygon, 892
convert polylines, 892, 893
converting to decimal degree, 861
convex hull buffer, 80
coordinate, 805
coordinate arrays, 1083
coordinate system, 57, 58, 193, 803, 804, 805, 807, 809, 813, 817, 824
coordinate system faq, 808
coordinate system for export, 1006
coordinate systems, 826
coordinates, 193, 463, 778, 817, 1123
coordinates in images, 778
copies, 1022
copy, 156, 917
copy server and layer name, 663
correlation, 214
correlogram, 415
count, 265
Count Property, 1095
covariance, 214
cover contours, 531
create a custom toolbar, 1047
create a data file with digitize, 775
create a grid report, 214
create a north arrow, 890
create colors, 960
create complex polygon, 913
create contour levels at regular intervals, 503
create custom line style, 931
create donuts, 913
create grid from contour map, 275
create grid from contours, 275
create islands, 913
create islands/lakes, 913
create lakes, 913
create multiple lines from one, 902
create multiple lines of text, 867
create multiple polygons from one, 912
create multiple polylines from one, 902
create template (text), 869
CreateObject, 1087
creating a 1-Grid vector map, 619
creating a 2-Grid vector map, 622
creating a blanking file with the digitize command, 777
creating a color spectrum file, 955
creating a contour map, 89
creating a cross section, 103, 438
creating a filled contour map and overlay on a wireframe, 587
creating a filled contour map containing blanked areas, 521
creating a grid file, 90
creating a level file containing contour level information, 506, 957
creating a new classed post map, 541
creating a new data file, 85
creating a variogram, 288
creating anchor nodes, 954
creating custom toolbars, 118
creating dialogs with Scripter, 1084
creating EPS files, 1227
creating index contours, 516
creating level files in the worksheet, 506, 959
creating post maps, 541
creating several maps in the same plot window, 771
criteria select, 795
critical value of K-S statistic at 90%, 95%, and 99% significance level, 186
crop, 375, 688
crop map, 688
cross section, 103, 438, 442, 723
cross validate, 211
cross validation references, 211
cross validation report, 214
cross X axis at, 733
cross Y axis at, 733
cross Z axis at, 733
CSV, 130, 1131

CTRL key when selecting, 971
cubic convolution, 447, 450
cubic model, 297, 306
currency, 1080
currency format, 165
current command, 34
current date, 1020
current map extents, 663
current session only, 1039
current time, 1020
current version, 70
cursor tracking, 798
curvature, 400, 405, 409
curve, 883
custom button appearance, 1047
custom colors, 960
custom contour label positions, 526
custom coordinate system, 193, 817
custom datum, 822
custom datum definition, 821
custom keyboard commands, 118, 1050
custom labels, 98, 526, 559
custom line style, 931
custom toolbars, 23, 1047
customize, 1047, 1054
customize commands, 1047
customize keyboard commands, 1054
customize menu, 1047
customize options, 1047
customize toolbars, 1047
customizing post map classes, 547
cut, 156, 423, 425, 918
cut and fill volume, 425
cut and fill volumes, 423
cycle spacing, 696
cyclic, 387
DAT, 88, 153, 390, 438, 1131, 1133, 1138
DAT Files, 130, 1131
data, 142, 289
data exclusion filter, 209
data export automation options, 1140
data export options dialog, 1138
data file, 18, 85, 86, 88, 130, 560, 561, 1123
data file base map, 458
data file formats, 130, 1123
data files, 42, 1123, 1131
data files used for posting, 560
data filters, 209
data import options, 153, 1133
data labels in the data file, 560
data layer information, 672
data link, 62
data location statistics, 265, 269
data menu commands, 157, 175, 193, 817
data metric, 269, 271
data metric other z statistics, 269
data metric terrain, 271
data metrics, 237, 265, 267, 268, 269
data metrics data location, 269
data metrics z moment, 268
data metrics z order, 267
data orientation, 229, 299
data page, 289, 621
data repeatability, 284, 312
data set size, 238
data sort, 174
data statistics, 214
data statistics report, 214
data to color mapping, 950
data transform, 86, 170
data used, 214
database, 62
database linking, 798
date, 867, 878, 1080
date format, 80, 165, 878, 942
date formats, 133
date time, 80, 131
date time format, 942
date time formats, 133
date values, 131
date/time, 131, 165
date/time format, 878, 942
datum, 822
datums, 813, 820, 821
DBF automation import options, 1178
DCX, 1123
DCX export, 1002
DCX import, 465, 999
DDF, 1123
debug: browse, 1078
Debug.Print, 1064
debugging scripts, 1064
decimal degrees, 861
decimal digits, 942
default angle post map, 537
default attributes, 1034, 1038
default columns for XYZ data, 192
default gridding method, 237
default linear variogram, 311
default map scale, 682
default path, 1025
default settings, 1020, 1025, 1034, 1036, 1039, 1041
default symbol, 1038
defaults, 1033, 1034, 1036, 1039, 1041, 1042, 1043, 1044, 1045, 1046
define coordinate system, 813
degree units, 621
degrees minutes seconds, 195, 861, 942
delete, 160, 918
delete anchor nodes, 954
delete contour label, 526
delete contour labels, 98
delete contour levels, 503
delete map in overlay, 54
delete method, 1095
delete selected cells, 160
deleted duplicates, 214
delimiter, 153, 1131, 1133, 1136, 1140
DEM, 43, 390, 492, 590, 1123, 1184, 1187
density, 265
depressions, 613
derivatives, 400
derived objects, 1094
descending, 174
descending axis, 733
deselect all, 974
destination column, 170
detrend, 291
detrending method, 214
deviation distance, 898
device independent bitmap, 916
dialog defaults, 1039, 1041, 1042, 1043, 1044, 1045, 1046
dialog editor, 1084

DICOM import options dialog, 1190
difference of polygons, 905
difference polygons, 905
different coordinate systems, 807
different projections, 807
differential and integral operations, 397, 412
differential and integral operators, 397, 412
diffuse, 572, 745
digital elevation model [.DEM], 43, 1123, 1184
digital filter, 380
digital filtering, 383, 385
digital terrain elevation data [.DTED], 1200
digital terrain elevation data [.DTED] file description, 1200
digitize, 775, 777
digits, 1046
dim, 1080
direction axis, 733
directional derivatives, 397, 400, 403, 405
display area, 762
display attribute on base map, 487
display distance, 762
display grid node labels, 338
display hidden columns, 139, 147, 149, 168
display hidden rows, 139, 147, 149
display labels, 338
display labels on grid nodes, 338
display perimeter, 762
displaying a filled contour map without contour lines, 521
displaying date/time on maps, 131
displaying dates on maps, 131
displaying different coordinate system, 807
displaying different projections, 807
displaying hidden columns or rows, 149
displaying times on maps, 131
distance, 265, 293, 762
distance scale, 696
distance to farthest, 265
distance to nearest, 265
distance weighting filter, 383
distortion, 686
DLG, 1123, 1184, 1193, 1197
DLG import, 465, 999
dms, 195, 942
dms format, 942
dms to dd, 195
dock object manager, 27
dock toolbars, 1047
docking mechanism, 27
Document Object, 1094, 1096
documentation, 74
Documents Collection, 1095, 1103
don't draw blanked nodes, 567
double, 1080
down then across, 1020
download, 663
download base map, 663
download base maps, 80
download data, 663
download maps, 663
download online maps, 663
download worldwide data, 663
downloads, 70, 74
drag an object, 974
drag and drop map layer, 54, 768
drag labels, 526
drag toolbars, 1047
drape maps on wireframes, 54
draw contours, 38
draw limits, 757
draw lines of constant, 570
draw map limits, 757
draw menu, 866, 880, 881, 882, 887, 888, 889
draw mesh lines on a 3D surface, 570
draw polyline, 880
draw text, 867
drawing contours, 492
drawing grid, 996
drawing tips, 880, 881, 882, 887, 888, 889
drift type, 291
DTED, 1200
duplicate data, 175, 209
duplicate window, 996
DXF, 800, 1002, 1123, 1203, 1205, 1206, 1208, 1211
dynamic linking, 798
E00, 1123, 1213, 1215, 1216
e00 DEM, 1216
e00 grid, 1216
EatWhiteSpace, 1136
eckert, 832, 833
eckert IV, 832
eckert VI, 833
ECW, 1217, 1218
edge effect, 397
edge of grid, 380, 387
edit: properties, 1076
edit class module properties, 1076
edit contour, 38
edit contour labels, 98, 526
edit contours, 38
edit grid, 38
edit grid value, 38
edit group, 989
edit labels, 526, 559
edit major tick spacing on an axis, 737
edit map source, 672
edit maps in an overlay, 54
edit menu, 32, 151, 156, 161, 164, 914, 915, 916, 917, 918, 973, 974, 975
edit overlay, 54
edit polygon, 895
edit polyline, 895
edit post labels, 559
edit properties, 32
edit server, 672
edit surface, 38
edit template (text), 869
edit text, 867
editing a color scale bar, 713
editing an existing 1-Grid vector map, 619
editing an existing 2-Grid vector map, 622
editing an existing classed post map, 541
editing cell contents, 141
editing color zones, 583
editing contour label positions, 526
editing the data files in the worksheet, 145
editing the worksheet, 157
editing variogram properties, 292
editing worksheet cells, 36, 140, 141
editing z levels zones, 581

