



Oasis montaj How-To Guide

Complete Workflow for Oasis montaj



The software described in this manual is furnished under license and may only be used or copied in accordance with the terms of the license.

Manual release date: April-29-14.

Please send comments or questions to info@geosoft.com

© 2013 Geosoft Inc. All rights reserved. Geosoft is a registered trademark and Oasis montaj is a registered trademark of Geosoft Inc. Other brand and product names mentioned herein are properties of their respective trademark owners. No part of this publication may be reproduced, stored in a retrieval system or transmitted, in any form, or by any means, electronic, mechanical, photocopying, reading, or otherwise, without prior consent from Geosoft Inc.

The software described in this manual is furnished under license and may only be used or copied in accordance with the terms of the license. OM.h.2013.04

Windows™, and Windows NT are either registered trademarks or trademarks of Microsoft Corporation.

Geosoft Incorporated
Queen's Quay Terminal
207 Queen's Quay West
Suite 810, PO Box 131
Toronto, Ontario
M5J 1A7
Canada
Tel: (416) 369-0111
Fax: (416) 369-9599

Website: www.geosoft.com

E-mail: info@geosoft.com

Support

For obtaining technical support, email support@geosoft.com

If you wish to speak to a technical support representative in your region, please visit the Geosoft Support page at: www.geosoft.com/about-geosoft/contact-us/world-offices for contact information.

Table of Contents

Support	ii
Table of Contents	iii
Geosoft License Agreement	xiv
Finding Help Information	xix
Contacting Technical Support	xix
Getting Started with Oasis montaj	1
Oasis montaj Mapping and Processing System	1
Software and Hardware Requirements	2
Geosoft Connect and Geosoft ID	3
Create your Geosoft ID	3
Create Geosoft ID:	3
Welcome to the Oasis montaj Environment	3
Opening Oasis montaj	3
Oasis montaj System Files	4
Oasis montaj Projects	5
Creating a Project	5
Closing a Project	7
To Close a Project	7
Oasis montaj Menus	7
Loading Applications and Tools	7
To Load an Application or Extension Menu	8
Oasis montaj Settings	8
To Change the Default Settings of the Oasis montaj Environment	9
Common Default Settings	9
Auto Save Time	10
Metadata	10
Metadata Viewer	11
To View and/or Edit Metadata	11
Metadata Tool	11
To View the Metadata Tool	11
Shortcut Keys	12
Oasis montaj eLicensing	14
About eLicensing Tools	14
To Install eLicense	15

First time use:	15
Show Computer ID	15
To Show Computer ID:	15
View your eLicense	16
To View your eLicense:	16
Update eLicense	16
To Update your eLicense:	16
To Park (or Move) your eLicense	16
To park (move) license from your computer to the Geosoft license server:	16
Re-installing your license:	17
Transfer eLicense	17
To Transfer your eLicense:	17
Lock eLicense	18
To Lock your eLicense to the current computer:	18
UnLock eLicense	18
To unlock you eLicense from the current computer:	18
Working with Geosoft Databases in Oasis montaj	19
The Geosoft Database	19
Database Lines/Groups	19
Oasis montaj Supported Data Formats	20
Database Menus	21
Drag and Drop Geosoft Database Files	22
Creating a Geosoft Database	22
To Create a New Database	23
Create a New Line or Group	23
To Create a New Line or Group	23
Line Types	24
Edit Geosoft Database Lines and Groups	24
To Edit a Line	24
Edit a Group	25
Save Changes to Databases	25
To Save Changes to Your Database	25
To Save Database As	26
Oasis montaj Coordinate Systems	27
Coordinate Systems in Oasis montaj	27
Define X, Y Channels	28
Defining Channels	28
To Define X and Y channels	28
Projected (X,Y) Coordinate System	29
The Coordinate System Dialog	29
Setting a Projected Coordinate System for a Database	29

To Set a Projected (X, Y) Coordinate System for Database Channels	29
Set Projected Coordinate System of a Grid	31
To Modify or Define a Projected Coordinate System of a Grid	31
Create New Map from Projected (X, Y) Coordinates	33
To Define a Map Coordinate System for a New Map from Projected (X,Y) Data	34
To Scan a Database for Data Range and Coordinate System Information	34
To Scan a Grid for Data Range and Coordinate System Information	34
To Define the Coordinate System	35
Geographic (long, lat) Coordinate System	36
The Coordinate System Dialog	36
Setting a Geographic Coordinate System for a Database	36
To set a Geographic (Long, Lat) Coordinate System for a Database	37
New Map from Geographic Coordinates	38
To Define a Map Coordinate System for a New Map from Geographic Coordinates	39
To Scan a Database for the Data Range and Coordinate System Information	39
To scan a grid for the data range and coordinate system information	40
To Define the Coordinate System	40
Displaying Data with Different Coordinate Systems on a Map	41
To Display Different Coordinate Systems on a Map	41
Create and Apply Warps on a Grid	42
To Define a Warp	42
To Apply A Warp to a Grid	43
Warping Image Files	44
Converting Elevations	45
Importing and Exporting Databases in Oasis montaj	46
Import menu	46
Export menu	46
Drag and Drop Geosoft Database Files	47
Importing ASCII Files	47
The Import Wizard	47
To Import ASCII Files	47
Importing Databases	50
To Import a Single Table into a Database	51
Importing All Tables into a Database	53
To Import All Tables into a Database	53
Exporting Geosoft Databases	54
Export to CSV or Other File Formats	54
To Export to CSV or Other Formats	54
Exporting to Geosoft File Formats	55
To Export to an XYZ File	55
Exporting to Microsoft Access Database Format	55

To Export to a Microsoft Access Database	56
Working with Spreadsheets in Oasis montaj	57
Spreadsheets	57
How the Spreadsheet Displays Project Data	57
Displaying Data in the Spreadsheet	58
Channels	58
Current X, Y and Z Channels	58
To Set Current X and Y Channels	59
Show and Hide Channels	59
To Hide a Channel (Column)	59
To Display a Hidden Channel	59
Protect and Unprotect Channels in a Spreadsheet	60
To Remove Channel Protection	60
To Protect a Channel	60
Create Working Channels in the Geosoft Database	60
To Create a New Channel	60
Delete a Channel from the Database	61
To Delete a Channel	62
Master Sample	62
To Activate the Master Sample	62
To Remove the Master Sample	62
Displaying Data in the Profile View	63
Display Profiles	63
To Display a Profile	63
To Display the Profile of Different Line	64
Set Profile Options	64
Y Axis Options	65
To Set Y Axis Options	65
X Axis Options	66
To Select X Axis Channel	66
Edit a Channel using Profile View	67
To Edit Data in a Channel Using the Profile View	67
Processing Data in the Spreadsheet	68
Calculate Statistics	68
To Calculate Basic Statistics on a Channel	68
Edit Data using Conditional Expressions	69
Example	69
To Use Conditional Expressions	69
Use the Channel Math Expression Tool	70
Common tasks	70
Expression File	70

To Access the Channel Math Expression Builder	70
Interpolate Data	71
To Fill Gaps in a Data Channel Using Interpolation	71
Dynamic Links: Geographic Connections to your Data	72
Working with Dynamic Linking	72
Link Data Dynamically	72
To Link Spreadsheet, Profile and Map Data	72
Oasis montaj Database Maintenance	74
View Database Properties	74
To View the Properties of a Database	74
Growing, Verifying and Repairing a Database	75
Defragmenting the Database	76
To Defragment a Database	76
Adding Database Comments	76
To Add Comments to a Database	76
Basic Mapping in Oasis montaj	77
Maps in Oasis montaj	77
Understanding the Parts of a Map	78
Base and Data Views	78
3D Views	78
Groups	78
Map Menus	78
Map Preparation	79
Selecting Data for Mapping	79
To Select Data for Processing	79
Creating a New Map	80
Creating a map from X and Y Coordinates	80
To Create a New Map from X,Y	81
Map Templates and Base Maps	82
Creating MAPPLOT Map Templates (manual)	82
To Create Your Own MAPPLOT Map Template	82
Creating a Base Map	84
To Create and Plot a Figure Style Base Map	85
To Create and Plot a Map Style Base Map	87
Plotting Survey Lines	89
To Plot Survey Lines	90
Plotting Profiles	90
To Plot Profiles	91
Moving Around the Map	92
Map Layout	92
Map Tools	94

Map Pop-Up Menus	94
Map Window Status Bar	94
Configurable Hotkeys	94
Oasis montaj Gridding	95
Grid and Image Menu	95
Drag-n-Drop Geosoft Grid Files	96
Copying and/or Converting Grid Files	96
Copying and/or Converting Grid Files	96
Gridding Methods	96
Minimum curvature	96
Bi-Directional	97
Kriging	98
Anisotropic Kriging	99
Tinning	99
Inverse Distance Weighting Gridding	100
Anti-aliasing	100
The Inverse Distance Weighting Function: Defining the Weighting Slope and Power	100
Direct Gridding	101
Sample Density Gridding	101
Trend Enforcement Gridding	101
Gridding using the Bi-Directional Gridding Method	101
To Apply Bi-Directional Gridding to XYZ Data	101
To Apply Bi-Directional Gridding to XYZ Data Using a Control File	102
Multiple Channel Gridding	103
To Grid Multiple Channels	103
Displaying a Grid	104
To Display a Single Grid on a Map	104
Recolour Grids	105
Contouring	105
Plotting Contours	105
To Quick Contour a Grid	106
3D Gridding Methods in Oasis montaj	107
3D Menu	107
Creating a Voxel using the Kriging Gridding method	108
To Create a 3D Voxel model using Kriging:	108
Creating a Voxel Model using the Direct Gridding method	109
To Create a 3D Voxel model using Direct gridding:	109
Creating a Voxel Model using the Inverse Distance Weighted (IDW) Gridding method ...	111
To Create a 3D Voxel model using Inverse Distance Weighted Gridding:	111
Oasis montaj Trend Enforcement Gridding	113
Accessing the Trend Enforcement Menus	114

To Access the Trend Enforcement Menu	114
Finding Trends	114
To Find Trends	114
Accessing Trend Lines in the Database	115
To Access Trend Lines in a Database	115
Plotting Trend Lines	115
To Plot Trend Lines	116
Adding Your Own Trend Lines	116
To Add Your Own Trend Lines	117
Gridding Trend Enforced Data	117
To Apply Bi-Directional Gridding to Trend Enforced Data	118
Comparing the Two Grids	118
Deselecting/Selecting Trend Lines	119
To Deselect/Select Trend Lines in a Database	119
Applying Filters in Oasis montaj	120
Filter Menus	120
Database Tools Filters menu	120
Grid and Image Filters menu	120
Apply a Filter to a Database Channel	120
Displaying a Profile	121
To Display a Profile	121
Displaying a Profile of Different Line	121
To Display the Profile of Different Line	121
Applying a Non-Linear Spatial Domain Filter	122
To Apply the Non-Linear Filter	122
To View the Profile of the Filtered Channel	122
Applying a Filter to a Grid	123
Applying Grid Filter	123
To apply a filter to a grid	123
Applying Grid Math	124
To Subtract Grids	124
Oasis montaj Map Editing and CAD Tools	127
Displaying an Interpretation Map	127
To Open a Map	127
Map Groups	128
View/Group Manager Tool	128
Rendering Options	129
Other Tool Options	129
Creating a New Group	129
To Create a New Group	130
Selecting and Editing Groups	130

Editing Using the View/Group Manager Tool	130
Editing Using the Pop-Up Menu	131
Render Images	131
Hiding and Moving Map Groups	131
To Hide a Group on a Map	131
To Move a Group	132
Masking a Group	133
To Create a Rectangular PLY File	133
To Apply a Mask to a View	134
To Mask a Group	134
Lines, Poly-Lines and Polygons	134
Drawing Poly-Lines	135
Drawing Poly-Lines to Identify Linear Features	135
Using Snap to Point and Snap to Line	136
Changing Line Attributes	136
Changing Line Attributes	137
Creating Polygons from Lines	138
To Create Polygons from Lines	138
Creating Polygon Fills	139
Editing Polygons and Legend Boxes	140
Using the Polygon Attributes Tool	141
Get Polygon Attributes	141
Apply Polygon Attributes	142
To Copy Polygon Attributes from one Polygon to Another	142
Polygon Edit Tools	142
Add/Remove Vertices on a Polygon	144
To Access the Edit Polygon Vertices Option	144
To Add Additional Vertices on a Polygon	144
To Remove Vertices from a Polygon	144
Using the Windows Clipboard with Maps	144
Copying Georeferenced Information between Maps	145
Using the Clipboard to create a Montage Map or Poster	146
Using the Clipboard to Paste a Picture from Another Application on a Map	147
Using the Clipboard to Export Graphics from a Map to Another Application	148
Oasis montaj Advanced Mapping	150
Creating Custom Base Maps	150
Using MAPPLOT to create a Custom Base Map	150
To Create a Custom Base Map from a MAPPLOT Control File	150
To View and Edit a Control File	151
Gridding Random Data	155
Grid Random Data Using Minimum Curvature	155

To Perform Minimum Curvature Gridding	157
Gridding Geochemical Data	157
Kriging – Quick Overview	158
Variograms	158
Create a Variogram File	159
Variogram Models and Parameters	161
To Define a Variogram Model	162
To Grid the Data	163
Plotting Kriged Data	164
Symbols, Colour Bars and Legends	165
Plot Symbols, Colour Bars, and Legends	165
To Plot Location Plot Symbols	165
To Post Data Values on a Map	167
To Plot Proportional Sized Symbols	168
To Plot Zone Coloured Symbols	170
To Plot Colour Range Symbols	172
To Plot a Colour-Range Symbol Legend	174
Oasis montaj Colour Imaging	177
Creating Colour Shaded Images	177
To Create a Colour Shaded Image	178
Interactive Colour Imaging and Shadowing	178
To Activate the Colour Tool	179
To Manipulate the Colour Spectrum / Percentile Groups	179
To Activate the Interactive Colour Shading Tool	180
Oasis montaj Advanced Imaging	180
Create Advanced Images	181
2-Grid Composite Map	182
3-Grid Ternary Plot	182
To Add a Ternary Legend to a Ternary Map	183
Oasis montaj 3D Viewer	185
Opening the 3D Viewer	185
Navigating the 3D Viewer	185
3D Tools	186
3D Viewing Window	186
Status Bar	187
Notes about 3D Views	187
3D Views in Oasis montaj	187
Understanding Views	187
Creating a New 3D View in the 3D Viewer	188
To Create a New 3D View	188
Working with Planes	189

To add a Plane to an Existing 3D View	189
To Set a Plane as the Default Drawing Plane	190
Plotting 3D Colour Range Symbols	193
To Plot 3D Colour Range Symbols	193
Working with Voxels	195
To Display a Voxel in a 3D View	195
3D Gridding Methods in Oasis montaj	1
3D Menu	1
Creating a Voxel using the Kriging Gridding method	2
To Create a 3D Voxel model using Kriging:	2
Creating a Voxel Model using the Direct Gridding method	3
To Create a 3D Voxel model using Direct gridding:	3
Creating a Voxel Model using the Inverse Distance Weighted (IDW) Gridding method	5
To Create a 3D Voxel model using Inverse Distance Weighted Gridding:	5
How to Merge Voxels in Oasis montaj	1
To Invoke the Merge Voxels Tool	1
How to Filter a Voxel in Oasis montaj	1
To Smooth a Voxel	1
To Emphasize the Areas of Rapid Change	4
To Apply a Gradient Filter	5
Wireframing Geological Interpretations in 3D	6
Geosurface Files	7
Wireframing Options	8
Number of triangles	8
Extrude off-section vertices	8
New Segments	9
Remove Wireframes for Active Feature	9
Wireframe Ends for Closed Surfaces	10
Viewing Total Volume and Total Surface Area	10
Creating Isosurfaces from a Geosoft Voxel	11
Voxel Math Expressions in Oasis montaj	13
Use Closed Surfaces in Voxel Math Expressions	13
Import and Export 3DV Files in Oasis montaj	15
Export a 3D View as a 3DV File	15
Importing a 3DV file to a Map	15
Geosoft Seeker Tool in Oasis montaj	16
Searching for Data	16
To Use Search Tools	17
Viewing Results	18
To Use the View Results Tools	19
Downloading Selected Datasets	20

To Download Selections	20
Desktop Cataloger	21
Working with Scripts in Oasis montaj	22
Some typical uses for scripts include:	22
Batch Processing with Scripts	22
To Create a Script File	22
To view a Script	23
Example	24
Exporting and Printing Maps in Oasis montaj	27
Exporting Maps	27
Exporting Maps to a File	28
To Export Maps to a File	28
Printing Maps	29
Page and Printer Setup	29
To Configure Your Page Setup	29
To Configure Your Printer Setup	30
Change or Add New Printer Configuration	30
To Add a New Printer Configuration	30
Print Preview	31
To Preview a Map Before Printing	31
Print a Map	32
To Print a Map	32
Printing in Batch	33
To Print in Batch	33

Geosoft License Agreement

Should there be a separate signed agreement between you and Geosoft, or between your company and Geosoft, pertaining to the licensed use of this software, that agreement shall take precedence over the terms of this agreement.

1. DEFINITIONS:

In this Agreement:

"Licensed Program(s)" means the actual copy of all or any portion of Geosoft's proprietary software technology, computer software code, components, dynamic link libraries (DLLs) licensed through the Geosoft license server, including any modifications, improvements or updates provided by GEOSOFT.

"Effective Date" is the date the Geosoft license is installed. This date is recorded by the Geosoft License server when the Licensed Program(s) is installed.

"Services" means the Services described on Section 4.

"Termination" means the occurrences contemplated by Section 6 and 7.

2. LICENSE:

If you have paid the license fee for a single user license of the Software, GEOSOFT grants to me a non-transferable and non-exclusive license to use the Licensed Program(s) for my own purposes and solely in executable code format, whereby the Licensed Program(s) are being used only by myself, on one computer, at any one time.

If you have purchased a Subscription License for the Software by paying the fee therefor, GEOSOFT grants to me a terminable, non-transferable, and non-exclusive license to use the Licensed Program(s) for my own purposes and solely in executable code format, whereby the Licensed Program(s) are being used only by myself, on one computer, at any one time for the duration of the subscription service purchase, typically one year. Under the Subscription License, the Licensed Software is licensed only for the intended duration. If you do not renew the Subscription beyond the duration, you agree to stop using the software, and remove the software from your systems.

Title and all intellectual property rights in and to the License Program(s), including, without limitation, copyright, trade secrets and trademarks, shall remain with GEOSOFT and its licensor(s). I agree to refrain from raising any objection or challenge to such intellectual property rights, or from assisting or causing or permitting other(s) to do so, during the term of the Agreement and thereafter.

I may not assign this Agreement or any part thereof or sub-license the rights granted herein, or lend, rent, time-share, sell or lease the software without the prior written consent of GEOSOFT.

I may not attempt to reverse engineer, de-compile or disassemble the software.

I may not copy the Licensed Programs other than for a single archival copy and reasonable number of backup copies.

I may not use the Licensed Program(s) for commercial network services or interactive cable or remote processing services.

I may not make any attempt to circumvent the License Manager that controls the access to the software use.

I may not remove or obscure any copyright, trademark, or legend or notice.

3. TERM:

The Term of this Agreement shall commence on the Effective Date and shall continue until termination, as described in Section 6.

4. SERVICES:

(a)

(i) According to the terms of my initial purchase, GEOSOFT shall make available to me, without additional fees such corrections and improvements to the Licensed Program(s) as may be generally incorporated into the Licensed Program(s) by GEOSOFT. (Normally this will be for a period of twelve (12) months).

(ii) GEOSOFT has a strong commitment to customer service and product support. GEOSOFT offers me, subject to applicable Service Charge(s), continuing support in the form of email or telephone advice and other assistance in problem diagnosis and the correction of errors or faults in the Licensed Program(s) during the life of this License. When a problem occurs which appears to be related to errors or faults in the Licensed Program(s), I may contact GEOSOFT and GEOSOFT will make an honest effort to solve the problem. However, GEOSOFT cannot guarantee service results or represent or warrant that all errors or program defects will be corrected. Also it is to be noted that each Licensed Program is designed to operate on a Windows XP (32 bit) or Windows Vista Business (32 or 64 bit), Windows 7 (32 or 64 bit) or Windows 8 platforms.

(iii) Further, if I request service relating to the modification of the Licensed Program(s) to meet a particular need or to conform to a particular operating environment, GEOSOFT may, at its discretion, modify the Licensed Program(s) to meet these particular needs, subject to applicable Services Charge (s). However, all intellectual property or other rights which may arise from such modifications shall reside with GEOSOFT.

(b)

(i) According to the terms of my Subscription License, GEOSOFT shall make available to me, without additional fees such corrections and improvements to the Licensed Program(s) as may be generally incorporated into the Licensed Program(s) by GEOSOFT. (Normally this will be for a period of twelve (12) months).

(ii) GEOSOFT has a strong commitment to customer service and product support. GEOSOFT offers me, subject to applicable Service Charge(s), support in the form of email or telephone advice and other assistance in problem diagnosis and the correction of errors or faults in the Licensed Program(s) during the life of this subscription. When a problem occurs which appears to be related to errors or faults in the Licensed Program(s), I may contact GEOSOFT and GEOSOFT will make an honest effort to solve the problem. However, GEOSOFT cannot guarantee service results or represent or warrant that all errors or program defects will be corrected. Also it is to be noted that each Licensed Program is designed to operate on a Windows XP (32 bit) or Windows Vista Business (32 or 64 bit), Windows 7 (32 or 64 bit) or Windows 8 platforms.

5. PROTECTION AND SECURITY OF LICENSED PROGRAM:

I agree that all additions, modifications, revisions, updates and extensions to the Licensed Program(s) shall be subject to all of the terms and conditions in this agreement.

I acknowledge that all copies of the Licensed Program(s), provided by GEOSOFT or made by me pursuant to this Agreement, including, without limitation, translations, compilations, partial copies, modifications, derivative materials and/or updated materials, are proprietary, and the property of Geosoft or its licensor(s), and may not be distributed to any other persons, without GEOSOFT's or its licensor(s) prior written consent.

I will not provide or otherwise make the Licensed Program(s) available to anyone in any form without GEOSOFT's prior written consent.

6. TERMINATION:

This agreement shall terminate upon the termination date, if any, specified in your purchase agreement with Geosoft.

This agreement may be terminated only upon thirty-day's prior written notice to GEOSOFT.

GEOSOFT may terminate this Agreement upon prior written notice effective immediately if I fail to comply with any of the terms and conditions of this Agreement.

This Agreement shall terminate automatically upon the institution, or consenting to the institution of proceedings in insolvency or bankruptcy, or upon a trustee in bankruptcy or receiver being appointed for me/us for all or a substantial portion of my/our assets.

7. EVENTS UPON TERMINATION:

I shall forthwith discontinue use of the Licensed Program(s), on the day Termination shall occur and agree not to resume such use in the future without written authorization from GEOSOFT.

I shall uninstall and remove all software from my computer. Within thirty days after Termination, I shall destroy all physical and digital copies of the Licensed Program(s) and certify such destruction in writing to Geosoft within 10 days thereof.. This obligation relates, without limitation, to all copies in any form, including translations, compilations, derivatives and updated materials, whether partial or complete, and whether or not modified or merged into other materials as authorized herein.

8. WARRANTY:

GEOSOFT, nor its licensor(s), does not warrant that the functions contained in the Licensed Program will meet my requirements or will operate in the combinations which may be selected for use by me, or that the operation of the Licensed Program will be uninterrupted or error free or that all program defects will be corrected.

Each Licensed Program shall be furnished to me in accordance with the terms of this Agreement. No warranties either express or implied are made to me regarding the Licensed Program.

THE FOREGOING WARRANTIES ARE IN LIEU OF ALL OTHER WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

9. LIMITATION OF REMEDIES:

I agree to accept responsibility for the use of the programs to achieve my intended results, and for the results obtained from use of said Program(s). I therefore accept complete responsibility for any decision made based on my use of the aforementioned Licensed Program(s).

In no event shall GEOSOFT, or its licensor(s) be liable for any damages, or loss of any kind, whether special direct, indirect, incidental or consequential, arising from use of, or performance or non-performance of the Licensed Program(s), including but not limited to any lost profits, lost savings or other consequential damages, or liability for use of the Licensed Program in a high-risk activity. Even if GEOSOFT or its licensor (s) has been advised of the possibility of such damages, or for any claim against me by any other party.

10. GENERAL:

I agree that this Agreement is a complete and exclusive statement of the agreement with GEOSOFT.

This Agreement supersedes all previous Agreements with respect to the Licensed Programs, with the exception of a current signed Technical Service Agreements.

GEOSOFT is not responsible for failure to fulfill its obligations under the Agreement due to causes beyond its control.

Should any part of This Agreement for any reason be declared invalid, such declaration shall not affect the remaining portion which shall remain in full force and effect as if this Agreement had been executed without the invalid portion thereof.

The relationship between the parties is that of independent contractors. Nothing contained in this Agreement shall be deemed to constitute or create a partnership, association, joint venture or agency.

The provision of this Agreement shall be binding upon me and GEOSOFT and my respective successors and permitted assigns.

This Agreement will be governed by the laws of the Province of Ontario and applicable laws of Canada.

11. ACCESSING DATA ON 3RD PARTY SITES:

Licensee shall not use the software to transfer or exchange any material where such transfer or exchange is prohibited by intellectual property laws or other applicable laws. Geosoft neither endorses nor assumes any responsibility for the safety, accuracy or availability of data sites or the information contained on them. Data sites, their operation and content are the responsibility of the person or entity for which they were created or maintained (the "Owner"). Both your use of a data site, and your right to use or reproduce information or materials from a linked site, are subject to the terms of use governing that particular data site. Any comments or inquiries regarding a data site must be directed to its Owner.

12. THIRD PARTY PRODUCTS

MICROSOFT BING MAPS

Microsoft Bing Maps Terms of Use can be found at: [Microsoft Bing Maps and MapPoint Web Service End User Terms of Use and Embedded Maps Service Terms of Use](#)

Microsoft may collect information such as, but not limited to, an end user's IP address, requests, time of submissions and the results returned to the user, in connection with transaction requests to the Services. All such information collected will be subject to, and used in accordance with, the data practices described in the then-current Microsoft Online Privacy Statement, a current copy of which is available at [Microsoft Online Privacy Statement](#).

Finding Help Information

There are several functions included in the basic Oasis montaj help system that may be useful to your work. The entire documentation for the system is available through the Application Help system. This electronic library of information enables us to constantly update the information and provide you with the most up-to-date information available. The best way to find information in this system is to use the Search tab to perform a full-text search across all help topics.

Contacting Technical Support

<p>North America</p> <p>Geosoft Inc., Queens Quay Terminal 207 Queens Quay West Suite 810, PO Box 131 Toronto, ON Canada M5J 1A7</p> <p>Tel +1 (416) 369-0111 Fax +1 (416) 369-9599 Email: support@geosoft.com</p>	<p>Europe and North Africa</p> <p>Geosoft Europe Ltd. 20/21 Market Place, First Floor Wallingford, Oxfordshire United Kingdom OX10 OAD</p> <p>Tel: +44 1491 835 231 Fax: +44 1491 835 281 Email: support@geosoft.com</p>
<p>South America</p> <p>Geosoft Latinoamerica Ltda. Praça Floriano 51 / 19º Andar CEP: 20031-050, Centro Rio de Janeiro, RJ, Brasil</p> <p>Tel: (55-21) 2111-8150 Fax: (55-21) 2111-8181 Email: support@geosoft.com</p>	<p>Australia and Southeast Asia</p> <p>Geosoft Australia Pty. Ltd 14/100 Railway Road Subiaco, WA, Australia 6008</p> <p>Tel +61 (8) 9382 1900 Fax +61 (8) 9382 1911 Email: support@geosoft.com</p>
<p>South and Central Africa</p> <p>Geosoft Africa Ltd. Southdowns Office Park Block A Ground Floor Unit 5 Cnr John Vorster Drive & Karee street Irene, Centurion South Africa 0062</p> <p>Tel: +27 12 676 9360 Fax: +27 12 665 0246 Email: support@geosoft.com</p>	

Getting Started with Oasis montaj

Oasis montaj Mapping and Processing System (MAPS) is Geosoft's core software platform for working with large volume spatial data. The platform provides the functionality required to locate, manage, visualise, manipulate, display and share located Earth Science data.

The Oasis montaj environment provides direct access to data contained in Oasis databases through a Spreadsheet window and an integrated Profile display window. The Oasis database is a high-performance database that provides efficient storage and access for very large spatial data sets.

Metadata (data about data) is captured by Oasis montaj from the first time the data is touched. The metadata information, which is based on ISO 19115 standards, is stored inside the data (if supported) or as a companion XML file.

The Interface provides visual Data links that enable you to dynamically connect data in the Spreadsheet, Profile, Map and ArcGIS MXD views. DAT Technology (for Accessing Grids and Images) enables the Interface to use a variety of grid and image formats in Oasis montaj.

The Oasis montaj core platform provides the basic resources for all Geosoft Applications and Tools. Geosoft provides a variety of systems that address specific applications in exploration geophysics, drillhole geology, exploration geochemistry and other areas. These systems consist of menus and corresponding Geosoft executables (GXs) that run on the core platform.

Oasis montaj Mapping and Processing System

The Oasis montaj Mapping and Processing System contains a variety of built-in data import, processing, analysis, visualization, mapping, and integration capabilities. This system enables you to perform complex processing, editing, mapping, and interpretation tasks including the ability to:

- Create, import and export: maps, databases, grids, voxels, MXD files, images and profiles
- Automatic Metadata Creation. Oasis montaj creates metadata automatically when you work with data, storing the name of the user, date, time and actions performed. A XML Metadata viewer and editor enables easy access to your metadata.
- Process data using Geosoft's gridding, contouring and 1D filter algorithms
- Advanced grid utilities and gridding toolkit
- 3-dimensional mapping that includes a range of options for visualizing data including; voxels, multiple surfaces and sections, each with own relief and contents, and each with its own orientation in 3D space
- Design and layout maps, consisting of base maps, grids, images, customized annotations, labels, colour symbols, multi-parameter symbol plotting, and other entities
- Advanced CAD tools to draw interpretation features on your maps
- Communication Service that enables interoperability between multiple instances of Oasis montaj, Target for ArcGIS and other third party applications
- Automate tasks using scripts

Get Started!

▶ [Getting Started with Oasis montaj](#)

Software and Hardware Requirements

Operating System	Windows 8 (32-bit and 64-bit) Windows 7 SP1 (32-bit and 64-bit) - Recommended Windows Vista Business SP2 (32-bit and 64-bit) Windows XP Service Pack 3 (32-bit) - Minimum requirement Windows 2000, 95, 98, ME, NT are not supported.
CPU	A Dual Core processor is recommended. An Intel Celeron processor is not recommended.
RAM	3+ GB recommended, 1 GB minimum required.
Graphics	Nvidia Professional 256MB 3D (OpenGL 2.0) Graphics Card recommended.
Printer/Plotter	Any Windows® supported colour printer. Hewlett Packard® large-format ink-jet plotters are recommended.
Installation	You must be logged on with Administrator permissions when installing Oasis montaj.
Installation Disk Space	2 GB or more of free space on the Program Files drive is required for the installation process.
Data Disk Space	Data disk space depends on the volume of project data to be processed and the printer driver you are using, however 100 GB is recommended. This is largely based on your business and data requirements.
Internet	<p>Internet Connection: To use VOXI, we recommend a medium to high-speed internet connection (512Kbps upload and 1Mbps download or higher). The faster your internet connection, the faster you will be able to upload your modelling information to the VOXI cloud servers and download your results.</p> <p>Internet Browser: To use the internet capabilities in Oasis montaj or Target, you will need to install Internet Explorer 8.0 or later. This does not mean that you have to have Internet Explorer as your default browser; Oasis montaj and Target just use the internet connection technology supplied in IE8.</p>
Supporting Software	<p>The following software prerequisites are required and will be downloaded and installed if not present:</p> <ul style="list-style-type: none"> • Microsoft .NET 4 • ESRI ArcEngine 10.2

Geosoft Connect and Geosoft ID

Geosoft Connect is a service available through the Oasis montajTarget for ArcGIS application. Geosoft users can sign in to their Geosoft ID, using Geosoft Connect, to access any online service that requires a secure sign in, including online licensing, online updates, VOXI Earth Modelling and Bing Map services. Geosoft Connect enables you to sign in from your desktop when you are connected to the Internet.

As a customer, your Geosoft ID is the only sign in you'll need to remember when accessing any of our services, resources, community activities or our web site.



For more information see the [Geosoft ID Common Questions](#) on the Geosoft web site.

Create your Geosoft ID

Your Geosoft ID is the only sign in you'll need to remember when accessing any of our secure online services, including online licensing, online updates, VOXI Earth Modelling and Bing Map services.

Create Geosoft ID:

1. Connect to the Internet.
2. Go to the, *Register Geosoft ID* (<https://www.geosoft.com/geosoftid/>) web page.
3. Click the **Create a Geosoft ID** button. The *Create a Geosoft ID* web page is displayed.
4. Enter your information as required and then click the **Create a Geosoft ID** button.



We prefer if you use your business email for your Geosoft ID. You may get a message saying 'A user account already exists for that e-mail address.', If you don't know or have forgotten the associated password, you may need to reset it.

5. Once you have registered, Geosoft will immediately send you an activation link to the e-mail address you provided (within 20 minutes).
6. When you receive the activation e-mail, click the **Click to Activate** link. Your Geosoft ID will be activated.



Geosoft Connect is a service available through the Oasis montajTarget for ArcGIS application. You can use this service to sign in to your Geosoft ID from your desktop, when you are connected to the Internet.

Welcome to the Oasis montaj Environment

This guide walks you through the basic steps to start working with your new software.

Opening Oasis montaj

Oasis montaj can be opened from the *Start* menu.

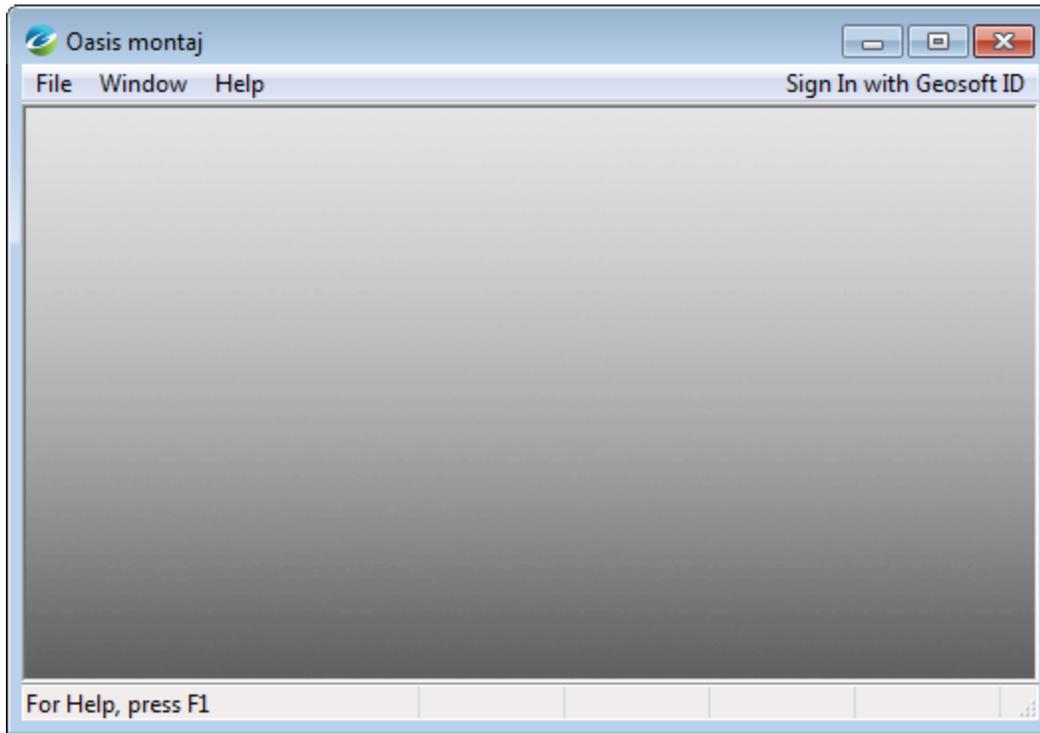


Oasis montaj should already be installed.

To Start Oasis montaj

1. On the *Start* menu, select *Programs*, and then select *Geosoft*.
3. Select *Oasis montaj* and then select **Oasis montaj**.

The Oasis montaj Graphical User Interface (GUI) opens with standard Windows style elements, including the Main Menu, Pop-up Menus and Application Help as shown below:



Oasis montaj System Files

As you work with the system, you will become familiar with a variety of standard files used for specific functions. The following list provides a short summary of the more important files.

Oasis montaj System Files	System File Extensions
Geosoft database file	*.GDB
Map file, including plots and grids	*.MAP
Geosoft grid file	*.GRD
Colour information for grids/images	*.AGG
Geosoft voxel file	*.GEOSOFT_VOXEL
ArcGIS MXD file	*.MXD
Oasis XML file	*.XML
Geosoft eExecutable	*.GX
Geosoft script file	*.GS
Oasis menu, Oasis sub-menu	*.OMN, * .SMN
Geosoft project file	*.GPF
Geosoft projection information file	*.GI
Geosoft map file (used in the MapInfo software to distinguish a Geosoft map file from a MapInfo (*.map) file)	*.GM

Oasis montaj Projects

Oasis montaj requires an open project. An Oasis montaj "Project" encompasses every item in your working project. This includes the data files in your project (databases, maps, and grids), the tools used (including auxiliary tools such as histograms, scatter plots etc.) and the project setup. A project includes the menus you have displayed and whether you are working on a map or profile and the state in which you left it the last time you used it.

Creating a Project

The project also controls your working folder. The system enables you to access files anywhere, but it is a good strategy to carefully organize your data (project information and files) before carrying out any processing.



A general rule to follow is to avoid working in the Geosoft folder.

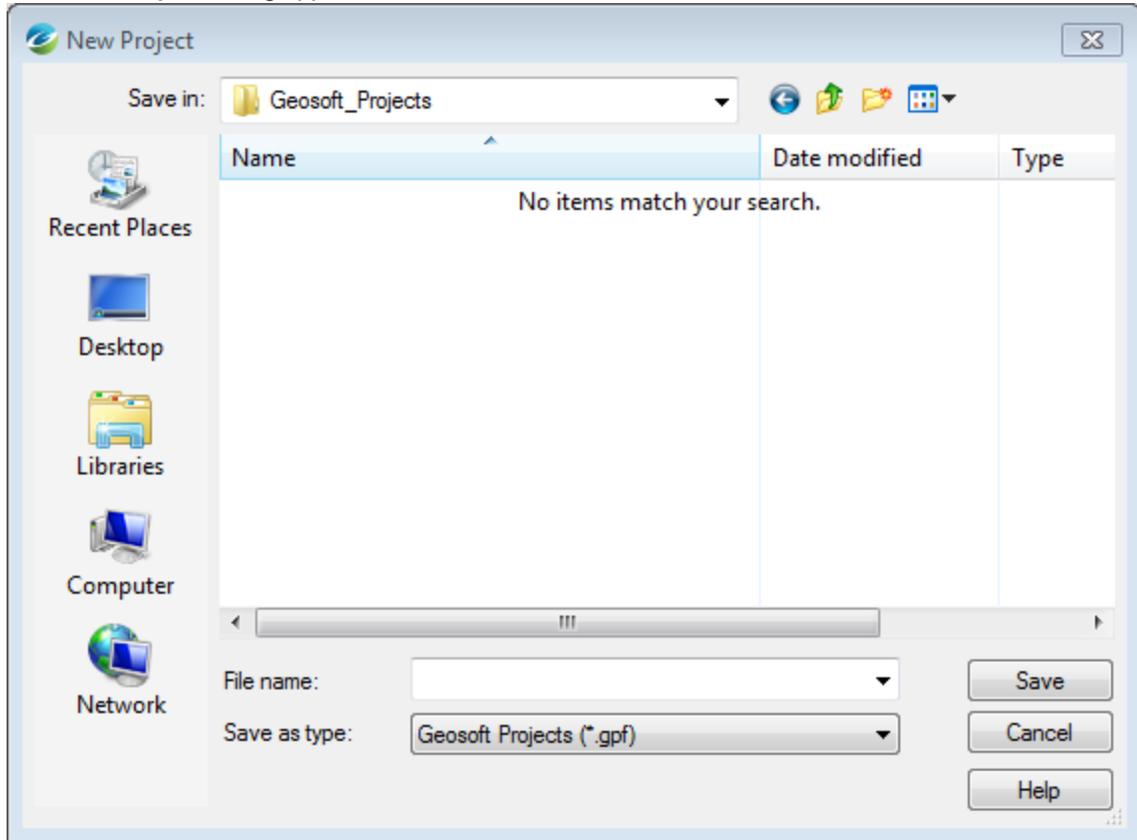


Use your Windows Explorer tools to create a working folder, for example E:\Geosoft_Projects.

Projects are saved as (*.gpf) files. If you open an existing project from a folder, the system assumes that all your project files are located in the same folder. To streamline your work, as well as keep it organized, you may wish to make sure that your project file is in the same folder as the other files you want to use. We recommend that each project you work on have its own project (*.gpf) file. If you use a number of applications or extension tools in Oasis montaj that have different menus, you can use the project to display only the menus you require.

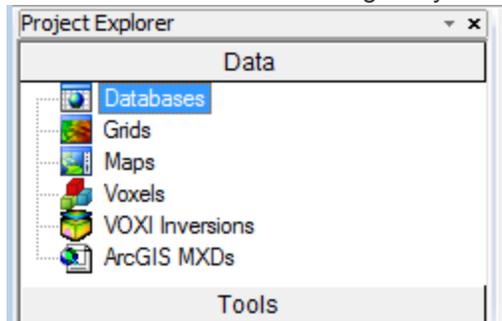
To Create a Project

1. Start Oasis montaj.
2. On the **File** menu, click **Project** and then click **New**.
The *New Project* dialog appears.



Oasis montaj assumes that your data is in the folder containing this project.

3. Specify a *File name* and folder for the project.
4. Click **Save**. The system saves the project and indicates it is open by adding menus to the menu bar, adding buttons to the Standard Short-cut bar and by displaying the Project Explorer window. These are visual clues indicating that you are ready to start working with the system.



The *Project Explorer* tool enables you to browse as well as open any data that has already been loaded into the project. The Project Explorer has two windows, the *Data* window that includes all data files

included in the project and the *Tools* window that organizes and maintains the project tools. To access the Tools window click the **Tools** bar on the bottom of the Project Explorer. To return to the Data window, click the **Data** bar on the top the Project Explorer.

Closing a Project

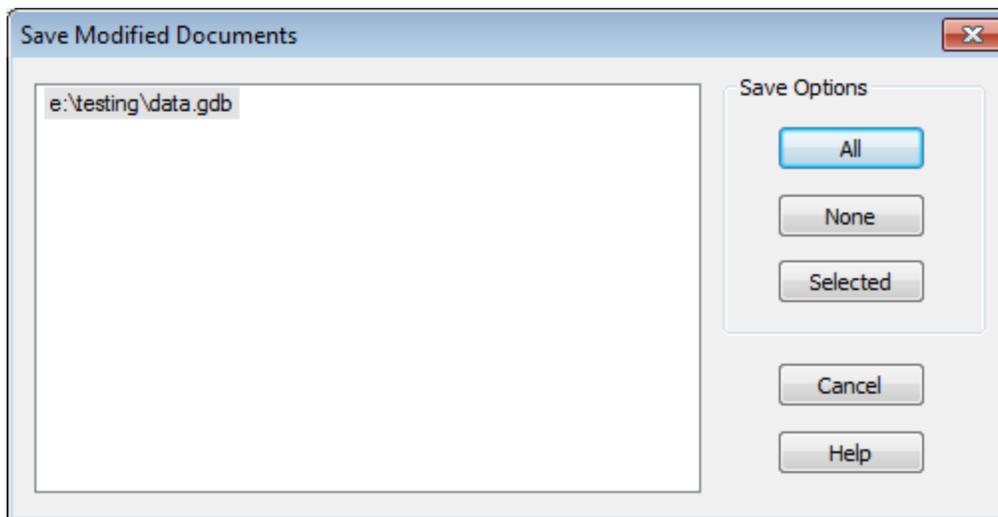
Closing a project will save all of your project databases, maps and profiles, plus information that tells the system whether you are working on a map or profile and the state in which you left it the last time you used it.

To Close a Project

1. On the *File* menu, click **Project** and then click **Close**.
The *Save Modified Documents* dialog appears.



Modified Documents' refers to items in your project that you have modified since the project was last saved, such as databases and maps.



2. Click **All** to save all open document(s) in your project. Alternately, you may select specific document(s) to save by highlighting the document(s) and click **Selected** or click **None** to abandon document(s) changes.

Oasis montaj Menus

The Oasis montaj Mapping and Processing System menus are displayed directly under the Title Bar when you open a project. These dropdown menus provide direct access to all of the systems capabilities.

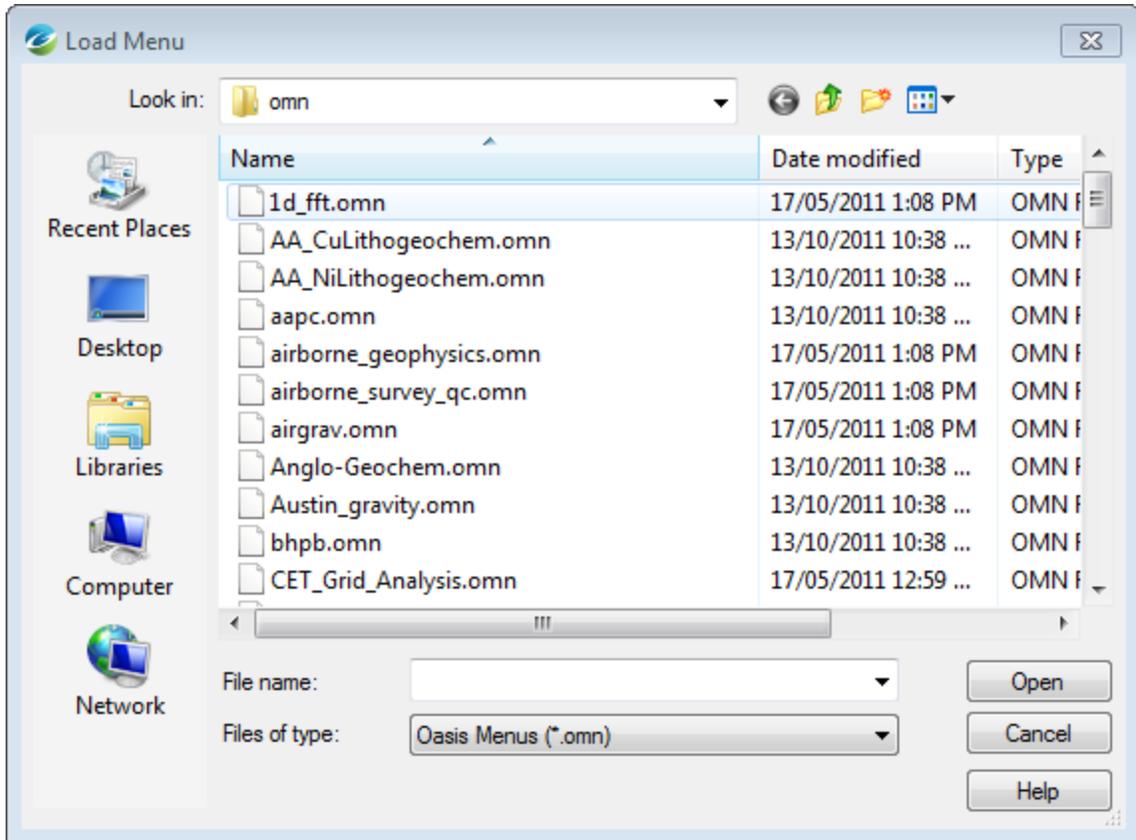
Loading Applications and Tools

In addition to the Oasis montaj base functionality, you may be licensed to access specialised applications, which we call "Extensions". Extensions are accessed by loading a menu file (*.omn), which in turn will load all of the menus and sub-menus associated with that extension. Examples include ((geophysical and geochemical analysis, 3D drillhole plotting, gravity and magnetic filtering, levelling, interpretation etc.)).

Your Oasis montaj project keeps track of which menus are displayed. This means that you can have a different set of menus displayed for each project. This is especially useful if you are working on multiple projects and using many applications or tools. For example, if you are working on a geochemistry project, you may want to display different menus than if you are working on an UXO project. By creating separate projects with differing menus you can customise the Oasis montaj environment to suit your current project.

To Load an Application or Extension Menu

1. On the GX menu, select **Load Menu**. The *Load Menu* dialog appears.



2. Select the menu file (*.omn) from the list of files and click **Open**. The new menu appears on your menu bar.



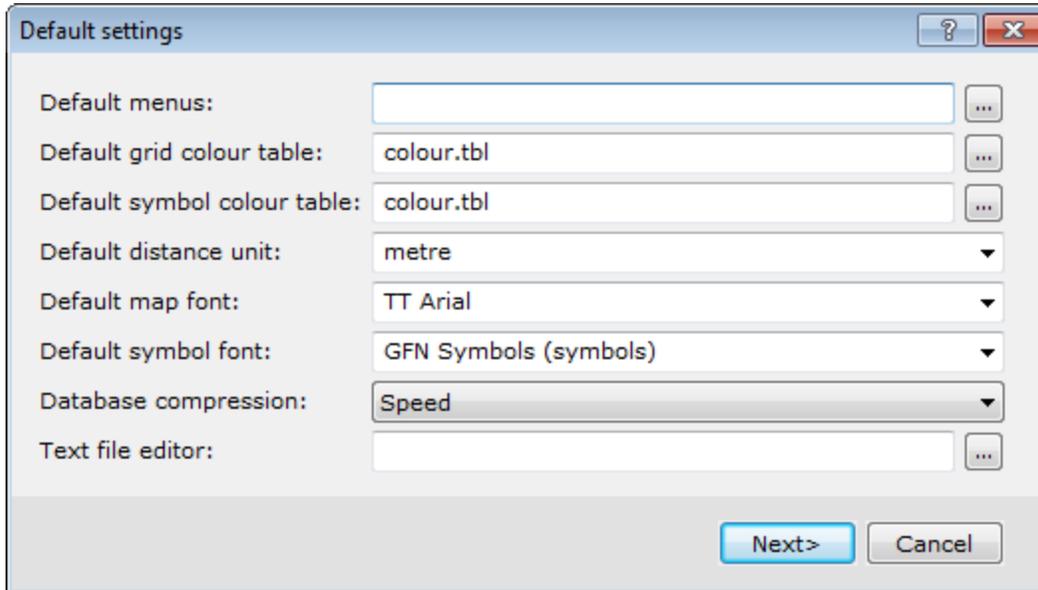
If you want to modify your default menus (i.e. load a specified menu with every new project) you can use the GX|Global Settings|General menu to display the Default settings dialog to change your settings. If you are really interested in system configurations and want to view all of the System settings, open the Advanced Settings file from the GX|Global Settings|Advancedmenu. The Advanced Settings will be displayed in the Metadata Browser.

Oasis montaj Settings

The Oasis montaj program has standard default settings which are designed to work well in most environments. However if you have particular personal or computing requirements you may overwrite the defaults to suit your needs.

To Change the Default Settings of the Oasis montaj Environment

1. On the GX menu, select **Global Settings** and then click **General**. The *Default settings* dialog appears.



2. Select the desired choices from the available selections.
3. Click **Next** to navigate through the *Default settings* dialogs.
4. Click **Finish** when complete.



*For more information on the available selections, click the **Help** button on your dialog of interest.*

Common Default Settings

The following table summarizes some of the more frequently changed settings.

Default menus	Select the menu files (.omn) to load as the default menus when creating a new project. If more than one menu is selected, each menu must be separated by the character.
Default grid colour table	Select the default colour table to use for displaying grids.
Default symbol colour table	Select the default symbol colour table (*.tbl).
Default map font	Select the default map font. The file TRUETYPE.INI will be modified to map the "default" font from Geosoft PLT files to the TrueType font specified here. To change the mappings of other Geosoft fonts you can modify the TRUETYPE.INI file directly.
Click Next for <i>More settings</i>	
Default input grid format	Select the input grid format you would like to use by default.
Default output grid format	Select the output grid format you would like to use by default.
Default grid element type	This is the default grid element type to use for Geosoft grids. If grids will be used with Geosoft DOS programs, select "16-bit (DOS compatible)".
Error level reporting	If All errors are selected, you will see more error messages each time an error occurs. This is useful for GX programmers who may need to see more error context when debugging GX's.

Auto Save Time

Oasis montaj has an *Auto Save Time* feature that enables databases and maps to be saved automatically. When *AutoSave* is enabled a popup dialog will be displayed asking if you want to save the changes to all open documents.

The frequency of the *AutoSave* (in minutes) can be modified or shut-off from the *GX|Global Settings|Advanced* menu.



Changing this setting to zero (0) shuts-off the AutoSave feature.

Metadata

Metadata (data about data) is captured by Oasis montaj from the first time a dataset is opened by the program. The metadata information, which is based on ISO 19115 standards, is stored inside the data (if supported) or as a companion XML file. When you import or work with data in Oasis montaj, *User name*, *Date*, *Time* and any actions performed on the data will be maintained in the Metadata. When data is derived from other data, the Metadata is passed on and the lineage of what was done to the data is maintained in the Lineage section. Note that, memory intensive GDB processing tasks are not recorded in the lineage.



When working with script files you have the ability to turn off the XML metadata file creation.

Attributes (or metadata), information about data, can be simple or complex and the descriptive needs of different kinds of data are infinitely diverse. To allow for this and provide flexibility, Geosoft developed the *XML Metadata Viewer* and *Metadata Editor* to help you organize your metadata.

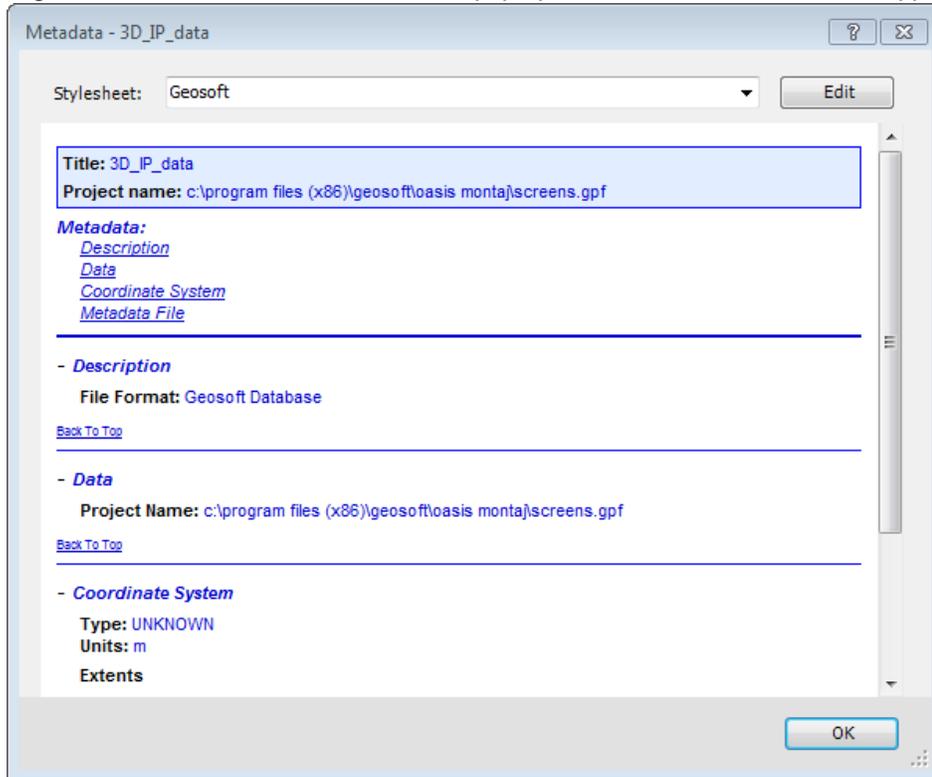
Even though each Geosoft data type (Database, Grid, Map, ArcGIS MXD, Voxel, etc) exhibits many different attributes, using these tools are easy. The current selection drives what exists inside the tool.

Metadata Viewer

Use the *Metadata Viewer* dialog to view the metadata associated with selected data files.

To View and/or Edit Metadata

1. On the *Project Explorer* window, select the data of interest.
2. Right-click and select **Metadata** from the pop-up menu. The *Metadata Viewer* appears.



3. Click **Edit** and the *Edit Metadata* dialog appears.
4. Make any necessary changes.
5. Click **Close** when finished and then click **OK** to close the Metadata Viewer.

Metadata Tool

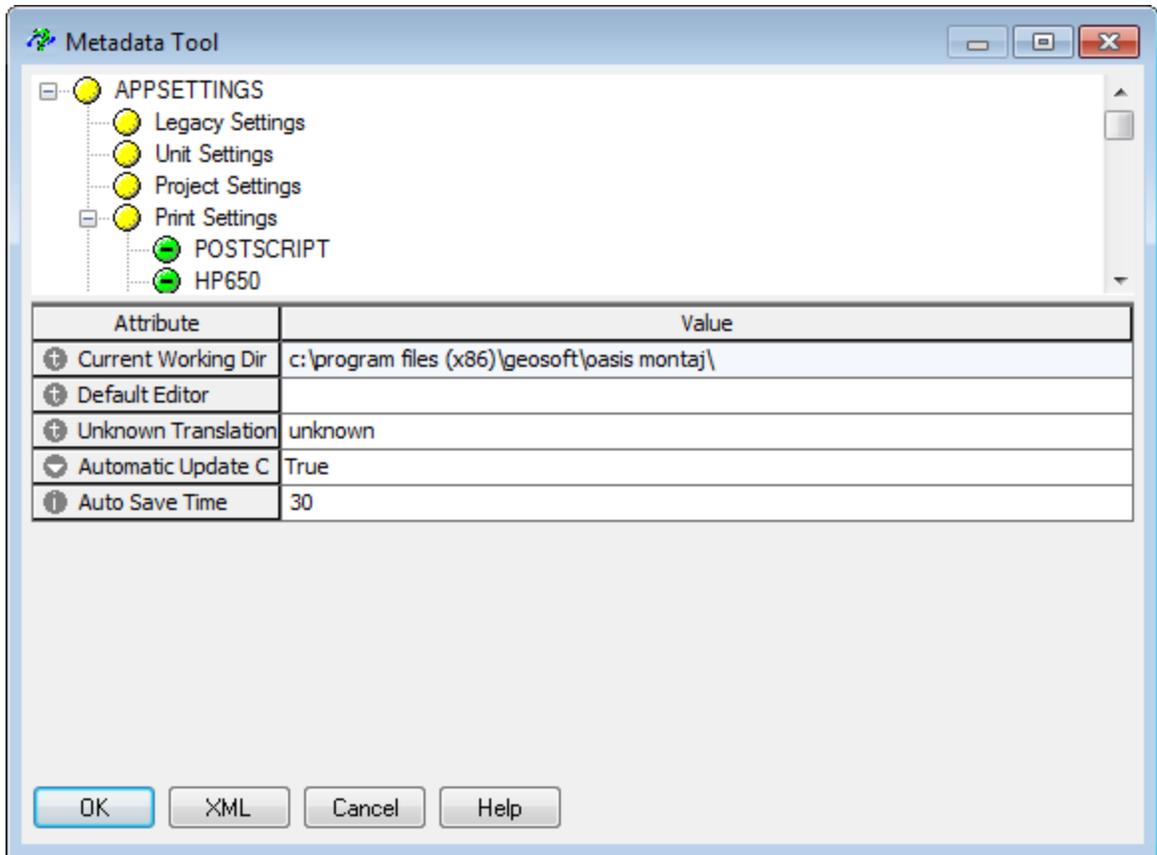
The *Metadata Tool* is a context-oriented, interactive method for viewing and editing attributes assigned to Geosoft Global Setting, DAP Data etc.



Values displayed in a grey box cannot be edited. Values displayed in a white box may be edited.

To View the Metadata Tool

1. On the *GX* menu, select *Global Settings* and then select **Advanced**.
The advanced global settings will be displayed in the *Metadata Tool*.



2. Make any necessary changes.
3. Click **OK** when finished or click **Cancel** to close the Metadata Tool.

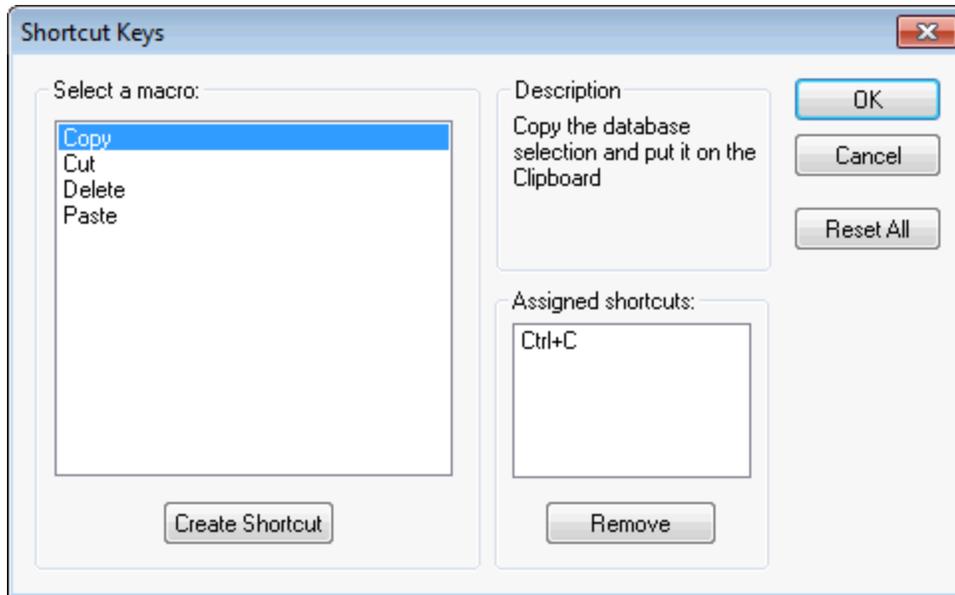
Shortcut Keys

Oasis montaj now provides configurable Shortcut Keys (Hotkeys) enabling quick and easy access to menu functions.

To view or modify these settings, you have two options:

- **Database Shortcut Keys:** Select *GX | Database Settings | Configure GDB Shortcut Keys*.
- **Map Shortcut Keys:** Select *GX | Map Settings | Configure Map Shortcut Keys*.

Both options provide an interactive dialog enabling you to configure your own shortcut keys (hotkeys) or view the provided defaults.



How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj eLicensing

With Geosoft's electronic licensing (eLicensing) model licenses are handled by the Geosoft license server. This system enables licenses to be moved between machines online, via email or if necessary by disk. Geosoft's eLicensing management system offers many tangible and real benefits, including:

- Regular upgrades and fixes. We release multiple upgrades a year (with interim patches when possible). You can only access these with a valid eLicense
- Easily transfer your license between office and home Geosoft installations
- Add extensions to your license instantly
- Evaluate extensions easily - for 14 days
- Easy to manage "1 user - 1 license" model
- Reduces costs by eliminating unauthorized use
- Rent extensions for specific time periods
- The eLicensing technology also enables us to add extra services and products on a regular basis

It may take some time to get used to the system and there are some restrictions that apply:

- Online licensing requires secure sign in of your Geosoft ID using Geosoft Connect
- Licenses are keyed to individual PCs. Therefore; you must park the License to switch it to another computer
- Licenses must also be parked before changing Operating System (for a new hard drive, for example)

Read the How-To Guide

▶ [Oasis montaj eLicense](#)

About eLicensing Tools

The Geosoft licensing system enables licenses to be updated in seconds and to be moved between computers. License updates or transfers can be done online, via email, or by disk if required.

The License options under *File/License* enable you to:

- **Show computer ID** – Displays computer ID
- **View eLicense** – Displays users license
- **Update eLicense** – Enables license to be updated from license server
- **Park eLicense** – Enables license to be parked on the Geosoft license server, and then retrieved later
- **Transfer eLicense** – Enables license to be transferred to a different computer ID
- **Lock eLicense** – Locks the license on the current computer
- **Unlock eLicense** – unlocks the license on the current computer with the use of the license key



To use the Internet capabilities in Oasis montaj, you will need to install Internet Explorer 7.0 or later. This does not mean that you have to have Internet Explorer as your default browser; Oasis montaj just uses the Internet connection technology supplied in IE7 to connect to the web.

Read the How-To Guide

 [Oasis montaj eLicense](#)

To Install eLicense

Your eLicense is installed from the Geosoft license server, online, via email or if necessary by disk.

First time use:

1. Connect to the Internet.
2. On the *Start* menu select, *Programs/Geosoft/Oasis montaj/License* and then select **Setup** (or double click on the **Oasis montaj** icon).
The *Geosoft License Manager* dialog will appear.
3. In the "License Key" box, enter your 8-digit alphanumeric license key (provided with your install DVD, or supplied to you by your Geosoft representative) and click the **[Internet]** button. Geosoft's license server will retrieve your license from our Internet server and download it to your computer.



*If you have not already signed in to your Geosoft ID, using Geosoft Connect, the Sign In with your Geosoft ID dialog will be displayed. Sign in to your Geosoft ID. For more information on your Geosoft ID see the topic **Create your Geosoft ID** in the Target Getting Started How-To Guide, or see the Geosoft ID Common Questions web page (<https://www.geosoft.com/geosoftid/common-questions>).*

4. All of the functionality you are licensed for will now be accessible to you.

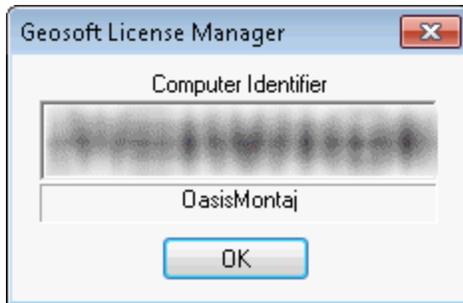
Get Started!

 [Getting Started with Oasis montaj](#)

Show Computer ID

To Show Computer ID:

1. On the *File* menu, select *License* and then select **Show Computer ID**.
The *Geosoft License Manager* dialog will appear.
2. The alpha-numeric ID displayed in this window is your Computer ID.
3. Click the **OK** button to close the window.



View your eLicense

You can view your Geosoft eLicense at any time to see what applications or extensions you are licensed for.

To View your eLicense:

1. On the *File* menu, select *License* and then select **View License**.
Your eLicense is displayed in your default web browser.
2. The eLicense information includes your License Key, Computer ID, User Name, Company, Support (Maintenance) Expiry date, Software build date, and other items as well as a list of your licensed products.
3. Close the window when you have finished viewing your License.

Update eLicense

You can update your eLicense (i.e. access additional applications or extensions) by first contacting your local Geosoft representative and then simply connecting to the Geosoft license server.

To Update your eLicense:

1. Contact your local Geosoft representative about adding additional applications or extensions to your license.
2. Connect to the Internet.
3. Use *Geosoft Connect* to sign in to your Geosoft ID.
4. On the *File* menu, select *License* and then select **Update License**.
Your eLicense is updated with the new licensed products.

To Park (or Move) your eLicense

To park (move) license from your computer to the Geosoft license server:

1. Connect to the Internet.
2. Use *Geosoft Connect* to sign in to your Geosoft ID.
3. On the *File* menu, select *License* and then select **Park License**.
The *Geosoft License Manager* dialog appears.



This is a warning message that tells you that parking a license places your license on the Geosoft license server so that it may be installed on a different computer for your own personal use. Your license is non-transferable, and use of the license by anyone

but you may be a violation of your license agreement.

4. It then asks you if you are sure you want to park your license? Click the **Yes** button. Your license is uploaded to the Geosoft license server to be downloaded at a later time and Oasis montaj closes.
5. Once the license is parked, it can be installed on another computer. After Installing Oasis montaj for the first time you will be prompted to enter your license key. You can choose to retrieve the license directly by clicking on the [Internet] button, or if you are not connected to the Internet, click on the **e-mail** button and follow the instructions. If you have parked the license, the next time you run Oasis montaj you will be prompted for your license key.



If you do not have access to the Internet, your eLicense can be transferred in a file using the Transfer License process.

Re-installing your license:

1. Connect to the Internet.
2. Use *Geosoft Connect* to sign in to your Geosoft ID.
3. On the *Start* menu select, *Programs/Geosoft/Oasis montaj* and then click **Oasis montaj**. The *Geosoft License Manager* dialog will appear.
4. In the "License Key" box, enter your 8-digit alphanumeric license key (provided when you "Parked" your license) and click the **[Internet]** button. Geosoft's license server will retrieve your parked license from our Internet server and download it to your computer.
5. All of the functionality you are licensed for will now be accessible to you.

Transfer eLicense

Your eLicense can be transferred from one computer to another (from your desktop to a laptop or visa-versa), using the computer ID.



*Oasis montaj must be installed on the destination computer to retrieve the Computer ID, which can be viewed from the **Start** menu, by selecting "Geosoft|Oasis montaj|Licenses|View Computer ID".*

To Transfer your eLicense:

1. Open Oasis montaj on your licensed computer.
2. On the *File* menu, select *License* and then select **Transfer License**. The *Geosoft License Manager - License Transfer* dialog appears.
3. In the *Target Computer ID* box, enter the computer ID of the destination computer (the computer you wish to transfer the license to) and click the **OK** button.
4. The file *OasisMontaj_Transfer.geosoft_license* file will be created in your C:\ directory.
5. Move this file via media (diskette, CD, USB drive) or network to the destination computer.
6. Using Explorer locate the file (*OasisMontaj_Transfer.geosoft_license*) on the destination computer, right-click and from the popup menu, select **Install License**. Your licensed products will be installed.



If you have access to the Internet, Parking your eLicense is the quickest and easiest method of moving your eLicense.

Lock eLicense

You can lock your eLicense to the current computer.

To Lock your eLicense to the current computer:

1. Connect to the Internet.
2. On the *File* menu, select *License* and then select **Lock License**.
The *Geosoft License Manager* dialog appears.
3. This is a warning message that asks, are you sure you want to LOCK the license on this computer? You will no longer be able to move or manage the license until it is unlocked using the license key.
4. Click the **Yes** button and the license will be locked on your computer until you unlock it using the license key.

UnLock eLicense

You can unlock your eLicense from the current computer by providing the license key.

To unlock you eLicense from the current computer:

1. Connect to the Internet.
2. On the *File* menu, select *License* and then select **UnLock License**.
The *Geosoft License Manager* dialog appears.
3. In the **License Key** box, enter your 8-digit alphanumeric license key (provided when you "Locked" your license) and click the **Internet** button.
4. Geosoft's license server will unlock your license and enable you to move and manage it again.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Working with Geosoft Databases in Oasis montaj

The purpose of this How-To Guide is to give you a general overview of how to create and edit Geosoft Databases (*.GDB).

One of the fundamental technologies in Oasis montaj is its unique database architecture. This architecture is designed to let you rapidly create and import data of many kinds (ASCII and binary) into high-performance Geosoft databases. After data is imported into a database, you have numerous options for editing, visualizing, processing and performing other tasks.

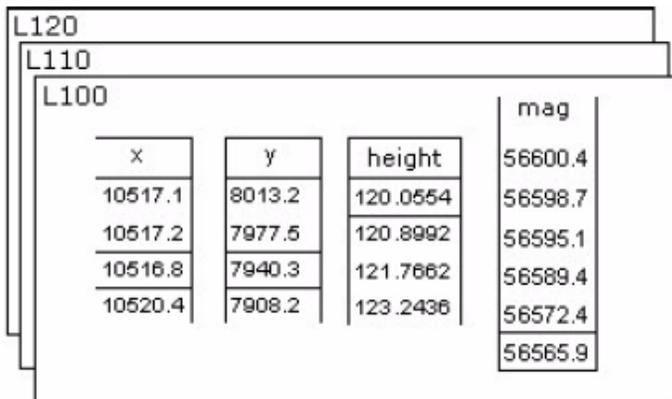
The “window” to the database in Oasis montaj is a specialized earth science spreadsheet that appears automatically when you create a new database. This spreadsheet provides access to a wide variety of data management, data and profile viewing, and processing capabilities.

The Geosoft Database

The Geosoft database is based on a proprietary binary database architecture that stores spatial data in a format that enables rapid access, processing and analysis of high volume data (tested up to 10 gigabytes).

Database Lines/Groups

Organized in lines (or groups), columns and elements, the database stores all data values of a particular type in individual columns or channels. This enables stand-alone processing of columns and eliminates the need to write results to interim storage areas and then re-write them after processing.



x	y	height	mag
10517.1	8013.2	120.0554	56600.4
10517.2	7977.5	120.8992	56598.7
10516.8	7940.3	121.7662	56595.1
10520.4	7908.2	123.2436	56589.4
			56572.4
			56565.9

The result is a significant increase in processing efficiency. This Geosoft file structure provides additional advantages including:

- Ability to process any number of lines/groups
- Ability to handle lines/groups with unlimited columns
- Ability to handle very large datasets (tested up to 10 gigabytes)

➤ Oasis montaj Supported Data Formats

Each database “line” has a line number, version number, line type, flight number (only required for airborne data), survey date and selection status.

A “group” is a special kind of line. It also stores data in channels, but instead of using a name based on line number, version number, line type and flight number, it can be named using any alpha-numeric string (with some exceptions).

Each database “group” line has an optional “CLASS” name. Like a regular line, it has a selection status.

While all the channels in a database are accessible inside a regular line, a group line can restrict the available, displayable channels to a smaller subset, excluding those not relevant to the group’s data.

A database may contain any number of lines and/or groups. If your survey data has not been collected on a line basis (i.e. is random), or cannot be organized conveniently into groups that can be thought of as lines (such as drillhole or UXO target data), you can store all data in a single “line”.



An additional benefit of the Geosoft database structure is that since columns are stored separately, this file format is capable of storing and manipulating data with different end point and sample intervals.

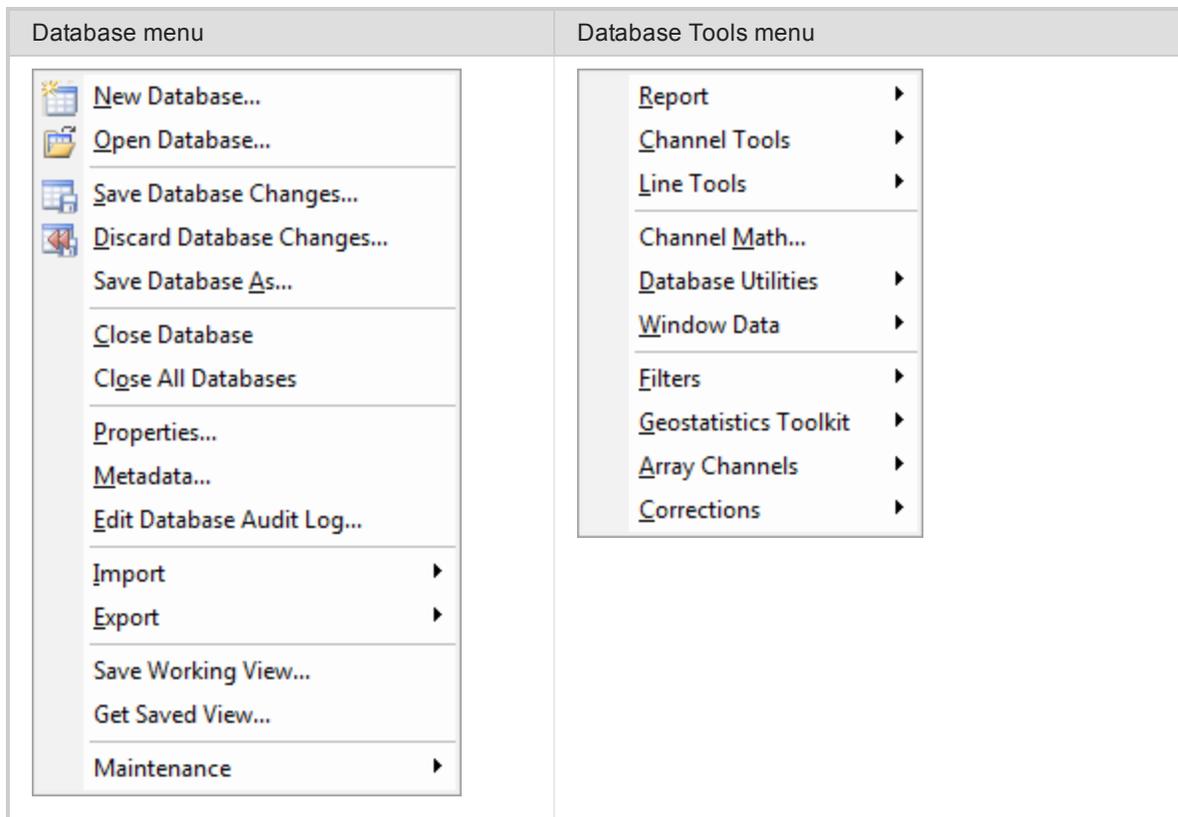
Oasis montaj Supported Data Formats

Oasis montaj provides seamless access to both original spatial data and processed information (grids, images and vector plots).

Spatial Data Import Formats	Processed Data Import Formats
<ul style="list-style-type: none"> ➤ AMIRA format files ➤ ASCII data files, CSV ➤ ASEG GDF files ➤ Blocked binary data files ➤ Database table files (single or all tables) ➤ Excel Spreadsheets ➤ Flat archive data files ➤ Geosoft binary data files ➤ Geosoft XYZ data files ➤ ODBC data files ➤ Picodas PDAS data files ➤ RMS data files ➤ RMS DAARC 500 binary file formats ➤ USGS data files 	<ul style="list-style-type: none"> ➤ ArcView shape files (SHP) ➤ AutoCAD DXF (DXF) ➤ DATAMINE (DM) ➤ Geosoft plot (PLT) ➤ Geosoft map files (MAP) ➤ Grid and image formats ➤ MapInfo TAB files ➤ Maxwell Plate files ➤ Microstation DGN files ➤ Surpac (STR, DTM) ➤ GoCad (VO)with rotation ➤ UBC (MOD, MSH, DEN, SUS) ➤ LAS files (LAS)
Common Grid Formats (GRD) including	Common image formats (IMG) including
<ul style="list-style-type: none"> ➤ DEM formats (GLOBE, ETOPO5, USGS) ➤ EOSAT Fast Format ➤ ER Mapper grid (ERS) ➤ ESRI Binary Raster (FLT) ➤ Landmark ZMAP (DAT) ➤ Geopack (GRD) ➤ Geosoft grid files (GRD) ➤ Geosoft Hypergrid (HGD) ➤ Grid eXchange Format (GXF) ➤ Surfer grid file (GRD) ➤ Texaco Startrax (GRD) ➤ USGS (DDF, DEM) ➤ World Geoscience (H) 	<ul style="list-style-type: none"> ➤ EOSAT MSS (Old 4 Band BIL) ➤ ER Mapper algorithm (ALG) ➤ ER Mapper compressed (ECW) ➤ GeoTIFF Image (TIF) ➤ GIF (GIF) ➤ IMG Image (IMG) ➤ JPEG 2000 (J2K and JP2) ➤ JPEG File Interchange Format (JPG) ➤ Landsat MSS (4 band BSQ) and Landsat TM ➤ PCIDSK Format (GIX) ➤ Portable Network Graphics Format (PNG) ➤ Tagged Image File Format (TIFF) ➤ Targa Image (TGA) ➤ Uncompressed PC Paintbrush (PCX) ➤ Windows Bitmap (BMP)

Database Menus

There are two main database menus, *Database* and *Database Tools*, as shown below:



Drag and Drop Geosoft Database Files

Oasis montaj includes the ability to drag Geosoft database files (*.GDB) from Windows Explorer and drop them into an open project.

The ability to drag Geosoft file formats directly from Windows Explorer and drop them into Oasis montaj for display is also available including grid files (*.GRD) and map files (*.MAP). Users can also drag XYZ files and drop them into an open database file (*.GDB). Items displayed in the Project Explorer window may also be dragged and dropped into the main Oasis montaj workspace.

Creating a Geosoft Database

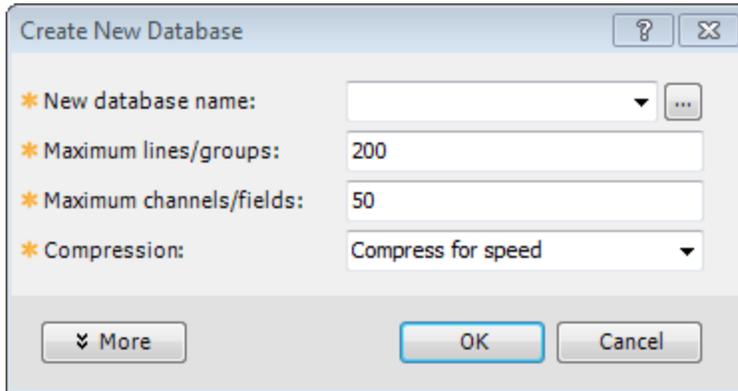
The *Create New Database* dialog enables you to create and define a new database for Oasis montaj. The dialog options enable you to set the initial size of the database and the database page size.

We recommend that you specify a *Maximum Line/Group* and *Maximum Channel/Field* number that is representative of the final estimated project size. This strategy ensures that you have enough space available in your project while not consuming excessive storage space. The size can be changed later via the *Maintenance* menu option from the *Database* menu, should your project expand in scope.

Additional parameters available from the **More** button control the database "page size", which is the smallest unit of memory that is read/written and stored on the database. In almost all cases, the default page size (1024 bytes) should be sufficient since this will support a total database size of up to 4 Terabytes in size and will enable individual channels on a line to hold up to 64 Megabytes of data.

To Create a New Database

1. On the *Database* menu, select **New Database**. The *Create New Database* dialog appears.



2. Specify a *New database name*.
3. Specify the *Maximum lines/groups* and *Maximum channels/fields*. The defaults are 200 and 50.
4. From the **Compression** drop-down list, choose the compression method.



You can choose either **Compress for speed**, **Compress for size** or **No compression**. This depends on which type better suits your needs. For example, you would more than likely compress for speed if you have a lot of hard drive space available. However, if space is limited, you may wish to compress for size.

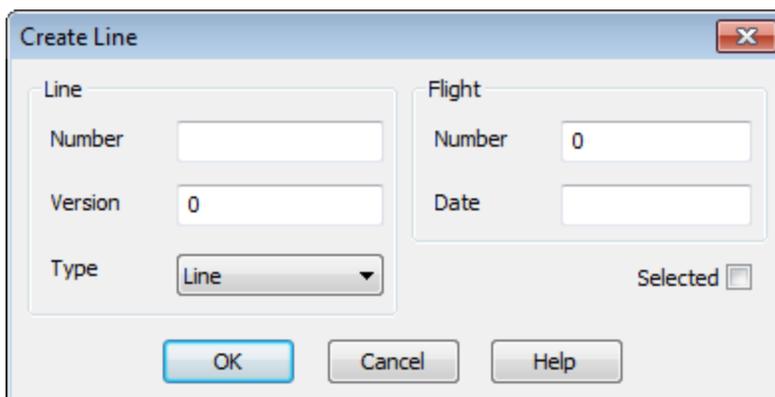
5. Click **More** to specify the way Oasis montaj works with data in the database file.
6. Click **OK**. The system creates a new database with the database name that you specified with a (*.GDB) extension and opens a Spreadsheet window with empty Channel Header Cells and Data cells.

Create a New Line or Group

Creating a new Line or Group is easy with the *Database* pop-up menu.

To Create a New Line or Group

1. From the current database, select the *Line* or *Group* header cell (top left cell).
2. Right-click and select **New Line** or **New Group** from the pop-up menu.
3. The *Create Line* or *Create Group* dialog appears.



4. Enter the parameters as required and click **OK**.

The new Line (or Group) will be created in your current database.

Line Types

- Line
- Base line
- Tie line
- Test line
- Trend line
- Special line
- Random line

Edit Geosoft Database Lines and Groups

Use the *Edit Line* dialog to edit a database line.

💡 To edit multiple lines, select **All Selected Lines**.

To Edit a Line

1. From the current database, select the *Line* header cell.
2. Right-click and select **Edit...** from the pop-up menu. The *Edit Line* dialog appears.

3. Specify the *Line Number* and *Line Version*.
4. From the drop-down list select the *Line Type*.
5. Specify the *Flight Number* and *Flight Date*. By default, it is the import date.

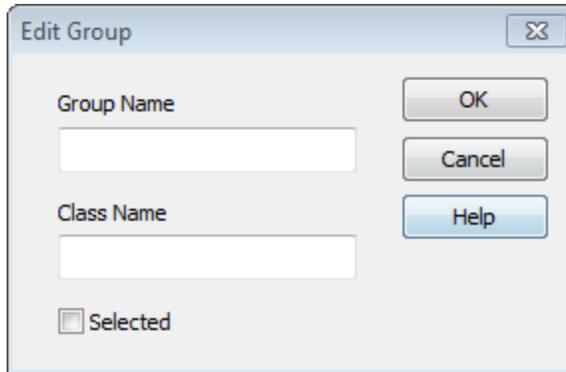
6. Check the *Selected* box to select the line, if desired.
7. Click **OK** when done.

Edit a Group

Use the *Edit Group* dialog to edit a database line.

To Edit a Group

1. From the current database, select the *Group* header cell.
2. Right-click and select **Edit...** from the pop-up menu. The *Edit Group* dialog appears.



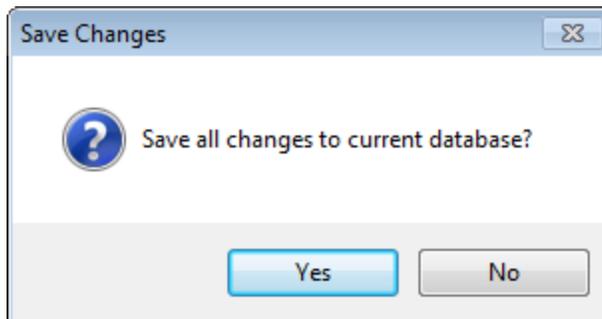
3. Specify the *Group Name* and *Class Name*.
4. Check the **Selected** box to select the group, if desired.
5. When satisfied with group edits, click **OK**.

Save Changes to Databases

Another advantage of the Geosoft database format is that the user can decide whether to discard or save any changes to data in a database without having to close and reopen the database.

To Save Changes to Your Database

1. On the *Database* menu, select **Save Database Changes**. The *Save Changes* dialog appears.



2. Click **Yes** to confirm the save. The system updates your database and the changes are now saved.

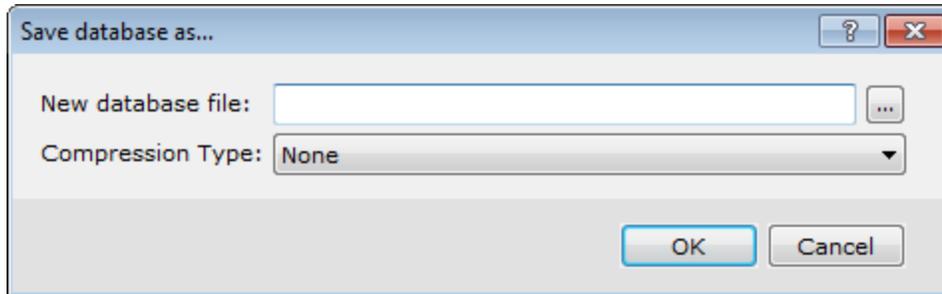


To undo changes, click the **Discard Database Changes** option on the *Database* menu. This will revert your database to the point where it was last saved.

To Save Database As

You may want to save your database with a different name.

1. On the *Database* menu, select **Save Database As...** The *Save database as...* dialog appears.



2. Specify the name of the database (*.GDB file) and then select the Compression Type (None, Speed or Size).
3. Click **OK** to confirm the save. The system saves your database with the new database name.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Coordinate Systems

In Oasis montaj, working with map coordinate systems requires that the map coordinate system information be attached to any database, grid or map. Once you have attached a coordinate system to a set of coordinates, Oasis montaj will deal with any re-projections that may be required to properly display data from different coordinate systems together on your maps.

When dealing with coordinate systems, it is important to be aware that the geographic location (X,Y) of any objects are either an implied or a known coordinate system. If you do nothing to define coordinate systems in your data, Oasis montaj will assume that all locations (X,Y) are in the same coordinate system, which is unknown.

You must define coordinate systems if you want to:

- Annotate maps with longitude, latitude locations
- Convert the location of information (data or grids) from one map coordinate system to another
- Display information on a map that is in a different map coordinate system
- Defined a warped coordinate system to fit data to a desired map coordinate system



If you do not need to do any of these things, you do not have to deal with projections.

Coordinate systems can be defined for any pair of channels in a database, for a grid coordinate system, and for a data view in a map. In most cases, all that is required is to define the coordinate system of the "X" and "Y" channels of an original database. This coordinate system will then be passed on to grids when data from that database is gridded, and maps created from this data will also inherit this coordinate system. Coordinate systems of data channels, grids and map views can also be viewed and modified at any time, although the modification of coordinate systems does require specific and accurate knowledge of the projection information.

Oasis montaj features "smart projections" that reproject data "on-the-fly" as it is displayed. This functionality enables maps to have several views; each containing a different set of projected coordinates. In addition, dynamic links between maps and data are maintained, even if different coordinate systems are used.

Coordinate Systems in Oasis montaj

In Oasis montaj a coordinate system is always attached or applied to something. This can be a coordinate pair on a database, a grid, or a map view. Before you can apply a coordinate system, you will need to know the following information about your data:

The type of projection coordinates that contain the location information for your data.

- **Projected (x,y)** - the coordinates are in a known projected coordinate system, such as UTM, or some other map-based coordinate system.

or

➤ Define X, Y Channels

- **Geographic (long,lat)** - the coordinates are longitudes and latitudes in degrees. Only a datum is required.



When importing data that includes an attached Geographic Coordinate system, note that Geosoft only supports decimal degrees (ddd.dd) or full Latitude-Longitude (ddd.mm.ss.ss).

Depending on the type of data, you may also need:

- The coordinate projection subset to use (e.g. UTM, USA State Plane)
- The units of measurement you want to use (e.g. feet, metres)
- The Datum (or ellipsoid) to use (i.e. WGS 84 / UTM zone 44N) and local datum to use



When you define (or modify) a coordinate system the coordinates of the data (database, grid or map view) remain unchanged. This process only sets the "coordinate system label". However, when you reproject data into another coordinate system the numerical representation of the coordinates will change.

Define X, Y Channels

Once your data has been successfully imported into Oasis montaj, you can set the coordinate system and projection information for your database.

Defining Channels

One of the first steps in setting a coordinate system is defining your X and Y channels. This can be done during the import process, or will be automatically set if your coordinate channels are called X and Y by default.



You cannot create a map until your X and Y channels are defined.

In Oasis montaj we have introduced "current" X and Y (and Z) channels. This feature enables users to select the channels that they would like to use as the current X and Y (and Z) channels (for example, East, North, and Relative Level).

Markers have been added to the channel header to indicate which channels are currently defined to be the "current" X, Y (and Z) channels. The markers are little rectangles on the right side of the header cell, and contain "x", "y" (or "z") in reversed display (like the triangle protection symbol on the left).

When you alter the current coordinate channels, these markers change too. Normally, you will just get "x" and "y", but if you have a "Z" channel it will get the "z". The "z" will be used in Drillhole databases, where DH_EAST = "x", DH_NORTH = "y" and DH_RL = "z".

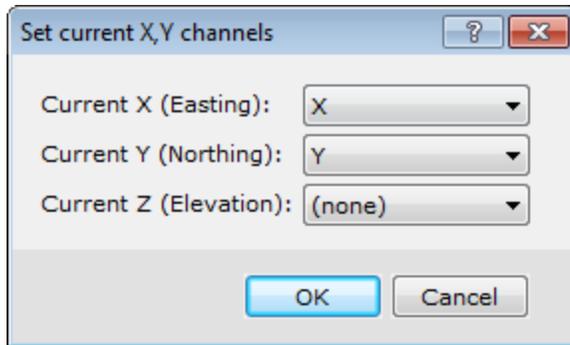


All grids, maps, and databases derived from a database that has coordinate system information attached to it, will inherit that coordinate system information.

To Define X and Y channels

When you first import your data, your X and Y coordinate channels may be called East and North (or any other valid naming system).

1. On the *Coordinates* menu, select **Set Current X, Y, Z, Coordinates**. The *Set current X, Y channels* dialog box appears.



2. Select *Current X* and *Y* channels from the drop-down list.
3. Select the *Current Z* channel, if applicable.
4. Click **OK**. Your database now has set X and Y channels which can be seen by the inverse x and y on the right side of the channel header cell.

Projected (X,Y) Coordinate System

The Oasis montaj *Coordinate System* interface has been designed to make creating, editing or viewing coordinate systems a simple, one-step process. Oasis montaj uses a single dialog to set and modify coordinate system settings. The *Coordinate System* tool also now supports ESRI projection files (PRJ).

The Coordinate System Dialog

Whenever you work with coordinate systems in Oasis montaj, the coordinate system information for the current database, grid, or view is displayed in the *Coordinate System* dialog. When the fields in the dialog are shaded grey, you can only view the information. If the dialog box fields are white, you may edit the values.

The dialog also includes a **Copy from** button that displays the **Copy Coordinate System From** dialog. Use this dialog to copy a coordinate system from another file. The file types supported include, Database, Grid, Geosoft Voxel, Geosoft projection (*.prj), ESRI projection (*.prj), Geosoft warp (*.wrp), Geosoft polygon (*.ply) and GMSYS model.

To define a coordinate system, you must know the type of coordinate system that is used for the data. If the wrong coordinate system type is specified, Oasis montaj may return strange results or take an unusually long time to process the data.



*If the database contains no coordinate system information, the **Unknown** radio button is automatically selected in the *Coordinate System* dialog box.*

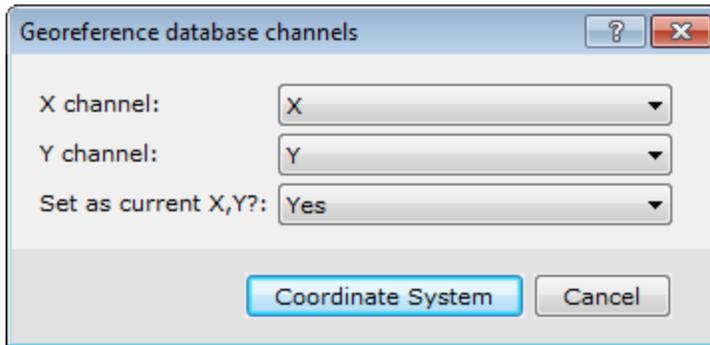
Setting a Projected Coordinate System for a Database

Use the Projected (x,y) coordinate system if your database coordinates are in a known projected coordinate system, such as UTM, or some other map-based coordinate system. A coordinate system can be defined from any pair of channels in a database. These two channels must contain coordinate information. Normally, the X and Y coordinate channels are used.

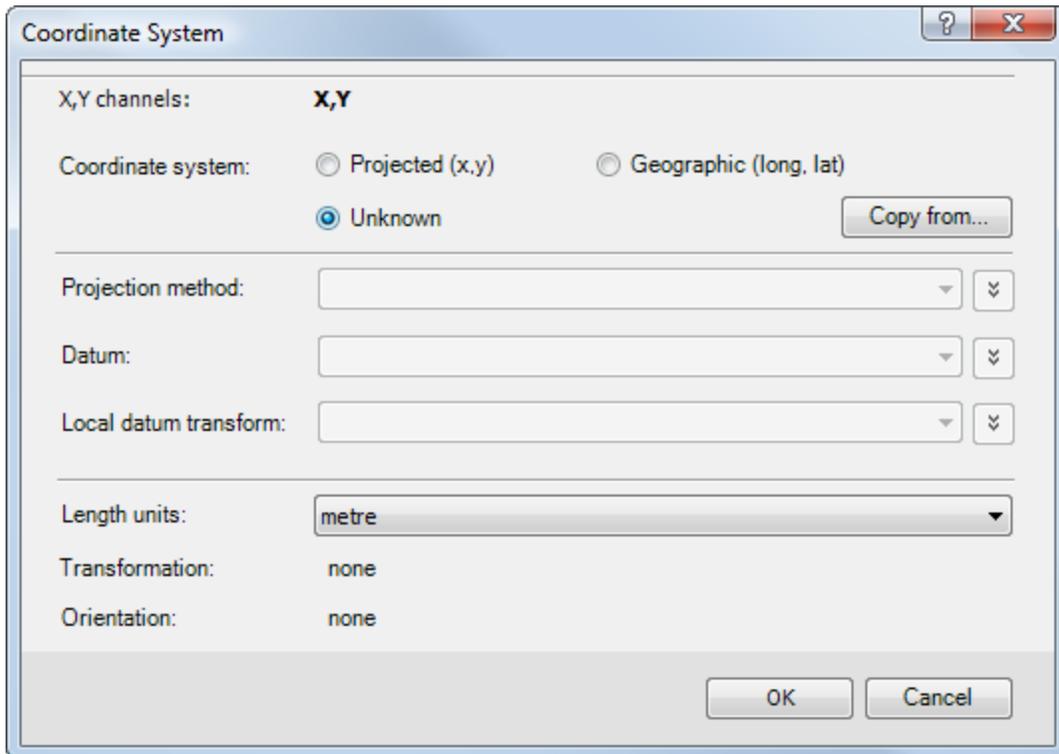
To Set a Projected (X, Y) Coordinate System for Database Channels

This procedure assumes you have an open a database which contains at least two channels with coordinate information (X, Y).

1. On the *Coordinates* menu, click **Coordinate System**. The *Georeference database channels* dialog appears.



2. Select the **X** and **Y** coordinate channels that you will use to set the coordinate system
3. From the **Set as current X,Y?** drop-down list, select **Yes**.
4. Click **Coordinate System**. The *Coordinate System* dialog appears. This dialog displays the current coordinate system information for the database.



5. Select the **Projected (x,y)** radio button. The appropriate dialog parameters are enabled.
6. Verify the **Length units** are correct.
7. Select the **Datum** and **Local datum transform** from the drop-down lists. The Local datum transform drop-down list provides all the local datum transforms, which apply to the selected *Datum*.
8. Select the **Projection method**. Choose one of the available pre-defined projections from the list. The list contains all projections currently defined by EPSG (v4.2). In cases where the coordinate

system can only be viewed, and not modified, this control is replaced with a label containing the name of the local datum transform.



*If the coordinate system method you want to use does not exist, but you know the parameters for it, you can click the **Projection method** drop-down menu, which activates a **New** button to access the New Projection Method dialog. Use this dialog to create a new (custom) coordinate system method.*

9. When all the Coordinate System settings are correct, click **OK**.

Set Projected Coordinate System of a Grid

Normally, when you create a grid from a database that already has location (coordinate) channels with a defined coordinate system, the coordinate system information in the database is automatically applied to the grid. You can modify the current coordinate system or define a new one from the *Grid and Image* menu.

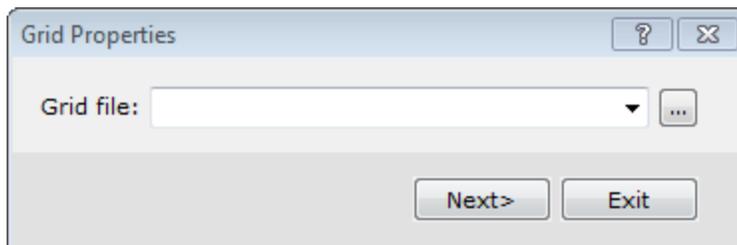


To ensure consistency between your databases and grids it is good practice to define a coordinate system for your database before creating grids from it.

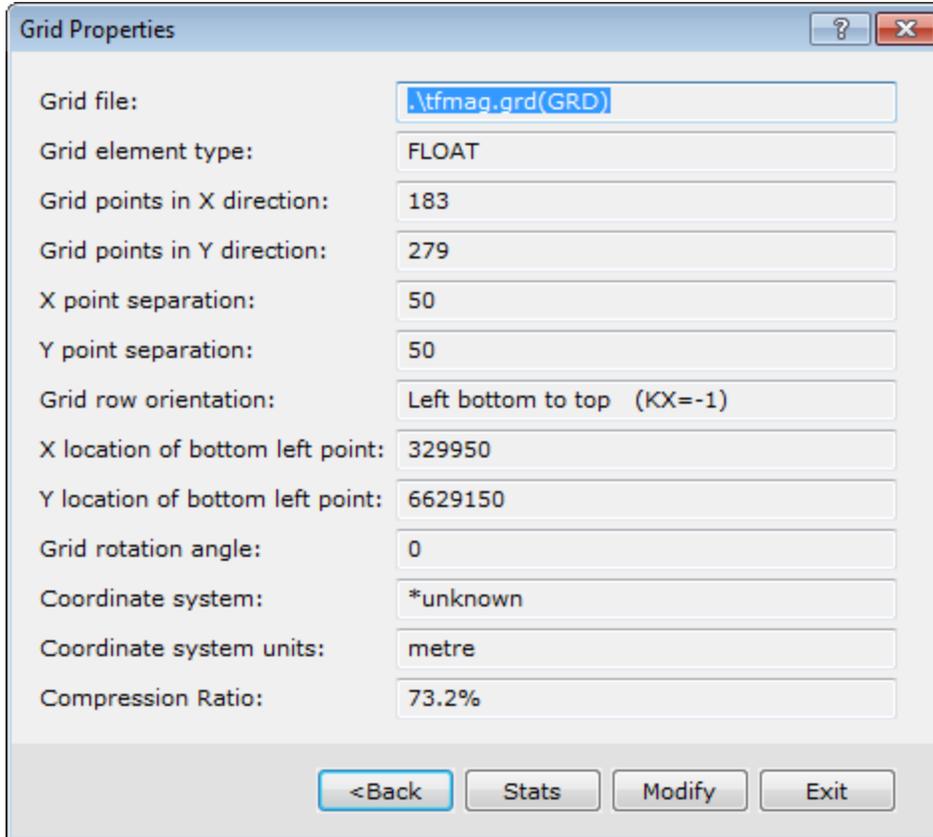
To Modify or Define a Projected Coordinate System of a Grid

When the fields in any dialog box are shaded grey, you can only view the information. If the dialog box fields are white, you may edit the values.

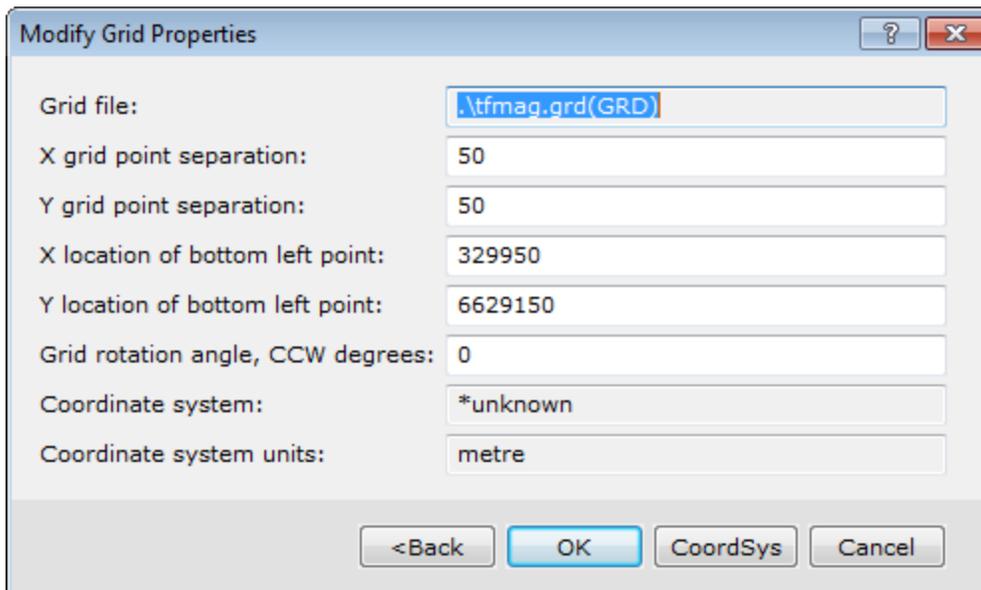
1. On the *Grid and Image* menu, select **Properties**. The *Grid Properties* dialog appears.