elevation, 589
elevation at a point, 447
elevation export, 1002
ellipse, 227, 889
ellipsoids, 818
email, 79
EMF, 1123, 1220, 1221, 1222, 1354, 1355
emf import, 1220
emf import automation, 1355
EMF import automation options, 1220
empty base map, 461
empty cells, 170
enable labels, 338
enable labels on grid node editor, 338
encapsulated postscript [.EPS] export automation options, 1225
encapsulated PostScript [.EPS] files, 1227
encapsulated postscript export options dialog, 1223
end, 1063
end sub, 1059
enhanced metafile, 916
enter group, 116, 989
enter group and add text to a base map layer, 116
entering data, 36, 140, 141, 142
entering data into a cell, 142
entire data source extents, 663
EPS, 1223, 1225, 1227
equal area, 842
equal intervals, 547
equal number, 547
equation template, 867
equations, 170
equidistant, 834, 835
equidistant conic, 834
equidistant cylindrical, 835
er mapper, 1218
ER Mapper [.ECW] Import Automation Options, 1218
ER mapper [.ECW] import options dialog, 1217
eraser, 362
error strings and special numeric values, 150
error variance, 312
ERS, 390
ESRI, 1123, 1327, 1328, 1330
ESRI ArcInfo export format (E00) import automation options, 1215
ESRI ArcInfo export format (E00) import options dialog, 1213
ESRI Shape Export Automation Options, 1330
ESRI Shape Import Automation Options, 1327
ESRI shapefile export options dialog, 1328
estimate, 211
estimator type, 293, 295
evenly spaced data, 281
exact and smoothing interpolators, 242
exact interpolators, 242
examples, 197, 285, 429, 431, 445, 447, 514, 516, 522, 527, 530, 626, 1102
examples - Scripter, 1086, 1102, 1103, 1104, 1106, 1107
excel export options dialog, 1359
Excel files, 68, 1136, 1140, 1360
Excel Import Options Dialog, 1359
exclude data within this distance, 211
excluded, 214
exit, 73
exit group, 989
expand, 389
expand a grid, 390
expand object manager, 1025
experimental curve, 287
experimental page, 293
explicit color names, 537
exponent zeros, 942
exponential format, 942
exponential model, 297, 306
export, 1002, 1005, 1006, 1009, 1123
export 3D contour lines, 99, 1002
export contour lines, 99, 800, 1002
export drawing objects in map units, 1006
export gsi, 1250
export kml, 1275
export map and non map objects, 1006
export multiple maps, 1006
export nc, 1296
export netcdf, 1296
export options - scaling page, 1005
export options dialog, 1005, 1009, 1268
export options string, 1140
export profile data, 723
export scaling, 1005
export variogram, 313
exporting a variogram, 313
extended Simpson's 3/8 rule, 427
extended Simpson's rule, 427
extended trapezoidal rule, 427
extract, 451
extract data, 772
extract grid, 451, 772
extract subset of grid, 451
extract Z value, 447
eyedropper, 365, 960
FaceName, 1045
faq coordinate system, 808
faq projection, 808
fault line properties, 493
faults, 233
faults and breaklines, 233
fax, 79
field of view, 678, 761
file: new module, 1075
file export, 1002
file format chart, 1123
file formats, 130, 1123, 1131, 1237, 1240, 1241
file import, 158, 465, 999
file list, 72, 158, 1025
file list (window menu), 72
file menu, 59, 60, 62, 67, 68, 72, 73, 693, 999, 1002, 1012, 1015
file menu - grid node editor, 38
file name, 68, 1020
file names, formats, and file extensions, 68
file preferences, 1025, 1036
file type, 1275, 1277
file type not recognized, 1001, 1130
file types, 42, 158

fill, 32, 423, 425, 518, 520, 522, 923, 925
fill and line properties, 567
fill blanks, 389
fill contours, 518
fill defaults, 1038
fill depressions, 613
fill dialog, 518, 925
fill edge, 387
fill in a sparse grid, 390
fill palette, 926
fill pattern, 518, 520, 926, 927
fill pattern defaults, 1044
fill pattern names, 927
fill pits, 613
fill polyline, 892
fill properties, 518, 521, 692, 923, 925
fill properties dialog, 518
fill property syntax, 1044
fill spectrum, 520
filled contours, 493
filled wireframe, 587
filter, 380
filter data, 175, 209
filter size, 380
filtering grids, 453
filters, 380, 383, 385, 387, 389
final grid size, 390
find, 161, 164
find and replace, 161, 164
find attribute, 795
find next, 161, 164
find objects, 795
find the Z value at a known X and Y coordinate, 447
finding the volume between two surfaces, 429
first, 514, 522, 527, 530
first derivative, 400, 403
first major tick, 733
fit criterion, 303
fit to, 1016
fit to page, 1015
fit to window, 991
fixed format, 942
fixed symbol size, 539
flat areas on wireframes, 590
FLD, 390, 1229, 1230
FLD automation export, 1230
flip, 191
flip axis direction, 733
flip columns and rows, 191
flip map, 733
flip rows and columns, 191
flow control, 1068
flow map, 613
FLT, 390
font, 867
font defaults, 1038, 1045
font properties, 919
font property syntax, 1045
footer, 1013, 1020
forecolor, 1044
foreground color, 518, 520, 923
format, 553, 1131
format builder, 878
format cells, 165, 166, 167
format column width, 168
format date time, 133
format dates, 133
format menu commands, 157
format row height, 169
format time, 133
formats for attribute values, 1042
formatting the data, 157
formulas, 170
forums, 74
forward, 980
fourier and spectral analysis, 397, 415, 417
fourier frequency, 417
frame, 692
free rotate, 987
frequency, 514, 522, 527, 535, 544, 570, 627
from edge, 1018
front, 979
full screen, 995
function, 279
function types, 260
functions, 170, 1070, 1097
gauss conformal projection, 856
gaussian low-pass filter, 383
gaussian model, 297, 306
gauss-kruger projection, 856
GCS, 836
general - preferences, 1025
general base map, 465
general gridding options, 43, 224, 229, 233, 299
general gridding recommendations, 237
general page, 291, 465, 493, 535, 544, 567, 578, 593, 599, 729, 947, 1025
general user-defined filter, 383
generate report, 214
geode, 663
geographic coordinate system, 825, 836
geometric forms of projections, 822
geoprocessing menu commands, 906
georeference, 779, 860
georeference an image base map, 778
georeference image, 779
geosoft, 1236
geosoft grd, 1236
geosoft grid file description, 1236
geostatistical reference, 332
get variogram, 314
GetFilePath, 1083
getting help, 74
getting started guide, 80
getting user input, 1083
ggf, 1236
GIF, 1123
GIF export, 1002
GIF import, 465, 999
global data, 1148
global elevation, 1148
global variables, 1082
GLOBE DEM data ?10g file description, 1148
gnomonic, 837
gnomonic projection, 837
golden software blanking export automation options, 1165
golden software blanking export options dialog, 1163
golden software blanking file description, 1160
golden software blanking import automation options, 1163
golden software blanking import options dialog, 1160
golden software boundary export automation options, 1248
golden software boundary export options dialog, 1247

golden software boundary import automation options, 1247
golden software interchange export automation options, 1252
golden software interchange import automation options, 1250
golden software PlotCall import automation options, 1305
golden software reference, 860
goodness of fit statistic, 187
google earth export, 1277
gouraud, 572, 745
gps, 1235
gps exchange format, 1235
gpx, 1235
gradational class symbol size, 552
gradational lines, 510
gradient method, 599
gradient operator, 412, 413
graphic files, 999
GRD, 390
grd geosoft, 1236
grid (drawing), 996, 1036
grid a data set, 197
grid assign coordinate system, 395, 817
grid blank, 362, 375
grid calculus, 397, 400, 406, 407, 408, 409, 410, 411, 412, 415, 454
grid calculus references, 454
grid contours, 275
grid convert, 371, 390
grid coordinate, 395, 817
grid data, 211
grid divisions, 1036
grid done notification, 1025
grid editor, 38
grid erase, 362
Grid Exchange [.GXF] Export Automation Options, 1255
grid exchange [.GXF] export options dialog, 1254
grid export blank line, 1242
grid export number of values, 1242
grid export values per line, 1242
grid extents, 447
grid extract, 451
grid feature classes, 275
grid file format, 1237, 1240, 1241
grid files, 42, 281
grid filter, 380, 383, 385, 387, 389, 453
grid filter references, 389
grid from base layer, 275
grid from contours, 275
grid from server, 274, 663
grid from vector, 275
grid function, 279
grid info, 367, 368, 380, 390, 397, 431, 435, 447, 453, 493, 567, 578, 593, 599, 621
grid line geometry, 447
grid lines, 738
grid math, 431, 434, 435
grid menu, 38, 279, 288, 313, 375, 380, 387, 389, 390, 397, 431, 435, 438, 447, 451
grid mosaic, 447
grid node, 38
grid node editor, 38, 340, 341, 344, 362, 367, 914
grid node editor commands, 35
grid node editor enable labels, 338
grid node editor labels, 338
grid node editor show labels, 338
grid node labels, 338
Grid Object, 1087
grid operations references, 454
grid options, 1244
grid periodogram, 417
grid point sample, 445
grid project, 395
grid report, 214
grid residuals, 445
grid slice, 438
grid spline smooth, 390
grid stats, 453
grid transform, 435
grid variogram, 288
grid vector data, 275
grid volume report, 423
gridded data, 281
gridding, 43, 237
gridding a function, 279
gridding example, 197
gridding faults, 233
gridding method comparison, 239
gridding methods, 237, 238, 239, 242, 258, 259, 260, 264, 265, 271
gridding overview, 43
gridding parameters, 90
gridding report, 214
gridding with breaklines, 233
gridding with faults, 233
gridlines, 1020
GridWindow, 1095
GridWindow Object, 1096
group, 988, 989
GS ASCII, 390, 1241
GS Binary, 390, 1240
GS PlotCall, 1305
GS_Surfer2, 916
GSB, 1123, 1247, 1248
GSI, 1123, 1250, 1252
gsi export, 1250
gsr, 860
gsr2, 860
GXF, 390, 1254, 1255
hachures, 530
handle size, 1033
handle size in pixels, 1033
handling duplicate data, 209
HDF, 390, 1257, 1258
HDR, 1265, 1266
head length, 629
header, 1013, 1020
height, 978
help, 73, 74
help menu, 70, 71, 1054
hidden columns, 139, 149, 168
hidden lines, 54
hidden rows, 149, 169
hide a map in an overlay, 54
hide object manager, 27
hiding columns, 139, 147, 149, 168
hiding rows, 139, 147, 149, 169
Hierarchical Data Format [.HDF] Export Automation Options, 1258
hierarchical data format [.HDF] export options dialog, 1257
hierarchy, 1087, 1100

histogram, 308
hole-effect model, 297, 306
honoring data, 237, 242
horizontal alignment, 867
horizontal distance, 438
horizontal light position, 572, 599, 745
horizontally align maps, 770
hotine, 838, 840
hotine oblique mercator, 838
hotine projection, 840
hotkeys, 1050, 1054
how contours are created, 492
how to use Surfer, 18
hue, 960
icon size, 1047
ID, 971, 975
identity, 974
ignore, 387, 389
image, 778, 916, 947, 1009, 1123, 1141, 1144
image base map, 778, 947
image coordinates, 778
image fill patterns, 927
image gallery, 74
image map, 593
image resolution to download, 663
imagery, 663
IMG, 390, 1265, 1266
immediate, 1057
import, 158, 465, 999, 1123
import (worksheet), 158
import database, 62
import file, 158
import file types, 1001, 1123
import gpx files, 1235
import nc, 1294
import netcdf, 1294
import options string, 1136
importing a base map, 465
improve automation performance, 1102
in, 991
increase down, 733
increase left, 733
indeterminate mode, 34
index, 1045
index contours, 514, 516
index number, 561
individual contour levels, 95
info, 74, 367, 447
ingridoptions, 1244
initial, 303
input grid file, 493, 567
InputBox, 1083
insert, 160
insert cells, 160
insert grid nodes, 390
insert points on line, 900
inserting levels, 503
inset, 906
integer, 1080
integral and differential operators, 412
integrated volume, 412, 414
internet, 74
internet maps, 663
internet update, 70, 1028
interpolate pixels, 593
interpolation, 242, 243
interquartile range, 265
intersect polygons, 904
intersection, 903, 904, 908
intersection points, 908
interval, 503
introduction to gridding methods, 237
introduction to level files, 957
introduction to map layers, 54
introduction to Surfer data, 142
introduction to the major Surfer Automation objects, 1089
introduction to the worksheet window, 36
introduction to volumes and areas, 419
inverse distance, 246
inverse distance filter, 383
inverse distance to a power, 237, 242
invert selection, 974
invisible contour lines, 521
Iris Explorer [.LAT] Export Automation Options, 1287
iris explorer [.LAT] export options dialog, 1286
islands, 913
islands/lakes, 913
isopach map, 434
italic, 867, 919
italic text, 867
Item, 1095
item method, 1095
join curves, 902
join lines, 902
join polylines, 902
JPEG, 1268
JPG, 1123
JPG export, 1002
JPG import, 465, 999
justify, 867
keep every nth vertex, 898
keep every other point in polyline, 898
keep object position, 978
keyboard commands, 118, 1050, 1054
keyboard map, 1050, 1054
kirsch filter, 385
kml, 1275, 1277
kml automation export, 1277
kml export, 1275
kmz, 1277
knowledge base, 74
Kolmogorov-Smirnov goodness of fit statistic, 187
kriging, 237, 238, 242, 291
kriging and variograms, 284
kriging references, 250
KSCriticalValue90 Property, 186
KSCriticalValue95 Property, 186
KSCriticalValue99 Property, 186
KSStatistic Property, 186
kurtosis, 184
label angle, 553
label format, 100, 553, 878, 942
label grid nodes, 338
label increment, 696
label plane, 553
label position, 526, 553, 559
label properties, 527
label property defaults, 1046
label set, 553
label source, 487
label template, 487
labels, 98, 100, 487, 553
labels from multiple columns, 553
labels grid node editor, 338
labels page, 553
lag direction, 293
lag width, 293, 297
lakes, 913
lambert, 842, 843