2. **Browse** to the grid you want to modify.
3. Click **Next**. The *Grid Properties* dialog appears. This dialog reports all of the basic information about the selected grid.



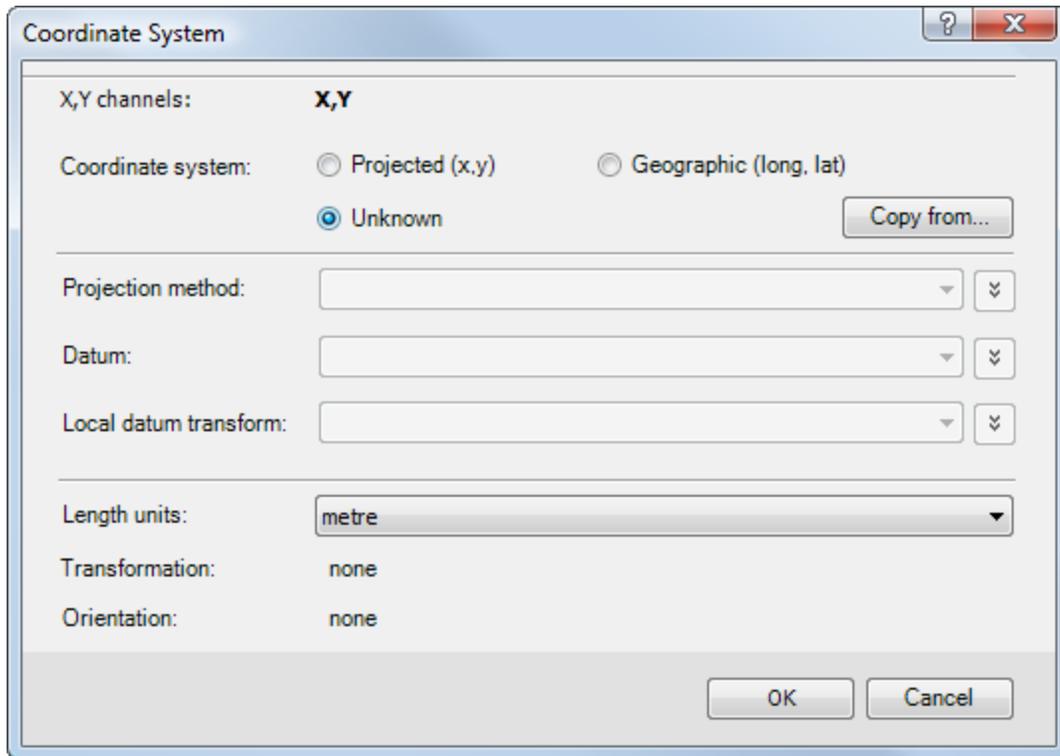
4. Click **Modify** to display the *Modify Grid Properties* dialog. This dialog displays the information that is used to locate the grid in a real coordinate system.



Changing anything in this dialog will have the effect of moving or changing the apparent size of the grid when displayed on a map.

5. Click **CoordSys** to modify (or define) the map coordinate system of the grid file. The *Coordinate*

System dialog appears. This dialog displays the current coordinate system information for the grid.



6. Select the **Projected (x,y)** Coordinate system radio button.
7. Verify the **Length units** are correct.
8. Select the **Datum** and **Local datum transform** from the drop-down lists. The Local datum transform drop-down list provides all the local datum transforms, which apply to the selected *Datum*.
9. Select the **Projection method**. Choose one of the available pre-defined projections from the list. The list contains all projections currently defined by EPSG (v4.2). In cases where the coordinate system can only be viewed, and not modified, this control is replaced with a label containing the name of the local datum transform.



*If the coordinate system method you want to use does not exist, but you know the parameters for it, you can click the **Projection method** drop-down menu, which activates a **New** button to access the New Projection Method dialog. Use this dialog to create a new (custom) coordinate system method.*

10. When all the Coordinate System settings are correct, click **OK**. The *Modify Grid Properties* dialog appears.
11. Click **OK** to apply the changes to the grid file and the *Grid Properties* dialog will be displayed.
12. Click **Exit** to close the dialog.

Create New Map from Projected (X, Y) Coordinates

In Oasis montaj maps have several Views that contain information about the map. Coordinate system information for a map is contained in the Data View. Base map information is contained in the Base View.

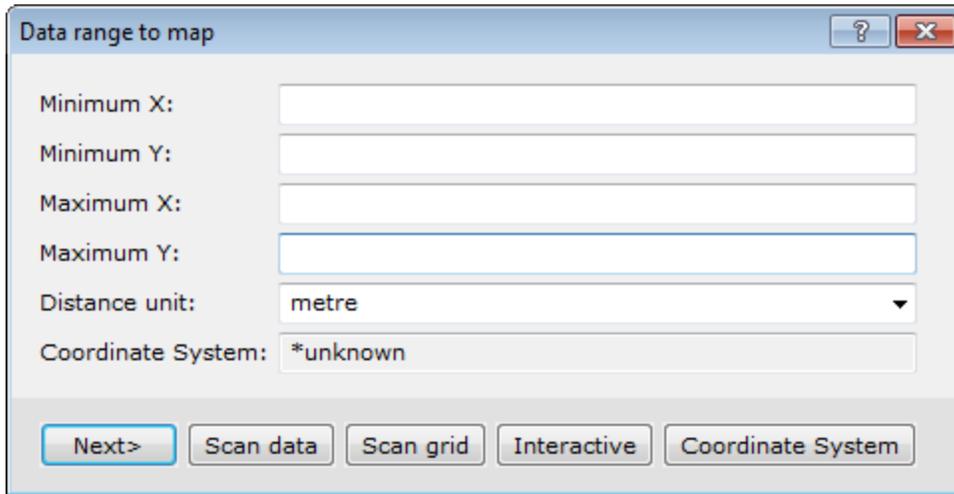
► Projected (X,Y) Coordinate System

When you create a new map, the system either reads the manually entered coordinates or scans a user-specified database or grid file and attaches their coordinate system and coordinate attributes to the specified map view.

Use the *New map from x,y* menu item to create a new blank map to fit the data range specified. The data range can be entered manually, or it can be determined from the range of selected data in a database or from a grid range. The first step is to create a new map using the coordinate system from an existing database or grid.

To Define a Map Coordinate System for a New Map from Projected (X,Y) Data

1. On the *Map Tools* menu, select *New Map* and then select **New Map from X,Y**.
The *Data range to map* dialog appears. This dialog shows the data range (coordinate), units and coordinate system information from the last coordinate system that was used.



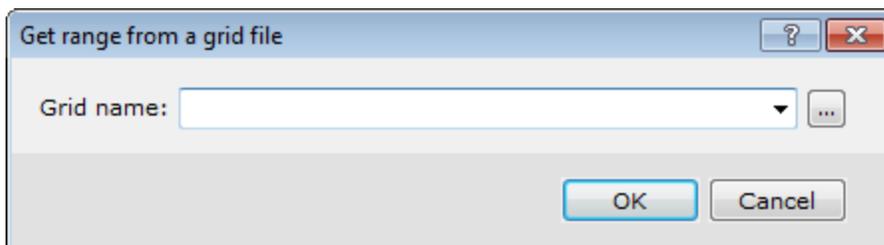
2. To change the coordinate system information you can scan an existing database or grid. Follow the procedures below depending on where the coordinate system information resides.

To Scan a Database for Data Range and Coordinate System Information

1. Click **Scan Data**. The system scans the current database open in your project. The new data range and coordinate system information obtained from the database appears in the *Data Range to Map* dialog box.

To Scan a Grid for Data Range and Coordinate System Information

1. Click **Scan Grid**. The *Get range from a grid file* dialog box appears.



2. **Browse** to the grid file.

3. Click **OK**. The new data range and coordinate system information, obtained from the grid, appears in the *Data Range to Map* dialog box.

To Define the Coordinate System

1. Click **Coordinate System** to view the coordinate system information. The *Coordinate System* dialog box appears. This dialog will display the known coordinate system information as defined in the scanned database or grid file.



When the fields in any dialog box are shaded grey, you can only view the information. If the dialog box fields are white, you may edit the values.

2. If all the coordinate system settings are correct, click **OK** to apply the coordinate system and the *Data range to map* dialog will again be displayed.
3. Click **Next** to continue. The *Create a New Map* dialog box will be displayed.

4. Specify a *Map name* for the new map.



You can also select a **Template** to use for your map layout.

5. To automatically calculate the map scale, click **Scale**. The default scale that will fit the defined data range to the specified template will be displayed. You can modify this value to a more appropriate scale.
6. Click **Finish** when you are done. The system will create a new (blank) map with coordinate system information in the Data View.

Geographic (long, lat) Coordinate System

The Oasis montaj *Coordinate System* interface has been designed to make creating, editing or viewing coordinate systems a simple, one-step process. Oasis montaj uses a single dialog to set and modify coordinate system settings. The *Coordinate System* tool also now supports ESRI projection files (PRJ).

The Coordinate System Dialog

Whenever you work with coordinate systems in Oasis montaj, the coordinate system information for the current database, grid, or view is displayed in the *Coordinate System* dialog. When the fields in the dialog are shaded grey, you can only view the information. If the dialog box fields are white, you may edit the values.

The dialog also includes a **Copy from** button that displays the *Copy Coordinate System From* dialog. Use this dialog to copy a coordinate system from another file. The file types supported include, Database, Grid, Geosoft Voxel, Geosoft projection (*.prj), ESRI projection (*.prj), Geosoft warp (*.wrp), Geosoft polygon (*.ply) and GMSYS model.

To define a coordinate system, you must know the type of coordinate system that is used for the data. If the wrong coordinate system type is specified, Oasis montaj may return strange results or take an unusually long time to process the data.



If the database contains no coordinate system information, the **Unknown** radio button is automatically selected in the *Coordinate System* dialog box.

Setting a Geographic Coordinate System for a Database

Use a *Geographic (long, lat)* coordinate system if your database coordinates are in longitudes and latitudes degrees and only a datum is required. A coordinate system can be defined from any pair of channels in a database. These two channels must contain coordinate information. Normally, the X and Y coordinate channels are used.

Before you begin, use the *Edit Channel* dialog to confirm the *Format* type is Geographic as shown below:



When importing data that includes an attached Geographic Coordinate system Geosoft only supports decimal degrees (ddd.dd) or full Latitude-Longitude (ddd.mm.ss.ss).

To set a Geographic (Long, Lat) Coordinate System for a Database

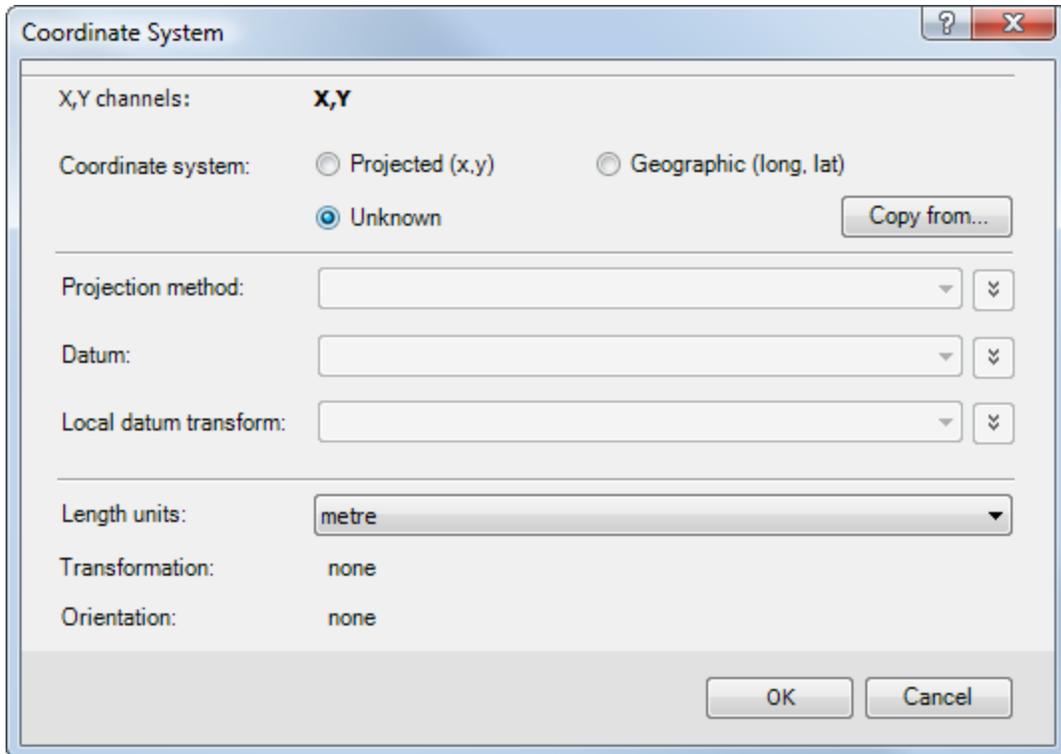
This procedure assumes you have an open a database which contains at least two channels with longitude and latitude information.



When you select a Geographic (long,lat) coordinate system the geographic coordinates will be displayed as a Cartesian system with units of degrees. This means that you will not be able to display latitude and longitude annotations in the base map because the X, Y coordinate system of the map will be represented by the long, lat coordinates.

1. On the *Coordinates* menu, click **Coordinate System**.
The *Georeference database channels* dialog box appears.

2. Select the X and Y coordinate channels that you will use to set the coordinate system and then click the **Coordinate System** button.
The *Coordinate System* dialog box appears.



3. Select the **Geographic (long,lat)** radio button. The appropriate dialog parameters are enabled.

 You will note that the *Length units* have been set to match the units used by the coordinate system and can not be changed.

4. Select the *Datum* and *Local datum transform* from the dropdown lists. The *Local datum transform* drop-down list provides all the local datum transforms, which apply to the selected *Datum*.
5. When all the Coordinate System settings are correct, click **OK** to apply the coordinate system.

 If your *Longitude, Latitude channels* display double asterisks (**), that only indicates that the channels are too narrow to display the data. Widen the channels and the data will be properly displayed.

New Map from Geographic Coordinates

In Oasis montaj maps have several views that contain information about the map. Coordinate system information for a map is contained in a data view. Base map information is contained in the base map view.

When you create a new map, the system either reads the manually entered coordinates or scans a user-specified database or grid file and attaches their coordinate system and coordinate attributes to the specified map view.

 The projected base map created from a latitude-longitude range will be windowed based on lines of latitude and longitude, not X and Y, and the data window will be oriented so that North is up at the centre of the map.

To create a new map from Latitude-Longitude requires a database that has a Projected (x,y) coordinate system.

When using latitude-longitude degrees as the coordinates of your data and/or a geographic coordinate system you need to use *New map from x,y* menu item. The resulting map will have latitude and longitude displayed in a right angled Cartesian coordinate system.



With a Projected (x,y) coordinate system, you can display Latitude and Longitude annotations on the basemap of a new map created from x,y coordinates.

To Define a Map Coordinate System for a New Map from Geographic Coordinates

The first step is to create a new map using the coordinate system from an existing database or grid. After creating a map from Longitude/Latitude the data view of the new map will have a warp attached to it. This is because when we create a Longitude/Latitude map we warp the view so that North is at the top and centred.



If you remove the warp on your map, new elements may not be positioned correctly.

1. On the *Map Tools* menu, select **New Map**.
2. Select **New map from Lat,Long**. The *Data range to map* dialog appears. This dialog shows the data range (coordinates), units, and coordinate system information from the current (selected) database in your project.

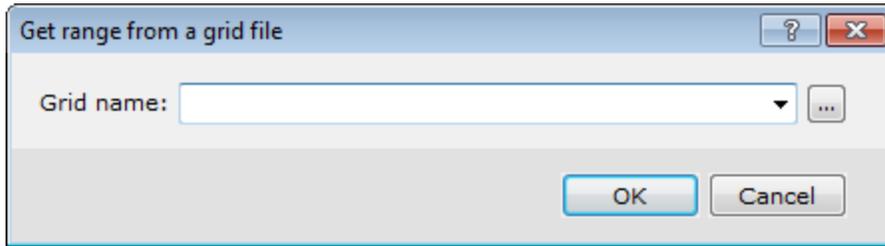
3. To change the coordinate system information, you will need to either scan an existing database or grid for the coordinate system information. Follow the procedures below depending on where the coordinate system information resides.

To Scan a Database for the Data Range and Coordinate System Information

1. Click **Scan Data**. The system scans the current database open in your project. The new data range, and coordinate system information obtained from the database appear in the *Data Range to Map* dialog box.

To scan a grid for the data range and coordinate system information

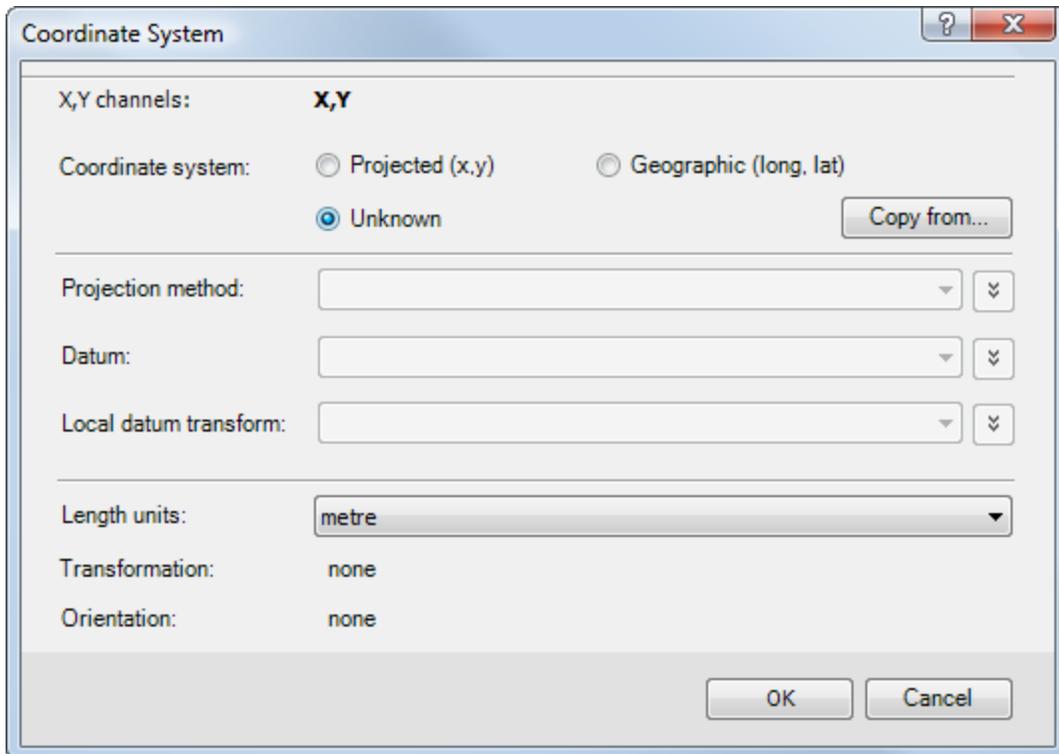
1. Click **Scan Grid**. The *Get range from a grid file* dialog box appears.



2. **Browse** to the grid file.
3. Click **OK**. The new data range, and coordinate system information obtained from the grid appear in the *Data Range to Map* dialog box.

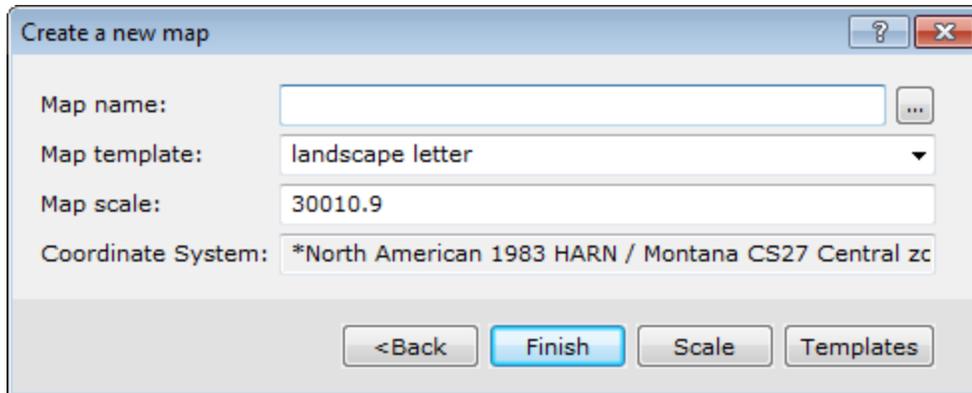
To Define the Coordinate System

1. Click **Coordinate System** to view the coordinate system information. The *Coordinate System* dialog box appears. This dialog will display the known coordinate system information as defined in the scanned database or grid file.



When the fields in any dialog box are shaded grey, you can only view the information. If the dialog box fields are white, you may edit the values.

2. If all the coordinate system settings are correct, click **OK** to apply the coordinate system and the *Data range to map* dialog will again be displayed.
3. Click **Next** to continue. The *Create a New Map* dialog box will be displayed.



- Specify a *Map name* for the new map.



*You can also select **Templates** to use for your map layout.*

- To automatically calculate the map scale, click **Scale**. The default scale that will fit the defined data range to the specified template will be displayed. You can modify this value to a more appropriate scale.
- Click **Finish** when you are done. The system will create a new (blank) map with coordinate system information in the Data View.

Displaying Data with Different Coordinate Systems on a Map

Since grids, images, data and maps can have different coordinate system information, you may want to display one or more types of coordinate systems on a map. The following is a list of data types that you can attach coordinate system information to and place on a map:

- > Grid or image
- > Located data
- > ArcGIS LYR
- > ArcView SHP
- > MapInfo data
- > Vector files such as DXFs, etc

To Display Different Coordinate Systems on a Map

- Identify the coordinate system for each data type (i.e. grids, images, maps, or located data).



Data types with different coordinate systems can be placed on the same map. However, make sure the data cover the same geographical area.

- Data that is imported into a map is re-projected into the map coordinate system when it is displayed. For example, if you place a grid with latitude/longitude coordinate system coordinates on a map defined in UTM coordinates, the system will display the grid on the map in the map's coordinate system while preserving the grid's coordinate system information. In this example, the lat/long meridians on the grid would appear curved, while the UTM meridians would appear straight.

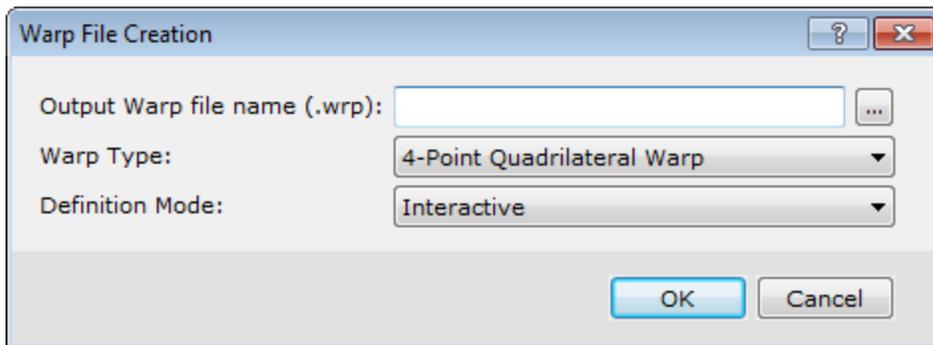
Create and Apply Warps on a Grid

Warping is useful when you have an un-referenced grid file in an unknown coordinate system or an un-referenced image.

- Relocate data, define one control point
- Scale and rotate data, define two points
- Scale in X and Y and rotate, define 3 control points
- Perform a quadrilateral warp, define 4 control points

To Define a Warp

1. On the *Coordinates* menu, select **Georeferencing** and then select **Define a Warp**. The *Warp file Creation* dialog appears.



2. Specify an *Output Warp file name (.wrp)*.
3. Select the **Warp Type** from the drop-down list.

You will be prompted to enter a number of control points, depending on the warp type. If *4-point Quadrilateral Warp* is chosen then the points must be specified in a clockwise or anti-clockwise direction to avoid a mid-point twist of the output data. Each control point will have an old coordinate and a new coordinate. The *Multipoint* warp (more than 4 points) is best for small translations relative to the units of distance on the map.

4. Select the **Definition Mode**.

In *Interactive* mode, both the old and new coordinates are defined using the mouse on the current map. In *Semi-interactive* mode the old coordinate is defined on the current map, but the new coordinate is entered in a dialog, using the old positions as default values. In *Manual* mode, both the old and new positions are entered in a dialog window.



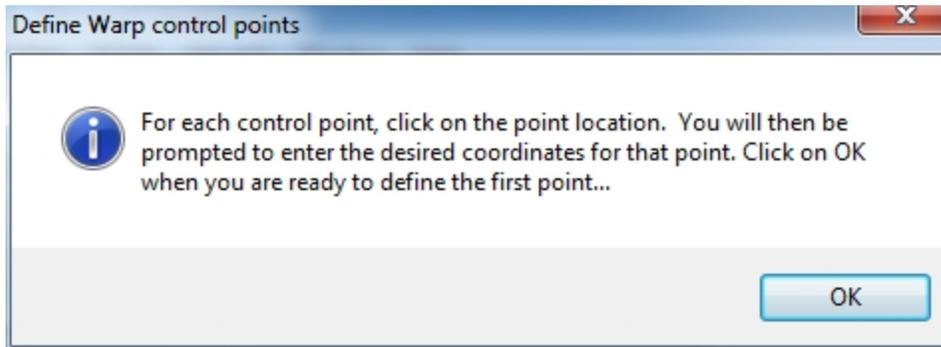
Semi-interactive mode enables you to select the points on the map that you are warping from (for example, the corners of a bitmap) and then manually input the points (in UTM) that you are translating to. If you find that you cannot accurately select these points, try the other modes (interactive and manual) to enter the four points. Remember, that you can use the right mouse menu to zoom, shrink, and pan around the map while you are selecting the points.

5. Click **OK**. The *Map projection* box appears asking you to *Define the output coordinate system?*
6. Click **Yes** if you wish to define the coordinate system of the map or click **No** to continue applying the warp.

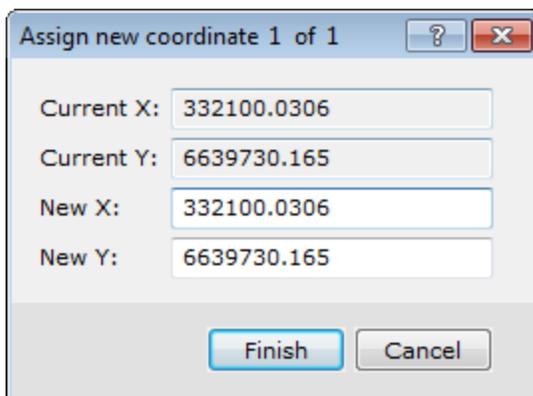


If you clicked **Yes** above, the *Coordinate System* dialog will appear. Refer to the *Application Help* files for details on defining the *Coordinate System*.

7. If you clicked **No** above or have defined the coordinate system, the *Define Warp control points* dialog appears.
8. This dialog tells you to click on the point location for each point in your warp file, you will then be prompted to specify the new “warped” location in the following dialog. Click **OK** to when you are ready continue. The cursor changes to a cross-hair enabling you to locate the first point.



9. **Left-click** to assign the first point. Once you have located the first point, the *Assign new coordinate* dialog appears.



10. Specify the *New X and Y* coordinates. Temporary circles will be placed on the locations that you select with the cross-hair cursor. These circles will be removed once all of the control points have been set and the warp control file has been created.

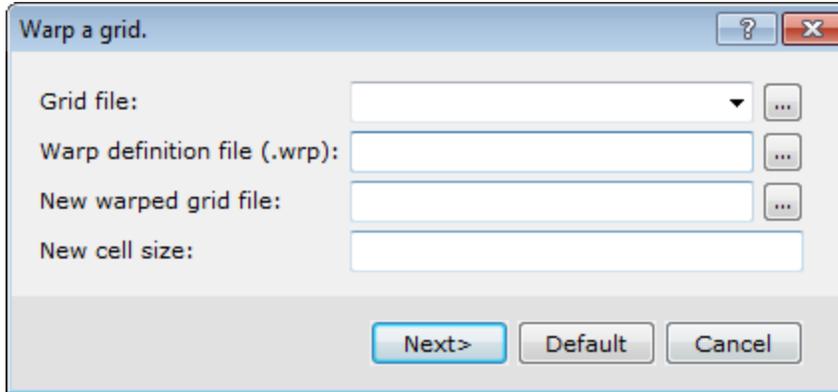


If you have multiple coordinates to define, click **Next** to specify the next coordinate point location. Continue this until you have specified all coordinates.

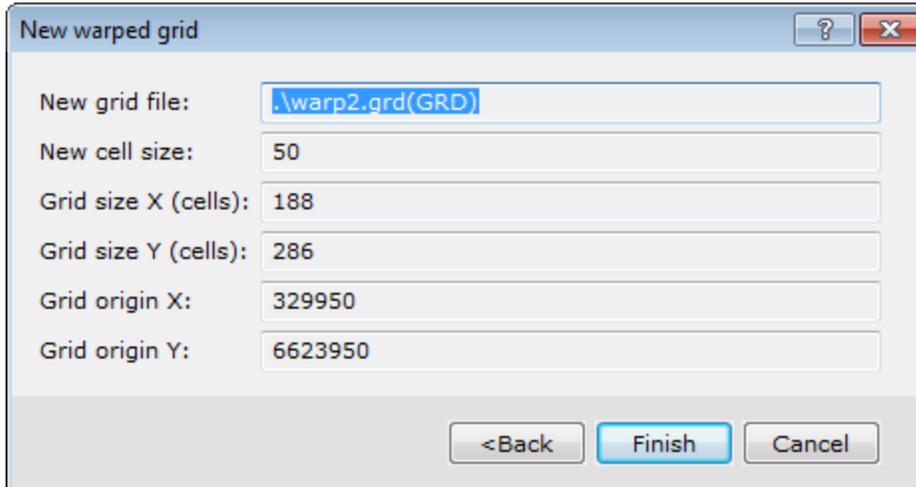
11. Click **Finish**. The warp is created.

To Apply A Warp to a Grid

1. On the *Coordinates* menu, select **Georeferencing** and then select **Warp a Grid**. The *Warp a grid* dialog appears.



2. From the drop-down list, select the **Grid file** to warp.
3. *Browse* to the *Warp definition file (.wrp)* from your project directory.
4. Specify the *New warped grid file* name and a *New cell size* (optional), if required.
5. Click **Next** to continue. The *New warped grid* dialog appears.



6. When all parameters are correct, click **Finish** to warp the specified grid file.

Warping Image Files

If you want to warp an image file (i.e. tiff, bmp) and the file is large, you may only want to attach the warp to the file. "Attaching" the warp is suitable if you have a very large file that is stored on a CD, for instance. The warp process is repeated every time you display the file to a projected map but the original file remains unchanged.

When you warp an image, the warped output image **must** be a Geosoft COLOUR grid (*.grd) *File type* in order to keep the colours of the original image.

If the image of the new warped grid appears twisted or is an incorrect size, check the warp points are in a circular order and the points are correctly located within the defined map area.

Converting Elevations

The *Geoid height* menu option enables you to convert GPS elevations to elevations relative to the geoid. Geoid height (GEOID GX) will create a channel of the geoid height relative to the GRS 80 spheroid for a given pair of coordinate channels. This GX requires a grid of the geoid model, which is the elevation of the geoid in metres relative to the GRS 80 spheroid, which is found in the *Oasis montaj/etc* directory. For more information, please refer to the GEOID GX help topic.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Importing and Exporting Databases in Oasis montaj

Oasis montaj provides a variety of importing and exporting capabilities. This How-To Guide covers the basics of importing and exporting common file types.



*For more detailed information on specific file formats, please refer to the Application Help files or click the **Help** button found on most dialog screens.*

Oasis montaj offers you the ability to:

- Read directly from common databases (Access, FoxPro, Dbase, Paradox and ODBC)
- Import ASCII data interactively via Excel-style import wizard
- Export your data as Microsoft Access database format (*.mdb)
- Export ASCII data

The following menus show you the importing and exporting options available to you in the system.

Import menu	Export menu
<ul style="list-style-type: none">Ascii...Database Table ▶Excel Spreadsheet ▶Geosoft XYZ...Geosoft Database Merge...Geosoft Binary...Flat Archive...Blocked Binary...ODBC...MapInfo TAB or MIF File(s)...ArcView Shape File(s)...ASEG-GDF ▶RMS...RMS DAARC500 binary dataUBC Mag3D Model(s)...Picodas PDAS ▶USGS ▶AMIRA ▶Run SEG-Y Reader...	<ul style="list-style-type: none">CSV etc...Geosoft XYZ...Geosoft GBN...Subset GDB...Microsoft Access...ASEG-GDFShape File(s)Mark...

Drag and Drop Geosoft Database Files

Oasis montaj includes the ability to drag Geosoft database files (*.GDB) from Windows Explorer and drop them into an open project.

The ability to drag Geosoft file formats directly from Windows Explorer and drop them into Oasis montaj for display is also available including grid files (*.GRD) and map files (*.MAP). Users can also drag XYZ files and drop them into an open database file (*.GDB). Items displayed in the Project Explorer window may also be dragged and dropped into the main Oasis montaj workspace.

Importing ASCII Files

Oasis montaj provides a variety of options for easily importing ASCII data using the Geosoft *Import Wizard* technology.



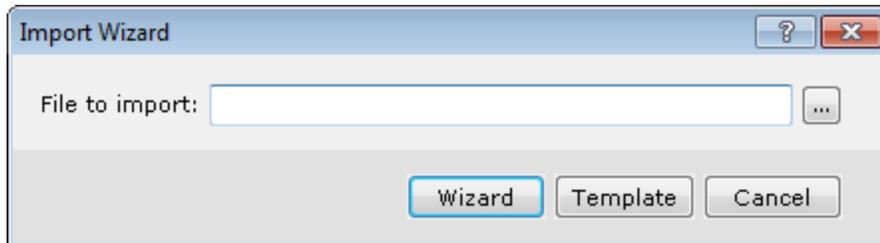
*For additional information on importing files, please refer to the Application Help files. You can also click the **Help** button on the dialog screen.*

The Import Wizard

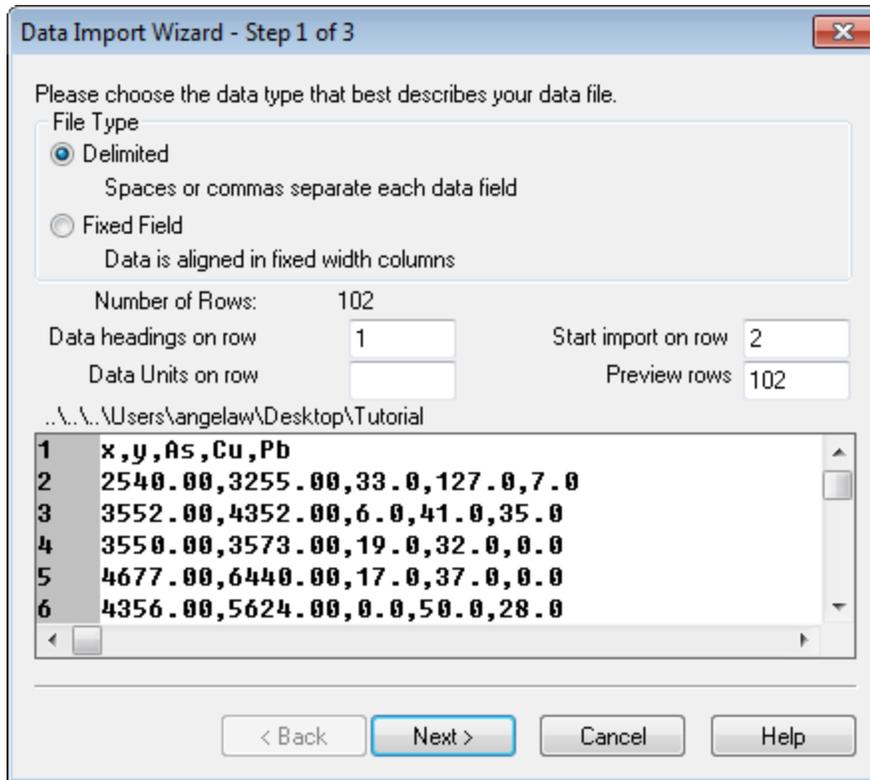
The *Import Wizard* enables you to easily import data from any ASCII spreadsheet or data file. The *Data Import Wizard* supports both Delimited and Fixed Field ASCII files. The *Data Import Wizard* also imports Microsoft Excel Comma Separated Value (CSV), Comma Delimited, White Space Delimited and Tab Delimited data files. The window at the bottom of the dialog box shows the file that is being imported.

To Import ASCII Files

1. On the *Database* menu, select *Import* and then select **Ascii**.
The *Import Wizard* dialog appears..



2. Browse to the *File to import*.
3. Click **Wizard** to begin the ASCII import wizard.
4. The system scans the file and displays the first of three *Data Import Wizard* dialog boxes.



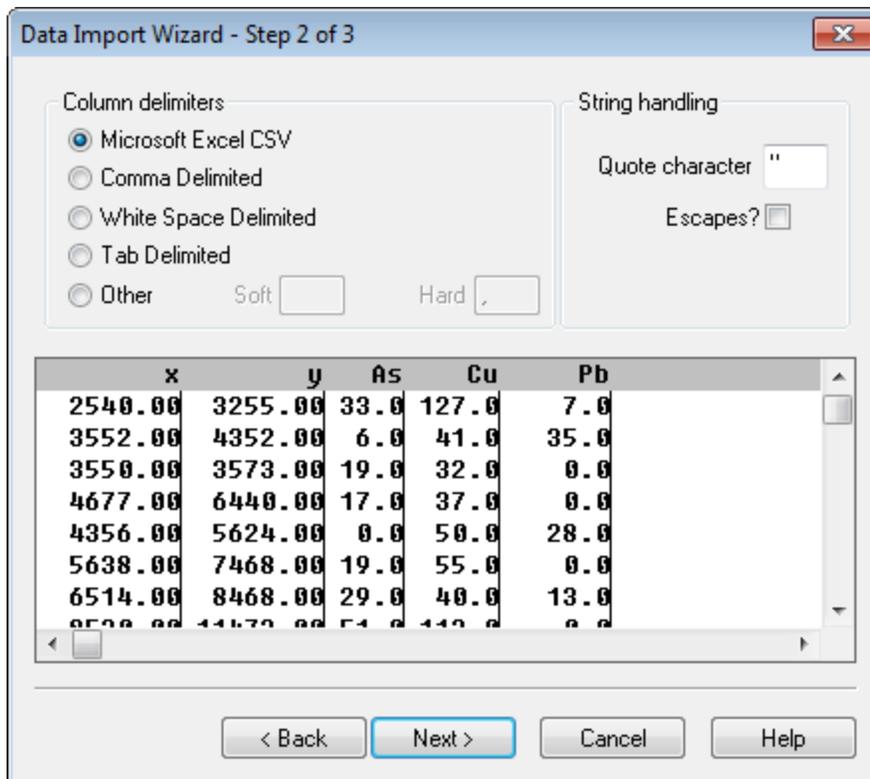
The wizard displays a preview of the data file and automatically determines the *File Type* containing the data (*Delimited* or *Fixed Field*).

- The four fields in the middle of the dialog box specify which line in the file contains the data headings (i.e. channel names), data units (“m” or “ft” ect.), which line to begin importing data on and the number of lines to display in the preview rows.

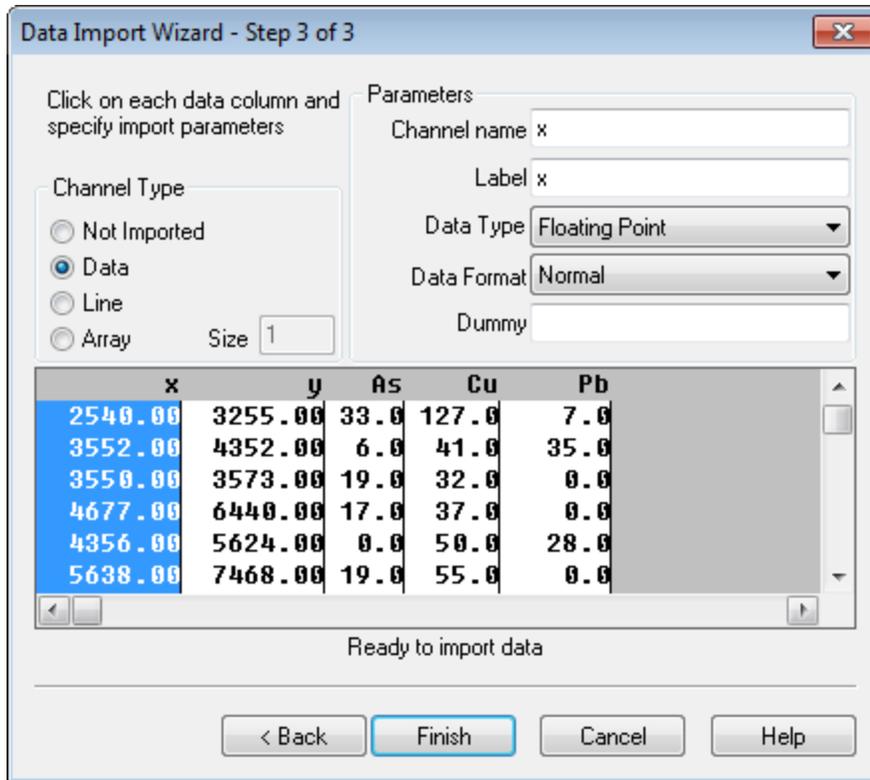


If your import data has several columns of data, you can use the horizontal scroll bar at the bottom of the display window to view the columns.

- Click **Next** to continue. The second dialog box in the *Data Import Wizard* appears.



7. Specify the *Column delimiters* for the type of character used to separate the column text. The system displays the data in columns by drawing lines in the preview window indicating the way in which it is preparing to import your data.
8. Click **Next** to continue. The system displays the third dialog box in the *Data Import Wizard*.



9. The wizard has scanned your data and determined the *Channel Type* of the data with which you are working.



It is always good practice to review your data to ensure that the wizard has selected the correct columns.

10. The *Parameters* area in the dialog box shows the name, label, type and format of data of the column highlighted in the preview window. Make any adjustments necessary.
11. Click **Finish** when you have confirmed the data import format is correct. The system displays the *Save the template?* dialog.
12. Click **Yes** or **No** to save the import template. An import template enables you to apply the settings chosen in the *Data Import Wizard* to future imports of the same type.



*If you select **Yes**, you will be prompted to provide a name and target directory for the template.*

13. The system displays the *Import* dialog confirming you want to import the data into the current database. Click **Yes** to import the data into the open database. The data is imported into the selected database. Click **No**, to create a new database. The *Create New Database* dialog opens.

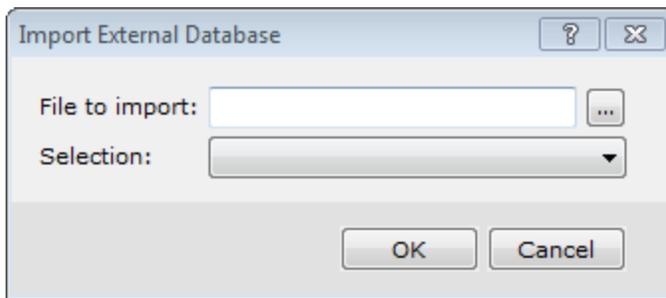
Importing Databases

Oasis montaj enables you to import a database table into a previously created database or to create a new database as part of the import process. The import tool assumes that the database is a Microsoft Access database (*.mdb). If it is not, the system prompts you for a database type (any of the following):

dBase III	FoxPro 2.0	Paradox 3.x
dBase IV	FoxPro 2.5	Paradox 4.x
dBase 5	FoxPro 2.6	Paradox 5.x

To Import a Single Table into a Database

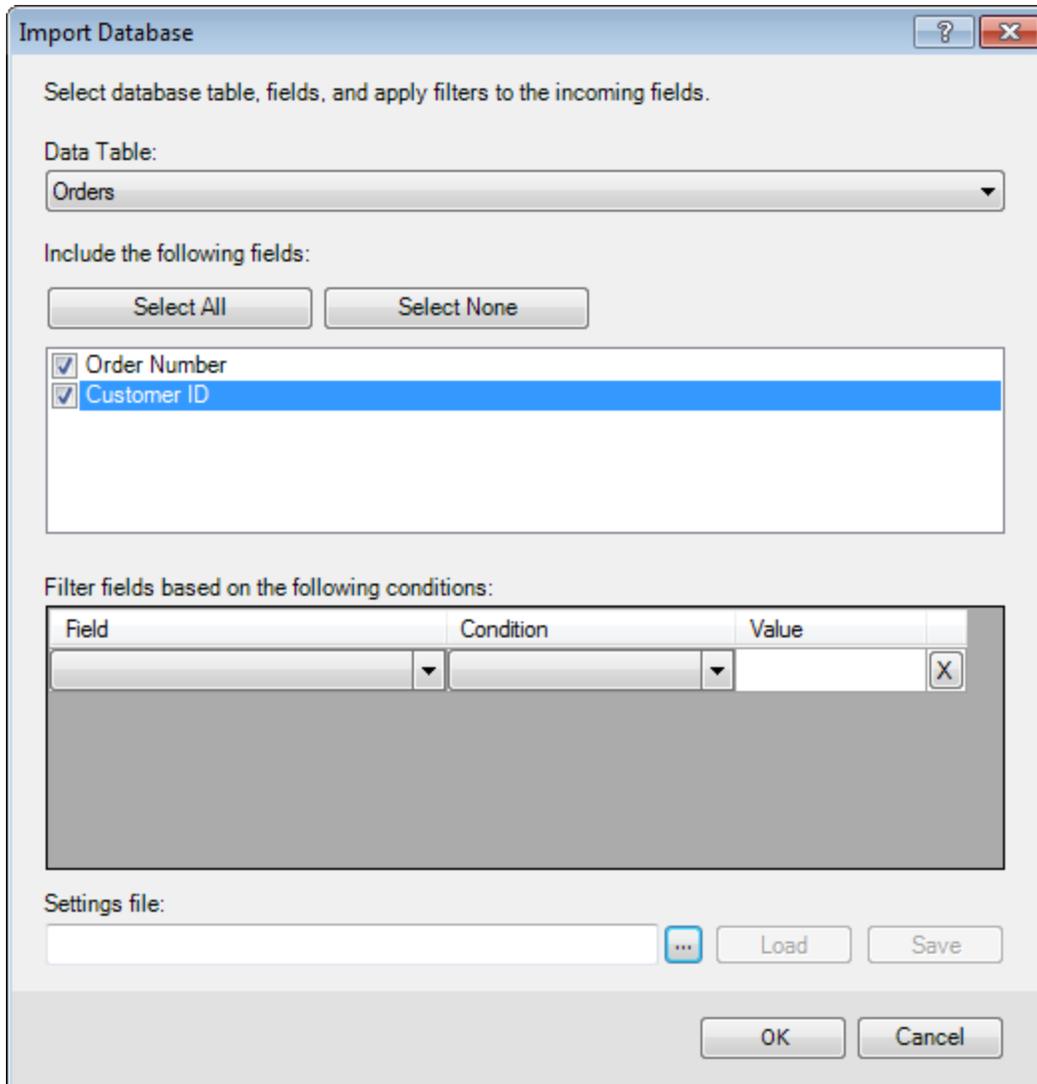
1. Open a previously created database in your current project.
2. On the *Database* menu, select *Import*, then select *Database Table* and then select **Single Table**. The *Import Database* dialog appears, confirming you want to import the data into the current database.
3. Click **Yes** to import into the open database. The *Import External Database* dialog appears.



 If you click **No**, the *Create New Database* dialog will open.

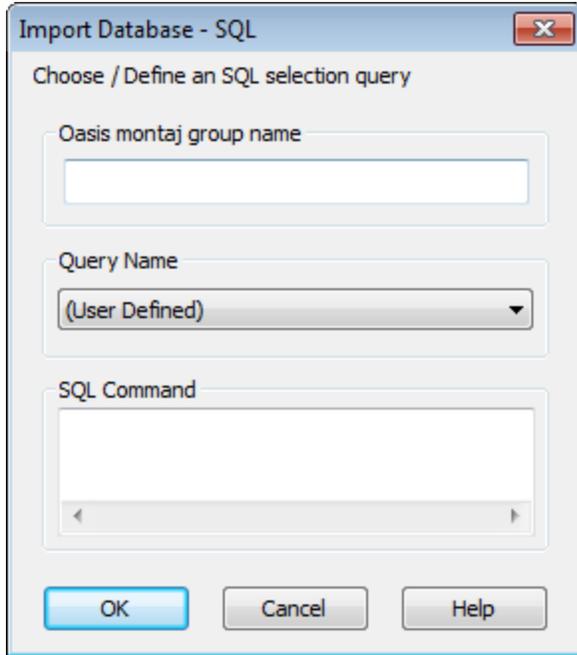
4. Browse to the *File to import*.
5. From the *Selection* dropdown list, choose *Selected Table & Fields* or *SQL Selection*.
6. Click **OK**. The *Import Database* dialog appears.
Depending on the selection above, you will see either:

Table & Field Option



or

SQL Option



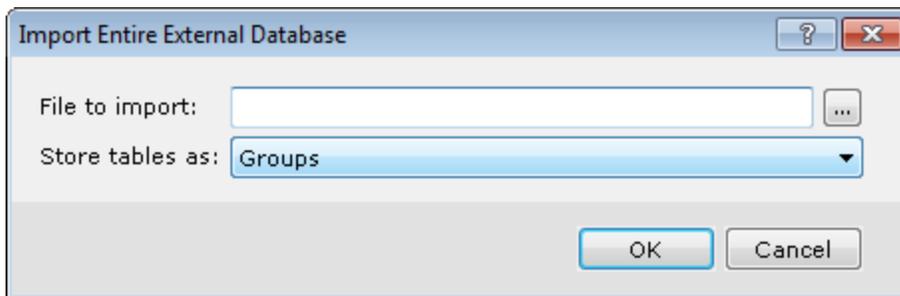
8. Complete the *Import Database* dialog for the import type.
9. Click **OK**. The data is imported into the selected database.

Importing All Tables into a Database

Use the *Import Entire External Database* dialog to import all tables in an external database into a database as lines.

To Import All Tables into a Database

1. Open a previously created database in your current project.
2. On the *Database* menu, select *Import*, then select *Database Table* and then select **All Tables**. The *Import Database* dialog appears, confirming you want to import the data into the current database.
3. Click **Yes** to import into the open database. The *Import Entire External Database* dialog appears.



💡 If you click **No**, the *Create New Database* dialog will open.

4. Browse to the *File to import*.
5. Select to *Store tables as* (*Lines* or *Groups*).

- › *Line* storage is most efficient when tables share the same channels.
 - › *Group* storage is useful for storing tables with different channels.
6. Click **OK**. The data is imported into the selected database.

Exporting Geosoft Databases

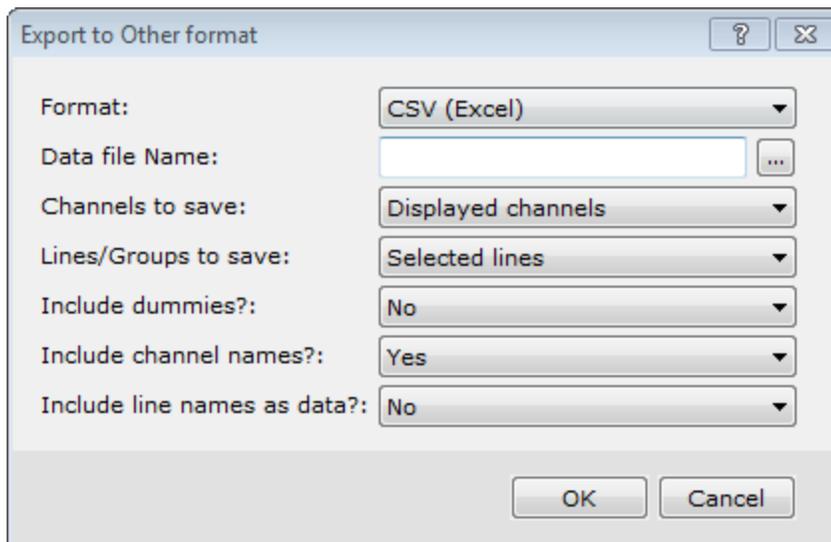
Oasis montaj enables you to export Geosoft databases to Microsoft Excel, Microsoft Access, Geosoft Files and other common data file types.

Export to CSV or Other File Formats

An important capability in Oasis montaj is the ability to export Geosoft databases to a CSV file or other useful formats. This is timesaving feature which enables you to quickly export your processed data from Oasis montaj and perform specialized analysis or processing in another program.

To Export to CSV or Other Formats

1. In your project, open the database to export .
2. On the *Database* menu, select *Export*, then select **CSV etc....**
The *Export to Other format* dialog appears.



3. Select the export *Format* type:
 - › CSV (Excel)
 - › ODDF (USGS)
 - › POST PC (USGS)
 - › POST UNIX (USGS)
4. Specify a *Data file Name*.
5. Select to save *Displayed channels*, *All database channels* or *Select channels from list*.
6. Select to save *Displayed line*, *Selected lines* or *All lines*.
 - › *Displayed Line* - Works for a single group or line, and only the displayed group or line is output.
 - › *Selected Lines* - If lines and groups are selected, only lines will be output. If there are only groups selected, then all groups with the same group class as the first group will be exported.
 - › *All Lines* - If both lines and groups are present, only lines will be output. If there are only groups present, then all groups with the same group class as the first group will be exported.
7. Select whether to *Include dummies*.

Exporting to Microsoft Access Database Format

8. If you wish to *Include channel names*, select *Yes*.
9. Select whether to *Include line names as data*. If *Yes*, the line names will be included as the first "column" of data.
10. Click **OK**. The database is exported in the format selected.

Exporting to Geosoft File Formats

Oasis montaj enables you to export your database to several Geosoft file types. This is a timesaving feature which enables you to quickly export your processed data from Oasis montaj and perform specialized analysis or processing in another program.

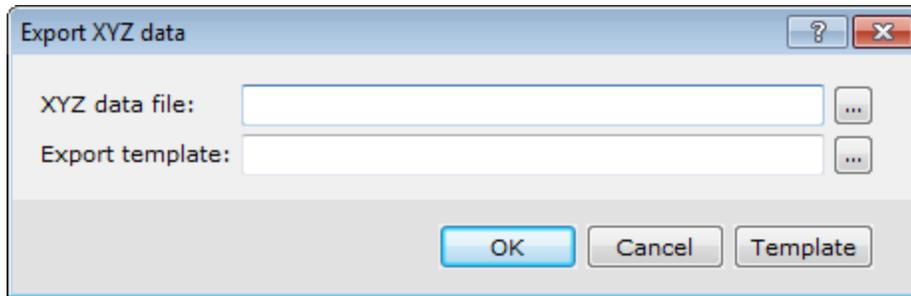


*For additional information on exporting to Geosoft file formats, please refer to the Application Help files. You can also click the **Help** button on the dialog.*

To Export to an XYZ File

Here we show the *Export XYZ data* dialog as an example.

1. In your project, open the database to export .
2. On the *Database* menu, select **Export**.
3. Select the target output type, for example Geosoft XYZ. The specified *Export* dialog appears.



4. Follow the dialog prompts to export the database to a file.



*To apply a template to the export, click the **Template** button.*

5. Click **OK** to export the database to the specified format.

Exporting to Microsoft Access Database Format



Before working with the database export capabilities, please be aware that Geosoft currently supports table export only to Microsoft Access databases.

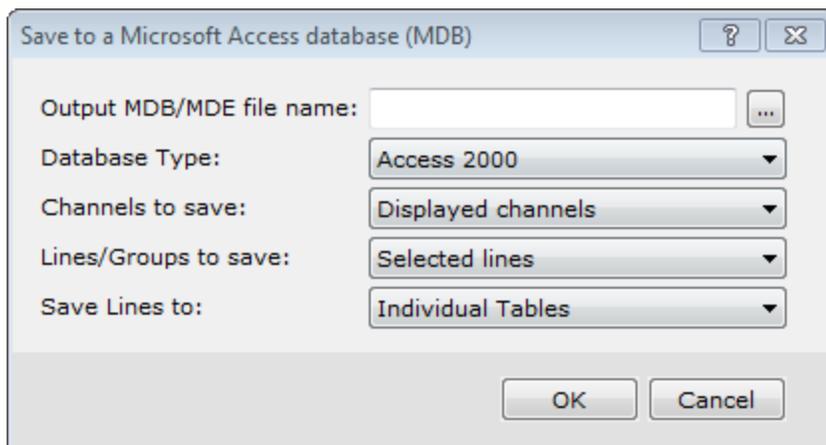
Typically this should not hinder you in practice since Access is widely supported by other database applications. However, we apologize for any inconvenience this limitation may cause. If you have any questions or concerns, please do not hesitate to contact any of our Technical Support groups around the world.

Microsoft Access Export Options	
Individual tables	Each line (or group in the case of drillhole data) has its own table in the output database.
Single table	All lines are stored in one table in the output database.

As a rule of thumb, if you are working with multiple lines, we recommend that you export lines as *Individual tables* so that you can keep track of them in the future. If you are working with a single line (i.e. geochemical data), it is more likely that you will export the line as a *Single table*.

To Export to a Microsoft Access Database

1. On the *Database* menu, select *Export* and then select **Microsoft Access**. The *Save to a Microsoft Access database (MDB)* dialog appears..



2. Specify an *Output MDB/MDE file name*.
3. Select the *Database Type* (*Access 2000* or *Access 97*) from the dropdown list.
4. Select to save *Displayed channels* or *All database channels*.
5. Select which *Lines/Groups to save* (*Displayed line*, *Selected lines* or *All lines*) from the dropdown list.



If you choose Selected lines, you can make sure the lines you wish to export are selected according to your specifications. To do this, select lines via the Line Selection Tool, which is accessed from the line header cell pop-up menu.

6. Select to export the database to *Individual Tables* or a *Single Table*.
7. Click **OK**. The system displays a progress dialog box as it creates the new Access database.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Working with Spreadsheets in Oasis montaj

This How-To Guide walks you through displaying, editing and processing Geosoft database data in the Spreadsheet view.

Spreadsheets

When you create or open a database, you see a spreadsheet. The Spreadsheet view is your “window” to the Geosoft database and it also provides you with flexibility in setting up your working environment. All data is stored securely in the underlying database (*.GDB) — you simply decide which data you want to display in the spreadsheet and keep all other data in the background, hidden from view.

The spreadsheet is organized in rows, channels (columns) and lines (also called groups in other applications such as drillhole plotting). Rows and columns work similar to standard spreadsheets in that you can edit and delete them as needed. The system also enables multiple “worksheets” in your database. Depending on the type of data you are working with (line, random line or drillhole), the worksheets have different names. To see what type of name is used, look at the Line Header Cell located at the top left corner of the spreadsheet. An “L” for example, indicates that you are working with line data. If your data has not been collected on a line basis, such as Geochemistry data, you can store all data in a single line.

General spreadsheet capabilities include:

- The ability to display data from your database through a spreadsheet window
- Handling of any sample interval
- Full editing and mathematical processing capabilities
- Process selected samples, selected channels and selected lines or groups

How the Spreadsheet Displays Project Data

The spreadsheet does not display your actual data, but rather a view of the data. This enables you to work with data in the spreadsheet without actually making changes to the data until you decide to save the database. After you save the database, your changes are permanent.

Depending on your type of project, the spreadsheet will display your data as either values or arrays.

- For surveys where a single value is recorded at each station (for example, magnetic surveys) each data cell will contain a single value.
- For surveys where multiple readings were recorded at each station (for example, an Induced Polarisation or electromagnetic survey) each data cell will contain multiple values. This is what we refer to as an Array database.

Line Number	Column	Channel Header Cell	Protected Channel	
EH61cart	X	Y	Fit X	Fit Y
0.0	705120	3913839	705120	3913839
1.0	705120	3913826	705120	3913826
2.0	705121	3913821	705122	3913822
3.0	705122	3913822	705122	3913822
4.0	705122	3913827	705122	3913827
5.0	705122	3913835	705122	3913835
6.0	705123	3913828	705123	3913828
7.0	705124	3913824	705124	3913824
8.0	705127	3913838	705127	3913838
9.0	705128	3913828	705128	3913828
10.0	705129	3913835	705129	3913835
11.0	705131	3913819	705131	3913819
12.0	705133	3913840	705133	3913840
13.0	705137	3913828	705137	3913828
14.0	705139	3913813	705139	3913813
15.0	705140	3913819	705140	3913819
16.0	705141	3913840	705141	3913840
17.0	705142	3913829	705142	3913829

Displaying Data in the Spreadsheet

The system automatically displays all of your data in the Spreadsheet window.



When the system creates a database, it automatically creates line 0 (L0:0). During the data import process, if the data that you import does not contain a line 0, and therefore there is no data in this line, the line is automatically deleted. If the line contains data it is retained.

Channels

The cells immediately right of the ID Cell are called *Channel Header Cells*. These cells are used to identify a unique type of data contained in a channel (column) of the spreadsheet. The data for a channel appears below the channel header.

The channel headers provide a visual indication of the current status of the displayed channel. If the top left corner of the header cell is marked by a black triangle, the channel is read-only, and it may not be modified. A red line at the bottom of the header cell indicates that the channel has been re-sampled in order to match the sampling rate of data from one of the other channels in the database.

Current X, Y and Z Channels

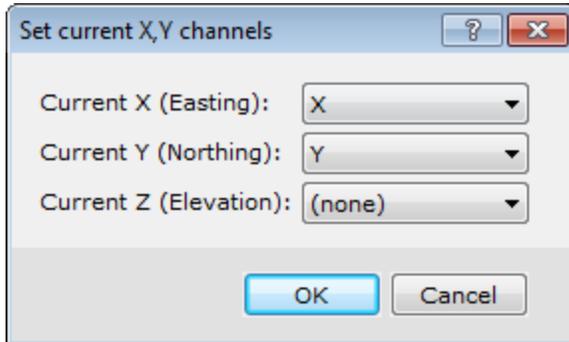
Current X, Y (and Z) channels are used to define which coordinates will be used for creating maps. For example, you may have two sets of coordinates in your database (e.g. Geographic and projected, or local grid); in this case you need to specify which to use when creating maps. Blue x, y (and z) markers in the channel header cells indicate which channels are the current X, Y (and Z) channels.



Typically, you will use only X and Y, but you can also assign a Z. The Z is used in Drillhole databases, where $DH_EAST = X$, $DH_NORTH = Y$ and $DH_RL = Z$.

To Set Current X and Y Channels

1. On the *Coordinates* menu, select **Set Current X,Y, Z Coordinates**.
The *Set current X, Y channels* dialog appears.



2. Select the names of the new coordinate channels.
3. Click **OK**. This makes these channels the new current X, Y (and Z) channels. The corresponding "x", "y" or "z" appears highlighted in blue on the right of the header cell. When you change the current coordinate channels, these markers change.



You can also use this menu item to review the current settings for the X, Y, and Z channels.

Show and Hide Channels

When you import data, all of your data is automatically displayed. You may want to hide certain channels from display in the spreadsheet window.

To Hide a Channel (Column)

When a column is hidden, the database still contains all the data.

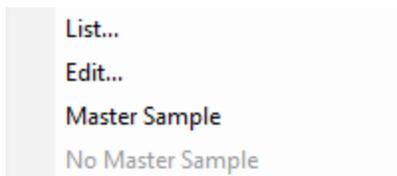
1. Click once on the channel header cell to select it.
2. Press the **Space bar**. The channel is hidden from the Spreadsheet view.



You can also right-click in the channel header cell and select **Hide Column** from the popup menu.

To Display a Hidden Channel

1. Move your cursor to the top of the first empty channel header cell.
2. Right-click and select **List** from the popup menu.



A box appears beneath the empty channel header listing all the available channels that are not currently being displayed in the Spreadsheet window.

3. Select the channel and click **OK** to display. The data channel appears.



If you know the name of the data channel already, you can simply position the cursor on a specific Channel Header Cell, type the name, and press **Enter**.



If you see *******, this indicates that the data are too wide for the spreadsheet column. To change the width of a column, place the cursor on the dividing line between the column headers. The mouse becomes a double arrow. Click and drag the line to the right to increase the column width. Release the mouse button when done.

Protect and Unprotect Channels in a Spreadsheet

A black triangle in the upper left-hand corner of a channel indicates the channel is protected (i.e. read-only) and cannot be edited by the user until the protection is turned off. Original imported data are automatically stored as protected data.

To Remove Channel Protection

1. Select (highlight) the channel header cell you want to unprotect.
2. Right-click and uncheck **Protected** from the popup menu.

The system removes the black triangle from the top-left corner of the channel header cell, indicating the channel is no longer protected.

To Protect a Channel

1. To protect a channel, select the channel header cell to protect.
2. Right-click and select (check) **Protected** from the pop-up menu.

A black triangle appears at the top-left corner of the channel header cell, indicating that the channel is protected.



Using the database popup menu you can also select to **Protect All** or **Protect None** of the database channels. Check the **Protect All** option to protect all of the channels in your database and **Protect None** to remove the protection from all of the channels.



Create Working Channels in the Geosoft Database

You can create new channels in your database to store the results of filters, mathematical expressions, or any other processing operations that you may apply to your data. We recommend that you create a new channel for storing any changes you make to the original imported data.

To Create a New Channel

1. From the top of an empty column, select the channel header cell (single click).
2. Type the name of the new channel.
3. Press **Enter**.
The *Create Channel* dialog appears.

4. The Name of your channel should be displayed automatically. If required, enter the remaining details, such as channel class, units, and display format.
5. Click **OK** to accept the defaults or changes you have made. The system creates a new channel in the database and fills this channel with dummy values (i.e. *).



*You can also select the channel header cell, right-click and select **New** from the popup menu.*

Delete a Channel from the Database

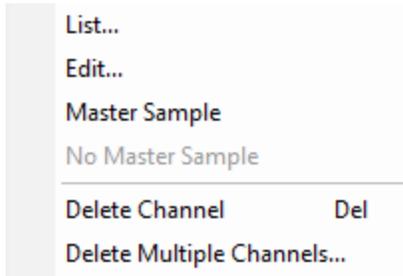
Before you can delete a channel from the Spreadsheet window, you need to verify that the channel is not protected. If you try and delete a channel that is protected, the system will not allow you to proceed with the deletion process.



*It is important to realize that deleting a channel is very different than just hiding a channel from Spreadsheet view. When you hide a channel from the spreadsheet, it is still in the database and can be retrieved simply by listing the channel again. When you delete a channel, it is removed from the Spreadsheet view and **also** deleted from the database.*

To Delete a Channel

1. Right-click on the channel header cell and select **Delete Channel** from the popup menu.



2. A *Delete* dialog box appears. This dialog box is a safety feature confirming that you really wish to delete a channel from the database.
3. Click **Yes** to permanently delete this channel from your database.



*The default is **No**. This is a secondary safety feature preventing you from pressing the **Enter** key by mistake and deleting the channel.*

Master Sample

The *Master Sample* option is used to force all other database channels to be sampled at the same fiducial rate as the selected channel. The Master Sample channel is identified by a blue line under the channel header.

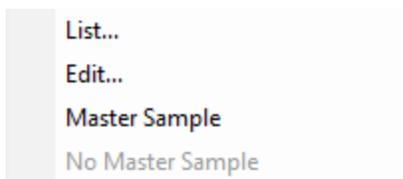
This feature is useful when you want to edit data from a channel that has a larger fiducial rate (data sampled less often) than other channels in the database.

By default, the spreadsheet window displays data using the channel with the smallest fiducial (most frequent sampling) rate.

For example, consider a database with two channels. Channel 1 has a fiducial sampling rate of 10 readings/second. This channel is considered to be smaller (more frequent) than Channel 2, which was sampled at 1 reading/second. By making Channel 2 the Master Sample channel, Channel 1 is displayed at the same rate so that only every tenth fiducial is displayed.

To Activate the Master Sample

1. Right-click on the database channel header.



2. Select **Master Sample** from the popup menu. A blue line is displayed under the channel header.

To Remove the Master Sample

1. Right-click on the database channel header.
2. Select **No Master Sample** from the popup menu. The blue line is removed from under the channel header.

Displaying Data in the Profile View

After you open a spreadsheet and display one or more channels, you have the option of displaying them in graphical (profile) form in a *Profile* window. The Profile window is composed of sub-windows called panes, which appear directly below its corresponding database. You can display up to five separate panes with 32 channels of data in each pane.



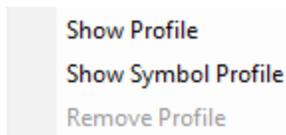
For additional information, see the application help *Help/Help Topics*.

Display Profiles

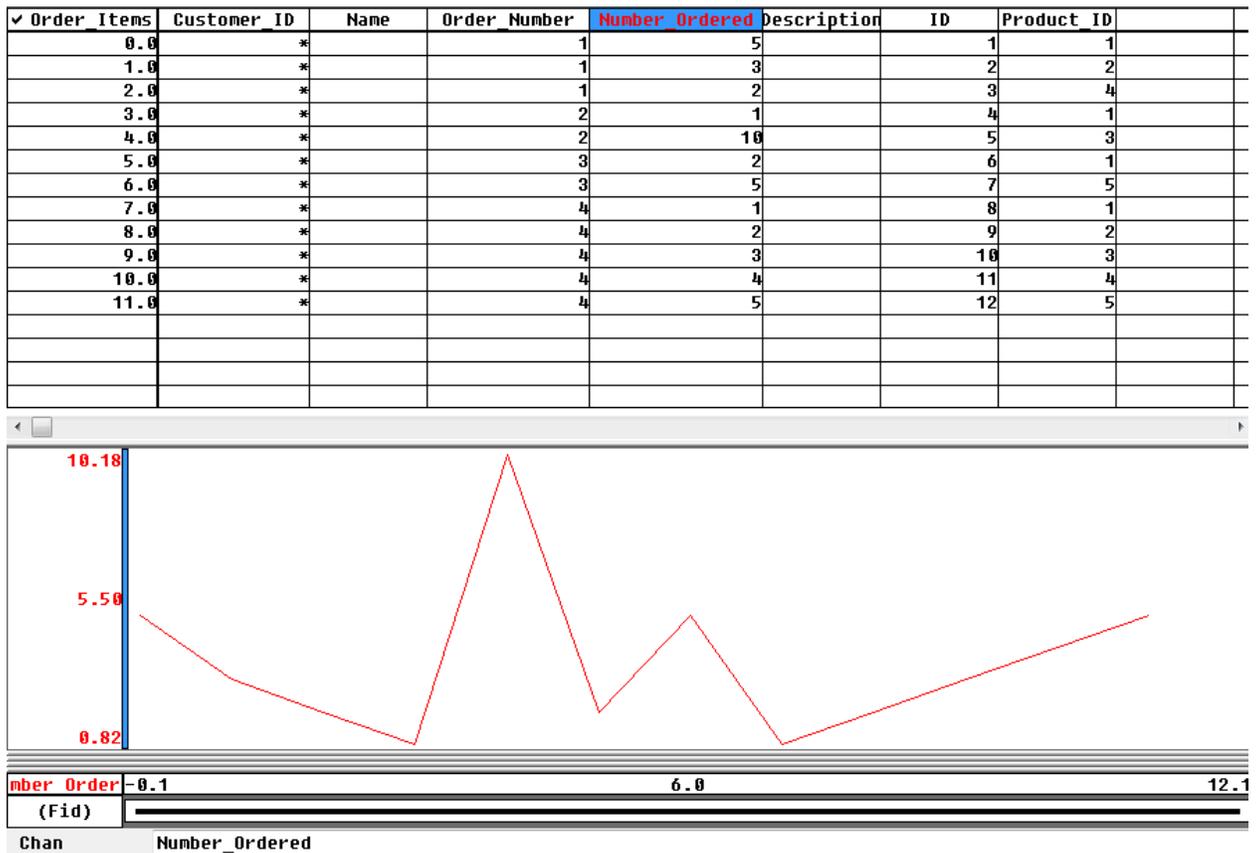
The Profile view is your "graphical window" to the Geosoft database. Profile windows are linked dynamically to their corresponding database. When you select a value or range of values in either the database or profile window respectively, they are also highlighted in the other window.

To Display a Profile

1. Select (highlight) the channel header cell of the channel you wish to view in the profile window.



2. Right-click and select **Show Profile** from the popup menu. The profile of the channel data displays in the *Profile* Window immediately below the spreadsheet.

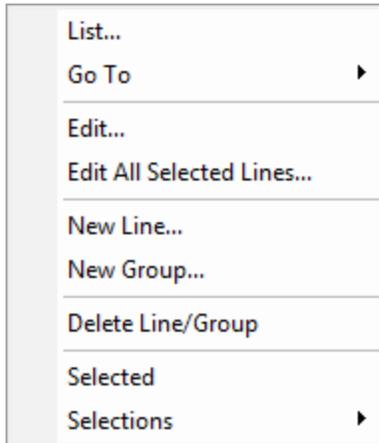


3. To see where values are located with regards to the profile line, simply select a value (or a range of

values) in the channel and the system will highlight the corresponding area on the profile (or vice versa).

To Display the Profile of Different Line

1. Select (highlight) the Line Header Cell in the top left corner of the spreadsheet.
2. Right-click and select **List** from the popup menu.



A list of all the line numbers in the current database will be displayed. You can think of each line number as a worksheet in the Spreadsheet window. When you view a new line number, a new worksheet is opened in the Spreadsheet window with corresponding data for that line number.

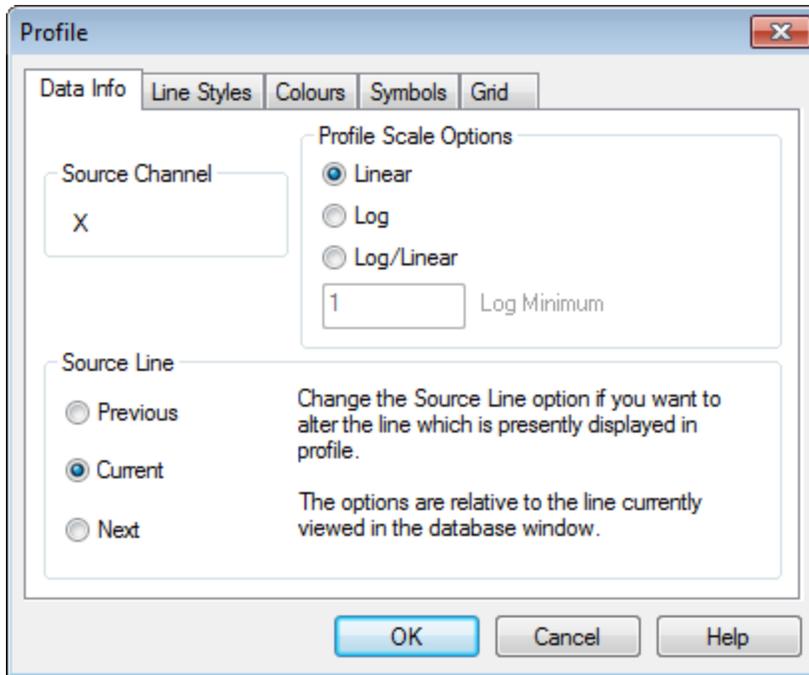
3. Click the line to display. The *Profile* window now displays the selected line.



*When the Line Header Cell is highlighted, you can use the Database Tools Bar to scroll through the lines by selecting **Previous Line/Group** or **Next Line/Group**.*

Set Profile Options

Use the tabbed *Profile* dialog to specify the options for the current profile. By default, the options will apply to the selected channel, in the image shown below this is the 'X' channel. This dialog is accessed by right-clicking inside the Profile window and selecting **Profile Options** from the popup menu.



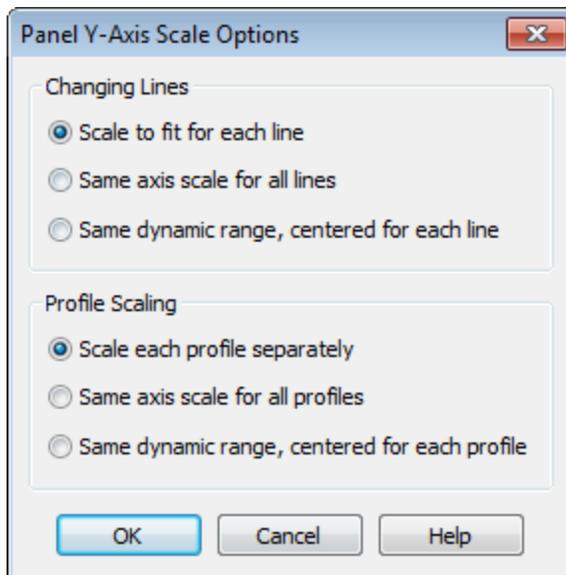
This dialog includes five tabs: Data Info, Line Styles, Colours, Symbols, and Grid.

Y Axis Options

The *Y Axis Options* menu enables you to set scale options for the Y axis of the profiles in the profile window.

To Set Y Axis Options

1. On the *Profile* window, right-click.
2. Select **Y Axis Options** from the popup menu. The *Panel Y-Axis Scale Options* dialog appears.



3. Select the method for **Changing Lines**, using the table below as a guide.

➤ Set Profile Options

4. Select the **Profile Scaling**, using the table below as a guide.
5. Click **OK** to change the Y Axis options.

The following table summarizes the different scaling options available to you:

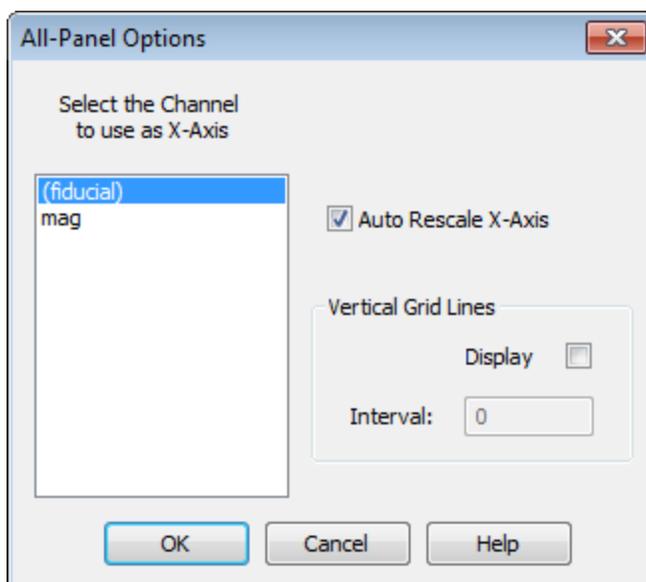
Changing Lines	
Scale to fit for each line	Adjusts the scale in the profile box to fit each line that is displayed.
Same axis scale for all lines	Uses the same axis scale for all the profiles that are displayed.
Same dynamic range, centred for each line	Uses the same dynamic range, centred for each line that is displayed.
Profile Scaling	
Scale each profile separately	All profiles (including those from different channels) are set to a different scale.
Same axis scale for all profiles	All profiles (including those from different channels) are set to the same scale.
Same dynamic range, centred for each profile	Uses the same dynamic range, centred for each displayed profile.

X Axis Options

The *X Axis Options* selection enables you to select the channel to use as the X-axis. You can also choose to automatically rescale the X-axis and/or display vertical grid lines.

To Select X Axis Channel

1. On the *Profile* window, right-click.
2. Select **X Axis Options** from the popup menu. The *All-Panel Options* dialog appears.



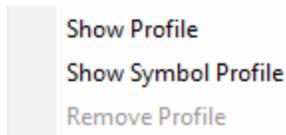
3. Select the *X-Axis* to use in the profile window.
4. You can also select to **Auto Rescale X-Axis**.
5. Select to display *Vertical Grid Lines*, if desired, and at what *Interval*.
6. When finished, click **OK**. Your profile is updated.

Edit a Channel using Profile View

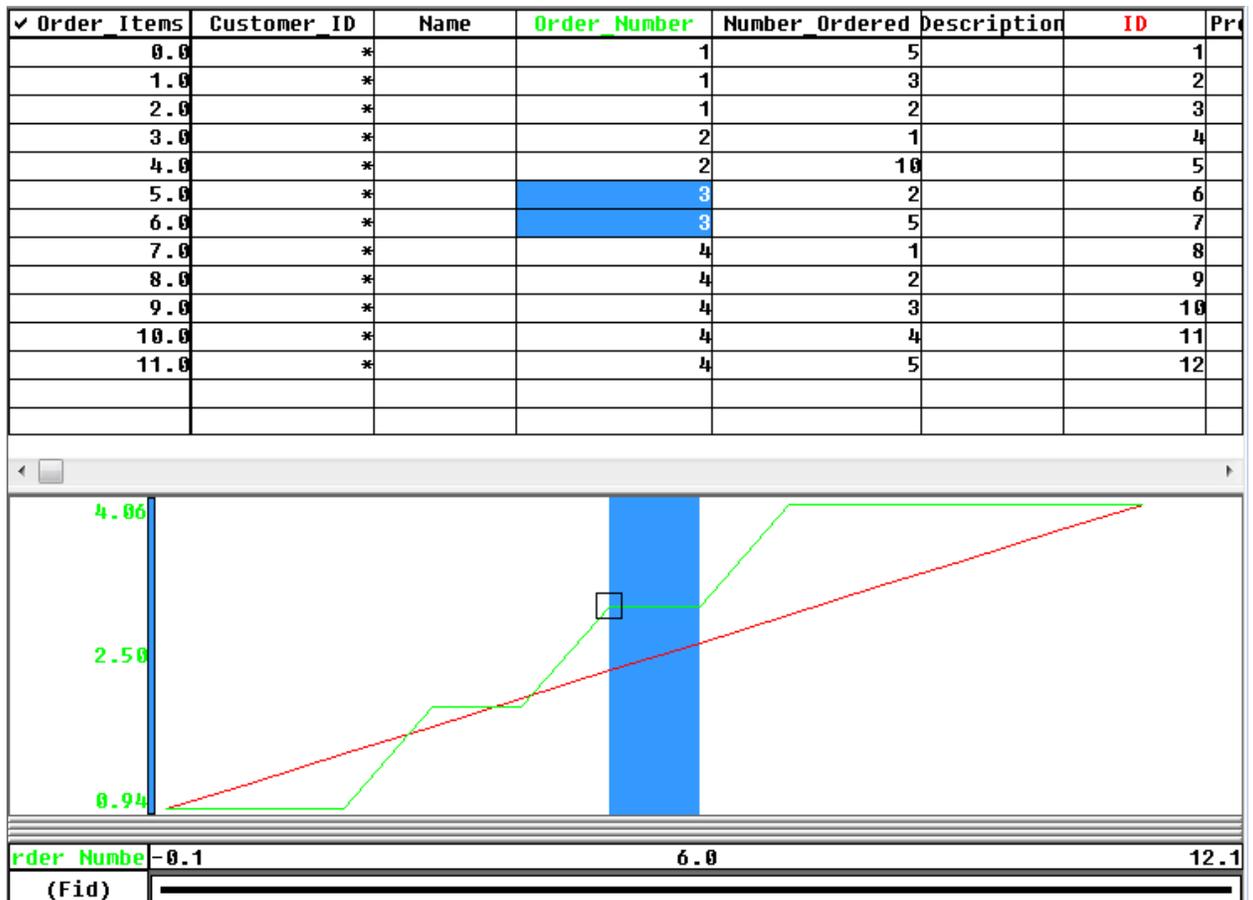
Before you can edit a channel, you must verify that the channel is not protected.

To Edit Data in a Channel Using the Profile View

1. Select the channel and right-click.



2. From the popup menu, select **Show Profile**. The system displays a profile of the data in the Profile window below the spreadsheet.
3. Move to the Profile window and click the data to be edited. You will notice that the corresponding data value is highlighted in the Spreadsheet view and a small box is drawn in the Profile window.





Clicking anywhere in the Profile window will display the corresponding value in the Spreadsheet view. You can click-and-drag to select a section of data in the Profile view.

4. Make the desired edits to the data in the spreadsheet.

Processing Data in the Spreadsheet

After you open a spreadsheet and display one or more channels, you have the option of processing the data using the Geosoft database tools. These tools help you apply formulas and perform database operations between channels for individual groups of points, lines or the entire database. Oasis montaj offers many options for processing and editing data in the spreadsheet. This guide shows some of the commonly used processes.



For more information about the processing tools, please refer to the Application Help files or click the **Help** button on the dialog of interest.

Calculate Statistics

Statistical information about your data can be very useful for general quality control and to ensure the values are in range with what you would expect.

To Calculate Basic Statistics on a Channel

1. Select the data to perform the statistics on using either a double or triple click of the mouse.



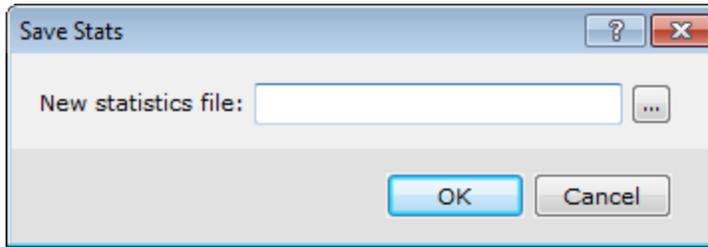
Clicking (highlighting) the channel header once selects the header cell, clicking twice selects the channel and clicking three times selects the channel in all of the selected lines (groups) in the database.

2. On the *Database Tools* menu, select *Report* and then select **Mark Statistics**. The *Stat Report* dialog appears.

Field	Value
Channel:	Number Ordered
Line(s):	[All]
Fid Range:	[All]
Num of items:	39
Num of dummies:	0
Minimum:	1.00
Maximum:	10.00
Mean:	3.64
Standard deviation:	2.40
Arithmetic sum:	142.00

3. Click **Save Stats** to save a copy of the statistical report.

The *Save Stats* dialog appears.



4. Specify a name and location for the *New statistics file*.
5. Click **OK** to save the file.
6. Click **OK** to close the dialog box.

The following list summarizes how to obtain results on specific parts of your database:

Click once on the channel header cell to highlight the header cell.	No statistics can be calculated.
Click twice (double click) on the channel header cell to highlight the data in the current line (group).	Statistics are calculated for that specific line (group) of the database.
Click three times on the channel header cell to highlight the data in all of the selected lines (groups) in the database.	Channel statistics are calculated for all of the selected lines (groups) in the database.

Edit Data using Conditional Expressions

Conditional math expressions can be used to edit data as required. For example, data spikes may be removed from a dataset.

Example

```
mag<9000?mag:dummy
```

This formula states that if the mag value is less than 9000, the channel value will equal the mag value; otherwise it is set to the dummy value.

To Use Conditional Expressions

1. Select the channel by double clicking the channel header (to highlight the data in the current line/group) or triple click (to highlight the data in all the selected lines/groups in the database).
2. To edit press the equal "=" key on your keyboard. The *Formula* bar appears below the profile window.

Formula=

3. Enter the appropriate formula.
4. Press **Enter**. The formula is applied.

Use the Channel Math Expression Tool

Use the *Channel Math Expression Builder* to create, save, load and execute math expressions for your channel data. This easy-to-use standardized dialog provides numerous mathematical expressions within a single tabbed dialog. For detailed information on the *Channel Math Expression Builder*, please click the **Help** button.

Common tasks

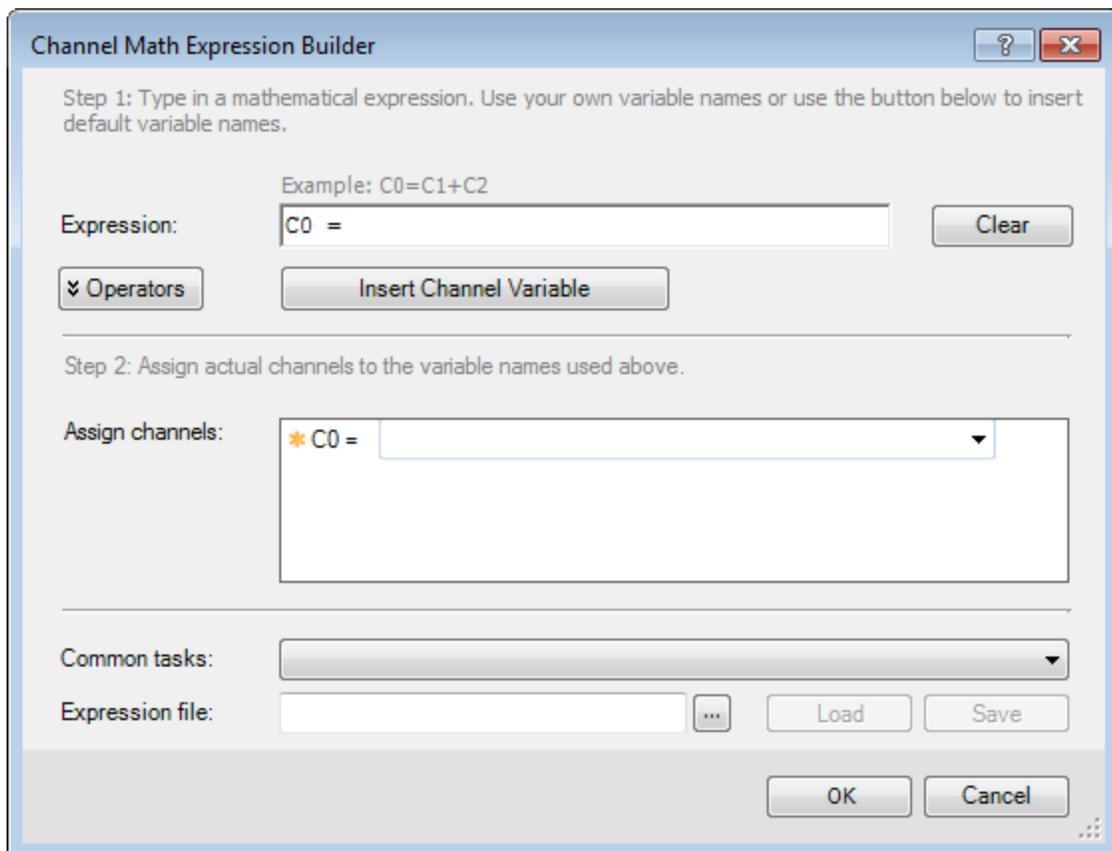
The *Channel Math Expression Builder* includes several pre-built **Common tasks**, accessible from the dropdown list.

Expression File

You can specify a pre-built expression file (*.exp) using the browse button on the *Expression file* field. You can also build an expression using the builder and save it to file using the Save option.

To Access the Channel Math Expression Builder

1. On the *Database Tools* menu, select **Channel Math**.
The *Channel Math Expression Builder* dialog appears.



The screenshot shows the 'Channel Math Expression Builder' dialog box. It has a title bar with a question mark and a close button. The main area is divided into two steps. Step 1: 'Type in a mathematical expression. Use your own variable names or use the button below to insert default variable names.' It features an 'Expression:' label, a text input field containing 'C0 =', and a 'Clear' button. Above the input field is an 'Example: C0=C1+C2'. Below the input field are two buttons: 'Operators' (with a dropdown arrow) and 'Insert Channel Variable'. Step 2: 'Assign actual channels to the variable names used above.' It features an 'Assign channels:' label and a dropdown menu showing '* C0 ='. At the bottom, there is a 'Common tasks:' dropdown menu, an 'Expression file:' text input field with a browse button (...), and 'Load' and 'Save' buttons. At the very bottom are 'OK' and 'Cancel' buttons.

2. Enter a mathematical expression in the *Expression* field.



To display the available Operators and Functions to help you build your channel math expression, click **Operators**. All of the Operators and Functions that are available to help you build your math expression are displayed.

3. Assign channels to the variable names used in step 2. Use the dropdown list to assign channels to the variable selected. An example of a simple channel addition ($C0 = C1 + C2$) expression is shown below:

Step 2: Assign actual channels to the variable names used above.

Assign channels:

C0 =	Addition	▼
* C1 =	X	▼
* C2 =	mag	▼

4. Click **OK**. The expression is applied.

Interpolate Data

In a situation where you remove cultural anomalies or "spikes" from magnetic data, the removed data points are replaced by dummy (*) values. Our 'Interpolate' function enables you to interpolate these dummy values.

To Fill Gaps in a Data Channel Using Interpolation

1. On the *Database Tools* menu, select *Channel Tools* and then select **Interpolate**. The *Interpolate Dummies* dialog appears.

2. Select the **Channel to interpolate** from the dropdown list.
3. Select the **Output interpolated channel** from the dropdown list. You can overwrite an existing channel by selecting it from the dropdown, or write the interpolated data to a new channel by entering a new channel name.
4. Select the **Interpolation method**. For more detail on the available methods, refer to the *Help* dialog..
5. Select Yes or No to **Interpolate dummy edges**.
6. Click **OK**. The system will interpolate the gaps created in the data.



*If chose to specify a new channel, the new channel with the interpolated data will not automatically display in the spreadsheet view, as it is hidden from view by default. Simply select the first empty Channel Header Cell, type the channel name, and press **Enter**. The channel will be displayed.*

Dynamic Links: Geographic Connections to your Data

Dynamic data links are "virtual" georeferenced connections between spreadsheets, profiles and maps. Links are graphical connections that connect what you see on a map to the same location in other displayed Map windows and to the original data in *Spreadsheet* and *Profile* windows. These links work in multi-directions so you can select any window and see the equivalent data in the other windows you are using.

Dynamic linking helps you to build knowledge quickly and make more informed decisions, through enabling real-time and visual quality control, processing and analysis of your data.

Working with Dynamic Linking

At the most basic level, you can use dynamic linking to link spreadsheet data and profiles.

1. When you work with *Spreadsheet and Profile* windows and select one or more cells in the spreadsheet, the system automatically shows you and locates the corresponding data in the *Profile* window (or vice-versa).
2. When you work with spreadsheet, profile, and map windows and select a point on the map, the system automatically highlights the corresponding data in the other windows (or vice-versa).
3. You can access the data linking tools on the Map Tools toolbar . The Shadow Cursor button () enables you to view data for a single line/group. To view data for multiple lines/groups use the Shadow Cursor Data Linking button ().
4. You can also create links between map extents (i.e. pan and zoom multiple maps will occur simultaneously). This functionality is enabled via the Change Extent tools (), on the Map Tools toolbar.

Note that, you can only use database linking if your map contains line/group linking items (i.e. survey lines, symbols, down hole sections or an ArcGIS LYR file).The system scrolls through the corresponding database, updating the Spreadsheet and Profile windows automatically to reflect the currently selected line/group.

Link Data Dynamically

Dynamic links are "virtual" georeferenced connections between spreadsheets, profiles and maps. These links work in two-directions so you can select any window and see the equivalent data in the other windows you are using.

To Link Spreadsheet, Profile and Map Data

Desktop linking enables you to place a special Link cursor on one or more maps and locate a specific part of the data.

1. Make sure only the desired map and the database are open in your project.



Do not remove them from your project, just close (minimise) any other map, grid or database items that are currently open in your project.

2. From the *Window* menu, select *Tile Vertically*. The system arranges your database window and map window side by side.
3. The system automatically turns on a map link and activates the *Shadow Cursors* button on the *Map Editing* toolbar. You can tell a button is activated because it will appear to be depressed more than the other buttons on the toolbar.
4. Click on a value in the database or on a point in the profile view, and you see that the Link cursor (large black cross) moves to the same location on the map. This type of linking enables you to quickly identify discrete anomalies on a map and view their corresponding values in the database.
5. You can display multiple maps and use this function to locate and visualize the same anomaly locations for different data types. It is quite useful for quality control (searching for anomalous noise spikes, for example) and for multi-parameter interpretation.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Database Maintenance

There are a number of utilities available for database maintenance. These utilities are provided to enable the user to check the status of the database, perform maintenance tasks and add comments to the database log file. The following is a list of the data maintenance utilities:

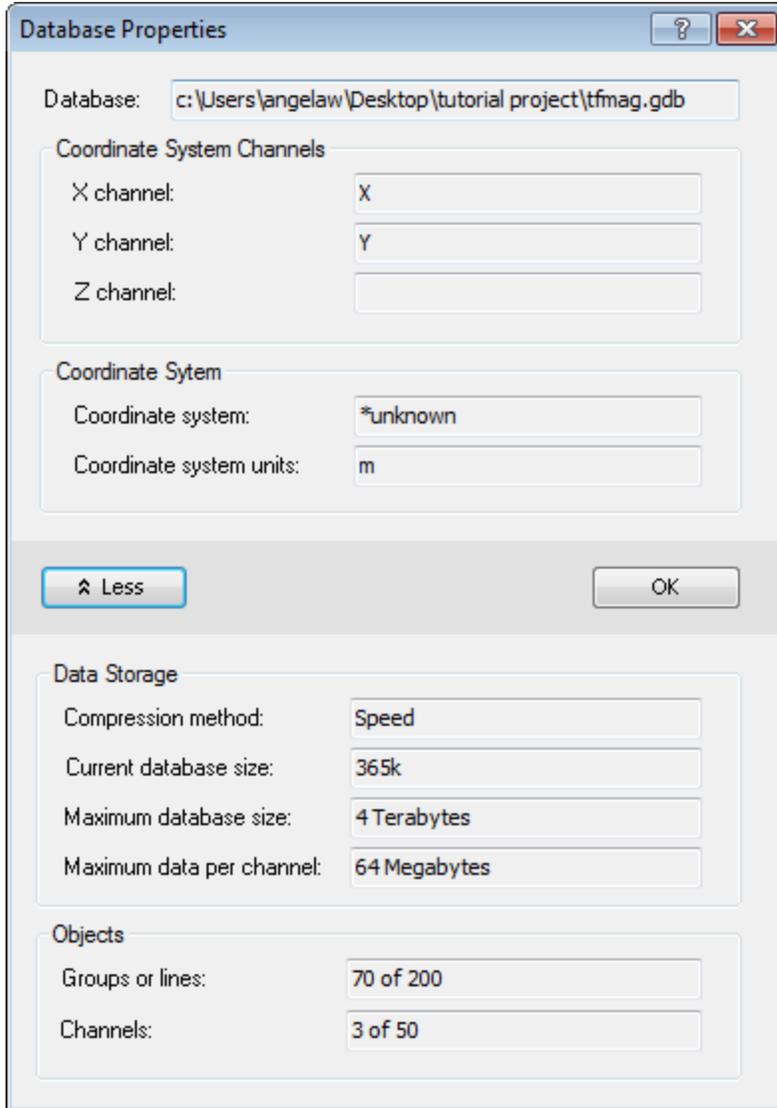
- Database properties
- Grow database
- Verify database
- Repair database
- Defragment database
- Add database comments

View Database Properties

From time-to-time, you may want to examine the database properties.

To View the Properties of a Database

1. Select the database.
2. On the *Database* menu, click **Properties**.
The *Database Properties* dialog appears.



 Click **More** to expand the database properties displayed.

We are mostly concerned with whether we are close to exceeding the limits of our *Groups or Lines and Channels*. If the unused or lost space seems excessive (say greater than 20%) we might consider *Defragmenting* the database.

 For more information about these statistics, click the **Help** button on the Database Properties dialog.

3. Click **OK** to close.

Growing, Verifying and Repairing a Database

Database utilities are accessed from the *Database | Maintenance* menu. If you find that the original size limits that were selected for your database (lines/groups and channels/fields) were too small, you can

increase these limits using the **Grow** command. If you wish to check the integrity of your database, you use the **Verify** command. If your database has been corrupted (for example, if the computer failed during a Defragment operation) you can run the **Repair** utility to try to recover some or all of the data.



*For each of these three commands, please refer to the detailed explanations provided by clicking the **Help** button on the dialog of interest.*



If you attempt to import data into a database which does not have the size capacity for that data, you will receive an error message which will prompt you to Grow the database prior to proceeding.

Defragmenting the Database

As you work with data in the database, new and modified data are saved to the end of the database file and to space freed within the database before the last time you saved the project. As the database grows, you can lose space due to fragmentation or from simply erasing data.

To Defragment a Database



Make sure you have backed-up the database before using this utility

1. On the *Database* menu, select *Maintenance* and then select **Defragment**.
The system removes all unused space by repacking the data as efficiently as possible and reduces the database file (*.GDB) size by the amount of freed space..

Adding Database Comments

Comments about your data can be added to the audit log of a database. Comments are useful as a place to hold information about the data collection procedures, instrumentation and surveyors used, any specific problems and the solutions to them, and other bits of information that may have formerly been kept as field notes, readme files, or other notes.

The comment block is an RTF file that can be edited with the Windows Wordpad application, or another text editor.

To Add Comments to a Database

1. On the *Database* menu, select **Edit Database Audit Log**.
This launches your editor and opens the file which is stored in your database.
2. Edit this file however you wish, adding text, pictures, etc.
3. When you are done editing, close the editor. You will be prompted to save the file. Click **Save**.
4. On the *Database* menu, select **Edit Database Audit Log** again. You will see the comments that you entered .

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Basic Mapping in Oasis montaj

Oasis montaj can be used to produce an intelligent entity called a *map*. Physically a map is a special graphic file (*.MAP) containing drawings and images. We call the map intelligent because it enables you to edit it interactively, apply dynamic linking and track the map creation processes. This How-To Guide walks you through basic map creation.

Maps provide a key tool for interpretation for many earth science applications. Oasis montaj provides a wide range of capabilities for performing both basic and advanced mapping. In addition, the map is an important part of the dynamic linking and imaging capabilities in Oasis montaj.

Topics covered in this How-To Guide:

- Map preparation
- Creating a new map
- Setting up map layout templates
- Creating a base map
- Plotting survey lines
- Plotting profiles

Other How-To Guides focus on custom map creation, dynamic linking and imaging, and adding special elements to maps.

Maps in Oasis montaj

In Oasis montaj, maps are special items that serve a number of purposes in the system. First, the map window provides the basic mechanism for creating maps, images and linking to other maps and data. Secondly, maps provide a means of editing graphics, adding text and creating highly customized final presentations for output to printers and plotters. To work effectively with maps, you need to be familiar with the purposes of maps in the system as well as the role of views and groups.

Oasis montaj supports on-the-fly image re-projection, so images may be used in a single map (previously, images that used plug-ins would not re-project and you had to create a separate map to display the image). Also the speed of redrawing re-projected data groups on maps has been improved significantly. True transparency has also been added and is automatically enabled, with video cards that support it.

For users who don't have true transparency supporting video cards, the 'Light table' display technology is used. With this technology, each group or layer in a map has individual transparency settings, controlled from the *View/Group manager*. These transparency settings affect both 2D and 3D raster images and vector line work.

When you create a new map the system opens up a new blank *Map Window*. After a *Map Window* is open, you can add a wide variety of data- and base map-related information.

Understanding the Parts of a Map

Oasis montaj uses views to organize information on a map. Map groups determine how (in what order) that information is displayed.

Base and Data Views

Geosoft Maps use views to organize and display information. A view divides the information on a map into a *base view*, which uses paper coordinates, and a *data view*, which uses ground coordinates. As you use the system, you will become familiar with the information that is stored in each type of view and how to manipulate them (for example, by turning off a map layer for plotting purposes).