lambert azimuthal, 842
lambert conformal conic, 193, 817, 843
lambert equal area, 842
lambertian reflection, 602
landscape, 1012, 1016
Laplacian operator, 412, 413
large data set, 238
large icons, 1047
last major tick, 733
LAT, 390, 1286, 1287
lat/long, 836
latitude, 836
latitude/longitude, 86, 193, 686, 817, 860, 861
layer, 54, 107, 695, 971
layer information, 672
layers, 768
layout, 997
least absolute value, 303
least squares, 303
leave, 389
left justify, 867
legend, 629, 702, 708, 713
legend entries, 702
legend layer, 702
Legend Object, 1094
length, 287, 297, 682
length (magnitude) data, 626
length component grid file, 621
length scale, 284
length units, 1025
level files, 344, 506, 586, 957, 958, 959
levels, 93, 495, 503, 512
Levels Collection, 1095
LGO, 1123
LGS, 1123
library, 870
library list, 1079
light color, 572, 745
light position angles, 572, 599, 745
lighted, 567
lighting, 572, 745
limits, 463, 631, 688, 733
line, 310, 508, 510, 880, 930, 932
line and fill properties, 692
line connections, 902
line continuation, 1067
line defaults, 1038
line dialog, 510
line format, 508, 932
line palette, 934
line properties, 94, 512, 513, 514, 515, 516, 589, 692, 880, 930, 931
line property defaults, 1043
line property precedence, 589
line property syntax, 1043
line reshape, 895
line simplification, 898
line smoothing, 898, 900
line spectrum dialog, 585
line style names, 934
line styles, 930, 931, 934
line thinning, 898
line to area, 892
line width, 930
linear, 574
linear convolution filters, 383
linear detrending, 291
linear drift type, 291
linear symbol scaling, 539
linear variogram, 311
linear variogram model, 297, 306
linearly scaled vectors, 629
link post maps with the worksheet, 798
LizardTech MrSID [.SID] Import Automation Options, 1334
load base map, 465
load database, 62
loaded, 1057
loading a classed post map class file, 547
loading a color spectrum, 950
loading a level file in a 3D wireframe map, 586, 958
local datum, 822
local polynomial, 237, 271
local polynomial math, 273
lock object, 978
lock position, 978
logarithmic, 80, 495
logarithmic contours, 495
logarithmic model, 297, 306
logarithmic scaled vectors, 629
lommel-seeliger law, 602
long, 1080
longitude, 836
lower color, 567
lower quartile, 214, 265
lower surface, 429, 578
low-pass filters, 383
lum, 960
LVL, 506, 586, 957, 958, 959
M.A.D., 265
magnify, 993
magnitude, 629
mail, 79
main, 35
major interval, 733
major Surfer automation objects, 1089
major ticks, 730, 737
map add, 121
map coordinate system, 58, 803, 804
map coordinates, 395, 817
map digitize, 775, 777
map frame, 692
map layer, 54, 107, 695
map layers, 54, 89, 121, 768
map limits, 688, 757
map menu, 461, 559
map menu commands, 541, 559, 619, 622, 777
map overlay, 54, 768
map projection, 58, 803
map properties, 678, 682, 688, 692
map rectangle, 757
map scale, 682, 685, 686, 688
map scale bar, 696
map servers, 663
map stack maps, 770
map trackball, 761
map types, 43, 541, 619, 622
map units, 682, 805
MapFrame Object, 1100, 1106, 1107
mapInfo interchange format [.MIF] export options dialog, 1291
mapinfo interchange format [.MIF] import automation options, 1291
MapInfo Interchange Format Export Automation Options, 1293
mapping date/time values, 131
mapping dates, 131
mapping time values, 131
margins, 976, 1012, 1018
mask, 375, 531

masking portions of a contour map with a base map, 531
masking with background, 772
material color, 106, 567
math, 157, 170, 431, 434
math text editor, 867
max lag distance, 214, 291, 293, 303
maximum, 265, 303, 733
maximum absolute value, 150
maximum bin value, 547
maximum columns, 150
maximum contour, 495
maximum data value, 214
maximum distance, 303
maximum iterations, 303
maximum lag distance, 214, 293
maximum number of data to use from each sector, 224
maximum rows, 150
MDB, 1151
mean, 187, 265
mean average deviation, 265
measure, 762
measure distance, 762
measure map, 762
median, 214, 265
median difference filter, 385
median distance, 265
memory requirements, 150
menu bar, 21
menu commands, 35, 1047
mercator, 193, 817, 844, 848
mercator projection, 844
Merge, 158
merge files (worksheet), 158
merge grids, 447
mesh, 105, 570
metafile, 916, 946
metafile export, 1002
metafile import, 465, 999
metafile properties, 946
methods, 1087, 1097
metric, 1025
metric system, 1025
metric units, 1025
metrics, 265
micro variance, 312
midpoint difference, 599
midrange, 214, 265
MIF, 1123, 1291, 1293
miller, 845
miller cylindrical, 845
minimum, 265, 303, 733
minimum absolute value, 150
minimum bin value, 547
minimum contour, 495
minimum curvature, 237, 238, 253, 254
minimum curvature math, 253
minimum data value, 214
minimum number of data, 264
minimum number of data in all sectors, 224
minimum number of data to use from each sector, 224
minimum number of points, 289
minor ticks, 730
minor ticks per major, 730
mirror, 387
mirror X, 435
mirror Y, 435
missing data, 593, 599, 723
mode, 1044
model, 297
model curve, 287
model line, 310
modified shepard's method, 237, 242
modify a color scale bar, 713
modify individual contour levels, 503
modifying axes, 1107
modifying the worksheet and data files, 168, 169, 174
module properties, 1076
module types, 1075
mollweide, 846
mollweide projection, 846
moment statistics filters, 385
mosaic, 447
most recently used list, 72
move backward, 981
move contour labels, 98, 526
move forward, 980
move object, 978
move points, 895
move to back, 980
move to front, 979
moving average, 237, 264
moving average filter, 383
moving individual post map labels, 101, 553, 559
moving the active cell, 142
multiline text, 867
multilog, 260
multiple labels, 80, 553
multiple lines text, 867
multiple polygons in one, 913
multiple select, 795
multiquadratic, 260
NAIP data, 663
name, 975
name an object, 971, 975
named and positional arguments, 1070
named arguments, 1069, 1070
national elevation database, 274, 663
national projection, 826
natural cubic spline, 260
natural neighbor, 237, 242
nc export, 1296
nc import, 1294
ndata, 211
nearest, 574
nearest neighbor, 237, 242, 258, 281, 574
nearest neighbor resampling, 447, 450
nearest neighbor statistics, 214
negative data and polar grids, 626
negative planar area, 423, 428
negative surface area, 423, 429
negative values, 626
negative values in grid, 435
negative volume, 423, 425
neighborhood, 380
nested variogram model, 297
net volume, 423
netcdf, 1294
netcdf export, 1296
netcdf import, 1294
netcdf import automation options, 1295
new, 59, 60
new coordinate system, 193, 813, 817
new features, 80
new module, 1059, 1075