- **Base view** - map surrounds, north arrows, scale bars
- **Data view** - map coordinates, contours, grids, images

3D Views

A 3D view can include data from an existing 2D view or group already displayed on a map and 2D grid files. Like 2D views, a 3D view also represents coordinates, but in this case in X,Y **and** Z. 3D views may also contain any number of drawing planes. Any group or drawing object that can be drawn on a 2D view can be drawn on a plane in a 3D view. Each plane in a 3D view can be oriented independently in the 3D coordinate system. A plane in a 3D view can be flat or it can have a surface relief defined by a grid file.

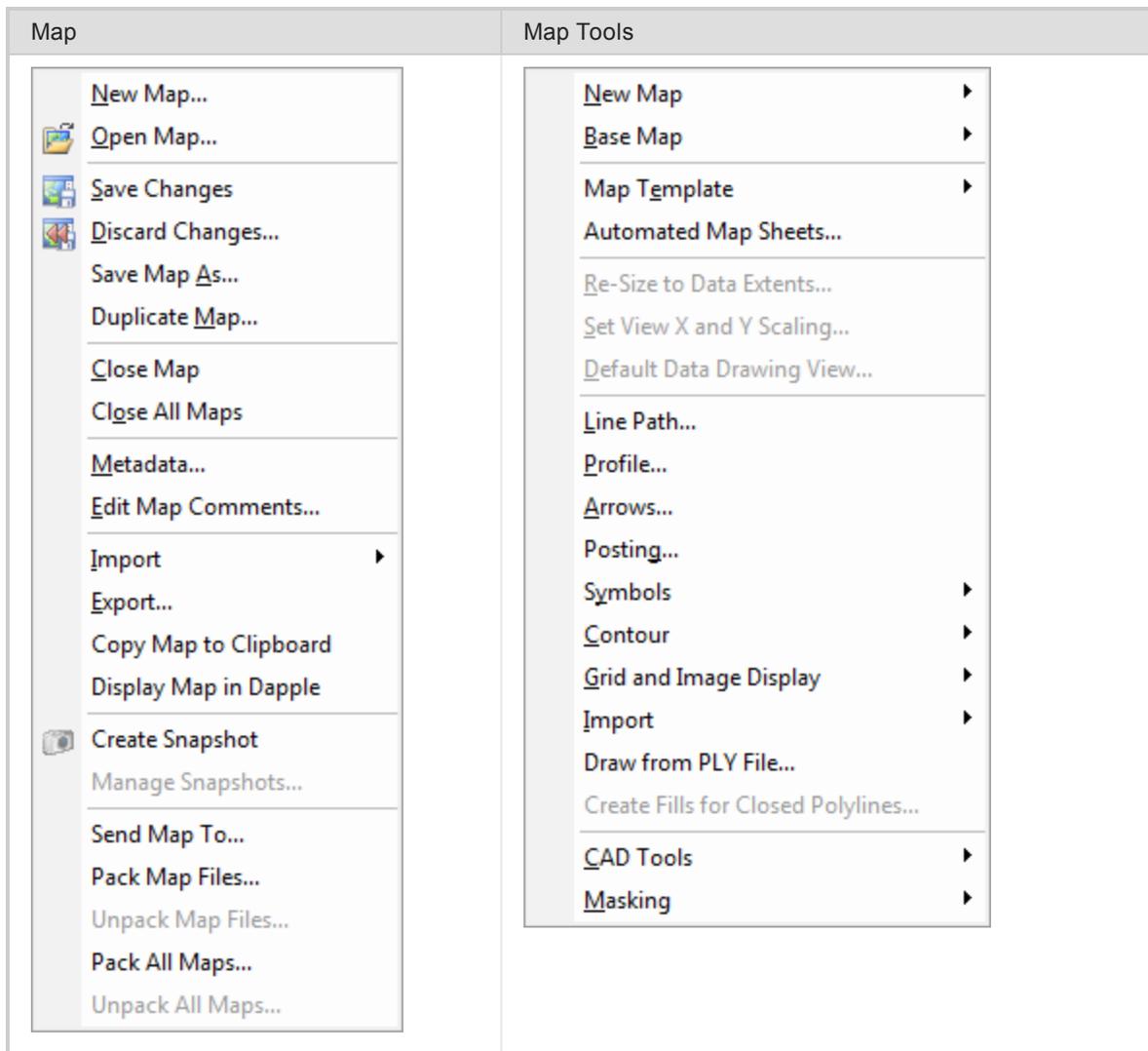
The *3D Viewer* provides all of your 3D-mapping tools, functions and settings in one integrated 3D environment. The 3D Viewer enables you to change the orientation of the 3D view, relative to your point of view, by rotating the coordinate system horizontally and vertically and by zooming and panning. The 3D view can be modified using the 3D Tools, including defining the orientation, setting relief grid controls and transparency levels of individual planes, and specifying axis labels.

Groups

Oasis montaj uses groups to determine the order in which objects are displayed (rendered) on a map. Examples of Groups in the Base view include a north arrow, title text box, or a scale bar. Examples of Groups in the Data view include lines, polygons, grids, and images. You can create, edit, move, hide, and mask groups. You can also move groups in front and behind each other. Any new object you add to a map, such as a polygon or line, is added to the current group. Map group names have a limit of **1024** characters.

Map Menus

There are two main map menus, *Map* and *Map Tools*, as shown below:



Map Preparation

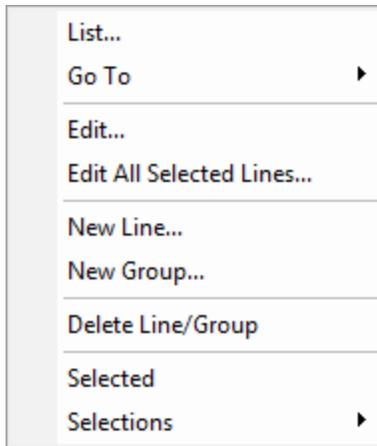
Oasis montaj creates base maps and map elements, and performs other processing (such as gridding) based on the lines (or groups) selected in the database.

Selecting Data for Mapping

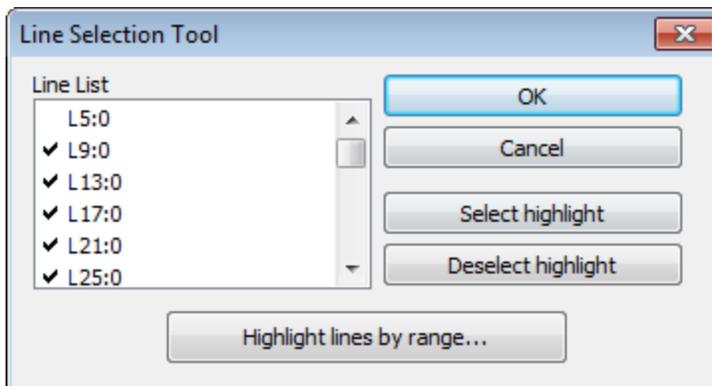
By default, when you import data, all lines are selected.

To Select Data for Processing

1. First ensure that the database is open and selected in your current project.
2. Select (highlight) the line header cell.
3. Right-click and select **Selections** from the pop-up menu.



4. Select **Selection Tool**. The *Line Selection Tool* dialog appears.



5. You can highlight the lines you want to select or deselect individually, or click the **Highlight lines by range** button to select lines by range. Lines that are selected will have a check mark beside them and lines that are not selected will have no check mark beside them.



*Clicking **Select highlight** or **Deselect highlight** either adds a check mark beside the highlighted lines or removes the checkmark.*

6. Once you have made your selections, click **OK**. The data for mapping is selected.

Creating a New Map

When you create or open a map in Oasis montaj, the system displays a *Map* window in the project. For a new map, the window is initially empty. For an existing map, all map elements are displayed when the window opens. For more information on mapping, map components and working with maps, click the *Help* button on the dialog of interest.

Creating a map from X and Y Coordinates

Maps can be created from a 'X,Y' coordinates, Long/Lat, or a template. The example below demonstrates how to create a new map from X and Y coordinates using data from the current database.

To Create a New Map from X,Y

1. Open a database in your current project.
2. On the *Map Tools* menu, select *New Map* and then select **New Map from X,Y**. The *Data range to map* dialog appears.

The screenshot shows a dialog box titled "Data range to map". It has a title bar with a question mark and a close button. The dialog contains the following fields and controls:

- Minimum X: [text input field]
- Minimum Y: [text input field]
- Maximum X: [text input field]
- Maximum Y: [text input field]
- Distance unit: [dropdown menu showing 'metre']
- Coordinate System: [text input field showing '*unknown']
- Buttons at the bottom: Next>, Scan data, Scan grid, Interactive, Coordinate System

The *Data range to map* dialog is used to create a new blank map to fit the data range specified. The data range can be entered manually, or it can be determined from the range of the selected data (**Scan data**) in the current database, from the range of a grid file (**Scan grid**), or interactively by defining an area on a map (**Interactive**).

3. Click **Scan data** to have the system query the database for the *Minimum* and *Maximum* coordinates and the *Distance* units.



If the scanned data or grid file does not have an established unit, "metres" is assumed. It is important that the distance units accurately reflect the units of the data range. This default value can be modified in the Default Settings dialog.

4. Click **Next**. The *Create a new map* dialog appears.

The screenshot shows a dialog box titled "Create a new map". It has a title bar with a question mark and a close button. The dialog contains the following fields and controls:

- Map name: [text input field with a browse button (...)]
- Map template: [dropdown menu showing 'landscape letter']
- Map scale: [text input field]
- Distance unit: [text input field showing 'metre']
- Buttons at the bottom: <Back, Finish, Scale, Templates

5. Enter a *Map name* and location.
6. From the drop-down list, select the correct **Map template**.
7. Click **Scale**. The system automatically calculates a scale that will fit the data on the map template. If no scale is specified, a scale will be chosen which will fit all of the data within the

chosen map sheet.



It's recommended to round the scale to a more appropriate number, for example 100000. (If you round the scale, you must always increase the number, thereby reducing the scale).

8. Click **Finish** to display the new blank map.

Map Templates and Base Maps

An important component of the map is the map template. Templates control placement of base map elements (surrounds), contours, grids, plots, images and any other graphics elements required to complete your map.

Creating MAPPLOT Map Templates (manual)

The system provides a variety of default templates that enable you to create your own standard maps with logos, custom layouts and other elements as required. For example, you may use the default A4 or A3 templates to create your own layout that you use regularly for making a map.



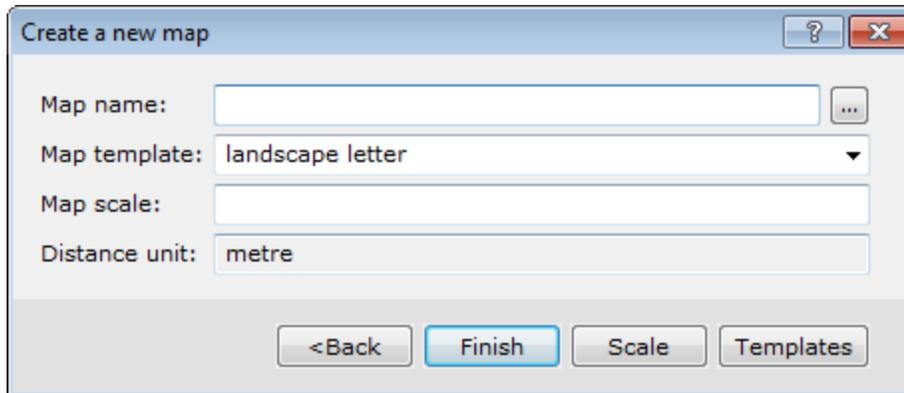
When you create a map template, it is saved permanently in your system. When you are learning the system, we suggest you experiment using the same template until you are ready to create a final version with a custom name. You may want to make up a testing template, called TEST, for example.

To Create Your Own MAPPLOT Map Template

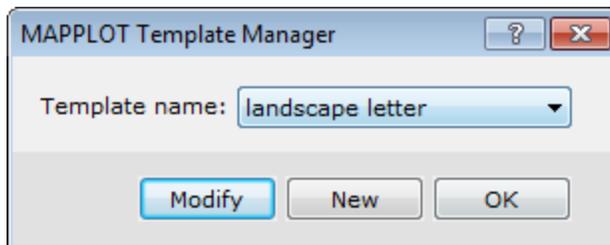
In this example we create a new map from X,Y coordinates and save the results as a MAPPLOT Map Template.

1. On the *Map Tools* menu, click **New map**.
2. Click **New map from X,Y**. The *Data range to map* dialog appears.

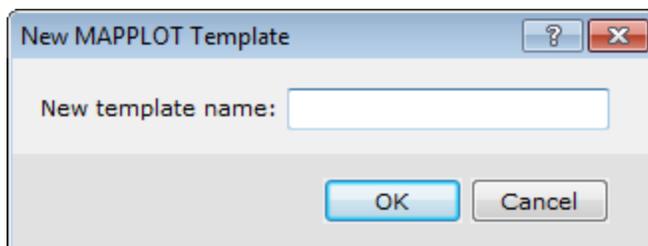
3. Confirm the X,Y Coordinates and Units.
4. Click **Next**. The *Create a New Map* dialog appears.



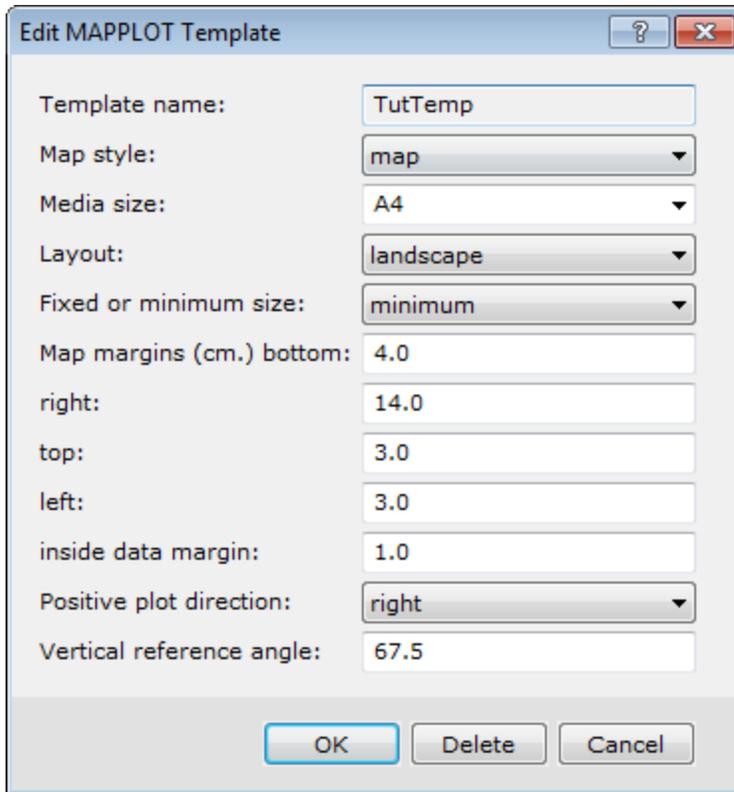
5. Specify a *Map name* and location.
6. From the drop-down list, select the *Map template*.
7. Click **Templates**. The *MAPPLOT Template Manager* dialog appears.



8. Click **New** to create a new template, so that the original template is not overwritten. The *New MAPPLOT Template* dialog appears.



9. Specify a *New template name*.
10. Click **OK**. The *Edit MAPPLOT Template* dialog appears.



11. Select the **Map style**.
Figure style maps have a layout with the title, scale bar and north arrow located at the bottom of the map, whereas *Map* style maps have a layout with the title box, scale bar and north arrow along the right hand side of the map.
12. Define the other Template settings. You can specify margins, and set the plot direction.
💡 For detailed information on each setting, click the **Help** button.
13. When complete, click **OK** to continue. The system returns you to the *MAPPLOT Template Manager* dialog.
14. Click **OK**. The *Create a new map* dialog reappears.
15. Click **Finish** to display the new blank map.

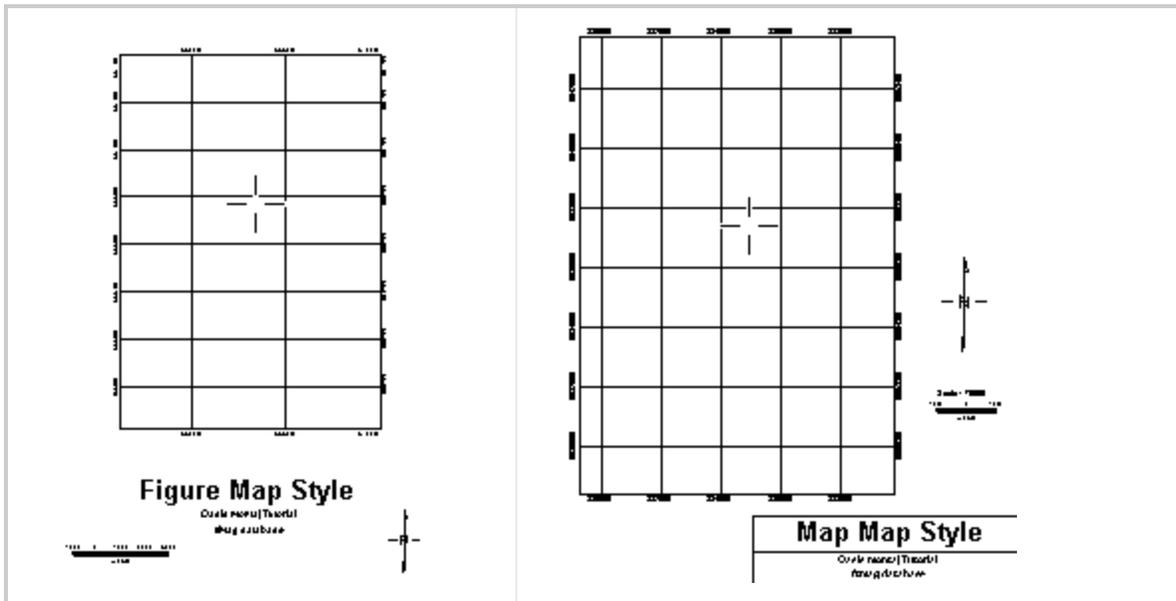
For more information on using MAPPLOT commands, see the *Geosoft Executables (GXs) | MAPPLOT Topics and Command Reference* topics, located in the Oasis montaj Application Help System (*Help/Help Topics*).

Creating a Base Map

Maps are composed of views. A view is a coordinate translation and a masking region and is used to organise and display information on a map. Maps consist of a *Base view*, which uses paper coordinates with an origin at the bottom left corner of the page, and a *Data view*, which uses ground coordinates.

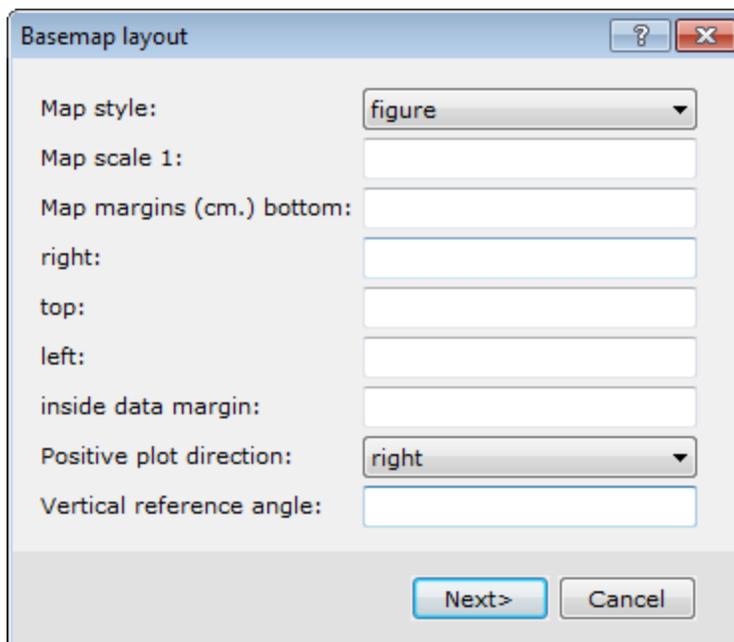
You can customize your base map with many elements, including map surround, north arrow, scale bar, reference grids, latitude/longitude annotations, text blocks/balloons and titles. The layout of your base map can either be map or figure style. Figure style maps have a layout with the title, scale bar and north

arrow located at the bottom of the map, whereas map style maps have a layout with the title box, scale bar and north arrow along the right hand side of the map. The following images show the difference in layout between the two styles.



To Create and Plot a Figure Style Base Map

1. Make sure your map file is open and selected in the project.
2. On the *Map Tools* menu, select *Base Map* and then select **Draw Base Map**. The *Basemap layout* dialog appears.



The 'Basemap layout' dialog box contains the following fields and controls:

- Map style:
- Map scale 1:
- Map margins (cm.) bottom:
- right:
- top:
- left:
- inside data margin:
- Positive plot direction:
- Vertical reference angle:

Buttons: Next>, Cancel

This dialog is the first dialog in the base map layout wizard. The type of *Map style* that you choose here will reflect the type of dialogs to follow.



For this example, we select the **figure** style.

3. Select the **figure** style.
4. Click **Next>** to continue.

The *Figure style base map* dialog appears.

5. Specify the display characteristics of the base map elements.



For detailed information on each setting, click the **Help** button.

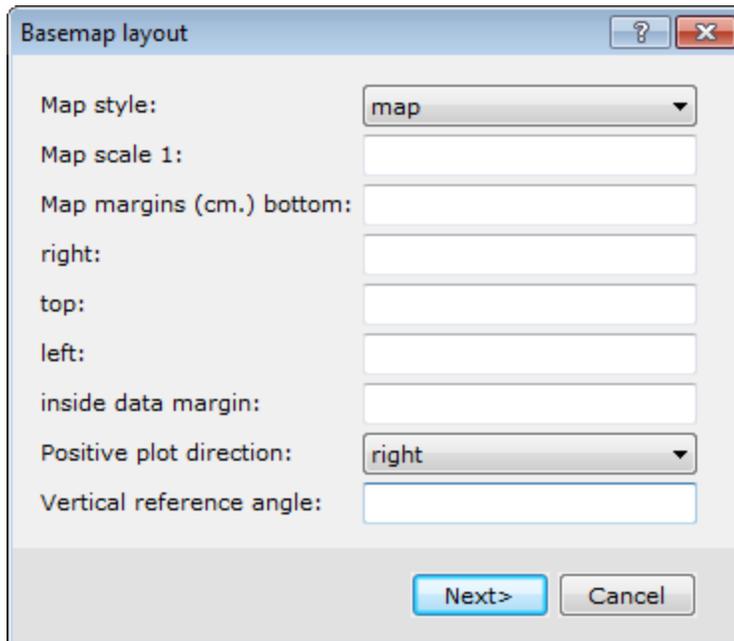
6. Set the *North direction*. If blank, no North arrow is drawn.
7. You can specify different types of display styles for the *Reference grid* via the dropdown list.
8. Configure the other base map options as desired.
9. Click **Next>** to continue.

The *Figure titles* dialog box appears.

10. You can specify the map titles and sub-titles, if desired.
11. Click **Finish** to plot the base map on your current open map.

To Create and Plot a Map Style Base Map

1. Make sure your map file is open and selected in the project.
2. On the *Map Tools* menu, select *Base Map* and then select **Draw base map**. The *Basemap layout* dialog appears.



This dialog box is the first dialog box in the base map layout wizard. The type of *Map style* that you choose here will reflect the type of dialog boxes to follow.



*For this example, we select the **Map** style.*

3. Select *map*.
4. Click **Next>** to continue.
The *Full map style base map* dialog appears.

Full map style base map

North direction (deg. Azimuth from Y):

Reference grid:

Reference grid spacing:

Line thickness (microns):

Line colour:

Y annotation orientation:

Add compass direction to annotations:

Longitude, latitude annotations?:

Longitude increment (deg.mmss):

Latitude increment (deg.mmss):

Line thickness (microns):

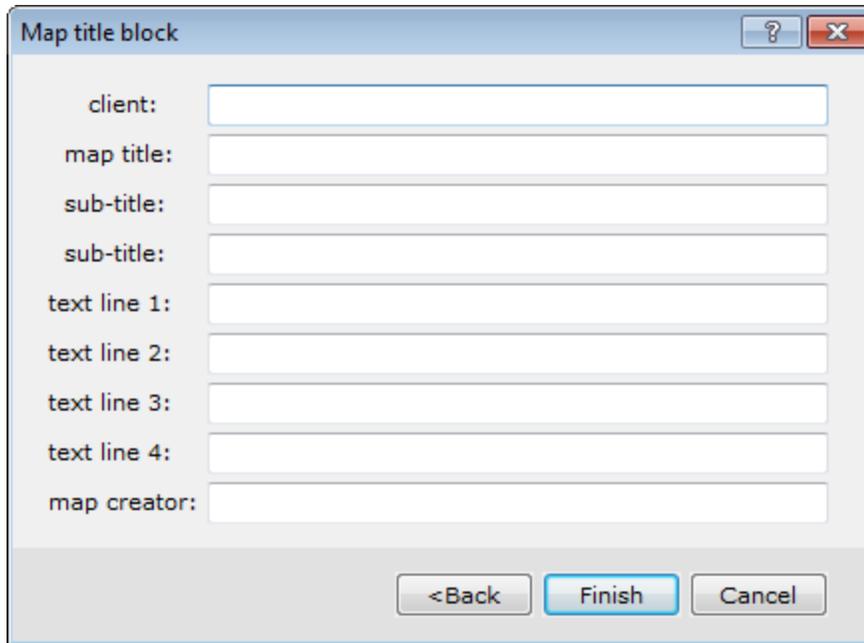
Line colour:

<Back Next> Cancel



For detailed information on each setting, click the **Help** button.

5. Set the *North direction*. If blank, no North arrow is drawn.
6. You can specify different types of display styles for the *Reference grid* via the dropdown list.
7. Configure the other base map options as desired.
8. When complete, click **Next>**.
The *Map title block* dialog appears.



The image shows a dialog box titled "Map title block" with a question mark icon and a close button (X) in the top right corner. The dialog contains several text input fields for map metadata:

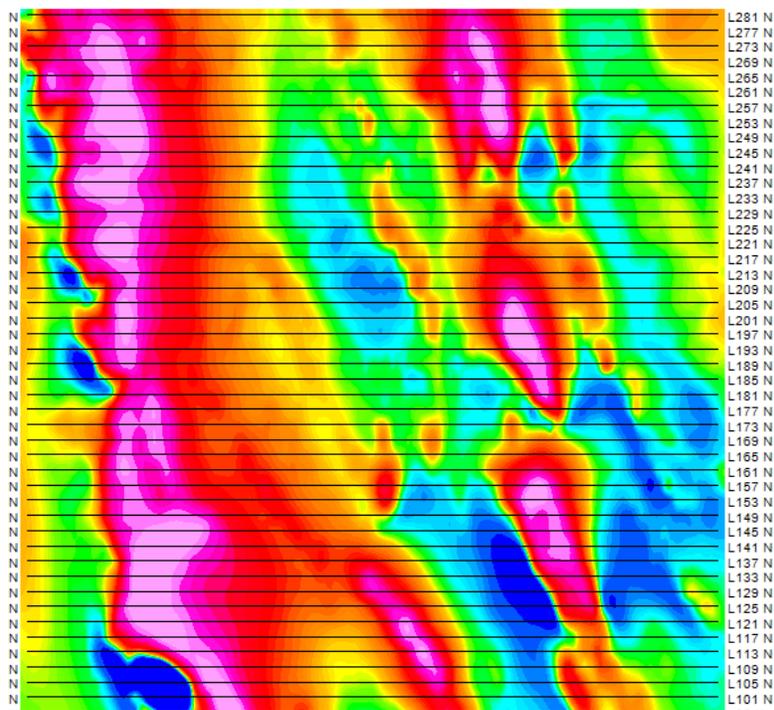
- client: [text input]
- map title: [text input]
- sub-title: [text input]
- sub-title: [text input]
- text line 1: [text input]
- text line 2: [text input]
- text line 3: [text input]
- text line 4: [text input]
- map creator: [text input]

At the bottom of the dialog, there are three buttons: "<Back", "Finish", and "Cancel".

9. You can specify the map titles and sub-titles, if desired.
10. Click **Finish** to plot the base map on the current open map.

Plotting Survey Lines

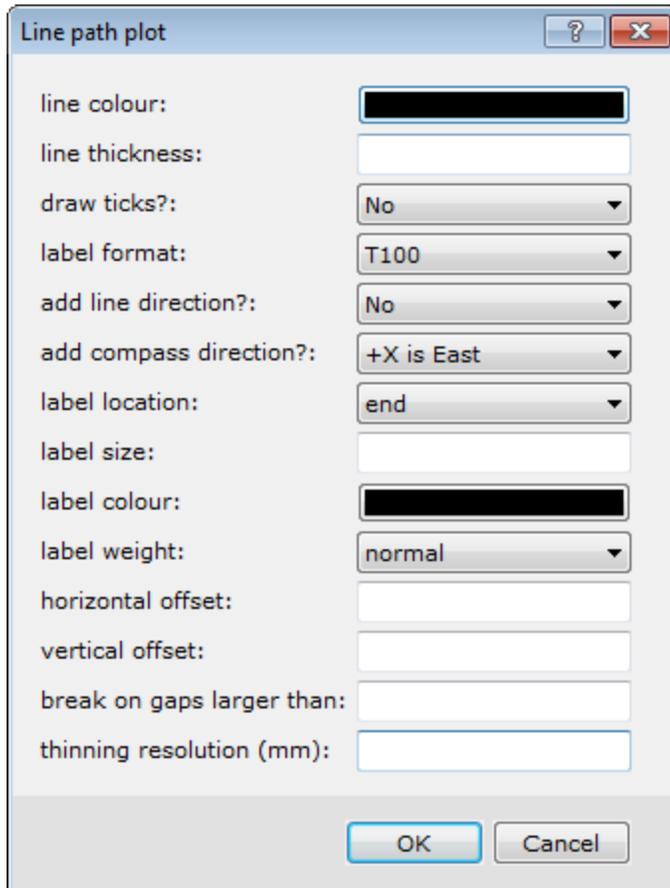
For line-oriented Geoscience surveys, you may want to display survey lines (flight lines for airborne surveys). An example is shown below.



To Plot Survey Lines

When plotting survey lines, make sure that the lines you want to plot are selected in your current database (see *Selecting Data for Mapping*).

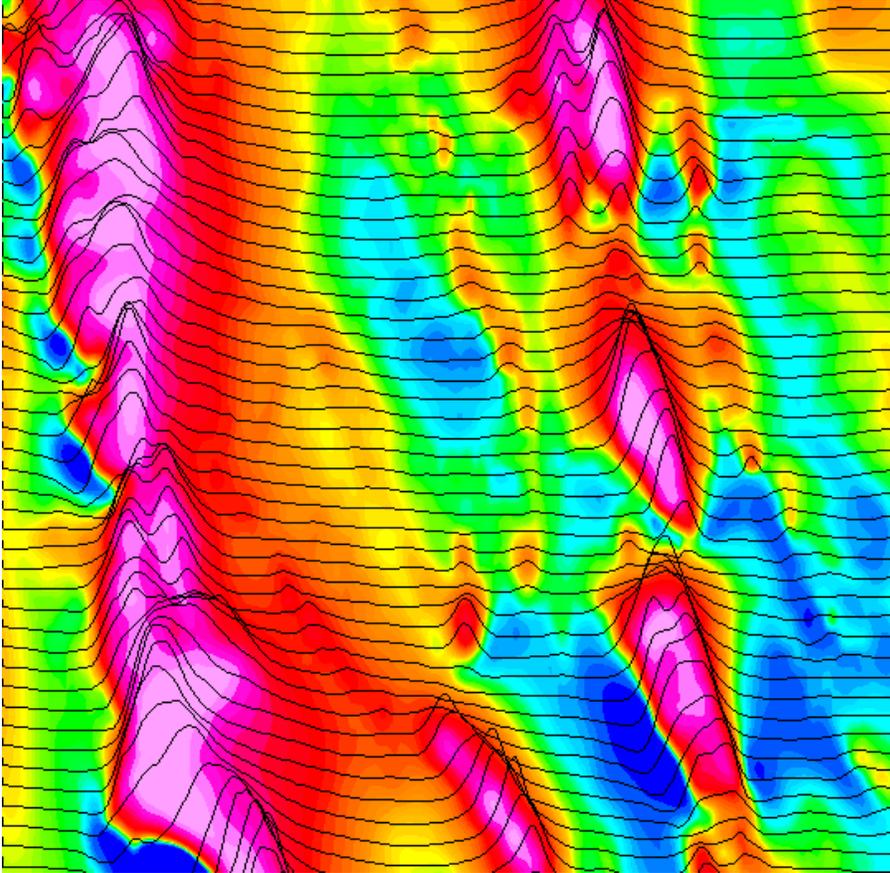
1. Select (highlight) the map.
2. On the *Map Tools* menu, select **Line Path**.
The *Line path plot* dialog appears.



3. Specify how the line paths should appear on your map. You can select the line colour, label size and colour, and other display settings. For more information on line path settings, click the **Help** button.
4. Click **OK**. You may be asked to confirm the map scale. The survey lines appear on your map.

Plotting Profiles

For line-oriented Geoscience surveys, you can display plan view profiles of the data values for each survey line. An example is shown below.



Click the **Help** button for more information regarding the parameter selections available in the Profile plot dialog box.

To Plot Profiles

When plotting profiles, make sure that the lines you want to plot are selected in your current database (see *Selecting Data for Mapping*).

1. Make sure the map is open and selected in your project.
2. On the *Map Tools* menu, select **Profile**.
The *Profile plot* dialog appears.

3. Select the channel to profile using the *Profile channel* dropdown list.
4. Specify the profile's vertical scale in units/mm.
5. Specify how the survey lines should display. You can define line colours, styles, and other options.
6. Click **OK**. The plan profiles of the selected channel appears on your map.

Moving Around the Map

Once you have your map displayed, you may want to zoom into the map so that more detail is displayed. You may also wish to move around the map using other options.



You can also access these commands by right-clicking on a map and selecting the options from the pop-up menu.

Map Layout

The following is a list of some of the different types of viewing options available for moving around and viewing the map with ease:

 Zoom Box	<p>Click this button to activate the Zoom Box. Click on the map, then while holding down the left mouse button and move your cursor to box in the area of the map you want to zoom. Click to select the area. Click the mouse button again to zoom to the area selected.</p> <p> <i>Once the box has been defined, the cursor and the box are linked and by moving the cursor around the map you can move the box to another location, and then click the left mouse button to zoom to the boxed area.</i></p>
 Pan	<p>Click this button to Pan around in the currently selected map. Click the left mouse button and while holding the button down, move the hand cursor to pan around the current map area.</p>
 Interactive Zoom	<p>Click this button to activate the interactive zoom. Click on the area of the map you want to zoom (e.g. an anomalous location on a grid), then while holding down the left mouse button, move your cursor left to zoom out and right to zoom in on the selected area.</p>
 Zoom In	<p>Click this button to zoom in and increase the displayed map features by 50%.</p>
 Zoom Out	<p>Click this button to zoom out and shrink the displayed map features by 50%.</p>
 Full Map	<p>Click this button to display the whole map area in the map window.</p>
 Zoom to Selection	<p>Click this button to zoom to the selected map view or group.</p>
 Previous Extent	<p>Click this button to undo all changes that you have made since the last time you saved the map. This command is useful for removing text and graphics that you may have added accidentally.</p> <p>When working on a map, you may prefer to use this command periodically so that you undo only a few changes at a time rather than your entire set of map edits and other changes.</p>
 Next Extent	<p>Click this button to redo the latest changes that you have made since the last time you saved the map. This command is useful for adding text and graphics that you may have removed accidentally.</p>
 Redraw	<p>If you manually resize the map window or perform editing on the map you may need to redraw the map to refresh its current view</p>

Map Tools



The *Map Tools* menu provides several options which enable you to control how your map appears and which menus are active. The last three buttons on the toolbar determine on which maps zooming occurs:

- › **This Map Only** button will zoom only the current map, which will be highlighted.
- › **All Maps** button will zoom all the maps, which share the same coordinate system.
- › **Other Maps Only** button will zoom all the other maps, which share the same coordinate system, with the exception of the current map. This enables the use of one map as a reference.



To find out the function of the other buttons on the *Map Tools Bar*, hold the cursor over the button until a description appears.

Map Pop-Up Menus

Map Views	Map Groups
Views ▶	Groups ▶
Coordinate System...	Edit This Group E
ArcGIS Transformations...	Next Group W
Next View W	Previous Group Q
Previous View Q	Grid Value ▶
	Delete Group Del

Map Window Status Bar

The projection information in the status bar is cleared when there are no active maps.

Configurable Hotkeys

Oasis montaj provides configurable *Hotkeys* enabling quick and easy access to menu items including all map tools. To access the *Shortcut Keys* dialog, an interactive dialog enabling you to set your own Hotkeys or view the defaults, select the *GX|Database Settings|Configure GDB Shortcut Keys* menu item.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Gridding

Gridding refers to the process of interpolating data onto an equally spaced “grid” of cells in a specified coordinate system, such as X-Y. The following gridding methods are available:

- Minimum curvature
- Bi-Directional
- Kriging
- Tinning
- Inverse Distance Weighting Gridding
- Direct Gridding
- Trend enforcement Gridding

Other gridding options also available are:

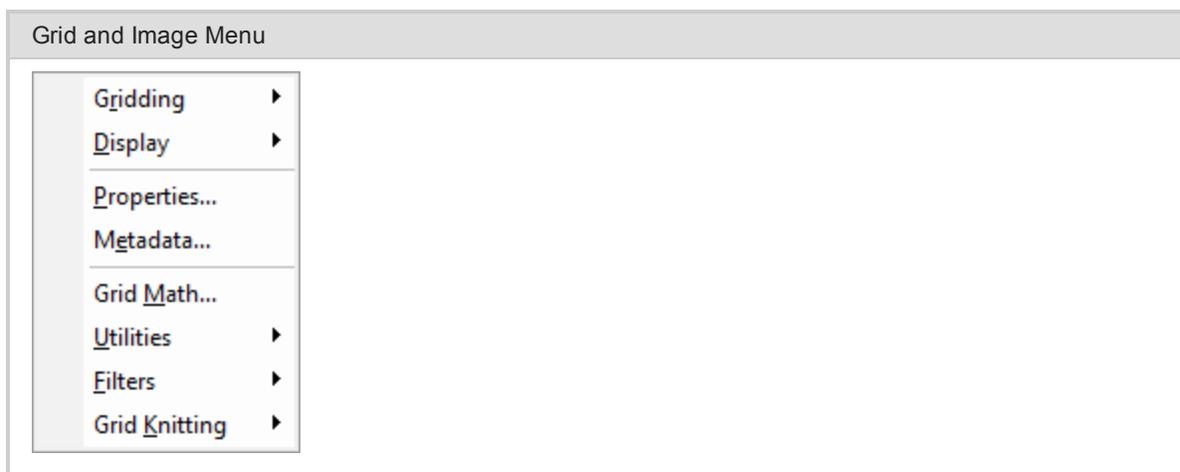
- Plot Variogram (Kriging)
- Grid Multiple Channels
- Re-Grid a Grid
- Grid from a Control File



Map elements (grids and images) are stored using relative paths. If you open a map with grids or images that have been moved from the original locations you used to be prompted to locate each grid or image. Now, when manual input of the file locations is required, the map will look for the other raster files in the same location, so you will only be asked once when there may be many grids displayed on the map

Grid and Image Menu

There is a main menu for grids and images, *Grid and Image* as shown below:



The *Grid and Image* menu includes options for creating, processing, displaying, analyzing, and converting grids and images.

Drag-n-Drop Geosoft Grid Files

The ability to drag Geosoft grid files (*.GRD) from Windows Explorer and drop them into an open project is now included in Oasis montaj.

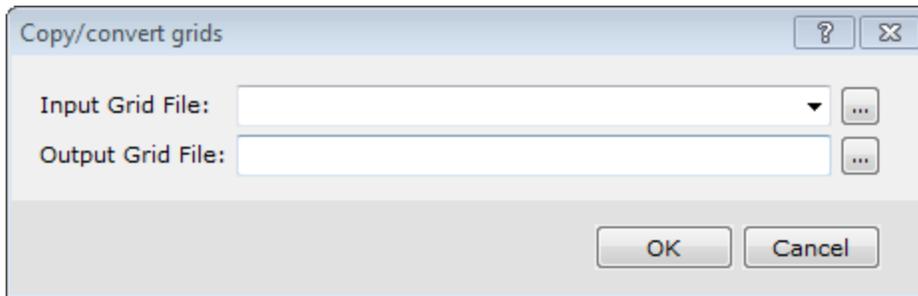
The ability to drag other file types directly from Windows Explorer and drop them into Oasis montaj for display is also available including; database files (*.GDB), map files (*.MAP), and users can also drag XYZ files and drop them into an open database file (*.GDB). Items displayed in the Project Explorer may also be dragged and dropped.

Copying and/or Converting Grid Files

Oasis montaj enables you to copy and/or convert any supported grid format to any other supported grid format.

Copying and/or Converting Grid Files

1. On the *Grid and Image* menu, select *Utilities* and then select **Copy/Convert**. The *Copy/convert grids* dialog appears



2. Using the **Browse** button, select the *Input Grid File*.
3. Specify the *Output Grid File*.
4. Click **OK**. The grid is copied and/or converted as specified.



For more information on copying and /or converting grid files, click the Help button on the Copy/convert grids dialog.

Gridding Methods

Oasis montaj provides seven different gridding algorithms that interpolate data to produce a grid. These methods are summarized below:

Minimum curvature

The Minimum curvature or random gridding method (RANGRID GX) fits a minimum curvature surface to the data points using a method similar to that described by Swain (1976) and Briggs (1974). A minimum curvature surface is the smoothest possible surface that will fit the given data values.

Minimum curvature first estimates grid values at the nodes of a coarse grid (usually 8 times the final grid cell size). This estimate is based upon the inverse distance average of the actual data within a specified search radius. If there is no data within that radius, the average of all data points in the grid is used. An iterative method is then employed to adjust the grid to fit the actual data points nearest the coarse grid nodes.

After an acceptable fit is achieved, the coarse cell size is divided by 2. The same process is then repeated using the coarse grid as the starting surface. This is interactively repeated until the minimum curvature surface is fit at the final grid cell size.

A very important parameter in the Minimum curvature process is the number of iterations used to fit the surface at each step. The greater the number of iterations, the closer the final surface will be to a true minimum curvature surface. However, the processing time is proportional to the number of iterations.

Minimum curvature stops iterating when:

- It reaches a specified maximum number of iterations, or
- Certain percentage of the observed points are within a limiting tolerance of the surface

By default these limits are 100 iterations and 99% of points within 1% of the data range.

Minimum curvature gridding also has the following capabilities:

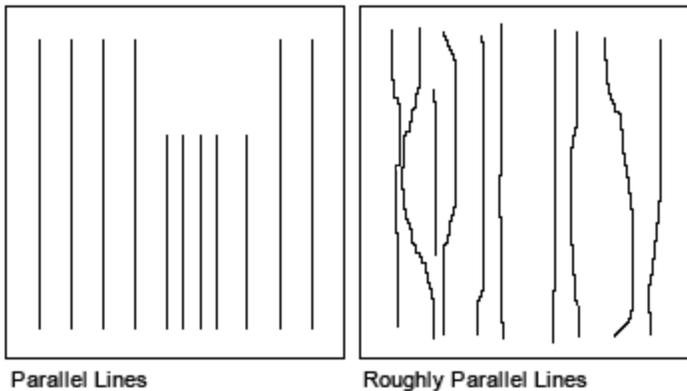
- Accessing unlimited number of input observation points
- Adjustable internal tension
- Applying de-aliasing filter
- Applying linear and logarithmic gridding
- Applying projections
- Blanking un-sampled areas
- Outputting grids up to any size



The actual maximum size of the grid that you can use depends upon the available free disk space of your computer.

Bi-Directional

Bi-Directional gridding or line gridding method (BIGRID GX) is a numerical technique for parallel survey lines or roughly parallel lines. Use Bi-Directional gridding, if the data is collected along lines that are roughly parallel, as in the following examples:



Bi-Directional gridding is not able to use the tie lines because of the way the gridding algorithm works. If the data on the tie lines is important, Minimum Curvature gridding or Kriging should be used.

The Bi-Directional gridding method is ideal for line oriented data because it inherently tends to strengthen trends perpendicular to the direction of the survey lines. In this way, Bi-Directional gridding can take advantage of the fundamental characteristics of line-based surveys.

The gridding process is carried out in two steps. First, each line is interpolated along the original survey line to yield data values at the intersection of each required grid line with the observed line. The intersected points from each line are then interpolated in the across-line grid direction to produce a value at each required grid point.

Geological trends in the data can be emphasized by the appropriate orientation of the grid so that the second interpolation is in the direction of strike. In addition to trend enhancement, Bi-Directional gridding enables the method of interpolation to be selected independently for the down-line and across-line directions. The interpolations available are linear, cubic spline (minimum curvature) or Akima spline.

Filtering of the line data before interpolation is also possible. Bi-Directional gridding can design and apply non-linear and/or linear numerical filters to the original line data. The use of the non-linear filter is a very effective way to remove data spikes (undesired high-amplitude short-wavelength features) from the original data.

Bi-Directional gridding has the following features:

- Unlimited number of data points per survey line
- Unlimited number of survey lines
- Unlimited output grid size



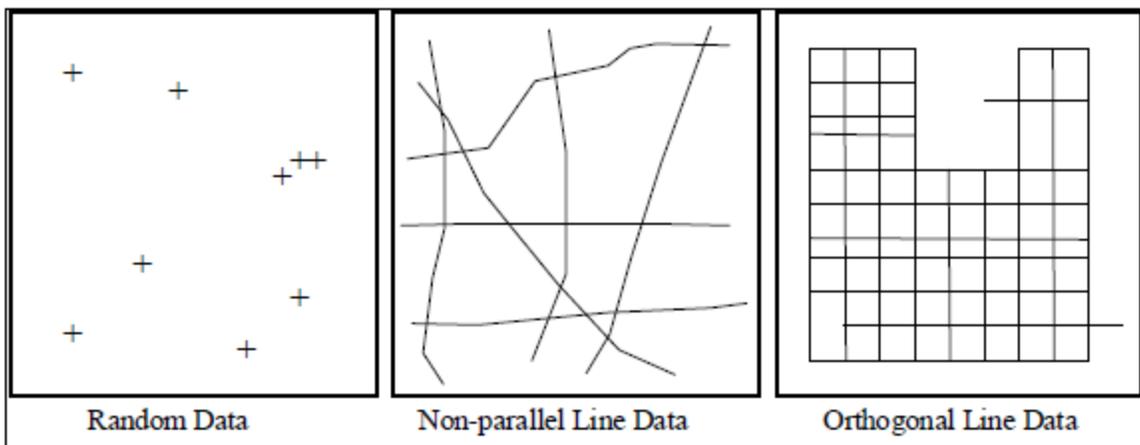
The actual maximum size of the dataset or grid that you can use depends upon the available free disk space of your computer.

Kriging

Kriging is a statistical gridding technique for random data, non-parallel line data or orthogonal line data.

Use Minimum Curvature gridding or Kriging when the XYZ data is not sampled along lines that run in roughly the same direction. Such data are often called random, because they give a random appearance when the data locations are plotted. Also, line data with survey lines that are orthogonal (or have random directions) should be gridded with Minimum Curvature gridding or Kriging.

The following figure illustrates these types of data:



If the data is relatively smooth between sample points or survey lines, then Minimum Curvature gridding should be used.

However, use Kriging, if the data is:

- Highly variable between sample locations
- Known to be statistical in nature (such as geochemical data)

- Poorly sampled
- Clustered

The Kriging statistical gridding method determines a value at each grid node based on the XYZ data. Kriging first calculates a variogram of the data, which shows the correlation of the data as a function of distance.

Simply speaking, the further data points become, the less correlation we expect between points. A variogram shows this phenomena for a given dataset, and based on the variogram, you are able to select a model that best defines the variance of the data. The technique of Kriging uses this model to estimate the data values at the nodes of the grid. A significant by-product of Kriging is the ability to estimate the error of the data at each grid node.

Geosoft supports ordinary Kriging as well as universal Kriging. Universal Kriging differs from ordinary Kriging in that it honours any regional trends that may be present in a dataset. For a more in-depth understanding of geostatistical analysis and Kriging, refer to Mining Geostatistics written by A.G. Journel and C.J. Huijbregts, 1978.

Kriging has the following capabilities:

- Applies de-aliasing filter
- Applies linear and logarithmic gridding options
- Blanks un-sampled areas
- Calculates a variogram from the input data channel
- Output grids up to any size
- Processes unlimited number of input observation points
- Supports linear, power, spherical, Gaussian, exponential and user-defined models



The actual maximum size of the grid that you can use depends upon the available free disk space of your computer.

Anisotropic Kriging

You can create a grid enhanced in a preferred direction using the strike direction and a weighting factor. This method enables you to incorporate the dominant geologic strike and define a preferential weighting in the strike direction, enhancing geological trends in your dataset.

Tinning

The ability to create a TIN (Triangular Irregular Network), and to use this TIN file to grid data using the Nearest Neighbour, Linear or Natural Neighbour methods is included in the Oasis montaj environment.

The TIN is created from a set of spatial data using the public domain *Sweep* algorithm implemented by Steven Fortune of Bell Laboratories (Fortune, S 1987). The TINDB GX applies the *Sweep* algorithm to the X, Y (Z-optional) data values in a Geosoft database (*.gdb) to create a binary TIN (*.tin) file.

When Z values are included in the (*.tin) file, a TIN grid can be created using the TINGRID GX. The TINGRID GX applies the *Nearest Neighbour, Linear or Natural Neighbour* (Sambridge, Brown & McQueen 1995) algorithm to the Z values in the (*.tin) file to create a grid.

The TIN gridding method requires one data point for each (X, Y) data location in the database. Tinning provides the ability to sum or average duplicate samples - data that have multiple Z values at single point locations.



When Z values are included in the (*.tin) file, only data point locations with non-dummy Z values are included.

Geosoft Tinning provides a number of ways of visualizing the TIN, including the ability to plot the TIN Nodes, the TIN Mesh (or Delaunay triangulation), the Convex Hull, and the Voronoi cells of your data.

Inverse Distance Weighting Gridding

The Inverse Distance Weighting (IDW) algorithm is a moving-average interpolation algorithm that is usually applied to highly variable data. For certain types of data (e.g. soil geochemistry, surface/groundwater chemistry) it is possible to return to an existing measurement site and record a measurement that is statistically different from the original, but within the general trend of all measurements within the area. In this case it is not usually desirable to honour local data minima or maxima, but instead to look at a moving average of surrounding data points and estimate local trends."

"IDW calculates a value for each grid node by examining surrounding data points that lie within a user-defined search radius. The node value is calculated by averaging the weighted sum of all the points, where the weighting inversely corresponds to distance from the grid node.

Anti-aliasing

Prior to gridding (and after any log or log-linear transformation), data is pre-processed using an anti-aliasing technique. All values falling inside any single grid cell are averaged, and the data is then represented by the single averaged value at the grid cell centre. Any error in the spatial representation of features introduced by this step will never exceed one-quarter of the Nyquist wavelength, which is equal to 2 cell sizes.

The Inverse Distance Weighting Function: Defining the Weighting Slope and Power

The inverse distance weighting function assigns averaging weights based on distance, out to the defined search radius:

$$1 / (\text{distance}^{\text{power}} + 1/\text{slope})$$

Where distance is in multiples of the horizontal cell size.

Using the default of 2 for the power, and 1 for the slope produces a bell-shaped weighting function. Requiring a slope > 0 ensures that the weight remains finite at zero distance. Decreasing the slope tends to flatten the bell, resulting in greater weighting of points away from the grid cell, and hence greater smoothing. Choosing a power less than 2, or a slope less than one, may result in over-smoothing the data.