new plot, 59
new projected coordinates, 193, 395
new script, 1059
new value, 505
new variogram, 288, 289, 291
new variogram properties, 289, 291
new view, 996
new window, 996
new worksheet, 60
new zealand map grid, 848
newsletter, 74
no search, 226
nodes, 950
nonlinear filters, 385
normal distribution, 187
north arrow, 890
note about the documentation, 74
nugget effect, 287, 297, 311, 312
number of classes, 547
number of cycles, 696
number of lags, 293
number of passes, 380
number of random points to validate, 211
number of values per line, 1242
numeric cells, 170
numeric format worksheet, 165
numeric label property syntax, 1046
numeric limits, 150
numeric precision, 150
numeric resolution, 150
numeric via colormap, 537
nzmg, 848
object, 1080, 1095
object browser, 1078
object dimensions, 34
object hierarchy, 1089, 1097, 1100
object ID, 971, 975
object list, 1098
object list - graphic, 1100
object manager, 19, 27, 971, 1025
object manager tree, 27
object model, 1087, 1089, 1100
object model chart, 1087, 1100
object module, 1076
object name, 34
object properties, 32, 678
object variables, 1081
object visibility, 27
objects, 1077
objects, 1078
objects, 1089
objects, 1094
oblique mercator, 848
offset, 435, 729, 923
offset along axis, 729
offset distance, 265
offset from axis, 729
one-grid vector map, 619
online help, 74
online map servers, 663
online maps, 663
opacity, 109, 110, 593, 599, 695, 708, 713, 919, 923, 930, 946, 947, 950
open, 158, 368, 999
open data dialog, 207
open file, 593
open grid, 368
open group, 989
open script, 1059
opening a worksheet window, 138
opening an existing data file, 86
opening data files, 142
opening surfer, 84
opening the colormap dialog, 949
opening the object manager, 27
opening windows, 19
Opening, Saving, and Closing Documents, 1103
operators, 1068
optional arguments, 1069
options, 1020, 1025, 1028, 1030, 1033, 1034, 1036, 1038, 1039, 1047
options menu - grid node editor, 38
options printing, 1035
order layer, 979, 980, 981
order objects, 979, 980, 981
order of transformations, 974
order statistics filters, 385
orientation, 229, 299, 976, 1012, 1016
original, 214
orthographic, 850
orthographic projection, 850
other z statistic, 269
other Z statistics, 265
out, 992
outgridoptions, 1244
overlap method, 447
overlay, 54, 107, 121, 767, 768, 807
overlay a masking base map, 531
overlay adjacent maps, 54
overlay maps, 54, 89, 91, 121, 463, 541, 767, 768, 971
overlay resolution, 574
overlay visibility, 574
overlaying maps, 54, 121, 768
overlaying maps in Scripter, 1106
overlaying two or more maps, 54, 121, 768
OverlayMaps, 1106
Overlays Collection, 1087, 1095
overlays page, 574
overview of Surfer objects, 1089
page, 991, 1016
page layout, 1012
page layout (worksheet), 1016
page margins, 985, 1012, 1018
page number, 1020
page order, 1020
page outline, 1030
page rectangle, 1030
page setup, 1012, 1016
page setup (worksheet), 1016, 1018, 1020
page size, 976, 1012, 1016
page transform parameters, 974
page units, 976, 1025
pair comparison files, 285
pairs, 287, 310
palette, 926, 959
pan realtime, 995
paper, 1012, 1016
parameter fit limits, 303
parent and application properties, 1096
Parent Property, 1096
parts of the worksheet, 138
paste, 151, 915, 916
paste special, 151, 915, 916
pasting data, 142

pattern, 520, 926, 1044
pause script, 1064
PCF, 285
PCT, 1123
PCT export, 1002
PCT import, 465, 999
PCX, 1123
PCX export, 1002
PCX import, 465, 999
PDF, 1123, 1298, 1299, 1301, 1303
pdf layer export, 1301
pentaspherical model, 297, 306
percent format, 165
percentage, 165
perimeter, 762
periodogram, 417
perspective, 678
peucker's approximation, 602
phone, 79
picture, 916
placing boundaries on other maps, 463
plan curvature, 406, 410
planar area, 428
planar area calculations, 428
planar grids, 265
plane, 729
plot, 310
plot cross sections, 442
plot document, 59
plot document commands, 35
plot lines of constant, 578, 587
plotcall, 1305
PlotCall Import, 465, 999
plotting cross sections, 442
PlotWindow, 1095
PlotWindow Object, 1096
PLT, 1123, 1305
PLT import, 465, 999
PNG, 1123
PNG export, 1002
PNG import, 465, 999
point, 882
point cloud, 635
point cloud density, 1034
point cloud points, 1034
point sample, 445
pointer coordinates, 34
points, 919
points to polyline, 894
polar, 826
polar data, 621, 626
poly polygons, 912, 913
polyconic, 851
polyconic projection, 851
polygon, 881
polygon reshape, 895
polygon to polyline, 893
polyline, 880
polyline connections, 902
polyline reshape, 895
polyline smoothing, 900
polyline to points, 893
polyline to polygon, 892
polynomial, 271
polynomial regression, 237, 238, 242, 259
population coefficient of variation (v), 186
population kurtosis, 184
population mean, 187
population mean deviation (md), 183
population skew, 185
population standard deviation, 188
population variance, 189
Portable Document Format [.PDF] Export Automation Options, 1303
portInt, 1080
portrait, 1012, 1016
position, 733, 954, 977
position object, 978
position relative to symbol, 553
position/size, 772, 976, 978
position/size toolbar, 976
positional arguments, 1069, 1070
positive planar area, 423, 428
positive surface area, 423, 429
positive volume, 423, 425
post labels, 559
post layer, 541
post legend, 702
post map, 89, 114, 442, 535, 539, 541, 553, 561
post map angle column, 561
post map changes, 80
post map connect points, 535
post map labels, 100, 553, 559
post map properties, 99
post map rows, 535
post map symbol, 537
post map symbol column, 561
post map symbol properties, 537
post map worksheet, 535
post map worksheet columns, 535
post map worksheet rows, 535
pour source, 613
Powell's search method, 303
power models, 297, 306
precedence, 589
precision, 150
predefined coordinate system, 193, 817
preferences, 1025, 1034, 1036
pre-filter, 175
prefix, 942, 1046
pregridded data, 281
preserve data values, 237
preset color spectrum, 950
prevent moving objects, 978
Prewitt filter, 385
print, 1015, 1022
print contour map, 1102
print method, 1015
print range, 1022
print setup (worksheet), 1016
print to file, 1022
printer, 1012, 1015
printing options, 1035
printing patterns, 1025
printing the online help, 73
printing the worksheet, 1022
PRN, 1022
producing a grid file from a regular array of XYZ data, 281
profile, 438, 723
profile curvature, 406, 409
profile data, 723
program statements, 1067
progress of task, 34
project coordinates, 193, 395, 817
projection, 58, 193, 803, 804, 805, 809, 813, 817, 824, 862
projection faq, 808
projection reference, 862
projection type, 827, 829, 830, 831, 832, 833, 834, 835, 836, 837,

838, 840, 842, 843, 844, 845, 846, 848, 850, 851, 852, 853, 854, 855, 856, 857, 859
projections, 57, 58, 803, 807
properties, 32, 99, 678, 1076, 1087, 1097
properties of multiple selections, 32
property get, 1077
property let, 1077
property manager, 32
property set, 1077
proportional, 537
proportional scaling, 539
proportional symbol, 537
proportional symbol size, 539
proportional XY scaling, 682, 685
public, 1077
pull up, 359
push down, 356
quadratic detrending, 291
quadratic drift type, 291
quadratic model, 297, 306
quartile, 214, 265
query, 795
query attributes, 795
query objects, 795
query string, 795
quick watch, 1057
R.M.S., 265
R2, 260
radial basis function, 237, 238, 242, 260
radial divisions, 214, 291
radian units, 621
radius 1, 227, 229, 264, 299
radius 2, 227, 229, 264, 299
range, 214, 265
range ring, 885
raster information, 1123
ratio, 229, 299
rational quadratic model, 297, 306
RAW, 390, 1312, 1315, 1316, 1318
RAW Binary Lattice [.RAW, .BIN] Export Automation Options, 1318
Raw Binary Lattice [.RAW] Import Automation Options, 1315
Raw Binary Lattice File [.RAW] Import Automation Options, 1315
RAW binary lattice files - export options dialog, 1316
RAW binary lattice files - import options dialog, 1312
read every, 451
realtime, 993
recalc grid, 390
recalculating a grid, 390
recent files, 1025
recently used list, 72
rectangle, 887, 994
rectangle must fully surround, 1033
redo, 915
redraw, 995
reduce grid file density, 390
references, 189, 211, 254, 259, 260, 312, 314, 389, 412, 423, 454, 599, 974, 1062
regional projection, 826
register serial number, 80
registration card, 80
regular contour intervals, 503
regularly spaced data, 281
relief parameters, 599
reload map data, 693
remap blank value, 431
remap blanked nodes, 567
remove duplicate data, 175, 209
remove hidden lines, 578
remove jagged line edges, 900
remove map from overlay, 54
remove negative values from grid, 435
remove space, 1136
removing a color scale bar, 708, 713
removing all contour labels, 528
rename, 975
replace, 161
replicate, 387
report, 211, 214, 291, 308, 423
report document, 214
reports, 214
resample grid, 380, 447
resample method, 574
resample methods, 450
reset window, 997
reshape, 895
reshape area, 895
reshape line, 895
reshape polygon, 895
reshape polyline, 895
residuals, 211, 445, 447
resize map, 682, 974
resize objects, 974, 978
resolution, 150
results file, 211
retain tool mode, 1025
retained duplicates, 214
retrieving items, 1095
reverse axis, 80, 733
reverse map, 733
reverse vector orientation, 629
RGB, 960, 962
right justify, 867
robinson, 852
Robinson 3-level filter, 385
Robinson 5-level filter, 385
robinson projection, 852
robinson sterling projection, 852
robinson-sterling, 852
root mean square, 265
rotate, 435, 761, 987
rotate worksheet, 191
rotation, 678, 974
rounded rectangle, 888
rounded square, 888
row, 147, 149, 150, 169
row and column headers, 1020
row and column label bars, 139
row formats, 169
row height, 147, 169
row labels, 139
row numbers, 138
rows to columns, 191
r-squared, 260
ruler, 1025
ruler divisions, 1036
rulers, 996, 1025, 1036
rulers and grid, 1036
run, 1063
running scripts, 1063
running scripts from the command line, 1063
running surfer, 84
sample coefficient of variation (v), 186
sample files, 3
sample kurtosis, 184
sample mean, 187

sample mean deviation (md), 183
sample skew, 185
sample standard deviation, 188
sample variance, 189
sat, 960
save, 68, 1103
save as, 68
save as previous version, 80
save classes, 547
save data, 772
save grid, 772
save grid as, 371
save profile data, 723
save script, 1059
save text grid files, 1242
saving a classed post map class file, 547
saving a color spectrum, 950
saving a data file, 68, 88
saving a level file in a 3D wireframe map, 586, 958
saving ASCII data files, 1131
saving the data file, 88
scale, 287, 435, 463, 682, 685, 686, 688, 696, 733, 923
scale (C), 297
scale bar, 109, 696, 708, 713
scale tracks, 696
scale vectors, 629
scale with lat/long maps, 686
ScaleBars Collection, 1095
scaled symbols, 539
scaling, 733
scaling method, 539, 629
scaling page, 1005
scaling proportional symbols, 537
scaling tab, 629, 733
scattered data interpolation dialog, 289
screen layout, 19
script examples, 1102
script failure, 1064
Scripter, 1059, 1062, 1063, 1068, 1080, 1083, 1084, 1086, 1087
scripter BASIC language, 1059
Scripter examples, 1102
Scripter windows, 1057
scripts, 1059
scroll control, 950
SDTS, 1180, 1182
SDTS import, 465, 999
SDTS topological vector profile import automation options, 1182
SDTS Topological Vector Profile Import Options Dialog, 1180
seamless geotiff, 1345
search, 224
search by attributes, 795
search ellipse, 227, 264
search for objects, 795
search rules, 224
search type, 226
search with gridding, 224
second derivative, 400, 404
seg, 80
segment length, 762
select, 142, 145, 973, 974
select all, 974
select area to download, 663
select by attributes, 795
select data source, 663
select entire worksheet, 142
select entire worksheet button, 138
select image resolution to download, 663
select map in an overlay, 971
select maps, 971
select tool, 973
select validation points within these limits, 211
selected, 993
selecting a column or row dividing line, 147
selecting a map layer and changing the object id, 971
selecting an object with block select, 973
selecting cells, 139, 142, 145, 146, 147
selecting objects, 27
Selection Collection, 1095, 1106
selection options, 1033
selection settings, 1033
selection tolerance, 1033
separate islands, 912
separate islands/lakes, 912
separate lakes, 912
separation angle, 285
separation distance, 285
separator, 1020
serial number, 71, 80, 84
server information dialog, 672
set, 514, 522, 527, 530
set all classes to use proportional sized symbols, 552
set all classes to use the same size symbol, 552
set all symbols to the same properties, 550
set all symbols to the same size, 552
set blank value, 380, 431
set dialog defaults, 1039
set drawing defaults, 1038
SET file, 1039, 1041
set general defaults, 1025
set labels to date/time, 942
set limits, 757
set map limits, 757
set number of values in a line, 1242
set number of values in a row, 1242
set page units, 1025
set paper size on printer, 1035
set ruler and grid defaults, 1036
setting axis scaling, 733
setting column options, 157
setting file, 1039, 1041
setting fill properties on a frequency basis, 522
setting hachures on a frequency basis, 530
setting label properties on a frequency basis, 527
setting line properties on a frequency basis, 514
setting map limits, 691
setting the 3D surface base properties, 567
setup, 1016
shaded relief map, 599, 602
shading method, 599
shading methods, 602
shadow cursor capability, 798
shaft length, 629

Shape Object, 1087, 1094, 1104
Shapes Collection, 1095
shear, 974
sheet, 1136
Shepard's method, 237
shininess, 567
shortcuts, 1054
show, 729
show base, 567, 578, 589
show color scale, 567, 593, 627
show contours, 341
show grid, 996
show grid node editor labels, 338
show grid node labels, 338
show labels, 338
show lower surface, 578
show margins, 1012, 1030
show nodes, 340
show page rectangle, 991
show rulers, 996
show scale legend, 629
show splash screen at startup, 1030
show streams, 613
show upper surface, 578
show vertical lines, 578, 589
SHP, 1002, 1123, 1327, 1328, 1330
SHP export, 99, 1002
SHP import, 465, 999
SID, 1332, 1334
significant digits, 942
sill, 287
simple level method, 495
simple levels, 495
simple shading method, 599, 602
simplification method, 898
simplify lines, 898
Simpson's 3/8 rule, 423, 427
Simpson's rule, 423, 427
single, 1080
single-use tool mode, 1025
sinusoidal, 853
sinusoidal projection, 853
size, 682, 867, 919, 977, 981, 1012, 1045
size and color page, 1009
sizing method, 537
skew, 185
skewness, 214
skip, 514, 522, 527, 530
skip missing data, 723
SkipExtraDelimiters, 1136
slice, 438, 442
slider button, 954
SLK Files, 130
slope, 287, 311, 406, 407, 409, 412
slope of surface, 404
small data set, 238
smooth, 353, 900
smooth grid, 353
smooth lighting, 572, 745
smooth line, 883, 898, 900
smooth lines, 80
smooth polyline, 898, 900
smoothing, 493
smoothing a grid file, 453
smoothing a variogram with lag width, 297
smoothing contours, 532
smoothing grids, 390, 453
smoothing interpolator, 312
smoothing interpolators, 242
smoothing wireframes, 590
snap position, 1036
snap to ruler, 1036
soft crop, 688
sort, 174
sort data, 174
source coordinate system, 58, 803, 804
sp1, 80
space delimit, 1131
spacing, 284
sparse grid, 390
spatial, 1006
spatial coordinate system, 1006
spatial filter, 175
spatial reference, 1006
SPCS, 826
special XLS characters, 1360
specify background properties for a map, 692
specifying advanced gridding options, 236
specifying cell coordinates, 1072
specifying contour levels, 503
specifying local polynomial advanced options, 271
specifying moving average advanced options, 264
specifying nearest neighbor advanced options, 258
specifying radial basis function advanced options, 260
specifying regression advanced options, 259
specifying the lines to draw on a wireframe, 587
specular, 572, 745
spherical model, 297, 306
splash screen, 1030
spline, 883
spline polygon, 900
spline polyline, 883
spline smooth, 390, 453, 532, 900
split complex polygon, 912
split complex poylgon, 912
split islands, 912
split islands/lakes, 912
split lakes, 912
split line, 902
split polyline, 902
square, 887
square root of the variance, 188
square root scaled vectors, 629
square root symbol scaling, 539
SRF, 42, 68
SRF file compatibility, 42, 68
stack, 1057, 1064
stack maps, 770
stacking a contour map over a wireframe, 770
standard colors, 960
standard deviation, 188, 214, 265
standard deviation filter, 385
standard error of the mean, 188
standardized variogram estimator, 295
starting surfer, 84
state plane, 193, 817, 826, 854
state plane coordinates, 854
statements, 1067
statistics, 157, 214, 289, 453
statistics note, 211
statistics page, 214, 308
statistics references, 189
status bar, 34
step, 1064
step amount, 293