The following table shows the effect of various slopes on the weighting given at various distances away from the centre cell. The weights have been normalized so the weight at the cell centre is equal to 1.

	weighting of centre cell	weighting 1 cell away	weighting 2 cells away	weighting 3 cells away	weighting 4 cells away
P = 2, S = 0.2	1	0.83	0.55	0.36	0.24
P = 2, S = 0.5	1	0.67	0.33	0.18	0.11
P = 2, S = 1	1	0.5	0.2	0.1	0.056
P = 2, S = 2	1	0.33	0.11	0.053	0.03

Clearly, as the slope increases, the weighting is more tightly concentrated about the centre cell. The search radius should also be chosen based on the fall-off of the weighting function. Increasing the search radius beyond where the weighting function is significant will have little effect on the results, and may result in large increases in processing time, since the processing time varies in proportion to the cube of the search radius. (Remember that the search radius is specified in ground units, not as a multiple of cell sizes.)

Direct Gridding

You can create a grid from highly sampled data without using any interpolation. This method is intended for use with over-sampled datasets such as LiDAR; it provides a quick gridded view of the datasets.

Sample Density Gridding

You can create a grid of the density of sample locations, rather than the value recorded at each location. This method is useful for determining the spatial distribution of highly sampled data and enables you to easily identify areas which are under-sampled.

Trend Enforcement Gridding

A traditional problem in gridding airborne geophysical data is the aliasing problem that occurs due to the presence of more samples “along survey acquisition lines” than across lines. This problem leads to obvious effects including ellipsoids or ellipsoidal “beads” between lines in gridded data. There are various solutions to these types of effects including filtering and gridding along trend directions. However, these techniques may reduce geologic signal or obscure local trends. The Oasis montaj “Trend Enforcement algorithm” is designed to provide a solution that preserves the character of local trends while eliminating aliasing effects.



*For additional information, you can click the **Help** button on any of the gridding dialogs.*

Gridding using the Bi-Directional Gridding Method

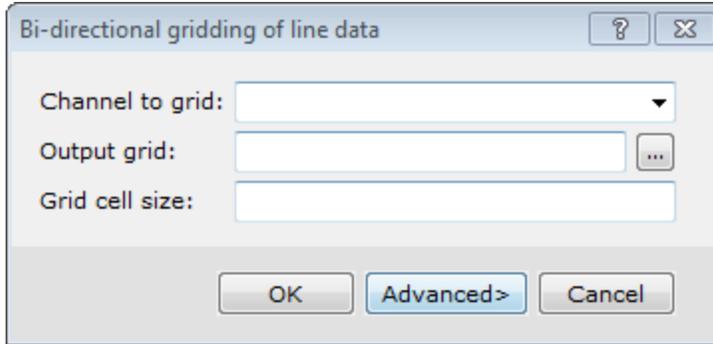
In this example, we use bi-directional gridding since the data are oriented in parallel lines and we expect to have cross-line trending. You can apply Bi-directional gridding by using the *Bi-Directional Line Gridding* menu option or running a Control File that contains the gridding commands and parameters. The procedures for both methods are provided. However, you will only need to follow one of the methods to perform the Bi-directional gridding successfully. If you are a new user, we recommend choosing the first method.



*The **Advanced** button enables you to specify advanced bi-directional gridding options.*

To Apply Bi-Directional Gridding to XYZ Data

1. Select the database.
2. On the *Grid and Image* menu, select *Gridding* and then select **Bi-Directional Line Gridding**. The *Bi-directional gridding of line data* dialog appears.



3. Select the *Channel to grid* from the dropdown list.
4. Browse to the *Output grid* location and provide a name. Note that the file extension (*.grd) will be appended by the system.
5. The *Grid cell size* should be 1/4 to 1/8 of the line separation distance.



The cell size is the distance between grid points in the X and Y directions. In most situations the cell size can be selected as 1/4 to 1/8 of the line separation, or the minimum station interval, whichever is greater. By "default" (i.e. if you leave the cell size field blank) BIGRID will choose a cell size based upon these criteria, although we do recommend that you choose the cell size directly.

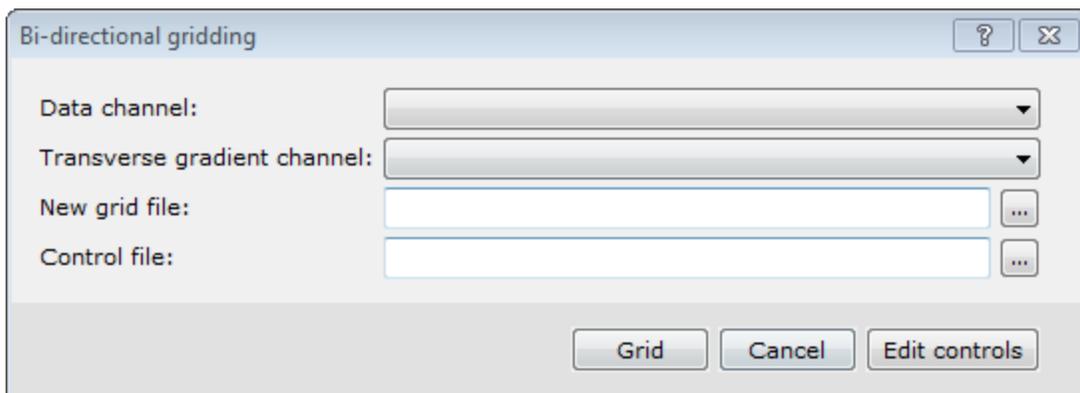
6. Click **OK**. The data is gridded and displayed in your current project.

To Apply Bi-Directional Gridding to XYZ Data Using a Control File

In the example below, we assume a control file exists. If no BIGRID control file exists, click **Edit Control** from the dialog and confirm the default text editor (such as notepad.exe). The new control file opens in the text editor with comments that explain how to use the **BIGRID** parameters. The rest of the control file is blank.

1. On the *Grid and Image* menu, select *Gridding*, then select *From Control File* and then select **Bi-Directional Control File**.

The *Bi-directional gridding* dialog appears.



2. Use the *Data channel* dropdown list to select the channel to grid.
3. If provided, use the *Transverse gradient channel* dropdown list to select a gradient channel.
4. Specify a name for the *New grid file*.

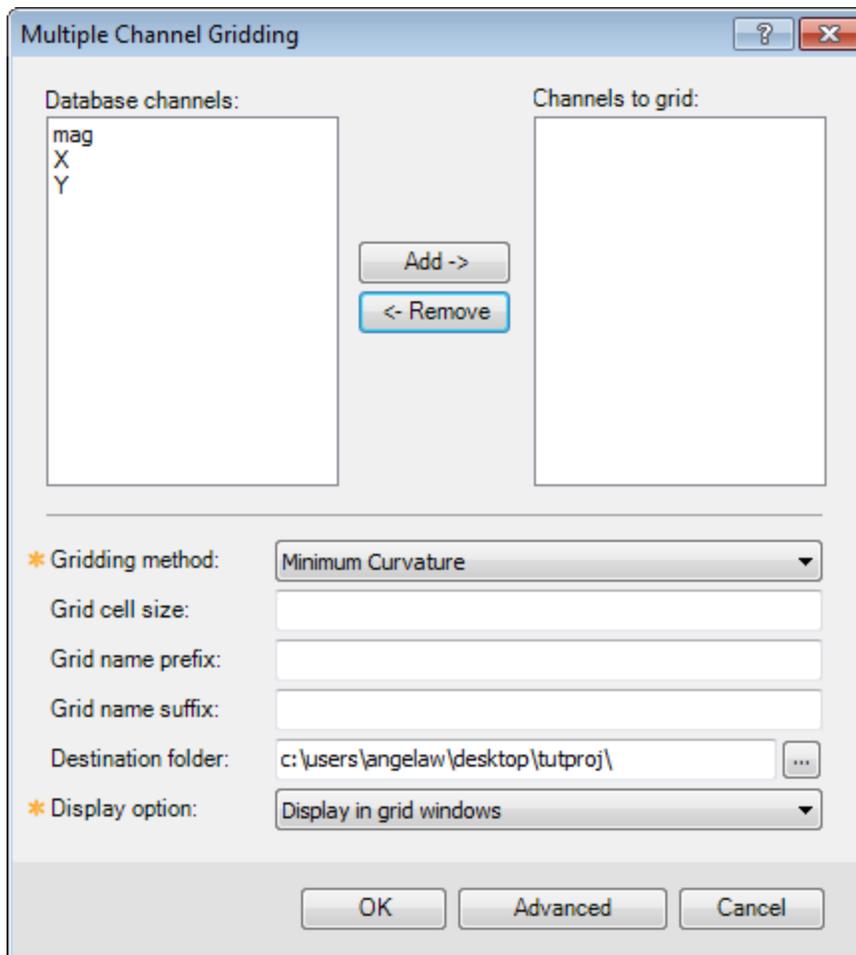
5. Browse to the *Control file*.
6. Click **Grid** to grid the data and display it in your current project.

Multiple Channel Gridding

Use *Multiple Channel Gridding* to grid multiple channels using the same gridding method. You can now, very efficiently, create grids of all of your selected data. The *Grid Multiple Channels* menu displays the *Multiple Channel Gridding* dialog. Use this dialog to select multiple data channels to grid. This can be used with Minimum Curvature and Bi-Directional gridding methods.

To Grid Multiple Channels

1. Open and select a database with multiple channels.
2. On the *Grid and Image* menu, select *Gridding*, then select **Grid Multiple Channels**. The *Multiple Channel Gridding* dialog appears



3. Select (highlight) the *Database channels* you want to grid.
4. Click **Add ->** to add the selected channels to the *Channels to grid* box.
5. From the *Gridding method* dropdown list, select the method.

► **Displaying a Grid**



The **Advanced** button enables you to define settings for the Gridding method (Bi-Directional or Minimum Curvature) selected.

6. Specify a *Grid cell size* if desired.
7. Specify a *Grid name prefix and suffix*, if desired.
8. Confirm the *Destination folder*.
9. From the dropdown list, select the *Display option*. You can display the grid in the current map, a grid window, or choose no display.
10. Click **OK**. The grids are created and appear under *Grids* in the Project Explorer.

Displaying a Grid

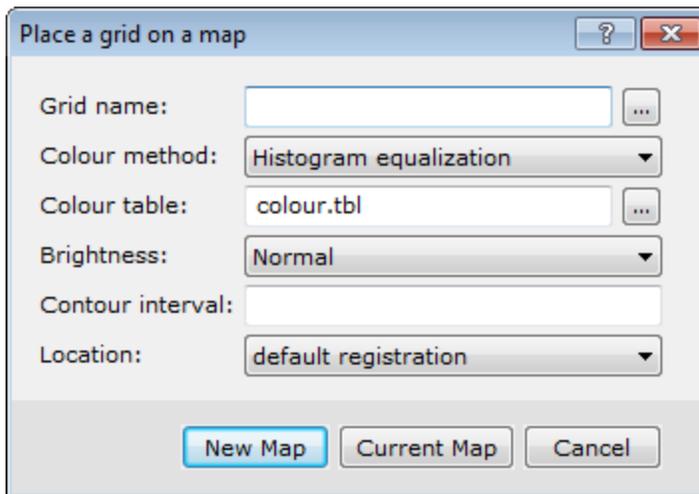
There are a number of options for displaying grids; these include displaying single grids, colour shaded grids, ternary image grids, two-grid composites (for shaded-relief maps), three-grid composites (for ternary maps) and four-grid composites. In this example, we will display a single grid.

Grids appear in the Project Explorer under "Grids". Opening a grid opens a view of the grid in its own internal "map". A grid map view is just like a real map and can accept any graphics that a map can. It can also be saved as a map, in which case the map will appear in the "Map" section of the project explorer.

When displaying grids, you must specify a colour table to use (supplied with the product). You may want to experiment with different colour tables to see how they affect your image.

To Display a Single Grid on a Map

1. Open a map.
2. On the *Grid and Image* menu, select *Display* and then select **Single Grid**. The *Place a grid on a map* dialog appears



3. Browse to select the *Grid name*.
4. From the dropdown list, select the *Colour method*.



The 'As last displayed' method will display each grid as it was displayed the last time the grid was viewed and the colour tables will be ignored.

5. Browse to select the *Colour table*.

6. From the dropdown list, select the *Brightness* level.
7. Specify the *Contour interval*, if required.
8. Select the *Location*. You may choose *default registration* or *fit to an area*.



If you choose 'fit to an area', you will be asked to define an area using the mouse.

9. Click **Current Map** to insert the grid on the open map. The grid is displayed on the map.



*If you click **New Map**, a new map is created with the same name as the selected grid.*

Recolour Grids

Click the **Auto Recolour Grids** button on the *Map Tools* bar to toggle the *Auto Recolour Grids* option on/off. This option recolours the displayed grid area as the grid extents change within a map window (e.g. when zooming or panning).



When turned ON the grid values stay the same. However, as the viewed extents of the grid change; the grid colour is recalculated and redrawn using the entire colour palette over the current displayed grid area. When Auto Recolour Grids is OFF the grids return to their original colours.

The *Auto Recolour Grids* option is a temporary viewing tool. To save/share a recoloured view of a grid you can Create a Snapshot or print the current map extents.



If the Auto Recolour Grid mode is turned ON when you print, it will apply to the printed page. Either turn OFF the mode when printing or print out a selected area and it will apply the recolouring.

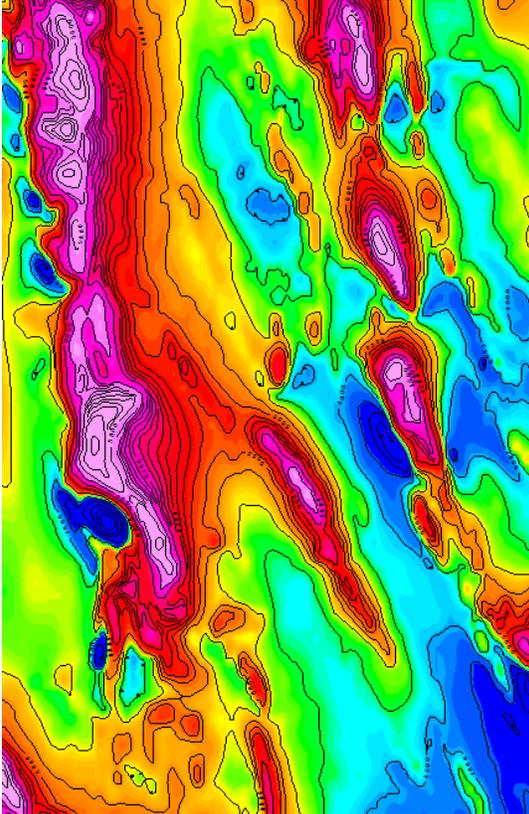
Contouring

Geosoft's basic contouring methodology is to thread contour lines through constant levels defined in a gridded data file. Gridded data files are usually created by one of the Geosoft gridding programs, or may be imported from other software systems that support the Geosoft Exchange Format (*.gxf). The grid file can contain any number of points. Contours are added directly to the map you are working with.

It is important that the grid cell size (the distance between nodes of the grid) is appropriate for contouring. We recommend that you use a cell size that is 5 mm at map scale when plotted. Grids, with cell sizes less than 1 mm, will require more processing time than is necessary to make a good contour map, while grid cells larger than 3 mm can produce contours that appear jagged. If the grid contains important information that is only honoured at cell sizes less than 1 mm, you should probably not be contouring the data, or you should be plotting the data at a smaller scale (make the same area on the ground larger on paper).

Plotting Contours

Contours refer to the lines drawn at specified intervals or multiples of intervals on your map. An example is shown below:



After you create a grid, you may want to contour your data and display these contours on your map. In Oasis montaj, contouring is specially designed to handle the very large dynamic data ranges that characterize Earth Science data sets. For more information on plotting contours, refer to the *Adding Contours to Maps* topic in the Application Help files.

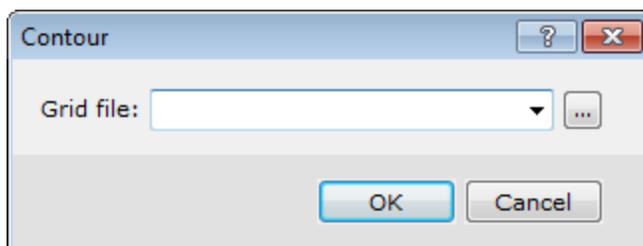
Contouring options include quick, custom, log and from a control file (for batch contouring).

- **Quick** - uses default parameters.
- **Custom** - uses parameters you specify the system to use. This method generates simple or log contours.
- **Log contour** - draws contours on a map using a specified grid in logarithmic mode
- **Have Control File** - ASCII control file enables full cartographic cosmetic control of plotting options

To Quick Contour a Grid

In this example, we will use the automatic (quick) method.

1. Make sure the desired grid is open and selected in the *Project Explorer*.
2. On the *Map Tools* menu, select *Contour* and then select **Quick**.
The *Contour* dialog appears.



3. Browse to the *Grid file* you want to contour. You may be asked to confirm the scale.
4. Click **OK**. The contours are drawn on the map.

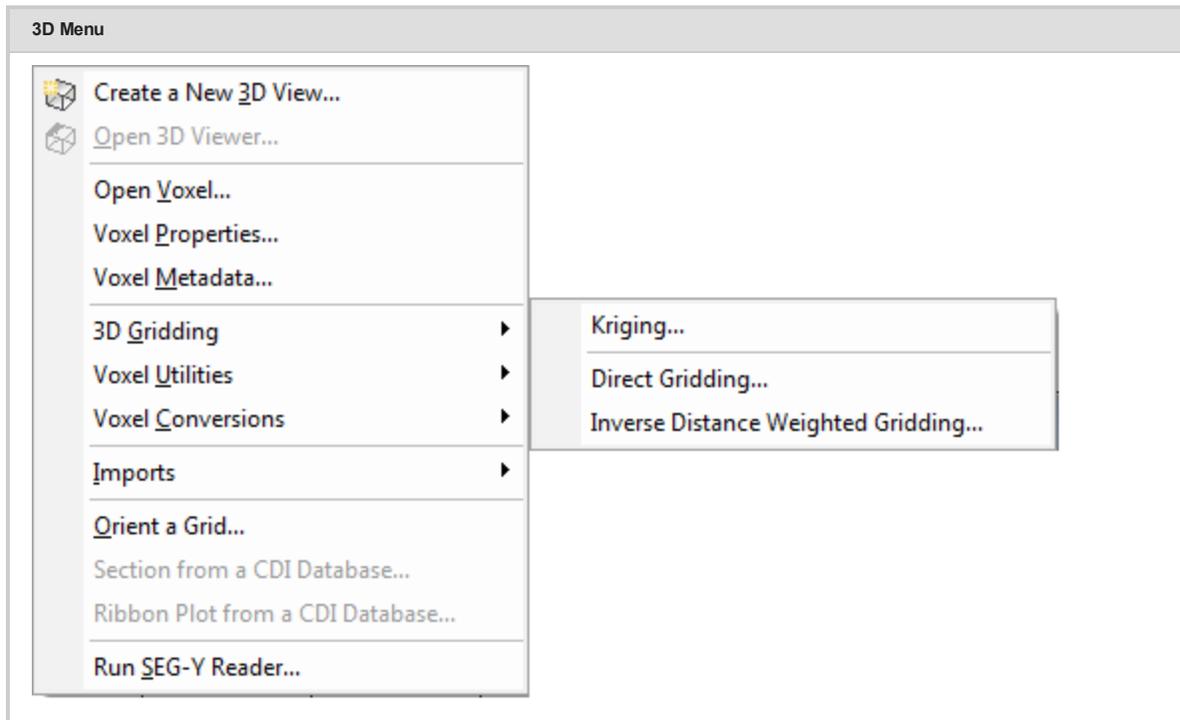
3D Gridding Methods in Oasis montaj

A "Voxel" is short for "volume pixel", the smallest distinguishable box-shaped element of a three-dimensional image. A Voxel is the 3D conceptual counterpart of the 2D pixel.

You can create a Voxel using the *3D Gridding* submenu, located on the *3D* menu or in the 3D Viewer from the *Voxel|3D Gridding* menu.

The *3D Gridding* submenu offers 3 different gridding algorithms, each with a different focus:

- The *Kriging* method best suits clustered data occurring in a region with otherwise sparse data distribution, such as geochemistry data.
- The *Direct Gridding* method on the other hand should be used to grid data uniformly and densely sampled throughout the region, such as GPR or seismic data.
- The *Inverse Distance Weighted Gridding* method is most suitable for gridding data collected along specific directions - such as borehole data.



The data to be gridded must have defined X, Y and Z coordinates. To define the coordinates use the menu, *Coordinates|Set Current X,Y,Z Coordinates*. The 3 dimensional coordinate system will locate the data in 3D space. The Z axis is interpreted as positive up. The **3D** workflow includes tools for creating, displaying and processing voxels. The *3D Viewer* enables visual inspection of voxels and comes with an extensive set of 3D viewing tools.



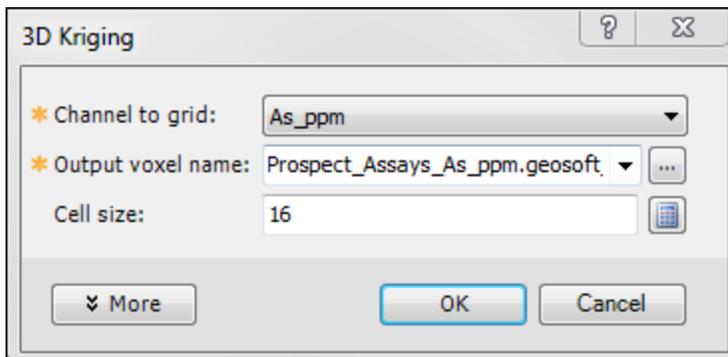
When applying 3D gridding, halving the cell size increases both the file size and processing time by a factor of 8 (2³).

Creating a Voxel using the Kriging Gridding method

From the *3D Gridding* menu, select the **Kriging** option to perform a statistical-based 3D gridding. The input data must be supplied in a Geosoft database (GDB file). The output will be a voxel model within the confines of the data extent.

To Create a 3D Voxel model using Kriging:

1. Open a Geosoft database containing X, Y, Z and the data channel you would like to grid. If the coordinate system is not completely defined, a message will pop up and you will not be able to continue until the corrective action is taken.
2. In the 3D Viewer, select *Voxel*, then select *3D Gridding* and then select **Kriging** (or on the main Oasis montaj *3D* menu, select *3D Gridding* and then select *Kriging*. The *3D Kriging* dialog appears.



If a database is not already open you will be prompted to open one.

3. From the dropdown list, select the *Channel to grid*. The first time through the process, the default *Output voxel file* name will default to the provided channel name. The subsequent invocations of this tool will remember the user specified file name. You can rename the output voxel file or accept the default name.
4. Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button to obtain a reasonable cell size based on the extents of your data. The cell size is calculated for a single cube, assuming the volume is divided into 10000 individual cubes. The value is then rounded down to give a more satisfactory value.



We recommend that your first attempt at gridding be using the defaults. After inspecting the outcome however you may decide to modify defaults such as the Blanking Distance to fill in the gaps, modify the Z cell increment, or generate the output on a logarithmic scale.

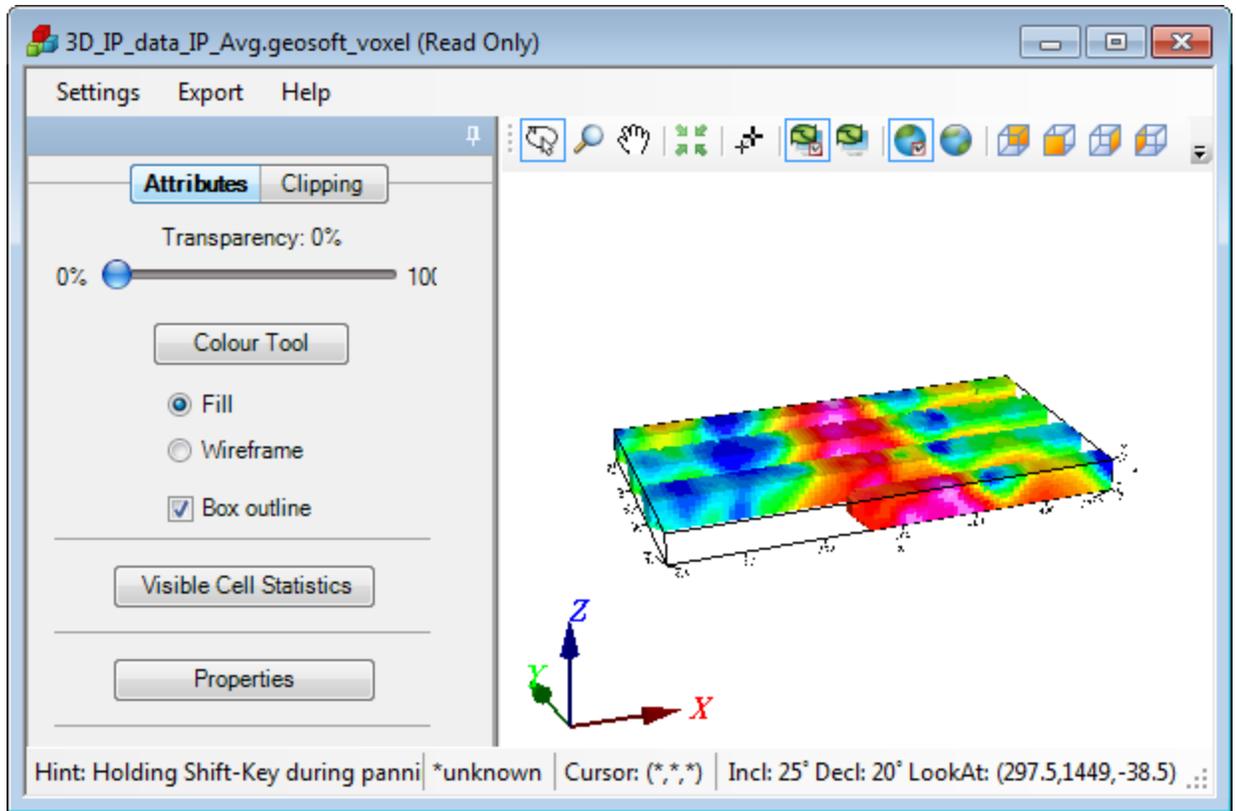


Additionally you can customize the variogram to best suit your data.



For more information, click the Help button on the 3D Kriging dialog.

5. When you are satisfied with your selections click **OK**. The Voxel is created and displayed in the *Voxel Viewer*. An example is shown below:

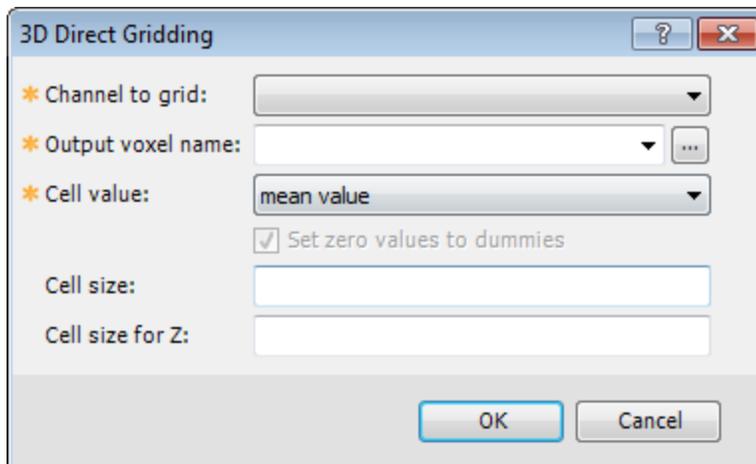


The *Voxel Viewer* provides additional viewing tools.

Creating a Voxel Model using the Direct Gridding method

To Create a 3D Voxel model using Direct gridding:

1. Open a Geosoft data database containing X, Y, Z and the array data channel to grid. If the coordinate system is not completely defined, a message will pop up and you will not be able to continue.
2. In the 3D Viewer, select *Voxel*, then select *3D Gridding* and then select **Direct Gridding** (or on the main Oasis montaj *3D* menu, select *3D Gridding* and then select *Direct Gridding*. The *3D Direct Gridding* dialog appears.





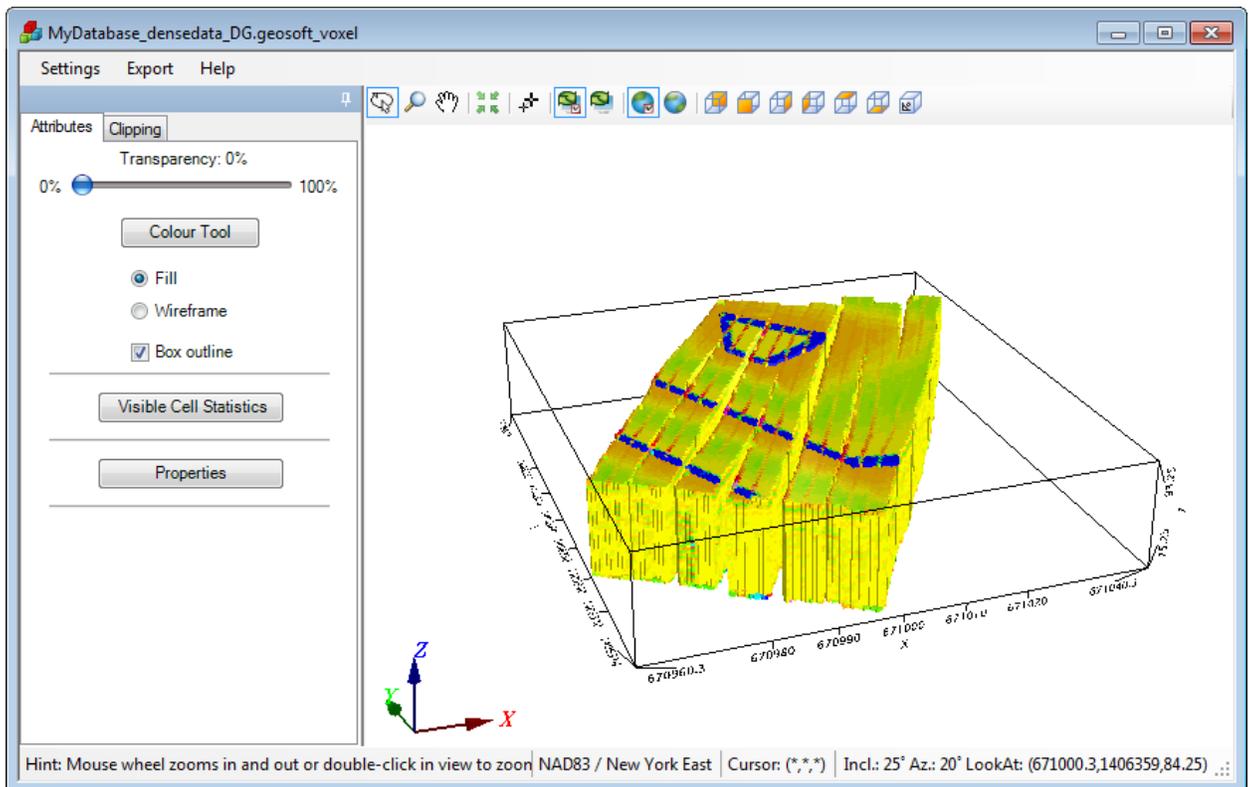
If a database is not already open you will be prompted to open one.

3. From the dropdown list, select the *Channel to grid*. Only array channels are eligible to be gridded. The first time through the process, the default *Output voxel file* name will be set to a combination of the current database and provided channel name. The subsequent invocations of this tool will remember the user specified file name. You can rename the output voxel file or accept the default name.
4. Specify the *Cell value*. You can choose the *minimum value* ; *maximum value* *mean value* or *number of items in a cell*
5. Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button, to obtain a reasonable cell size based on the extents of your data. The cell size is calculated for a single cube, assuming the volume is divided into 10000 individual cubes. The value is then rounded down to give a more satisfactory value.



For more information, click the **Help** button on the *3D Direct Gridding Help* topic.

6. The default vertical Z cell size is set to the same value as the horizontal cell. If this value is not reflecting the desired resolution, modify it accordingly.
7. When you are satisfied with your selections click **OK**. The Voxel is created and displayed in the *Voxel Viewer*. An example is shown below.



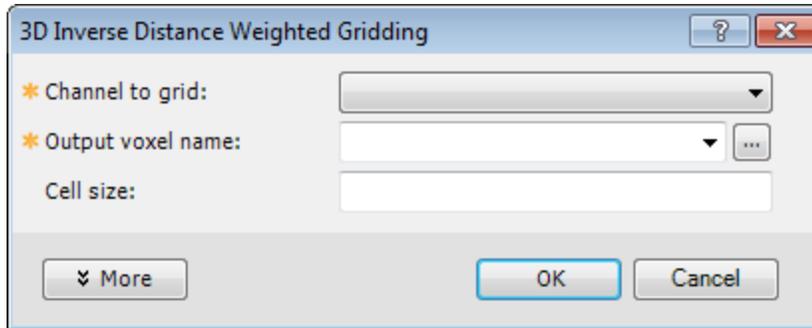
Direct Gridding of GPR data. Data courtesy of UIT

Creating a Voxel Model using the Inverse Distance Weighted (IDW) Gridding method

To Create a 3D Voxel model using Inverse Distance Weighted Gridding:

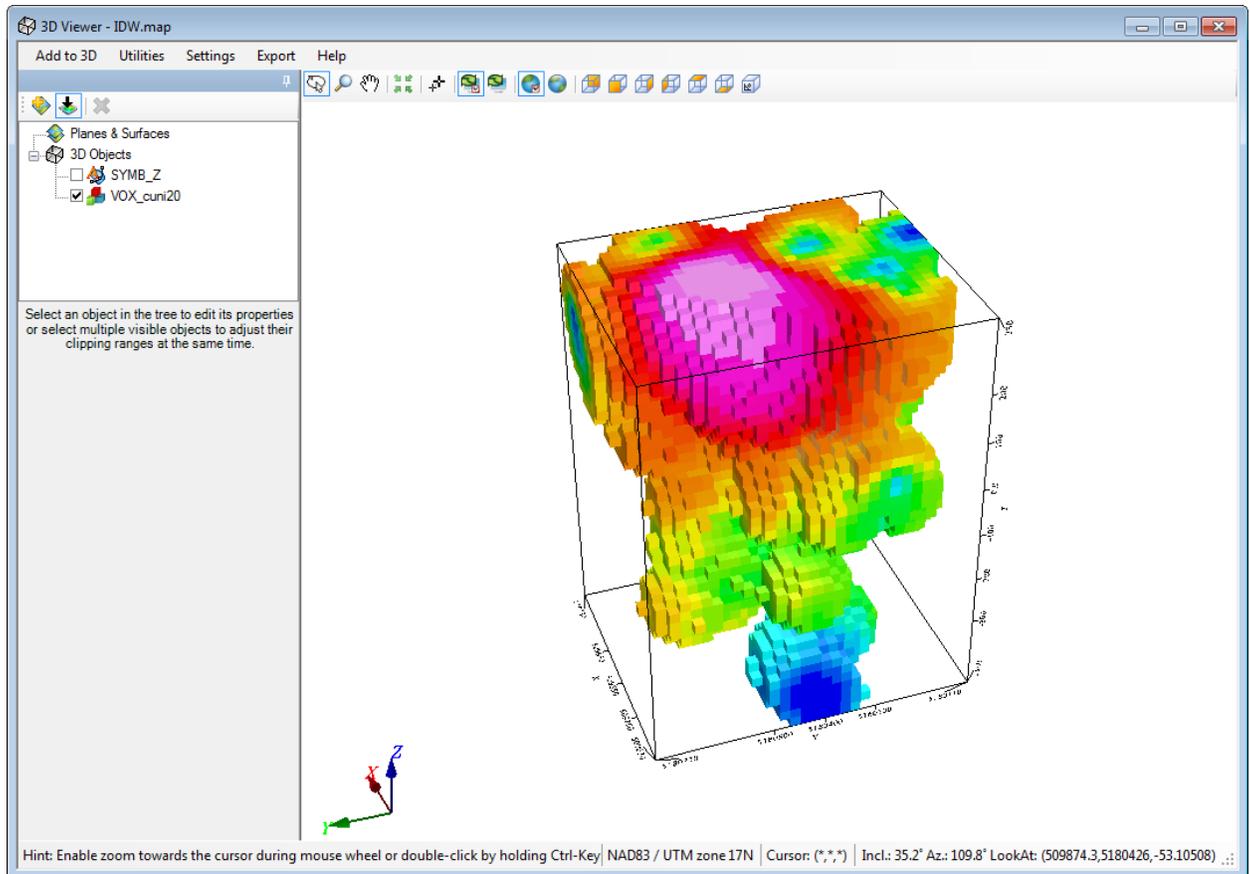
1. Open a Geosoft database containing X, Y, Z and the array data channel to grid. If the coordinate system is not completely defined, a message will pop up and you will not be able to continue.
2. In the 3D Viewer, select *Voxel*, then select *3D Gridding* and then select **Inverse Distance Weighted Gridding** (or on the main Oasis montaj *3D* menu, select *3D Gridding* and then select, *Inverse Distance Weighted Gridding*).

The *3D Inverse Distance Weighted Gridding* dialog appears.



If a database is not already open you will be prompted to open one.

3. From the dropdown list, select the *Channel to grid*. The first time through the process, the default *Output voxel file* name will default to a combination of the current database and provided channel name. The subsequent invocations of this tool will remember the user specified file name. You can rename the output voxel file or accept the default name.
4. Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button, to obtain a reasonable cell size based on the extents of your data.
5. When you are satisfied with your selections click **OK**. The Voxel is created and displayed in the *Voxel Viewer*. An example is shown below.



Inverse Distance Weighted Gridding of Borehole data



We recommend that your first attempt at gridding be using the defaults. After inspecting the outcome however you may decide to modify defaults such as the Weighting power & slope to alter the degree of influence of contributing data points relative to their distance , or the Blanking Distance to fill in the gaps, or generate the output on logarithmic scale.

- Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button, to obtain a reasonable cell size based on the extents of your data. The cell size is calculated for a single cube, assuming the volume is divided into 10000 individual cubes. The value is then rounded down to give a more satisfactory value.



For more information, click the Help button on the 3D Inverse Distance Weighted Gridding Help topic.

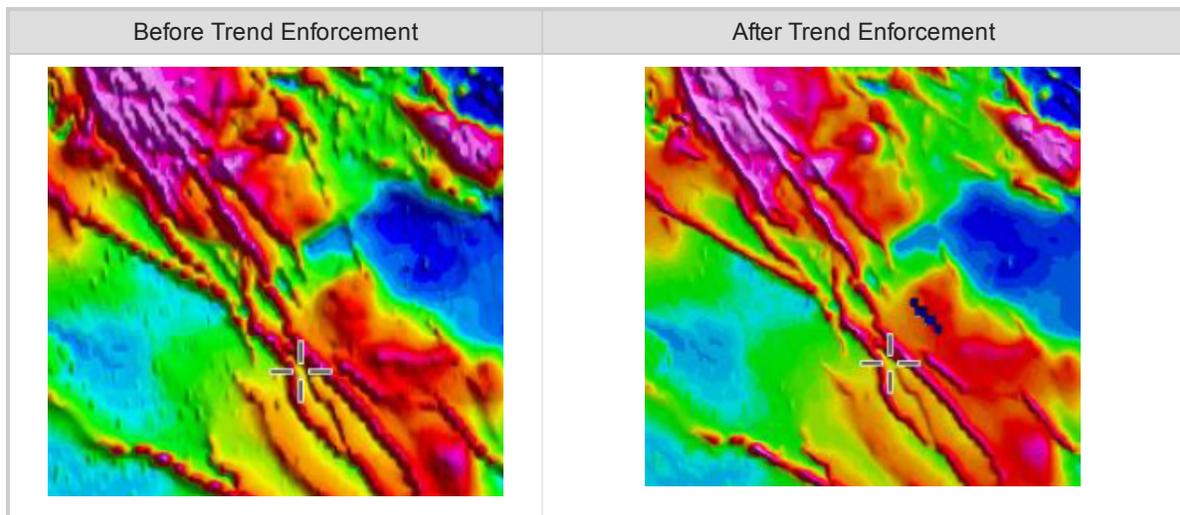
How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Trend Enforcement Gridding

This How-To Guide demonstrates how you can use the **Trend Enforcement Gridding Tool** to alleviate a traditional problem in gridding geophysical data, that is, the aliasing problem that occurs due to the presence of more samples “along the lines” than across lines. This problem leads to obvious effects including ellipsoids or ellipsoidal “beads” between lines in gridded data. There are various solutions to these types of effects including filtering and gridding along trend directions. However, these techniques may reduce geologic signal or obscure local trends.

A traditional solution is to add a trend line. This method works as long as you match high values and low values. The Trend Enforcement algorithm is designed to perform this process automatically – smoothly joining highs to highs, and lows to lows. The algorithm also makes sure that the highs and lows do not cross (i.e. backtrack).



The **Trend Enforcement Gridding Tool** algorithm is designed to provide a solution that preserves the character of local trends while eliminating aliasing effects. This guide describes how to:

- Access the Trend enforcement menus
- Grid your data using Bi-Directional Line gridding
- Display colour-shaded grid
- Find trends with the Trend Enforcement Tool
- Access trend line channels in the database
- Plot trends
- Add your own trends
- Grid trend enforced data
- Deselect/select trend lines

Accessing the Trend Enforcement Menus

The *Trend Enforcement* menu can be accessed from the *Grid and Image|Gridding* menu on the Oasis montaj main menu bar.

To Access the Trend Enforcement Menu

1. On the *Grid and Image* menu, select *Gridding* and then select **Trend Enforcement**.
The *Trend Enforcement* menu appears

Find Trends...	(trenddb.gx)
Plot Trends...	(trndpath.gx)
Add Trend Line...	(trnddigi.gx)

These menu items (GXs) enable you to eliminate aliasing effects while preserving the character of local trends.

Finding Trends

A traditional problem in gridding geophysical data is the aliasing problem that occurs due to the presence of more samples “along the lines” than “across lines”. This leads to obvious effects including ellipsoids or ellipsoidal “beads” between lines in gridded data.

To Find Trends

1. Open the database in which you want to find the trends.
2. On the *Grid and Image* menu, select *Gridding*, and then select *Trend Enforcement*, and then select **Find Trends**.

The *Find trend lines in a database* dialog appears.

Find trend lines in a database

Data channel for trends:

Window for Max-Min Search:

Preferred angle for trends:

Allowable deviation in angle:

Maximum length for joins:

Maximum deflection for joins:

Minimum length for trend line:

Resampling interval along trends:

Trend line breaking angle:

Output to database:

OK Cancel

3. Use the *Data channel for trends* dropdown list, to locate the channel in which to locate the trends.
4. In the *Window for Max-Min Search* field select the max-min value on either side of the selected point. The default is 100.
5. Specify the *Preferred angle for trends*, measured counter-clockwise from the horizontal (X-axis) in degrees. The default is 30.
6. Set the *Maximum length for joins*. This controls the length of the joins. The default is 500.
7. Set the *Maximum deflection for joins*. This controls the local straightness of the joins. If the field is blank, no check will be made for this condition.
8. Enter a value for *Minimum length of trend line* if you wish to ignore all trend lines shorter than the specified value. This test is applied after any resampling and before breaking of lines at the breaking angle. If the field is blank, this test is not performed.
9. Set the *Resampling interval along trends* field. The line may be splined at a fixed interval. For maximum effectiveness set this value to a fraction of the average line spacing. If the field is blank, this test is not performed.
10. Set the *Trend line breaking angle*, if appropriate. If the field is left blank, this test is not performed.
11. Select to *Overwrite* or *Append* the *Output to database*.



For more information on each of the fields in the "Find trend lines in a database" dialog, click the Help button.

12. When ready to continue, click **OK**. The system calculates the *Trend lines* and places this information in your database.



*When using **Minimum Curvature Gridding** as your gridding technique, you must specify a value in the "Re-sampling interval along trends" parameter. If no value is entered, the **Trend Enforcement Tool** will not calculate the appropriate trend lines.*

Accessing Trend Lines in the Database

You may want to review your trend lines before you plot them on a map or re-grid your data using the trend enforced data.

To Access Trend Lines in a Database

1. From your database, right-click the *Line Header Cell* (upper left cell).
2. From the popup menu select *List*. Another popup menu appears containing a list of the lines in the database.
3. Scroll down the list and select the line which contains the trend lines (usually labelled as R0, R1, etc.). The system displays the trend lines in the database.



When you click on a data value (X, Y or Z) in the database, you will notice that the linked map cursor moves to the selected location of the trend line on the map.

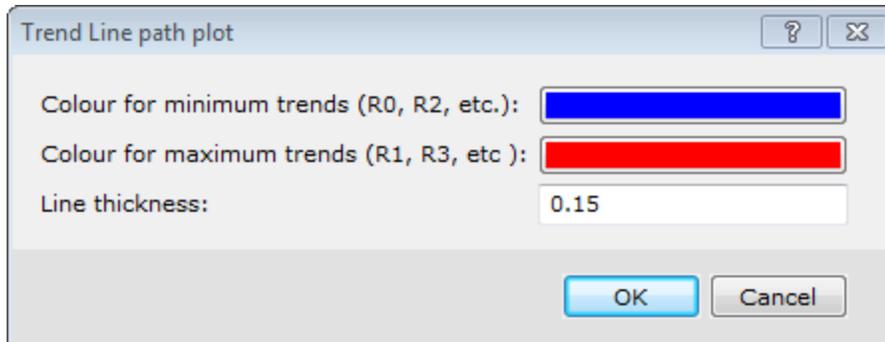
Plotting Trend Lines

The **Trend Enforcement Tool** displays the trend lines found in the current database.

To Plot Trend Lines

1. On the *Grid and Image* menu, select *Gridding*, then select *Trend Enforcement*, and then select **Plot Trends**.

[The *Trend Line path plot* dialog appears.

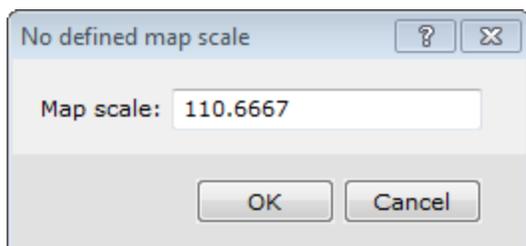


2. Use the **Colour for minimum trends (R0, R2, etc)** and the **Colour for maximum trends (R1, R3, etc)** boxes to select the colours to use to display the trend lines on the map.



Red and Blue are the default as these colours are the opposite of the gridding convention and show up clearly on the map

3. Set the *Line thickness*. The default is 0.15.
4. Click **OK**. The *No defined map scale* dialog appears.



5. Set the *Map scale*. The displayed default scale is chosen to fit the map to an area approximately 30x30 cm when plotted. You can adjust the map scale to a more appropriate value, for example 1:120.



This dialog is displayed whenever drawing to a map for which a map scale has not been defined. This situation most commonly occurs when a map has been opened just to display a grid (as applies here). A map scale is required to determine appropriate scaling for drawing text, patterned lines, and other map graphic details.

6. Click **OK**. The trend lines are plotted on the current map.

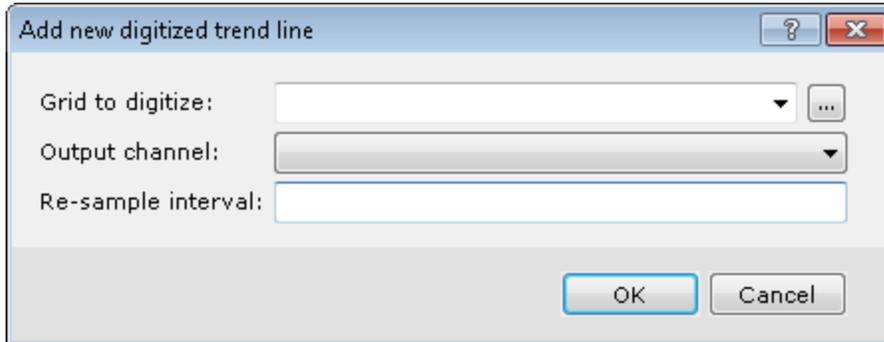
Adding Your Own Trend Lines

The **Trend Enforcement Tool** may not find all of the trend lines in your database. You can interactively digitise a trend line from a grid and add it to the end of a line in the current database. You also have the ability to resample the trend line to a specified sample interval (necessary when using the random gridding techniques).

To Add Your Own Trend Lines

1. Ensure that the database you want to add the trend lines to, is open in your current project.
2. On the *Grid and Image* menu, select *Gridding*, then select *Trend Enforcement*, and then select **Add Trend Line**.

The *Add new digitised trend line* dialog appears.

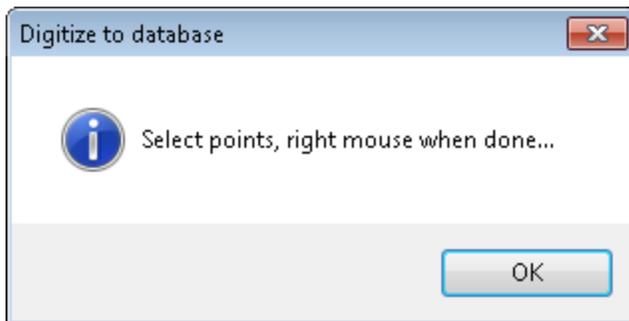


3. Use the *Browse* button to locate the *Grid to digitise*.
4. From the *Output channel* dropdown list, select the channel to place the digitised values from the grid. This should be the same data channel that will be gridded.
5. In the *Re-sample interval* field specify an interval at which to resample the data along the defined trend line.



It is recommended that the trend line be re-sampled at the same cell size as the grid.

6. Click **OK** to continue. The *Digitise to database* dialog box appears.



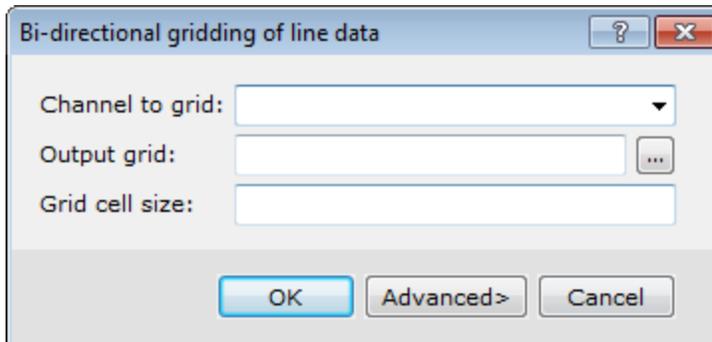
7. Click **OK** to begin digitizing.
8. Draw a *Trend Line* by selecting points (either minimum or maximum point values) on the displayed grid.
9. When you are finished with each line, right-click and select **Done**. The digitised trend line data values, which have been interpolated to the specified re-sample interval, will be added to the trend line(s) in the current database.

Gridding Trend Enforced Data

You can re-grid your data, using the same parameters as the original grid. This can be done to view the two grids side-by-side for comparison.

To Apply Bi-Directional Gridding to Trend Enforced Data

1. On the *Grid and Image* menu, select *Gridding*, and then select **Bi-Directional Line Gridding**. The *Bi-directional gridding of line data* dialog appears.



2. Use the dropdown list to select the *Channel to grid*.
3. Enter the name of the *Output grid*. This name should be different than the original grid.
4. Specify the *Grid cell size*. The cell size is the distance between grid points in the X and Y directions. In most situations the cell size can be selected as 1/4 to 1/8 of the line separation, or the minimum station interval, whichever is greater.



By default, BIGRID will choose a cell size based upon these criteria, but we recommend that you choose the cell size directly.