step CCW, 293
step CW, 293
step direction, 293
stereographic, 855
stereographic projection, 855
stock pattern, 927
stop editing group, 989
stop script, 1064
stream intersection, 613
streams, 613
stretch, 974
strikeout, 919
strikethrough, 867
string, 1080
style, 919, 1043, 1045, 1046
sub main, 1059
sub position, 867
sub-polygons, 912, 913
subroutines, 1070, 1097
subroutines and functions, 1070
subscript, 867
subset, 688
subset of grid, 451
subtitle, 310
suffix, 942, 1046
suggested reading, 314, 599, 1062
sum, 265
Sun Raster Image [.RAS, .SUN] Export Automation Options, 1144
superscript, 867
support, 79
surface, 567, 570, 572, 574, 708, 745
surface area, 429
surface area calculations, 429
surface material color, 106
surface mesh, 105
surface offset, 570
surface overlay, 107
Surfer 6 [.GRD], 1240, 1241
Surfer 6 binary grid file format, 1240
Surfer 6 grid file format, 1240
Surfer 6 text grid format, 1241
Surfer 7 [.GRD], 390, 1237
Surfer 7 grid file format, 1237
Surfer automation help, 1087, 1089
Surfer automation object model, 1100
Surfer automation objects, 1087
Surfer file, 68
Surfer files, 42
surfer help, 74
Surfer object model, 1087
surfer tutorial, 84
Surfer user interface, 19
Surfer.ini settings, 1039
switch, 191
switch columns and rows, 191
switch rows and columns, 191
symbol, 882, 941
symbol angle, 539, 544, 561
symbol angle values from the data file, 561
symbol column, 537, 561
symbol defaults, 1038, 1045
symbol frequency, 544
symbol height, 539
symbol index, 561
symbol number, 561
symbol origin, 629
symbol page, 627
symbol properties, 941
symbol properties dialog, 941
symbol property syntax, 1045
symbol set, 561
symbol size, 539
symbol size post map, 537
symbol specifications in the data file, 561
symbol style, 627
symbol tab, 627
symbol width, 629
symbology legend, 702
symbols, 310
symbols from data file, 561
symset, 1045
syntax, 1043, 1044, 1045, 1046
system file information, 71
system info, 71
tabbed document, 19, 24, 36, 38, 1030
tabbed view, 19, 24, 1030
tabbed windows, 19, 24, 1030
tabs, 24
tagged image [.TIF] export options dialog, 1345
Tagged Image File Format [.TIF] Export Automation Options, 1144
tangential curvature, 406, 411
target coordinate system, 58, 803
target precision, 303
task progress, 34
tech support, 79
technical support, 79
template, 867
template (text), 867
template coordinate systems, 827
template label, 487
template library (text), 869, 870
template projections, 827
templates, 827
tension factor, 900
terminate flow, 613
terrain aspect, 265, 406, 408
terrain modeling, 397, 406, 407, 408, 409, 410, 411, 412
terrain modeling references, 412
terrain slope, 265, 406, 407
terrain statistics, 271
test template (text), 872
text, 866, 867, 916, 919
text alignment, 867
text and font properties, 919
text cells, 170
text editor, 867, 869, 870, 872, 875
text editor template library, 870
text grid files, 1242
text grid format, 1241
text multiple lines, 867
text properties, 867, 919
text qualifiers, 1131
text strings, 942, 1068
text template, 867
text template editor example, 875
text to number, 190
TextQualifier, 1136, 1140
texture map, 574
TGA, 1123, 1342
TGA export, 1002
TGA import, 465, 999
the cross validation dialog, 211

the cross validation process, 211
The Object Browser, 1078
the variogram grid, 285
thin, 898
thin lines, 80
thin plate spline, 260
thin polyline, 898
three-minute tour, 3
threshold, 406, 613
threshold averaging filter, 385
tick length, 730
tick spacing, 733
ticks, 730, 737
TIF, 1123, 1345
tile, 1015
tile horizontal, 997
tile vertical, 997
tilt, 678, 761
time, 130, 867, 878
time date format, 942
time format, 80, 165, 878, 942
time formats, 133
time values, 131
tips on selecting cells, 145
title, 729
to back, 980
to draw a grid over the map, 738
to front, 979
tolerance, 293, 1033
tool persistence, 1025
toolbar positions, 1047
toolbars, 19, 21, 23
tools menu, 35, 1020, 1047
tools menu commands, 1020
tools options, 1025, 1028, 1030, 1034, 1035, 1036, 1038, 1039
tooltips, 35, 1047
total page count, 1020
total planar area, 423
trace, 1064
track cursor, 798
trackball, 761
training videos, 74
transform, 157, 170, 435, 974
transform grid, 435
transform references, 974
transforming data in the worksheet, 170
transparency, 109, 110, 593, 599, 695, 708, 713, 919, 923, 930, 950
transpose, 191
transpose columns, 80
transverse mercator, 193, 817, 856
transverse mercator projection, 856
trapezoidal rule, 423, 427
treat as blank, 170
treat as number, 170
treat as text, 170
triangulation with linear interpolation, 237, 238, 242
trim, 375
trim exponent zeros, 942
true type fonts, 919
truevision targa [.TGA] file description, 1342
Truevision Targe [.TGA] Export Automation Options, 1144
truncate, 1015
tutorial, 86
tutorial - changing tick labels, 102
tutorial advanced contour levels, 95
TVP, 1180, 1182
two-grid vector maps, 622
TXT, 1002, 1131
TXT files, 130, 1131
type, 1046, 1095
Type Library References, 1078
types of projections, 822
u.s. data, 663
unblank node, 914
uncombine maps, 777
underline, 867, 919
understanding how to use Surfer, 18
undo, 867, 914, 1025
undo levels, 1025
ungroup, 989
unhiding columns, 168, 169
unhiding rows, 169
uniform class symbol, 550
uniform lines, 510
union of polygons, 903
union polygons, 903
units, 229, 299, 423, 427, 1025
univariate statistics, 214
universal transverse mercator, 857
universal transverse mercator coordinate system, 826
universal transverse mercator projection, 857
unprojected lat/long, 825, 836
unprojected latitude/longitude, 836
update, 70, 1028
update a post or classed post map, 561
update layer, 367, 779
update statistics, 214
updating Surfer, 70
upgrade, 70, 80
upper and lower surface, 423
upper color, 567
upper quartile, 214, 265
upper surface, 429, 578
us data, 663
use affected levels with hachures, 530
use all of the data, 226
use caution when saving excel files, 68
use data limits, 629, 688
use different scaling in the X and Y dimensions, 685
use grid math to eliminate negative values from a grid file, 435
use overlay color only, 574
use scaling to minimize distortion on latitude/longitude maps, 686
use surface color only, 574
use the digitize command, 775
use the grid math command to generate an isopach map, 434
use the keyboard to simulate mouse operations, 1050
user defined, 547
user defined filters, 383
user input, 1083, 1086
user interface, 19, 1030
user interface style, 1030
user-defined types, 1082
UserDialog editor, 1083, 1084, 1086
UserDialog example, 1086
USGS, 1184, 1187
USGS Digital Elevation Model [.DEM] Export Automation Options, 1187

USGS digital line graph boundary import automation options, 1197
USGS digital line graph boundary import options dialog, 1193
USGS file import, 465, 999
using a preset color spectrum, 950
using a symbol column for post map symbols, 561
using an angle column for post map symbols, 561
using breaklines and faults when gridding, 233
using color spectrums in map series, 955
using cross validation, 211
using custom setting files, 1041
using grid math to eliminate negative values from a grid file, 435
using scaling to minimize distortion on latitude/longitude maps, 686
using Scripter, 18, 1060
using scripter help, 1061
using surfer, 16
using the color spectrum file in other maps, 955
using variogram results in kriging, 314
utm, 193, 817, 826, 857
validation point, 211
values per line, 1242
van der grinten, 859
van der grinten projection, 859
variable correlation, 214
variable covariance, 214
variables, 1064, 1080, 1081, 1082
variance, 189, 214, 265, 287
variance filter, 385
variance line, 310
variant, 1080
variation, 186
VarioComponent Object, 1104
variogram, 284, 288, 313, 314, 315, 317, 318, 321, 323, 326, 327, 332
variogram add component, 306
variogram anisotropy, 323
variogram autofit, 303
variogram components, 249, 284, 285, 297
variogram data page, 289
variogram FAQ, 327
variogram functions, 297
variogram grid, 285, 291, 318
variogram grid report, 214
variogram introduction, 315
variogram model, 287, 297, 310, 321, 323, 326, 327
variogram model graphics, 306
Variogram Object, 1104
variogram properties, 292, 293, 297, 303, 308, 310
variogram reference, 332
variogram report, 214
variogram tutorial, 314, 315, 317, 318, 321, 323, 326, 327
vector example, 626
vector information, 1123
vector legend, 702
vector map, 619, 621, 622, 627, 629
vector PDF export, 1301
version, 71
vertex, 895
vertex averaging, 898
vertical alignment, 867
vertical light position, 572, 599, 745
vertical scale, 293
vertices, 895
videos, 74
view, 678
view data source, 672
view full screen, 995
view map source, 672
view menu, 27, 34, 798, 991, 992, 993, 994, 995, 996
view menu - grid node editor, 38
view page, 991
view variables, 1064
view worksheet data, 138
view zoom selected, 993
viewshed, 658
viewshed analysis, 658
viewshed layer, 658
viewshed properties, 658
Visible Property, 1087
visual aid, 1047
Visual BASIC, 1059
visual tool kit [.VTK] export options dialog, 1351
volume, 412, 419, 423, 425, 427
volume between two surfaces, 429
volume of a lake, 431
volume of blanked grid, 430
volume reference, 423
VTK, 390, 1351, 1353
vtk automation export, 1353
vtk export, 1353
W** files, 130
warn on truncate, 1035
warp, 351, 779
warp grid, 351
warp image, 779
watershed, 613
wave model, 297, 306
wave number, 417
web map servers, 663
web mapping servers, 663
web maps, 663
web servers, 663
website URL, 672
weighted average interpolation, 243
weighted averaging, 242, 243
welcome to Surfer help, 74
wheel mouse, 993, 995
width, 1043
wildcards, 161
window menu, 996, 997
Window Object, 1094, 1096
window types, 19
windows, 19, 1057
Windows Collection, 1095
Windows Enhanced Metafile [.EMF] Export Automation Options, 1222
windows enhanced metafile [.EMF] export options dialog, 1221
Windows Enhanced Metafile [.EMF] Import Automation Options, 1220
Windows Metafile [.WMF] Export Automation Options, 1356
windows metafile import options dialog, 1354

windows patterns, 927
wireframe, 578, 581, 583, 586, 587, 589, 590, 713, 958
wireframe base, 589
wireframe blanking, 590
wireframe line property precedence, 589
wireframe lines, 510, 587
Wireframe Object, 1087
wksWindow, 1095
wksWindow object, 1096
WMF, 1123, 1220, 1354, 1355, 1356
WMF export, 1002
WMF import, 465, 999, 1220
wmf import automation, 1355
WMS, 663
working with date/time values, 131
working with dates, 131
working with scripts, 1059
working with times, 131
working with worksheet data, 142
worksheet, 36, 60, 142, 157
worksheet background, 167
worksheet columns, 535, 544
worksheet commands, 35, 36, 157, 161, 193, 817
worksheet data, 142
worksheet document, 36
worksheet equations, 170
worksheet error codes and special numeric values, 150
worksheet files, 1131
worksheet formatting, 1131
worksheet label column, 553
worksheet labels, 553
worksheet limits, 150
worksheet memory requirements, 150
worksheet name, 138
worksheet numeric format, 165
worksheet rows, 535
worksheet size, 150
worksheet specifications, 150
worksheet technical specifications, 150
worksheet transform, 170
worksheet transpose, 191
worksheet window, 36, 138, 142
world coordinate systems, 827
world projection, 827
worldwide data, 663
WPG, 1123
writing functions, 1070
writing scripts, 1062
writing subroutines, 1070
X component data, 626
X component grid, 626
X component grid file, 621
X coordinate, 438
X direction, 211
X offset, 974
X scale, 682, 974
XLS, 130, 1136, 1140, 1359, 1360
XLSM, 80
XLSX, 1136, 1360
XY coordinates in a data file, 560
XYZ columns, 192
XYZ data file, 85, 86, 88
XYZ files, 1131
XYZ points, 1131
Y component data, 626
Y component grid, 626
Y component grid file, 621
Y coordinate, 438
Y direction, 211
Y offset, 974
Y scale, 682, 974
Z direction, 211
Z level dialog, 503, 505
Z levels, 581
Z Levels, 589
z moment, 268
Z moment statistics, 265
Z order statistics, 265, 267
Z scale, 682
Z scale factor, 599
Z value, 438
Z value of grid at a point, 447
Z values in the data file, 560
zoom, 366, 991, 992, 993, 994
zoom actual, 994
zoom full screen, 995
zoom in, 366, 991
zoom out, 366, 992
zoom page, 991
zoom realtime, 993
zoom rectangle, 994
zoom selected, 993

Customer Service Resources

Before calling, please check the following available resources as your question may already be answered.

Registration: Register online at www.goldensoftware.com or fax to the number below.
Knowledge Base: support.GoldenSoftware.com or in the **Surfer** program using the **File | Online | Knowledge Base** command
Forums: support.GoldenSoftware.com or in the **Surfer** program using the **Forums** button in the online help.
Frequently Asked Questions: In the **Surfer** program using the **File | Online | Frequently Asked Questions** command
Tutorial: Complete the tutorial section in this guide or in the *Tutorial* book in the online help.
Online Help: surferhelp.GoldenSoftware.com or press F1 in the **Surfer** program.
Support Videos: support.GoldenSoftware.com for **Surfer** specific training videos and webinars, or check out the Golden Software channel on YouTube.

Business Hours

Technical Support:

Monday through Friday, 8:00 AM - 5:00 PM, Mountain Time

Product Sales:

Online orders available 24 hours, 7 days a week

Golden Software Contact Information

www.goldensoftware.com
SurferSupport@goldensoftware.com
phone: 303-279-1021
fax: 303-279-0909

Golden Software, LLC
809 14th Street
Golden, Colorado 80401 USA

Phone +1 303 279 1021
Fax +1 303 279 0909

www.goldensoftware.com

© Copyright 2017 Golden Software, LLC. All Rights Reserved. Golden Software®, the Golden Software® mark, Surfer® and the Surfer® logo marks are owned by Golden Software and may be registered. All other trademarks and logos are the property of their respective companies. Software developed in the USA.