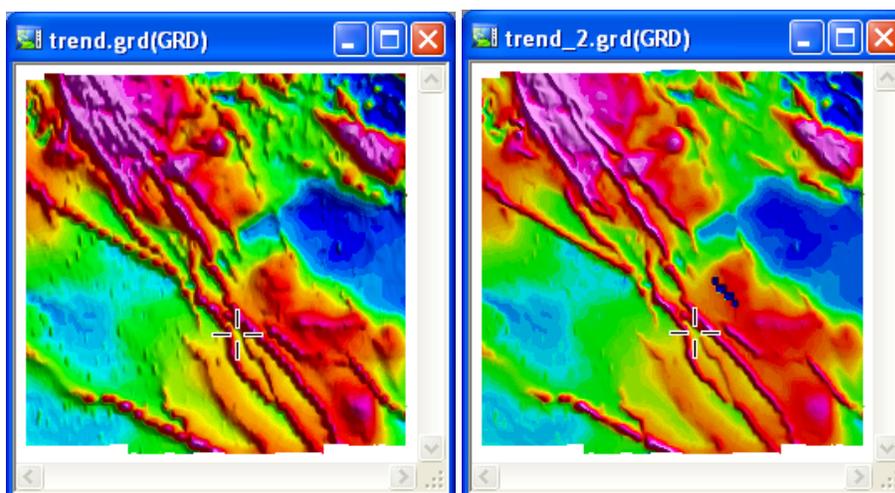
5. Click **OK** to grid the data and display the grid in a temporary map.



For information about the Advanced options, click the Help button on the Advanced Gridding options dialog, .

Comparing the Two Grids

We can display two grids side-by-side to see the difference the trend enforced gridding has on the data. An example is shown below.



The original grid "trend.grd" (left) the Trend Enforcement grid "trend_2.grd" (right).



The grid should be displayed as a colour shaded grid to properly compare the two grids.



The actual data values and locations are exactly the same in both grids. The **Trend Enforcement Tool** forces the gridding process to honour the trended data. Trend lines must be selected to be included in gridding. You can deselect the trend line(s) when performing additional processing with the database.

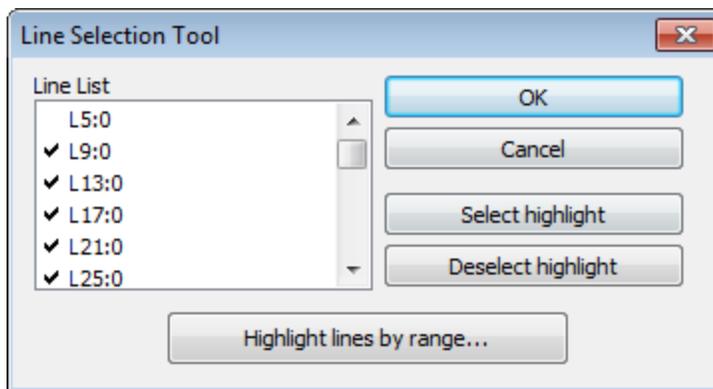
Deselecting/Selecting Trend Lines

You may want to deselect/select your trend lines for other processing applications.

To Deselect/Select Trend Lines in a Database

Before you begin, select your database.

1. Click on the *Line Header Cell* - the upper left cell.
2. Right-click and from the pop-up menu select *Selections*, then select **Selection Tool**. The *Line Selection Tool* dialog appears.



3. Scroll down the list and highlight the desired line.
4. Click **Deselect highlight**. The check mark beside the line is removed.
5. Repeat the previous step as needed.
6. Click **OK** to continue.
7. To select lines, repeat above procedure and click the **Select highlight** button. A check mark will be placed beside the line.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Applying Filters in Oasis montaj

Filters are usually applied to geophysical data in order to remove noise/errors, remove regional trends and/or enhance certain geological features. The objective is to select a filter and parameters that do not introduce additional noise into the data.

1D filters are applied to a channel of data in a database, while 2D filters are applied to gridded data and are commonly used during image processing.

This How-To Guide walks you through the basic steps for applying a 1D filter to a channel in a database and a 2D filter to a gridded dataset.

Filter Menus

There are two main Filter menus included with the Oasis montaj system. The database filters can be found on the *Database Tools|Filters* menu, and the grid filters can be found on the *Grid and Image|Filters* menu, as shown below.

Database Tools Filters menu	Grid and Image Filters menu
<ul style="list-style-type: none">Non Linear Filter...Convolution Filter...Rolling Statistics...Linear Regression...	<ul style="list-style-type: none">3x3 Convolution...5x5 Symmetric Convolution...7x7 Symmetric Convolution...9x9 Symmetric Convolution...Vertical Derivative Convolution...Shaded Relief...Horizontal Gradient...Trend...Automatic Gain Correction...Analytic Signal...

Apply a Filter to a Database Channel

Removing spikes or noise from data, while preserving significant or relevant geological features, is the most common application of Database Filters. The objective is to select the filter and filter parameters that accomplish this, while not introducing any artifacts into the data.

In this section you will apply the Non Linear Filter which can be used to remove very short wavelength, high amplitude features from the data.

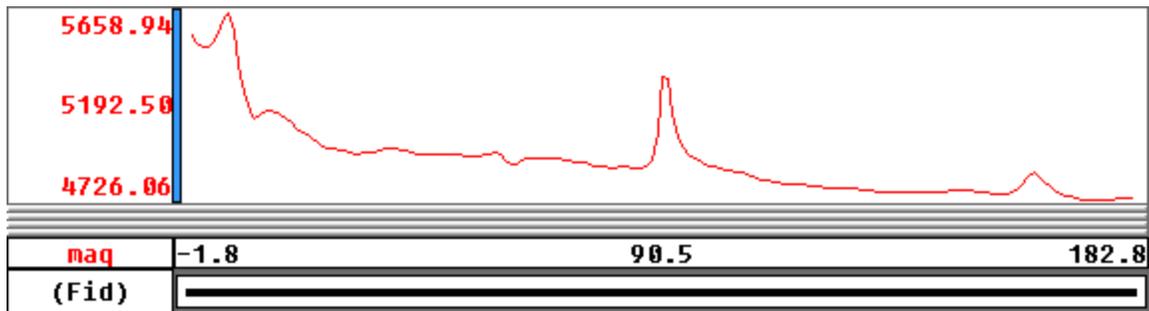
Displaying a Profile

After you open a spreadsheet and display one or more channels, you have the option of displaying them in graphical (profile) form in a *Profile* window. The Profile window is composed of sub-windows called panes, which appear directly below its corresponding database. You can display up to five separate panes with 32 channels of data in each pane.

To Display a Profile

1. Select (highlight) the profile window pane you want to display a profile in. The vertical bar on the left will be highlighted.
2. Select (highlight) the *Channel Header Cell* in your database you wish to profile, right-click and from the popup menu select, **Show Profile**.

The channel data will be displayed in the profile window.

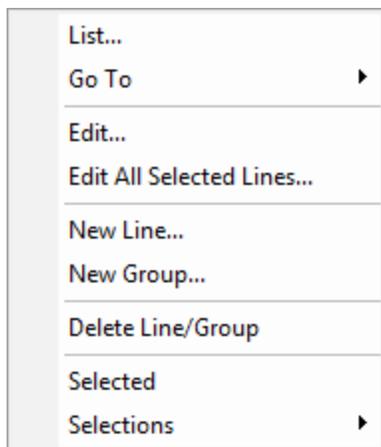


Displaying a Profile of Different Line

If you have line data, you will probably want to look at multiple lines of your data.

To Display the Profile of Different Line

1. Select (highlight) the *Line Header Cell* in the top left corner of the spreadsheet.
2. Right-click and select **List** from the popup menu.



A list of all the line numbers in the current database will be displayed. You can think of each line number as a worksheet in the Spreadsheet window.

3. Click the line to display. The *Profile* window now displays the selected line.



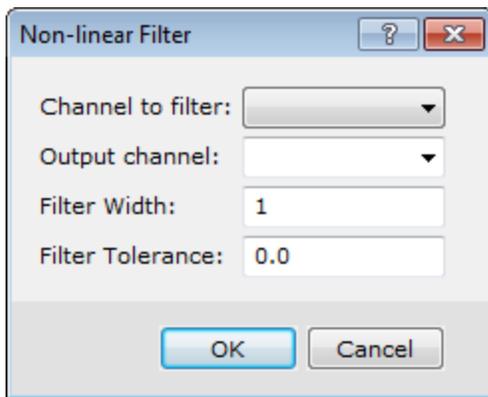
When the Line Header Cell is highlighted, you can use the Database Tools Bar to scroll through the lines by selecting **Previous Line/Group** or **Next Line/Group**. You can also use the **Page Up** and **Page Down** keys on your keyboard to scroll through the lines.

Applying a Non-Linear Spatial Domain Filter

The Non-Linear Spatial Domain Filter is a particularly effective way to remove spikes from magnetic data, but it can also be effective for removing short wavelength features, such as signal arising from unwanted near surface geological or cultural features. The non-linear filter is always applied before the linear filter.

To Apply the Non-Linear Filter

1. From the *Database Tools* menu, select *Filters*, then select **Non Linear Filter**. The *Non-linear Filter* dialog box appears.

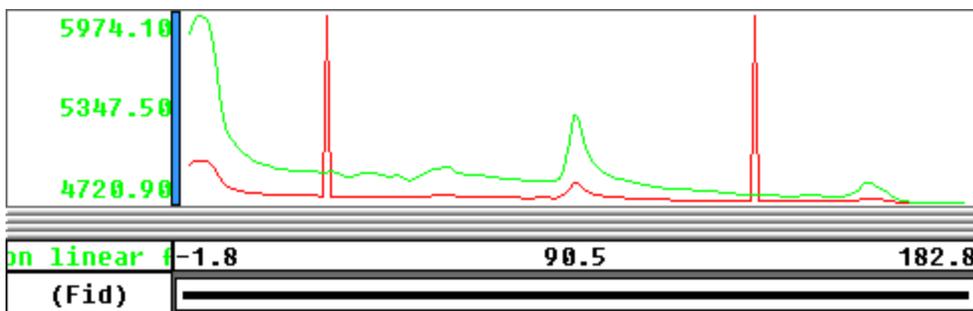


2. Use the dropdown list to select the *Channel to filter*.
3. Specify (type) the *Output channel* name, or select a channel using the dropdown list.
4. Enter a *Filter Width*. Note that, this is the maximum width of the noise measured in data points. Features that are wider than this width will not be altered by the filter.
5. Specify the *Filter Tolerance*. Only noise of greater amplitude than this tolerance value will be altered.
6. Click the **OK** button.

Now you can compare the filtered and unfiltered channel profiles.

To View the Profile of the Filtered Channel

1. Right-click in the header cell of the channel containing the filtered data, and select **Show Profile**. The filtered profile along with the original data profile will be displayed in the profile window.



Applying a Filter to a Grid

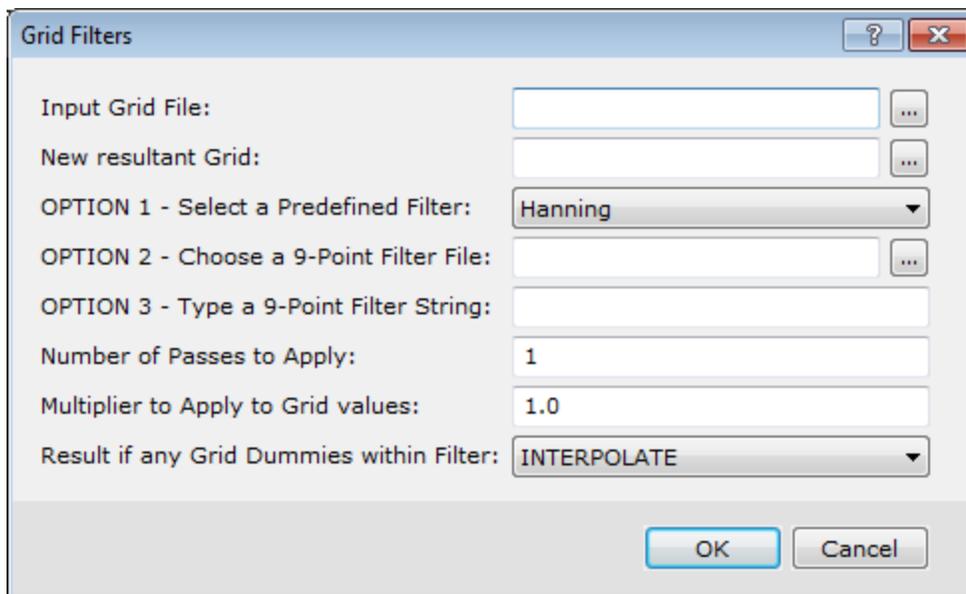
Grid filters are commonly used to smooth a grid before contouring in order to improve the appearance of the contours. In this application, 1 to 3 passes are usually sufficient. Another useful application is filtering to remove regional data trends by producing a curvature grid. To do this, a 3x3 Convolution (Hanning) filter is passed once over a gridded dataset and the output curvature grid is then subtracted from the original grid. This creates a residual gridded dataset, where the residual is proportional to the curvature in the original grid, with zero values indicating inflection points. The curvature grid may also be subsequently filtered to smooth noise.

Applying Grid Filter

In this section you will apply the 3x3 Convolution (Hanning) filter to remove the high frequency component of your data. You can then subtract this "smoothed" grid from the original grid to create a residual grid.

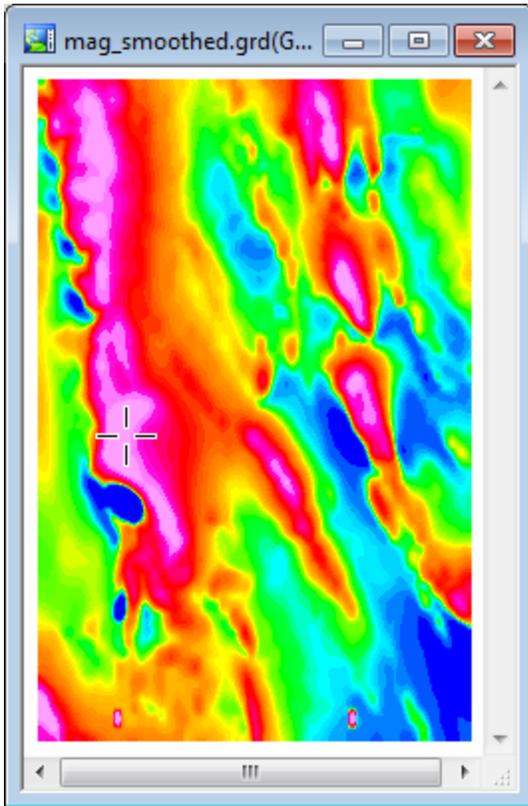
To apply a filter to a grid

1. From the *Grid and Image* menu, select *Filters*, and then select **3x3 Convolution**. The *Grid Filters* dialog appears.



2. From the *Input Grid File* list, select your input grid.
3. For the *New resultant Grid*, enter the name of the new filtered grid.
4. From the *OPTION 1 - Select a Predefined Filter* list, select **Hanning**.
5. Click the **OK** button.

The new smoothed mag grid opens in a display window.



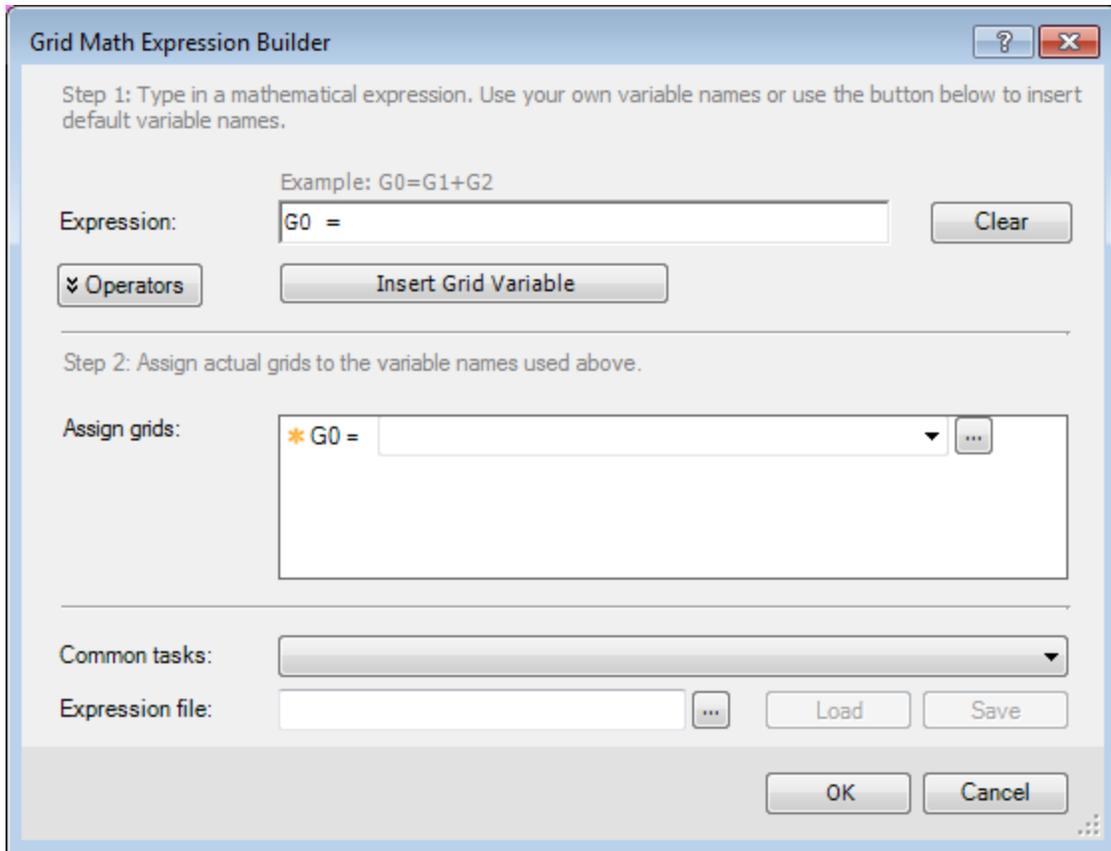
Applying Grid Math

With the Grid Math Expression Builder, you can create, save, load and apply mathematical expressions involving one or more grids in your project.

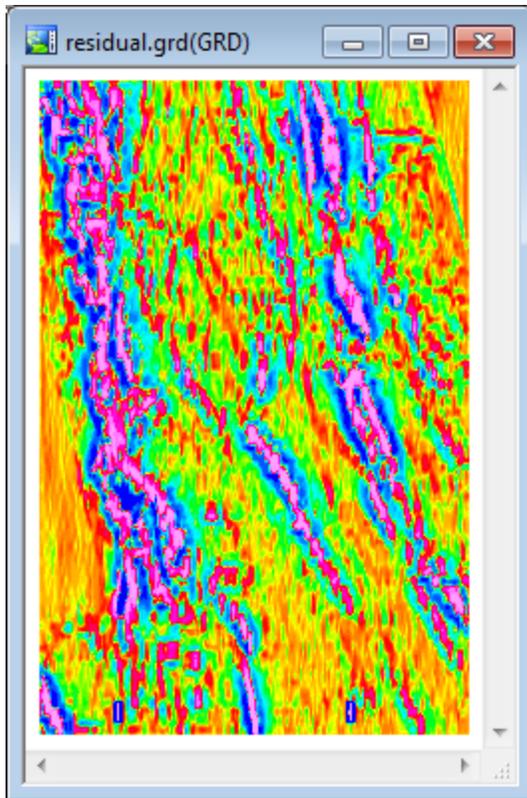
In this section you will create a residual grid by subtracting the filtered grid from your original grid.

To Subtract Grids

1. From the *Grid and Image* menu, select **Grid Math**.
The *Grid Math Expression Builder* dialog appears.



2. In the *Expression* box, enter **G0= G1– G2**.
3. In the *Assign grids* box, click in the list beside the **G0** channel and type the name of the new residual grid.
A new grid, with this name, will be created when this expression is computed.
4. From the list beside **G1**, select the original grid.
5. From the list beside **G2**, select the filtered grid.
6. Click the **OK** button.
A new residual grid is created and opens in a display window.



The residual grid represents the high frequency information in the magnetic data, which represents very shallow geological or cultural features. Note the shape and orientation of individual features is now evident within the larger areas of anomalous high magnetic intensity.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Map Editing and CAD Tools

This How-To Guide introduces you to the basic map editing and CAD functions in Oasis montaj. The procedures are designed to show you how you can use these tools to draw an interpretation from geoscience data. You will also learn to use the clipboard to copy, import, and export map groups and images.

Topics covered in this How-To Guide:

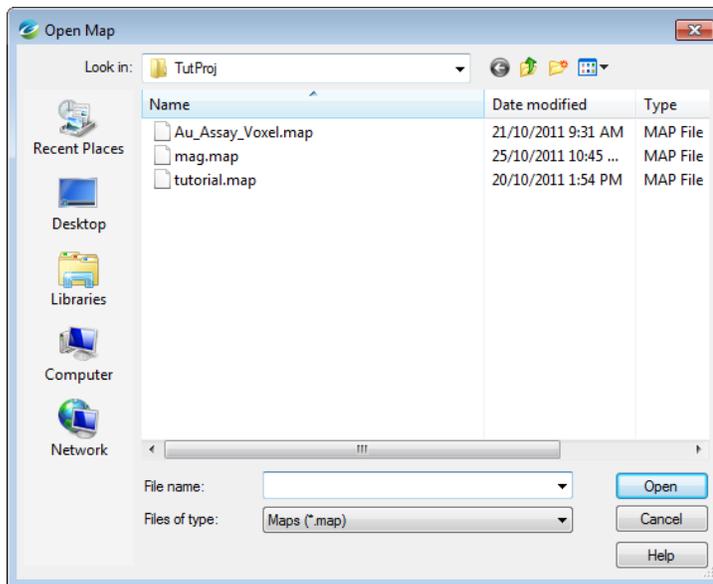
- Display an interpretation map
- Create and work with map groups
- Create and edit lines, poly-lines and polygons
- Use the Windows Clipboard to import and export graphics

Displaying an Interpretation Map

In order to edit a map, it must first be available in the *Project Explorer*.

To Open a Map

1. On the *Map* menu, select **Open map**.
The *Open Map* dialog box appears.



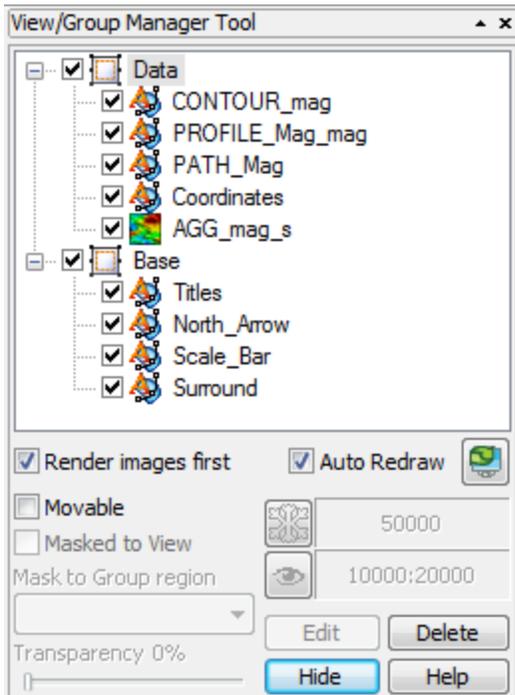
2. Select the map file in your project directory and click **Open**.
The map appears in the map window.

Map Groups

Map Groups organize map objects for display. You can use the *View/Group Manager Tool* to display and edit the views and groups in a map.

View/Group Manager Tool

This tool (see example below) consists of a 'tree' structure that contains branches representing the *Data* views and the *Base* views. Under each view, groups are listed according to their layer on the map.



Each group in a view shows an icon beside its name that identifies the group as either a map vector object or an aggregate object. Views can either be normal 2D views or 3D views.



The check box controls and indicates visibility of an item.

To select a single item in the tree, click on the item desired. Multiple selections of groups within a view can be achieved by holding the **<Ctrl>** key and clicking on the desired items. Changing the selection during group editing will end group editing without cancelling any current changes.

The effect of a double-click on any item depends on the state of the map. If in shadow cursor mode, the map will switch to either group or view selection mode and select the item that was clicked upon. If the map is already in either of these selection modes, a double-click has the same effect as hitting the **Edit** button or using the activate shortcut key (default **<Enter>** key).

The group on the top layer (closest to the front) is listed first, followed by the next layer behind it, followed by the rest of the layers to the bottom layer at the end of the list. It is possible to control the render order in the tree by using the Drag-n-Drop technology. The Drag-n-Drop capability enables you to move map groups up and down within views and also to move views relative to each other.

Rendering Options

Render images first	Render (draw) the images groups first.
Auto-Redraw	Automatically redraw the map when a change is made to it.
 Redraw	Redraws the map when changes are made to the groups in the View/Group Manager and the Auto-Redraw option is off.
 Frozen Scale	Freezes the scale of the currently selected map group, independent of the view scale of the map. For example, when zooming in the text size in a group will not grow but remain the same size on the screen.
 Visible Scale	Scale range in which the currently selected group will be visible. For example, individual groups may be made visible only at specific scales. Then, while zooming, if the map scale is outside the range, the group is not drawn.

The *View/Group Manager Tool* is a “modeless” dialog, which means it can remain open while you work on your map. It can be toggled on or off using the “**M**” hot key and can even be docked on the side of your screen or at the top with the toolbars.

Other Tool Options

The other Tool options include moving, masking, transparency settings, editing and deleting.

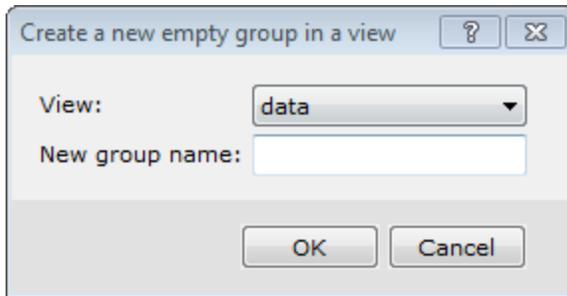
Moveable	Enables the movement of the selected group on the map using the cursor.
Masked to View	Masks the selected view using the mask applied to the view. This option enables you to apply a mask to selected groups within a view.
Mask to Group region	Select the extended view mask applied to the view. This option enables you to save the mask into the map and activate it on specific groups in the view.
Transparency	Enables the Transparency Slider (Transparent - Opaque). Move the slider to the transparency level wanted for the selected group.
Edit	<p>Click to edit (or activate) the selected item.</p>  <p><i>Double-click on a group or view in a map or using the activate shortcut key (default Enter key) to activate the edit mode for that item. The edit mode depends on the item; group edit mode for vector groups, colour tool for images and colour symbols and 3D Viewer for 3D views.</i></p>
Delete	<p>Click to delete the selected View/Group.</p> <p>If you delete a group by mistake or make any other undesired changes, you can click the Undo last map change button on the Standard Toolbar or use the shortcut (default Ctrl-Z) at any time to undo any changes and revert to the original map.</p>

Creating a New Group

The following procedure demonstrates how to create a new map group.

To Create a New Group

1. On the *Map Tools* menu, select *CAD Tools* and then select **New Group**. The *Create a new empty group in a view* dialog appears.



2. From the **View** drop-down list, select the existing view in which to place the new group.
3. Specify the *New group name*.
4. Click **OK**. The new group is created and selected in your current map window and the *Map Group Editing Bar* is enabled.

Selecting and Editing Groups

When you create a new group, this group becomes selected and you can edit it. A group is open for editing whenever you see the *Map Group Editing Bar* displayed.



Editing Using the View/Group Manager Tool

This tool provides a single dialog that enables you to switch between different groups, change their properties (moveable, hidden, masked), and change their layering order (move groups to the front or back).

1. Verify that *Map Group Mode* is selected.
2. On the *Map Tools Bar*, select **View/Group Manager**. The *View/Group Manager Tool* is displayed. The currently selected group is highlighted. To switch to another group, click on the group name so that it becomes highlighted.
3. Click **Edit** and the group is selected in edit mode on your map and the *Map Group Editing Bar* is displayed.
4. When you are done editing the group, right-click and select **End Editing** from the pop-up menu.



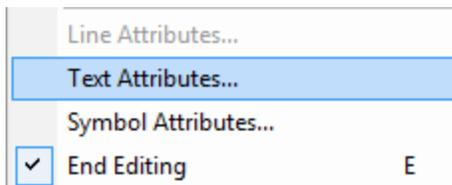
You can also edit groups using the Mapping pop-up menu. You may find this method quicker when you are editing and switching between groups frequently.

Editing Using the Pop-Up Menu

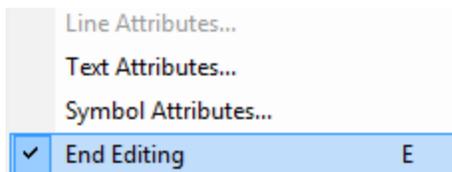
1. Verify that *Map Group Mode* is selected.
2. Right-click on the map and select **Groups** from the pop-up menu.



3. Select the name of the group you would like to select.
4. Right-click on the map again and select **Edit this group** from the pop-up menu. The group is selected in edit mode and the *Map Group Editing Bar* appears.
5. To edit Symbol or Text attributes, right-click again and select **Select All** from the pop-up menu. Then right-click again and from the pop-up menu, select to edit the *Attribute (Text or Symbol)*, as shown below.



6. When you are done editing the group, right-click and select **End Editing** from the pop-up menu.



Render Images

Check the *Render images* box to render (draw) the images groups first. This option is useful when you add an image to your map, and would like to draw this group first.

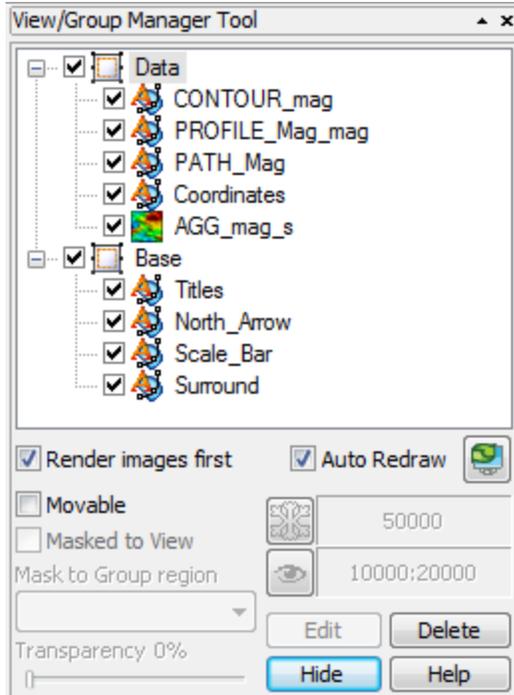
Hiding and Moving Map Groups

The procedures below demonstrate how to hide and move groups on a map.

To Hide a Group on a Map

When you have many groups on a map, you may not want to display all of them on your final map. The procedure below shows you how to hide a group on your map.

1. Verify that *Map Group Mode* is selected and then click **View/Group Manager**.
The *View/Group Manager Tool* appears.



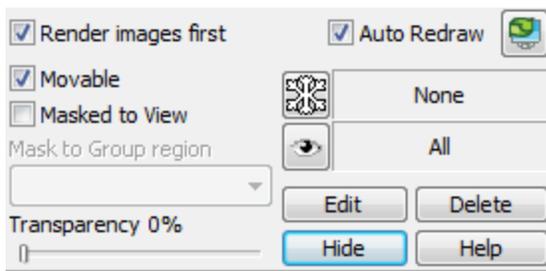
- 2.
3. In the *View/Group Manager Tool*, select (highlight) the group you want to hide.
4. Remove the check box from the box directly in front of the icon. The group is hidden.

⚠ *To display the group, add the check mark back again.*

To Move a Group

To move a group around a map with your cursor, you must first select the **Movable** option for that group. Once this option is set, you can move the group by selecting it and moving it with the cursor.

1. From the *View/Group Manager Tool*, select (highlight) the name of the group.
2. Place a check in the *Movable* box (as shown below).



💡 *You can also click **Movable** from the pop-up menu.*

3. Place the cursor anywhere inside the group on the map. The cursor will change into a set of directional arrows.
4. Click-and-drag the group in the direction you want to move it.

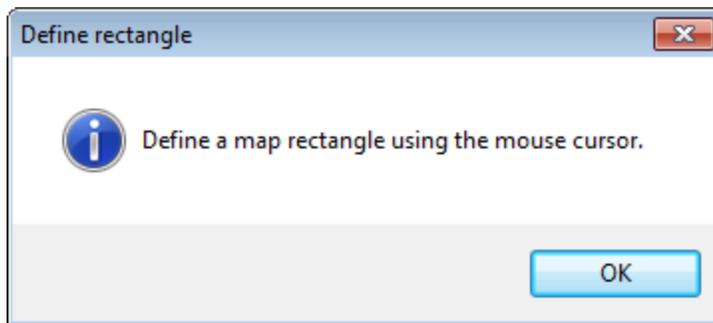
Masking a Group

Masking a group involves using a polygon or rectangular mask file to make only a specific area of a group visible. To mask a group, you must first create and apply a mask to a view. You can then use this mask on specific groups in the view areas of the group outside the mask. The following three procedures show you the steps to create a mask, apply it to a view, and turn masking on for a group.

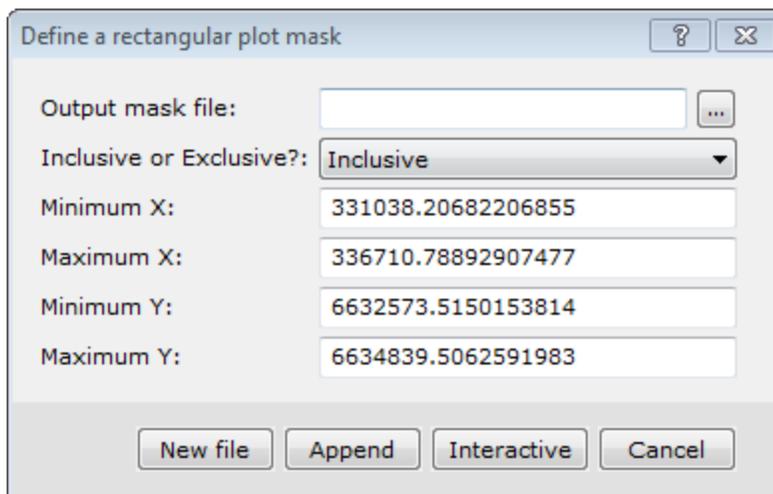
To Create a Rectangular PLY File

This procedure demonstrates interactive creation of a Rectangular PLY file. You can also create Polygon or Geographic PLY files. Please refer to the Application Help files for more information.

1. On the *Database Tools* menu, select **Window Data**.
2. Select **Create Rectangular PLY file**. The *Define a rectangular plot mask* dialog appears.
3. Click **Interactive**. The *Define rectangle* dialog box appears.



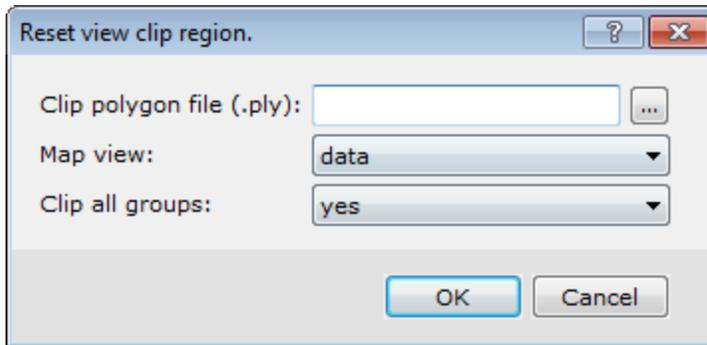
4. Click **OK** to continue.
5. Use the mouse to draw a rectangle on the map . The *Define a rectangular plot mask* dialog is then displayed with the coordinates of the rectangle, as shown in the example below.



6. Specify the *Output mask file*.
7. From the dropdown list select **Inclusive** or **Exclusive**. This determines what is visible when the mask is applied to a group. For example, if you select *Inclusive*, only features inside the rectangle mask will be visible.
8. Click **New File** to save the mask file. The file is saved.

To Apply a Mask to a View

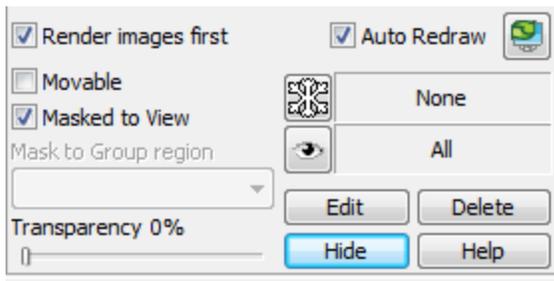
1. On the *Map Tools* menu, select **Masking**.
2. Select *Apply View Mask*. The *Reset view clip region*. dialog appears.



3. Browse to the *Clip polygon file (*.ply)* to apply.
4. From the dropdown list, select the *Map view* where the clip mask should be applied.
5. Select **yes** from the *Clip all groups* dropdown list to force all groups in the view to be clipped. Otherwise, select **no**.
6. Click **OK**. The mask is applied to the view.

To Mask a Group

1. Verify *Map Group Mode* is selected and select the map.
2. Click **View/Group Manager**.
3. Select (highlight) the group you want to mask.
4. Place a check mark in the *Masked to View* box (as shown below).



5. Click **Redraw**. The map is redrawn to display only the section of the grid inside the mask.



You can also turn Masking on or off by selecting a group in a map. On the *Map Tools* menu, select *Masking* and then select **Clear View Mask**. The mask will be cleared from the View.

Lines, Poly-Lines and Polygons

You can use the line and the poly-line tools to draw linear features such as dykes and faults. You can then connect these lines and create shaded polygons to mark different geological or geophysical anomalies in your project area. This How-To Guide demonstrates several common poly-line and polygon procedures.



For detailed information on using the *Map Group Editing Bar*, please refer to the *Application Help files*.

Drawing Poly-Lines

The procedure below shows you how to draw poly-lines on your map and how to create polygons from these lines.



A poly-line can change direction. Use the **Draw a Straight Line on Map** button to draw a straight line.

This procedure also shows you how to use the *Snap to point* and *Snap to line* tools, which are useful for ensuring that your lines connect each other without overlapping. The *Map Group Editing Bar* is used to create lines, polylines and polygons.



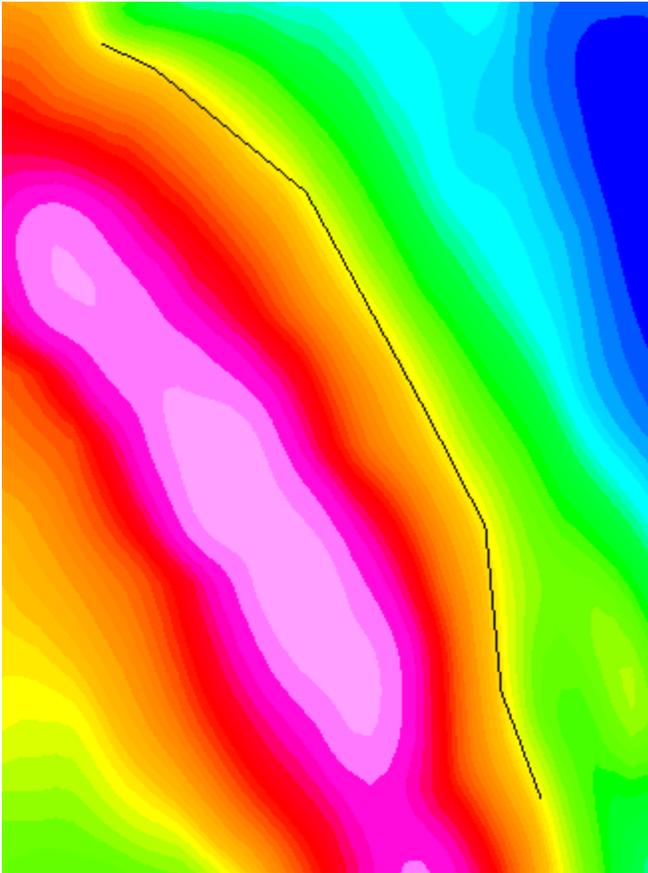
Drawing Poly-Lines to Identify Linear Features

1. Verify *Map Group Mode* is selected.
2. Right-click the map, and select **Groups** from the popup menu.



3. Select the group to be edited.
4. Right-click on the map and select **Edit This Group** from the popup menu. The *Map Group Editing Bar* is displayed.
5. On the *Map Group Editing Bar*, click **Draw a Poly-line on Map**.
6. Click the cursor on the place you want the line to start, and then move the mouse, clicking wherever you want to change direction.
7. When you are finished, right-click and select **Done** from the popup menu. The poly-line appears.

An example of a poly-line is shown below.



Using Snap to Point and Snap to Line

You can connect poly-lines to create polygons.

1. Click **Draw a Poly-Line on Map**.
2. Right-click and select **Snap to Point** or **Snap to Line** from the popup menu. This determines whether the new poly-line begins from a location along the existing line or a point on the line.

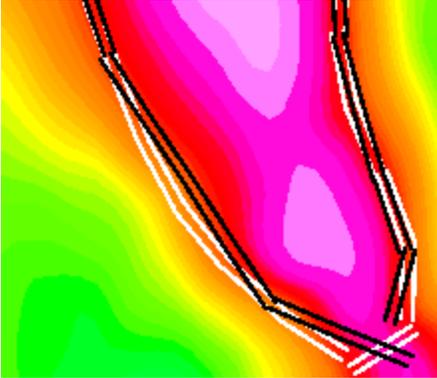


The cursor will change to a circle with a shaded centre. When you move the cursor over a point or node along the line, the cursor changes to a target symbol with the shaded circle indicating the location of the point.

3. Move the cursor and click-and-drag to define the shape of the line. Draw the poly-line(s) as desired.
4. When complete, right-click and select **Done** from the pop-up menu.
5. Click **Esc** to cancel the Snap To mode.

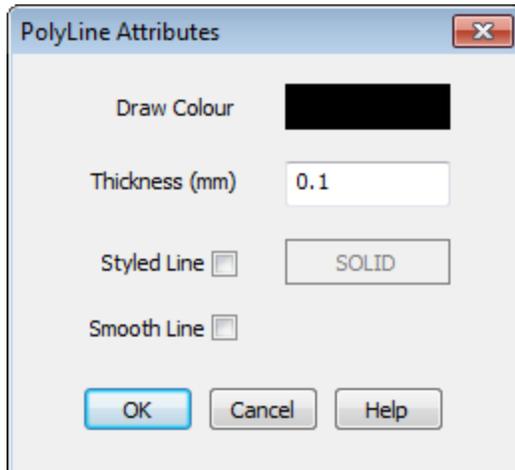
Changing Line Attributes

This procedure shows you how to modify the attributes (thickness, colour and pattern) of lines. An example is shown below.

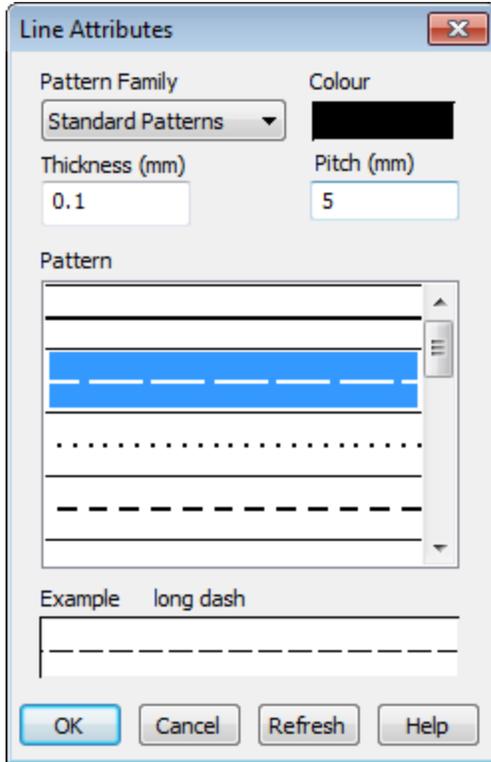


Changing Line Attributes

1. Left-click (select) the line you would like to edit. Sizing handles are displayed around the line, indicating it is selected.
2. Right-click and select **Attributes** from the pop-up menu. The *Polyline Attributes* dialog box appears.



3. In the *Thickness* box, you can specify the line thickness in millimetres (0.1 is the default).
4. To change the line colour, click on the **Draw Colour** box. A colour palette dialog box is displayed for you to choose a colour.
5. To change the line pattern, make sure there is a check mark in *Styled Line* check box. Double click in the *Styled Line* box to display the *Line Attributes* dialog box.



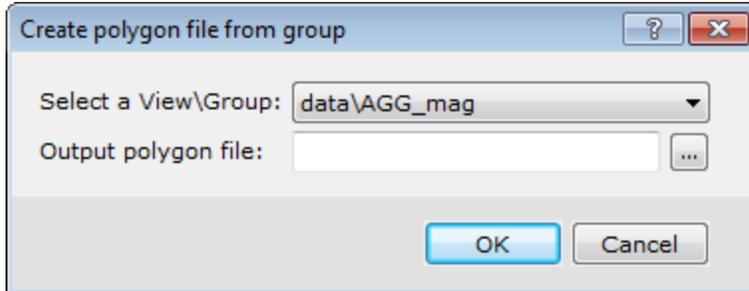
6. From the *Pattern Family* dropdown list, you can select from a variety of different sets of patterns for the line.
7. You can also set the *Pitch* (which is the distance between the symbols or dashes in the line) and the line *Thickness*.
8. Click **OK** when you are finished.
9. Check **Smooth Line** to smooth the rough edges in the poly-line into smooth curves.
10. Click **OK** to apply the changes to the line or click **Cancel** to exit the dialog without making any changes.

Creating Polygons from Lines

After delineating different areas of your map with poly-lines and/or straight lines, you can use these lines to create polygons. This process will convert all the lines in the current group into polygons. If you have lines that you do not want converted to polygons either create these lines in a different group or ensure that the lines are not connected to other lines to create polygons.

To Create Polygons from Lines

1. On the *Map Tools* menu, select *CAD Tools* and then select **Create PLY File from Map Group**. The *Create polygon file from group* dialog appears.

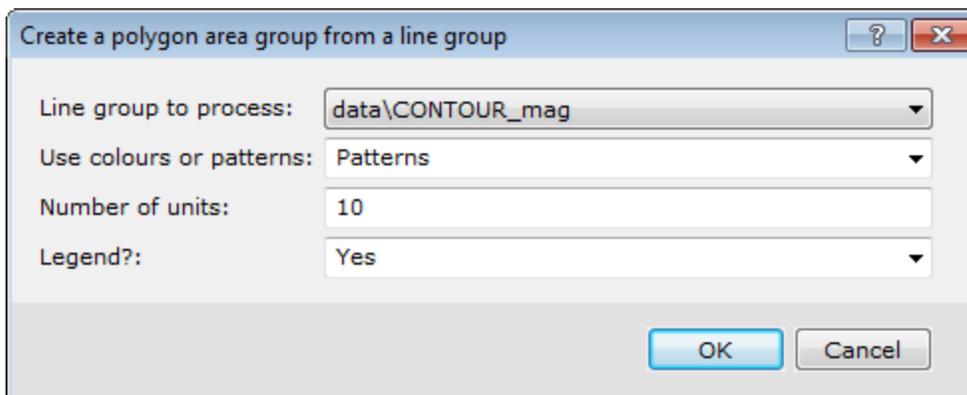


2. From the *Select a View\Group* dropdown list, select the View or Group to convert to polygons.
3. Specify the *Output polygon file*.
4. Click **OK** to create a polygon file from your lines in the selected group.

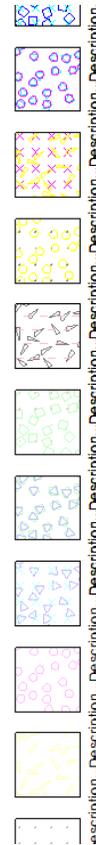
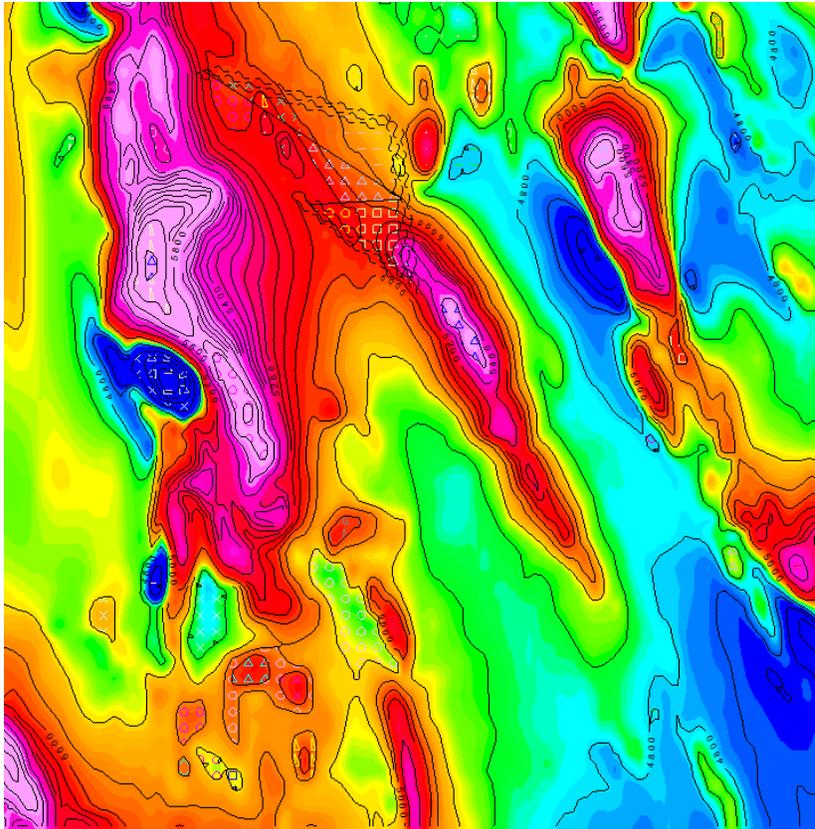
Creating Polygon Fills

You can create fills for the closed polygons you created above.

1. On the *Map Tools* menu, select **Create Fills for Closed Polylines**.
The *Create a polygon area group from a line group* dialog appears.



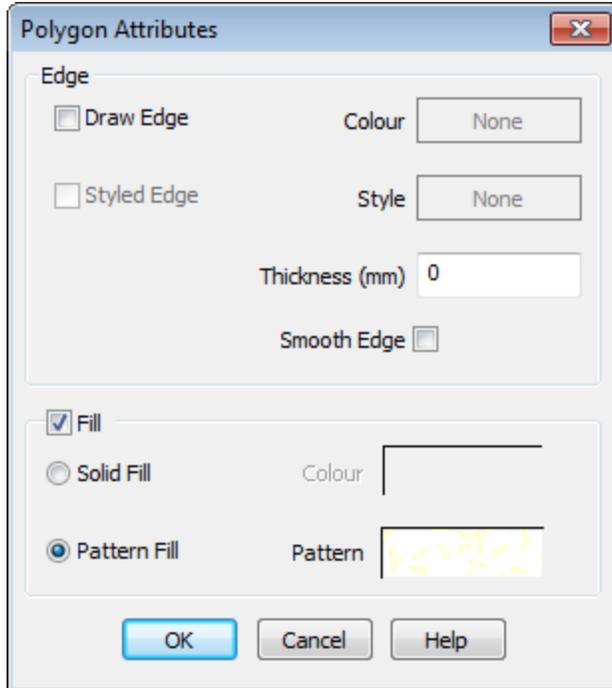
2. Use the **Line group to process** dropdown list to select the group of lines that you want to convert to polygons.
3. The **Use colours or patterns** dropdown list has three options for shading polygons: *Colours*, *Patterns* or *Both*.
4. The *Number of units* indicates the number of boxes of categories you want to create in the legend. The default is 10. You can edit these boxes later.
5. Select **Yes** if you wish to add a legend to the map.
6. Click **OK** to continue.
7. Specify the legend settings in the *Legend* dialog, if you are adding a legend.
8. Click **OK** to create polygons and a legend similar to the map below.



Editing Polygons and Legend Boxes

You can edit both polygons and their associated legend boxes in the same way.

1. Verify *Map Group Mode* is selected.
2. Double-click the polygon or box you want to edit. The *Polygon Attributes* dialog appears.



3. To change the line colour, click the Edge **Colour** box. A colour palette dialog box is displayed for you to choose a colour. To turn line colouring off (i.e. make the line invisible) remove the check mark from the *Draw Edge* check box.
4. You can select a line pattern and/or colour using the *Styled Edge* and *Style* boxes.
5. To display a polygon fill, check the **Fill** check box.
6. You can select a **Solid Fill** colour by selecting the radio button and choosing a colour using the *Colour* palette.
7. Select **Pattern Fill** to display the *Pattern Attributes* dialog and select a pattern fill and colour.
8. You can also set the *Tile Size* and *Tile Density*, which changes the space between the pattern symbols and the Line Thickness (% of Tile Size), which changes the size of the individual pattern symbols.



Click **Refresh** to see what the pattern will look like in the example box.

9. Click **OK** when you are finished selecting polygon attributes to apply the changes or *Cancel* to exit the dialog without making any changes.

Using the Polygon Attributes Tool

Using the *Get Polygon Attributes* tool (eyedropper) and the *Apply Polygon Attributes* tool (bucket) on the *Map Group Editing Bar*, you can copy fill colours and patterns from one polygon and apply them to another polygon.

Get Polygon Attributes

	The image cursor changes to an eyedropper.
--	--

Apply Polygon Attributes

	The image cursor changes to a bucket.
---	---------------------------------------

To Copy Polygon Attributes from one Polygon to Another

1. Click the **Get Colour** button on the *Map Group Editing Bar*. The image cursor will change to an eyedropper.
2. To get a colour (and pattern) from a polygon, click with the eyedropper on the polygon containing the colour you want to copy. The cursor will change to a bucket.
3. Click on the polygon to which you want to apply the fill. The polygon now contains the new colour (and pattern).

Polygon Edit Tools

A *Polygon Edit* toolbar is provided with Oasis montaj. These tools enable you to edit polygon vertices, add exclusion regions to polygons, and switch between editing the vertices of main polygons (*Islands*) and vertices of the exclusion regions polygons (*Holes*).



<table border="1"> <tr><td colspan="3">Attributes...</td></tr> <tr><td><input checked="" type="checkbox"/></td><td>End Group Editing</td><td>E</td></tr> <tr><td></td><td>Edit Vertices...</td><td>N</td></tr> <tr><td></td><td>Add Vertices...</td><td>Num +</td></tr> <tr><td></td><td>Delete Vertices...</td><td>Num -</td></tr> <tr><td colspan="2">Render Order</td><td>▶</td></tr> <tr><td colspan="2">Nudge</td><td>▶</td></tr> <tr><td colspan="2">Delete</td><td>Del</td></tr> </table>	Attributes...			<input checked="" type="checkbox"/>	End Group Editing	E		Edit Vertices...	N		Add Vertices...	Num +		Delete Vertices...	Num -	Render Order		▶	Nudge		▶	Delete		Del	<h4>Edit Vertices</h4> <p>The Edit Vertices option is available on the <i>Map Edit</i> pop-up menu when you are in Group Edit mode and have a polygon selected on your current map.</p>
Attributes...																									
<input checked="" type="checkbox"/>	End Group Editing	E																							
	Edit Vertices...	N																							
	Add Vertices...	Num +																							
	Delete Vertices...	Num -																							
Render Order		▶																							
Nudge		▶																							
Delete		Del																							

Polygon Edit Tools	Description
 Edit Vertices	Edits the vertices of a previously created polygon.
 Add Vertices	Adds new vertices to a polygon.
 Delete Vertices	Deletes vertices from a polygon.
 Add New Rectangular Island	Add a new rectangular “Island” to a previously created polygon.
 Add New Polygon Island	Adds a new polygon “Island” to a previously created polygon.
 Add New N-Sided Polygon Island	Draws a new N-sided polygon “Island” to a previously created polygon.
 Add New Rectangular Hole	Adds a new rectangular “Hole” to a previously created polygon “Island”.
 Add New Polygon Hole	Adds a new polygon “Hole” to a previously created polygon “Island”.
 Add New N-Sided Polygon Hole	Draws a new N-sided polygon “Hole” to a previously created polygon “Island”.
 Cycle to Previous Hole/Island	Cycles (switch) to the previous Hole or Island.
 Cycle to Next Hole/Island	Cycles (switch) to the next Hole or Island.
 Delete Current Hole/Island	Deletes the current Hole or Island.

Add/Remove Vertices on a Polygon

Vertices can be added or removed from polygons by using the plus (+) and minus (-) keys on your **Number Keypad**.



*The plus (+) and minus (-) keys on the main alpha-numeric keypad will not work; you **MUST** use the **Number Keypad** only.*

To Access the Edit Polygon Vertices Option

1. Click **Map Group Mode**.
2. Select the group on your map that includes the polygon you wish to edit.
3. While holding your cursor over your map, right click and select **Edit This Group** from the pop-up menu. The group will now be in *Edit* mode (you will see a box with hatch marks around the group, this is a visual clue that you are in Edit mode).
4. Select the polygon to edit.
5. Right-click and select **Edit Vertices** from the pop-up menu. The vertices on your map will change from grey to white.

To Add Additional Vertices on a Polygon

1. While in *Edit* mode, select **Edit Vertices** from the pop-up menu.
2. Place your cursor, between two vertices on the polygon, where you want to place the new vertice (s).
3. Press and hold the “+” key on your **Number Keypad**. A plus sign is added to your cursor.
4. While holding down the “+” key, use your mouse to select the location for the new vertice(s).



*You can also select **Add Vertices** from the Polygon Edit Bar. Your cursor will change to a cross-hair and it will be linked to the polygon lines. Left-click to add new vertices to the polygon.*

To Remove Vertices from a Polygon

1. While in *Edit* mode, select **Edit Vertices** from the pop-up menu.
2. Place your cursor on the vertice(s) you want removed.
3. Press and hold the “-” key on your **Number Keypad**. A vertices selection box and a minus sign are added to your cursor.
4. While holding down the “-” key, use your mouse to select the vertices to be removed.



*You can also select the **Delete Vertices** button from the Polygon Edit Bar. Your cursor will change to a vertices selection box with a minus sign added. Left-click to delete vertice(s) from the polygon.*

Using the Windows Clipboard with Maps

The Windows clipboard can be used to move maps and map component objects from one map to another within Oasis montaj. In addition, you can use the clipboard to copy and paste objects between Oasis montaj and other applications.

The Windows clipboard lets you perform the following tasks:

- > Copy text and graphics (as EMF objects) from Windows applications to Oasis montaj maps
- > Copy map objects from Oasis montaj to the clipboard as EMF picture graphics and bitmaps to be pasted into most word processing, presentation and graphic applications
- > Copy objects from one map to the clipboard and paste these into another map as either georeferenced information in an existing data view or in a separate view centred on the currently displayed window

The Windows clipboard enables applications to perform two separate operations:

1. Information can be copied (or cut) to the clipboard
2. Information on the clipboard can be pasted into an application document

When information is copied to the clipboard, it can be placed on the clipboard in a variety of formats. When information on the clipboard is pasted into another application, that application will choose a format on the clipboard that it understands. For example, when text is copied to the clipboard from the Microsoft Word program, it is placed as raw ASCII text, a formatted EMF picture of the text, and a Word document that includes formatting information. When this information is pasted into a simple text editor, only the ASCII text can be used. When this information is pasted onto a PowerPoint slide, the text and formatting is used, and when pasted onto an Oasis Montaj map, the EMF picture is used.

When information is copied from an Oasis Montaj map to the clipboard, the following will be placed on the clipboard:

- > A text block that contains geo-referencing information about the object. This includes the locations of the corners of the picture formed by the information and the map projection of the coordinate system.
- > A map view and its contained groups, or all map groups if nothing on the map was selected. If a map group was selected, only that group and the view that contained the group are placed on the clipboard.
- > An EMF picture of the map or view/group selected and clipped to the currently displayed window.
- > A bitmap of the map or view/group selected and clipped to the currently selected window.

Copying Georeferenced Information between Maps

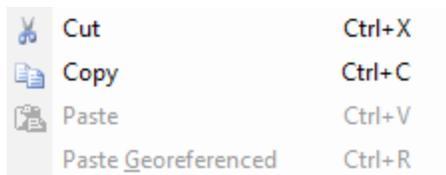
All data plotted in a map data view carry the geo-referencing information of the view. Georeference information includes the map coordinates of the data, a transform that translates the view coordinates to the map (paper) coordinates, and optionally a full map projection. When data is copied to the clipboard, all this information is included.

If you use the *paste georeferenced* command to paste the clipboard information into a map view on another map, the geo-referencing information will be used to locate this information correctly in the new data view. If the new data view is in a projection different from the original data view, the data will be re-projected to the coordinate space of the new view.

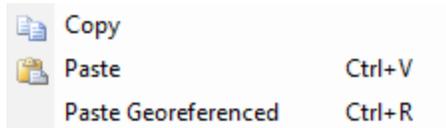
To Copy Georeferenced Objects (Groups) between Maps

You can copy entire maps, views, groups, or a single entity.

- > **Entire map** - Click *Shadow Cursors Tool*
- > **View** - From the *Map View Mode*, click the view to copy
- > **Group** - From *Map Group Mode*, click the group to copy
- > **Single Entity** - From *Edit Group Mode*, select the entity to place on the clipboard
 1. On the first map, select the entire map, a view or a group you want to copy. Reference the instructions above.
 2. Right-click and click **Copy** from the pop-up menu.



3. Select the map which will receive the information.
4. Right-click and select **Paste Georeferenced** from the pop-up menu.



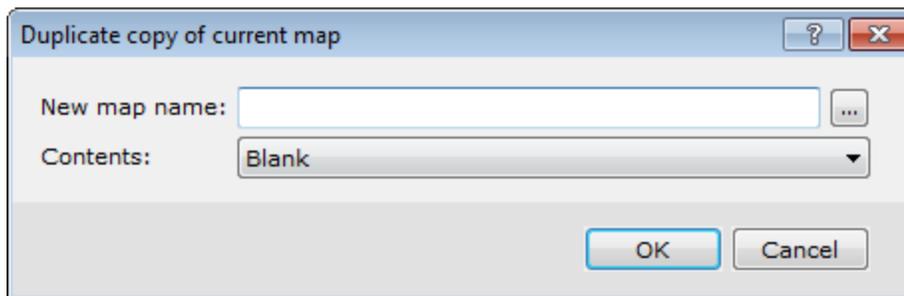
The map object is pasted with georeference information.

Using the Clipboard to create a Montage Map or Poster

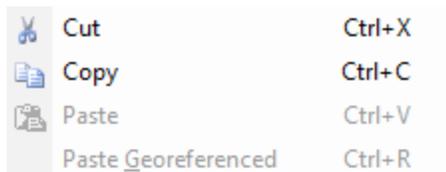
At times, you may want to make a single map that contains separate data views from multiple map presentations. This is often called a montage. Each data view may show different data for the same map area. This can be done using the cut and paste feature of Oasis montaj as shown below.

To Create a Montage Map from component maps

1. Create separate maps of all the different data sets that you want to show together on your montage.
2. Create a new empty map that will be the montage. It is easiest to create this map by duplicating an existing map without its contents.
3. On the *Map* menu, select **Duplicate map**.
The *Duplicate copy of current map* dialog appears.



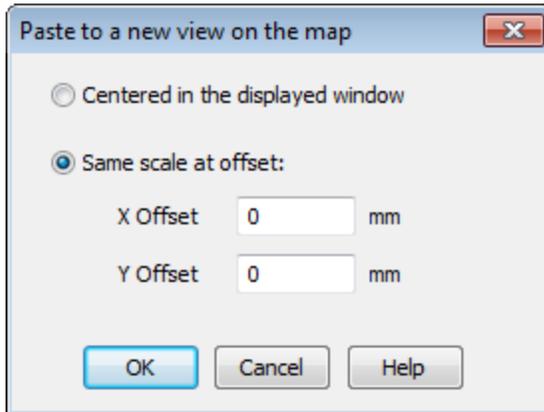
4. Specify a *New map name*.
5. Select *Blank* from the **Contents** dropdown list. The blank map that is created will be your montage map.
6. From the map that contains data, in *Map View Mode* mode, click the view you want to select.
7. Right-click and select **Copy** from the pop-up menu.



- In the blank montage map, right-click and select **Paste** from the pop-up menu.



- The *Paste to a new view on the map* dialog appears.



- Select **Same scale at offset**.
- Specify a location at which to place the view on the montage map. You can also simply move the view after pasting.
- Repeat steps 6-9 for each data view to be placed on the montage map.

When working with a montage map, you can also turn on the Map/Snap mode to limit the location of moved items to an even increment from the map origin. This helps you to exactly line up map views vertically and horizontally. Use the *Map/Snap resolution* option to specify the smallest increment to use for moving views.

You might also choose to create your montage map from the entire contents of other maps, including their base views. To do this, choose **Copy map to clipboard** instead of “Copy” in step 7. In step 9, always choose to offset the location of the inserted views as required. This is because once inserted, only individual views can be selected and moved on the map and it is difficult to move all map views and keep the alignment between views.

Using the Clipboard to Paste a Picture from Another Application on a Map

You can use the clipboard to paste an image onto a map from another application that supports copying Enhanced Metafile Format (EMF) or Bitmap information. For example, you might create marginal notes for your map in Microsoft Word. This information can be selected and copied to the clipboard, where it will be in EMF format (among others). This information can then be pasted on your map, sized and located as you like.

To Paste a Picture from another Application to a Map

- In the other application, select the graphical information and copy it to the clipboard.



From Word select the text, or in PowerPoint select a slide or an object on a slide.

- In Oasis montaj, select the map which will receive the graphic.
- Right-click and select **Paste** from the pop-up menu.



- You will be presented with a dialog that lets you choose the view for the receiving map.

Using the Clipboard to Export Graphics from a Map to Another Application

You can use the clipboard to copy maps, or parts of maps to other applications that can accept EMF or Bitmap graphics, or which can understand Oasis Montaj map objects on the clipboard.

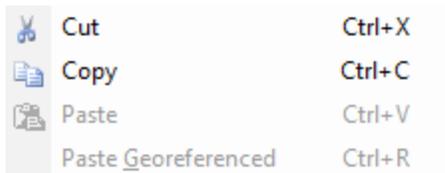
For example, you may wish to place your data view and the images it contains on a PowerPoint slide, or add this as a figure in a Microsoft Word document.



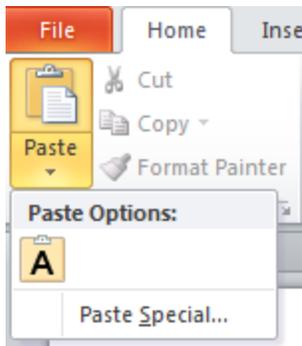
The example below demonstrates exporting a graphic to a PowerPoint Slide in Powerpoint 2010. You can use the same procedure in almost any Microsoft application. Other applications that support pasting of Enhanced Metafile or Bitmap information will have a similar procedure.

To Place a Grid Image on a PowerPoint Slide

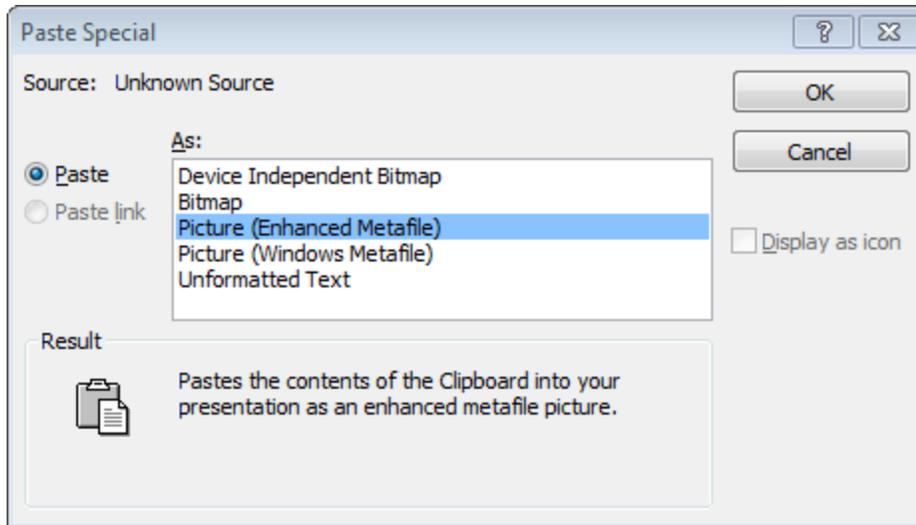
- In Oasis montaj, display the grid in a data view and add whatever graphics you want to include.
- Click **Map View Mode** and choose the view you want to place in PowerPoint.
- Right-click and select **Copy** from the pop-up menu.



- In PowerPoint, create a new slide, or open the slide to receive the picture.



- On the *Home* tab, click the **Paste** dropdown arrow.
- Select **Paste Special**.
The *Paste Special* dialog appears.



7. Select **Picture (Enhanced Metafile)**. The image is pasted into the PowerPoint slide.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Advanced Mapping

As more information becomes available in digital format, Earth science specialists are recognizing that it is essential to work with a variety of data from different sources. This approach not only spreads resources as far as possible but it also enables professionals to pool their knowledge and interpret results more effectively and make more informed decisions.

This How-To Guide investigates the integration of geochemical data in to an environment that has focused on a single type of data (i.e. geophysical).



This tutorial assumes you have a Geosoft project open with both geophysical and geochemical databases available.

Creating Custom Base Maps

Many users have the need to create custom base maps based on corporate standards or client requirements. Custom base maps can be prepared using the commands available in the MAPPLOT GX.



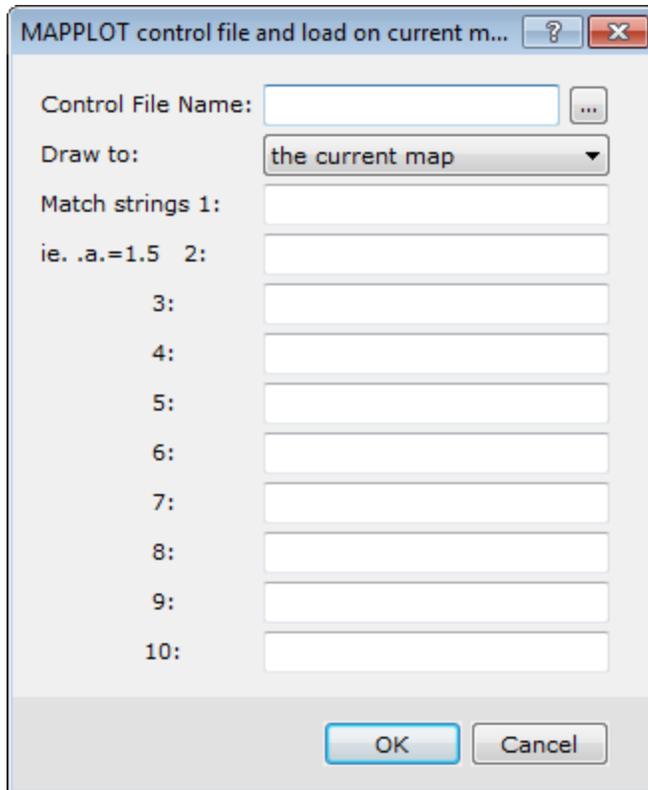
A MAPPLOT template file (mappplot.con) is available in the "C:\Program Files (x86)\Geosoft\Oasis montaj\etc" folder. You can use this file as a template to create your own custom MAPPLOT control file.

Using MAPPLOT to create a Custom Base Map

You can create a MAPPLOT custom base map from either a control file or a custom pdf.

To Create a Custom Base Map from a MAPPLOT Control File

1. On the *Map Tools* menu, select *Base map*, then select *MAPPLOT Control File* and then select **From Control File**.
The *MAPPLOT control file and load on current map* dialog appears.



2. Browse to and select the *Control File Name*.
3. From the *Draw to* dropdown list select a *new map* or *the current map*.
4. Specify the *Match strings* if required.

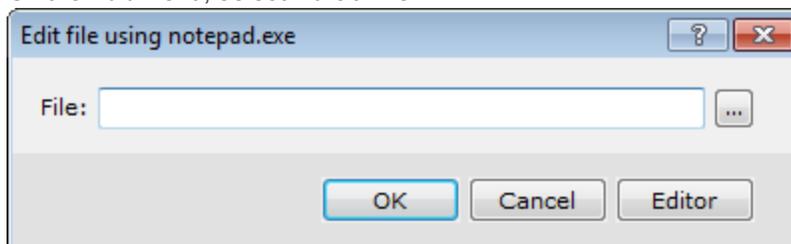


For more information on match strings, click the **Help** button.

5. Click **OK**. If you specified a new map above, the *New map* dialog appears.
6. Specify the *New map name* and click **OK**.
The map with the base map based on the MAPPLOT control file is displayed.

To View and Edit a Control File

1. On the *Edit* menu, select **Edit a file**.



2. Use the *Browse* button to locate the control *File*.

Example

An example control file is shown below. This file accesses several text files for legend information: **survey.leg** and **mag.leg**. It also places three images on the map: a Geosoft format grid file, a TIF file, (**Oasis montaj** logo); and a JPG file.



The four letter commands (for example, SURR) are all described in the help for the MAPPLOT GX.

```
MDF map04.mdf

DATT s1=kt100
DATT s2=kt200
DATT s3=kt500

DATT t1=kt100
DATT t2=kt200
DATT t3=kt300

MGRP Surround,0,0
SURR s3,0.15,s1
DATT s1

MGRP Coordinates,0,1
ANOY ,,,,,2000,0.15,-1,-1,-0.15,,1
ANOX ,,,,,2000,0.15,-1,-1,-0.15,,1

DATT refgrid=kt150
GRID 2

PROJ map.PRJ
DATT llgrid=kt050
GRID -3,0.05,0.0230
ALAT ,,0.5,,,-1
ALON ,,0.5

DATT s1
RECT 12
MGRP Scale_Bar,0,0
SCAL 1,6,2,,ft,,,1
SCAL 1,6,3,,mt

MGRP North_Arrow,0,0
NARR 9,-6,-4,0,0,4,s2,80.3,23.4

MGRP Index_Map,0,0
```

```

RECT 3,-10,7,-2.65,15
TEXT 3,-9.8,7.2,-1,"Index Map - Topography"
s1,,,,,Tl
RECT 3,-8,10,-6,13
s3
TEXT 3,-8.2,9.7,-1,"Survey Area"
s1,,,,,Tl

MGRP Color_Bar,0,0
CBAR 6,-14,-8,1.2,0.6,s1
mag.agg,,,,-1
"Magnetics"
"[nT]"

DATT s1
MGRP Legends,0,0
TXTB 3,-6,20,0,,mag.leg
s1
TXTB 3,-10,30,-1,,survey.leg
s1

MGRP Titles,0,0
TITL 3,0,0,12

b,"Total Field Magnetics",0,t3,.3
b,"Northern Province",0,t2,.3
b,"Canada",0,t2,.3
c,"Data collected by ",0,t1,.25
c,"Helicopter Magnetics System",0,t1,.25
c,"flown by",0,t1,.25
c,"Surveys 'R Us",0,t1,.25
d,"Gold Mapping Corp.",0,t2,.3,,,15
MGRP Mag,0,1
AGGR 0
AGG_mag
mag.grd,colour.tbl,3
MGRP Logo,0,0

```

```
AGGR 2,-5,1,18,6,1
AGG_logo
omontaj.tif(IMG;t=tif)
MGRP IndexMap,0,0
AGGR 3,-10,7,36,15,1
AGG_indexmap
indexmap.jpg(IMG;t=jpg)
```

Gridding Random Data

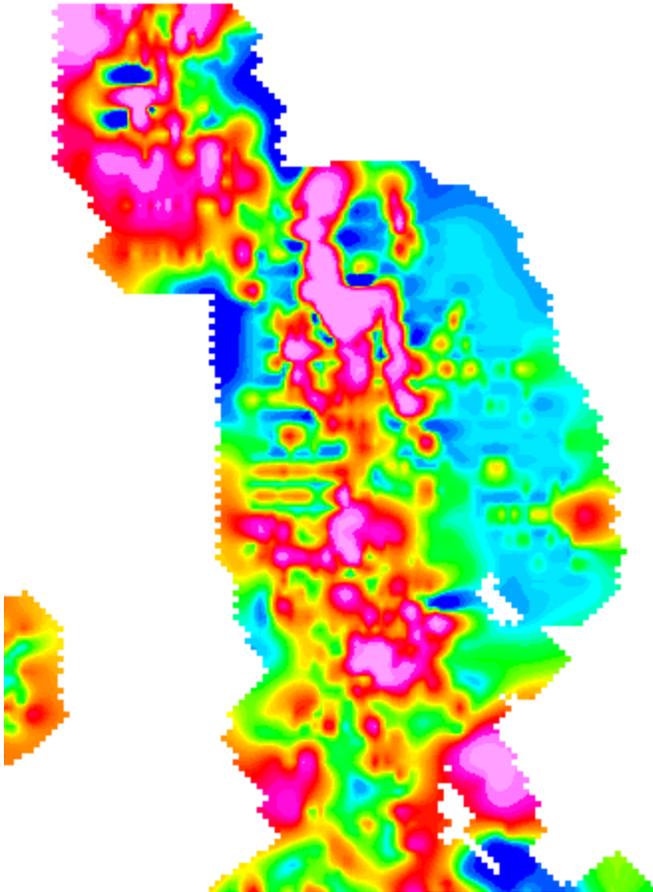
The random gridding method (RANGRID GX) fits a minimum curvature surface to the data points. An example is shown below.

Grid Random Data Using Minimum Curvature

If the data is relatively smooth between sample points or survey lines, minimum curvature gridding should be used. If the data is variable between sample locations, or is known to be statistical in nature (such as geochemical data), is poorly sampled or clustered, use the kriging method.

RANGRID first estimates grid values at the nodes of a coarse grid (usually 8 times the final grid cell size). This estimate is based upon the inverse distance average of the actual data within a specified search radius. If there is no data within that radius, the average of all data points in the grid is used. An iterative method is then employed to adjust the grid to fit the actual data points nearest the coarse grid nodes.

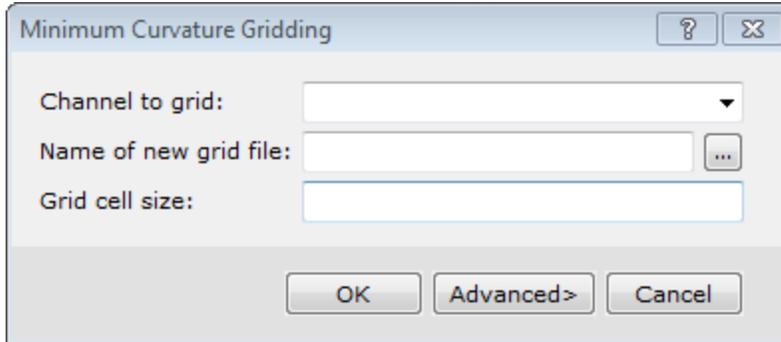
After an acceptable fit is achieved, the coarse cell size is divided by 2. The same process is then repeated using the coarse grid as the starting surface. This is iteratively repeated until the minimum curvature surface is fit at the final grid cell size.



For details on minimum curvature gridding, please refer to the Application Help files.

To Perform Minimum Curvature Gridding

1. Select the database.
2. On the *Grid and Image* menu, select *Gridding* and then select **Minimum Curvature**. The *Minimum Curvature Gridding* dialog appears.



3. Select the *Channel to grid* from the dropdown list.
4. Specify the *Name of new grid file*
5. Specify a *Grid cell size*.



For more information on the Advanced gridding parameters, click the **Help** button.

6. Click **OK**. The system processes the data and creates the output grid file.

Gridding Geochemical Data

There are six potential methods to use for gridding data:

1. **Bi-directional**
Suitable for highly sampled data surveyed along regular lines. Can work well with potential field geophysical data.
2. **Minimum Curvature**
Suitable for randomly oriented data with a relatively large number of samples. Can work well with geochemical data from comprehensive regional and local surveys.
3. **Kriging**
Suitable for clustered data in which clusters are characterized by a relatively large number of samples and may be widely spaced. Can work well with geochemical data from stream sediment surveys and similarly sampled data.
4. **Tinning**
Suitable for randomly oriented data with a relatively large number of samples, or clustered data in which clusters are characterized by a relatively large number of samples and may be widely spaced. Can work well with geochemical data.
5. **Inverse Distance Weighting Gridding**
Suitable for randomly oriented data with a relatively large number of samples. Works well with interpolating geochemical data, creating a grid from point data using an Inverse Distance Weighting algorithm.
6. **Direct Gridding**
Suitable for over-sampled datasets, such as, LiDAR; provides a quick gridded view of the datasets without using any interpolation.

Based on these very general criteria, Minimum curvature, Tinning, Inverse Distance Weighting and Kriging are all optimal methods for statistical data such as that from geochemical surveys, based on your data. Since geochemical data are statistical in nature and have a high range from sample to sample, it may also be appropriate to reduce the range of data through standard processing (i.e. application of a logarithmic transform).

In this brief tutorial, we provide an overview of Kriging and describe how to select an initial model (variogram, or more strictly, a semi-variogram) for creating a grid. Selecting a variogram model is the key to obtaining high-quality results – the actual gridding process is relatively routine. We also provide summaries of how to apply Kriging in Oasis montaj and create a map of your final results.



This tutorial assumes you have a database with clustered data suitable for Kriging.

Kriging – Quick Overview

This statistical gridding method uses the method of Kriging to determine a value at each grid node based on the located data you provide. A significant by-product of Kriging is the ability to estimate the error of the data at each grid node.

In Kriging, the approach is to first calculate a variogram of the data, which shows the correlation of the data as a function of distance. Simply speaking, the further apart the data points become, the less correlation we expect between points. A variogram shows this phenomena for a given dataset, and based on the variogram, you are able to select a model that best defines the variance of the data. There are five types of variograms you can generate in the system, including:

- › Power
- › Sphere
- › Gaussian
- › Exponential
- › User-defined

The technique of Kriging uses the model you select to estimate the data values at the nodes of the grid. In geochemical applications, the sphere and gaussian model are typically most effective. The spherical model is mathematically simpler and we typically recommend this model as your starting point. Advanced users may want to experiment with gaussian and user-defined variogram models.

Geosoft supports ordinary Kriging as well as universal Kriging. Universal Kriging differs from ordinary Kriging in that it enables the data to contain a regional trend. For a more in-depth understanding of geostatistical analysis and Kriging, refer to *Mining Geostatistics* written by A.G. Journel and C.J. Huijbregts, 1978.

Variograms

The semi-variogram (referred to as a variogram for convenience in Geosoft) shows the correlation (semivariance) of the data as a function of the distance away from the sampling point. The further apart the data points become the less correlation we expect between points.

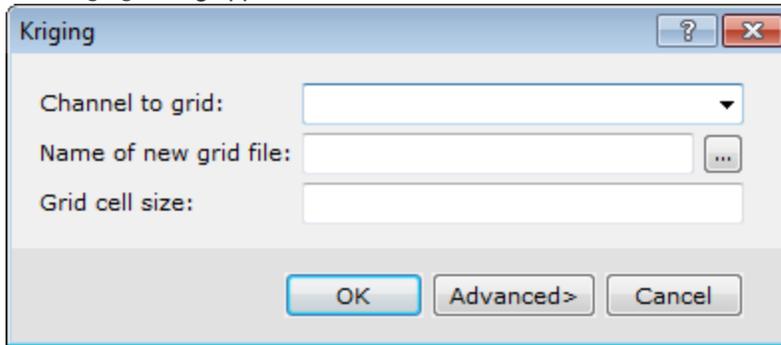
The normal procedure for creating a variogram is to:

1. Create a variogram file (*.var) of the channel you wish to display
2. Select a variogram model
3. Plot the variogram
4. Repeat this process for other models

Create a Variogram File

In this procedure, we will use the default model (linear) to simply generate a variogram of our data that we can then use to select the optimal variogram model and parameters.

1. On the *Grid and Image* menu, select *Gridding* and then select **Kriging**. The *Kriging* dialog appears..



2. Select the *Channel to grid* from the dropdown list.
3. Specify the *New grid file name*.
4. Click **Advanced**.
The *Kriging - Advanced Options* dialog appears.

Kriging - Advanced Options

Channel to grid: Cu

New grid file: .\Cu_grd.grd(GRD)

Name of error grid file: ...

Name of input variogram file: ...

Name of output variogram file: ...

Xmin,Ymin,Xmax,Ymax:

Log option: linear

Log minimum: 1

Low-pass desampling factor:

Blanking distance:

Remove trend (drift): No

Variogram model: spherical

Power:

Range/Slope:

Nugget:

Sill:

Strike:

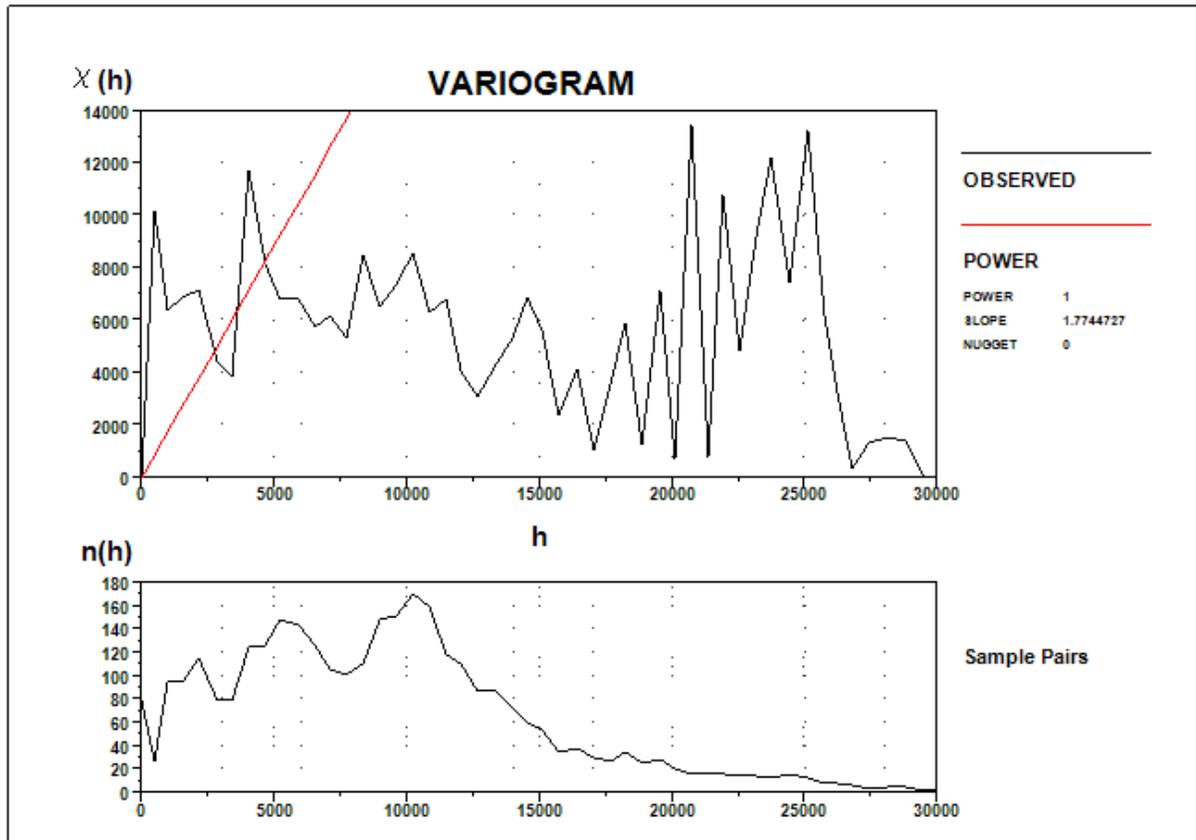
Strike weight: 1

<Back Variogram Only Cancel

5. Specify the *output variogram file* name.
6. Leave the rest of the settings as default.
7. Click **Variogram Only**. The system creates a variogram and displays it in a map in the current project.

Example

In the example below, we used the default model (linear) to simply generate a variogram of our data that we can then use to select the optimal variogram model and parameters.



We see a variogram based on the data with a linear model superimposed. The variogram shows the anticipated increase in variability as distance increases. At the right end, it may appear to decrease but this is usually the result of too few pairs for the statistics to be valid. Normally the variogram reaches a point (sill value) at which the semivariance plateaus. The corresponding h value is called the range value – beyond this distance there is no similarity in samples.

We also see a plot of sample pairs. This latter plot shows how many sets of samples occur within specified distances of each other – this curve may help you to refine your variogram in the selection of the data points used to visualize your model curve. Here, most of the data occurs within 10 000 m and data beyond this point should not be given as high an emphasis when building your variogram.

Variogram Models and Parameters

Selecting a model and parameters is the toughest part of your kriging decision and requires considerable experience. Typically, you would use kriging to grid data within clusters accurately and then truncate the model between clusters so that clusters do not affect the gridding of neighbouring clusters. In this application, data integrity is maintained both within and between clusters and you can examine the error grid to verify that this effect is maintained.



*For more information on advanced kriging options, click the **Help** button on the dialog box.*

Variogram Parameters

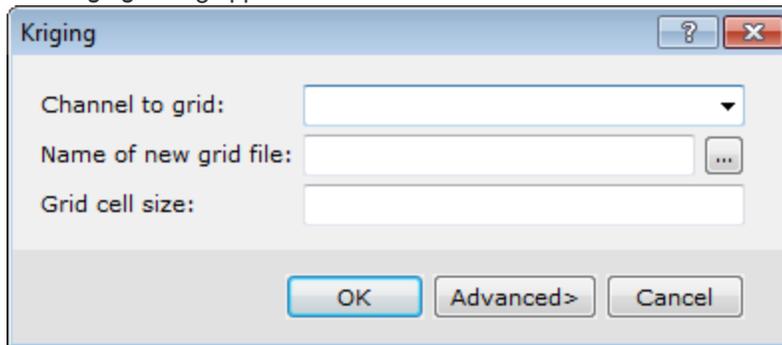
In general, you must specify a geochemical data channel, output variogram file, log option, log minimum and variogram model. In addition, depending on the model you are using, you may have to set additional parameters as follows:

Variogram Model Type	Parameters Defined
Power	Specify a power value to use.
Spherical, Gaussian and Exponential	Specify the Range/Slope, Nugget and Sill.
User-defined	Specify an input grid containing your custom variogram model and select the User-defined Variogram model.

- **Power** - This is the power for the power model. The default is 1, and the nugget and slope will be fit to the data by default if not specified.
- **Nugget (starting value)** - The nugget is the average error in each data point, and is indicated by the intersection of the variogram model with the $h=0$ axis. The default is 0 indicating that there are no repeated samples.
- **Sill** - This is the value corresponding to the point at which the variogram becomes uncorrelated and reaches its plateau point (or goes flat).
- **Range/Slope** - The range is the distance at which the variogram model reaches the sill value. Beyond the range, the data is uncorrelated.

To Define a Variogram Model

1. After displaying the starting linear model, evaluate the data and verify which Variogram model is appropriate.
2. On the *Grid and Image* menu, select *Gridding* and then select **Kriging**. The *Kriging* dialog appears.



3. Select the *Channel to grid* from the dropdown list.
4. Specify the *New grid file name*.
5. Click **Advanced**. The *Kriging - Advanced Options* dialog appears.

Kriging - Advanced Options

Channel to grid:

New grid file:

Name of error grid file: ...

Name of input variogram file: ...

Name of output variogram file: ...

Xmin,Ymin,Xmax,Ymax:

Log option: ▼

Log minimum:

Low-pass desampling factor:

Blanking distance:

Remove trend (drift): ▼

Variogram model: ▼

Power:

Range/Slope:

Nugget:

Sill:

Strike:

Strike weight:

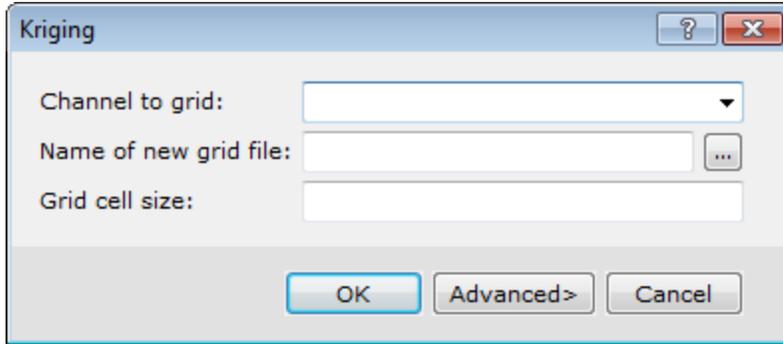
<Back Variogram Only Cancel

6. Enter the *output variogram file* name.
7. Enter the *error grid file* name, if desired.
8. Specify the other kriging options to suit your data. In particular, you will want to confirm the *Variogram model*, *Log option* and *Log minimum*.
9. You should then verify *Power*, *Range/Slope*, *Nugget*, and *Sill* are correctly entered for your variogram model type.
10. Click **Variogram Only**. The variogram is plotted and displayed in your current project.

To Grid the Data

You are now ready to grid the data using the model created above.

1. On the *Grid and Image* menu, select *Gridding* and then select **Kriging**. The *Kriging* dialog appears

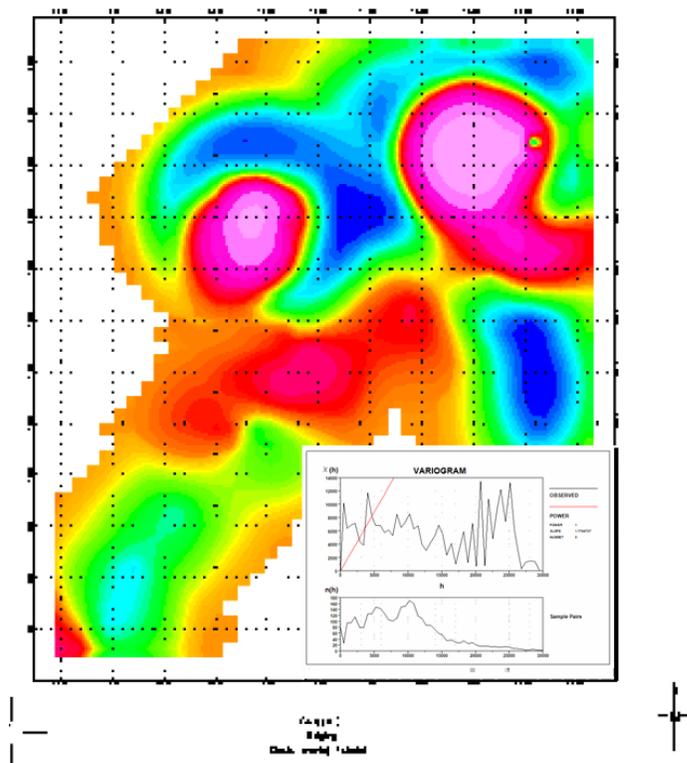


2. Select the *Channel to grid*.
3. Specify the *Name of new grid file*.
4. Click **Advanced>** to display the *Kriging – Advanced Options* dialog.
5. Specify the information as required from above.
6. Click **Back** to return to the *Kriging* dialog.
7. Click **OK** to grid the data and display it in a map window in your current project.

Plotting Kriging Data

The basic steps to display your Kriging grid are as follows. An example is shown below.

1. Create a new map
2. Display the grid
3. Display the variogram (using the *Grid/Gridding/Plot Variogram* menu option)
4. Display sample locations
5. Draw the Base map

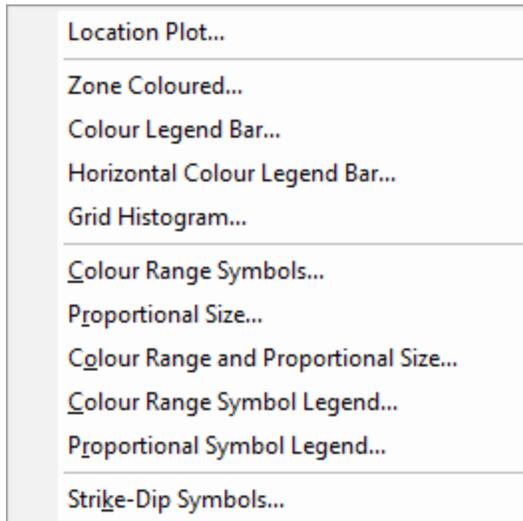


Symbols, Colour Bars and Legends

Oasis montaj provides the ability to create a variety of symbol type plots of spatially located Earth Science data. This functionality is ideally suited to the presentation of data sets that are not line-oriented, such as geochemical survey results.

Plot Symbols, Colour Bars, and Legends

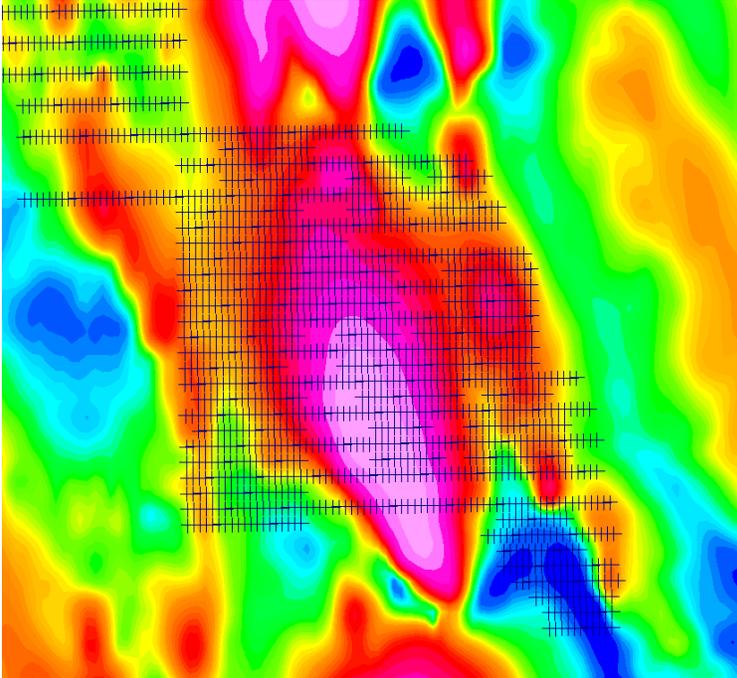
The types of symbol plots available in the system include:



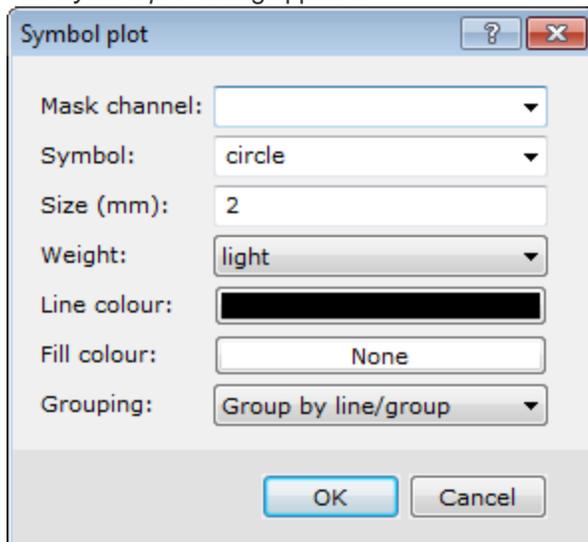
An effective method for geochemical data is to plot coloured symbols on a map. Oasis montaj enables you to create fixed size coloured symbols or symbols that are sized in proportion to the data values. Colours can be based on a variety of colour files, including zone (*.ZON), aggregate (*.AGG), table (*.TBL) or image transform (*.ITR). Symbols can have different fill and edge colours. The fill and edge colours are independent, so they can be set to different values for different channels.

To Plot Location Plot Symbols

The most basic type of symbol plotting in Oasis montaj is to draw location plot symbols on a map for all data points in the database. An example is shown below.



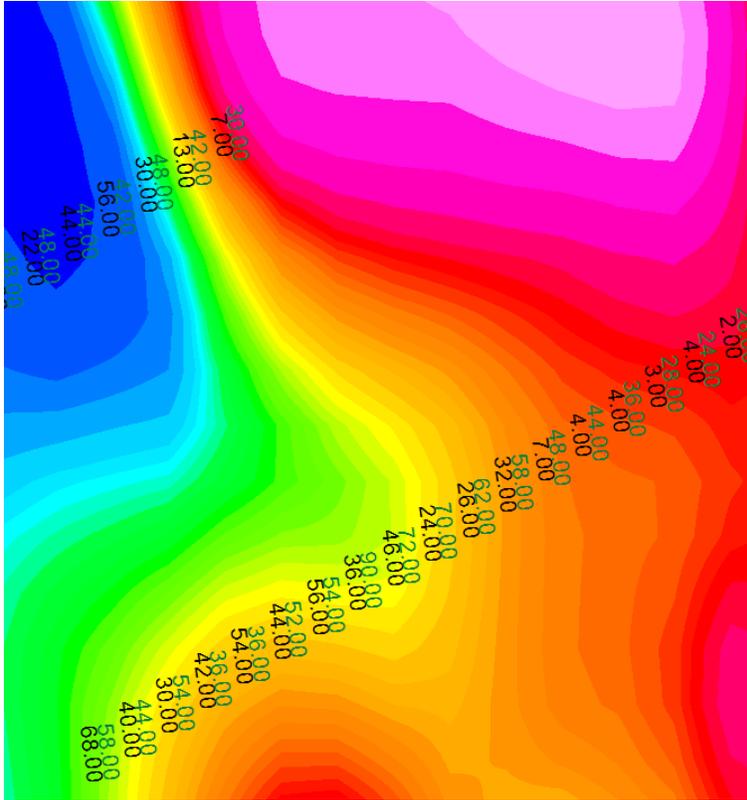
1. Verify the map is selected (highlighted).
2. On the *Map Tools* menu, select *Symbols* and then select **Location Plot**. The *Symbol plot* dialog appears.



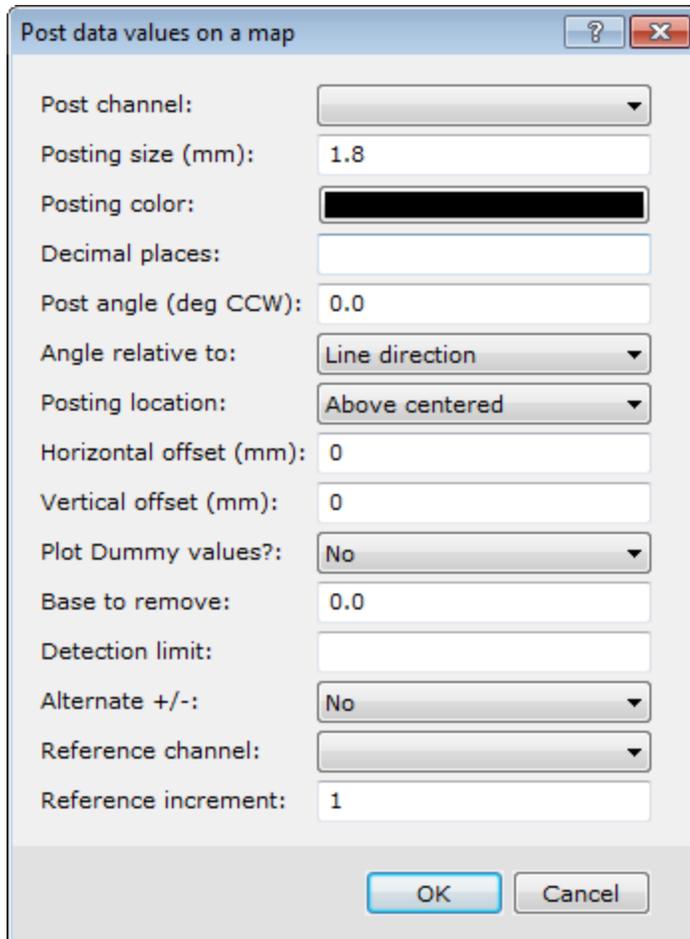
3. Select a *Mask channel* from the drop-down list. If the use of the mask channel is enabled, and set to this channel, only those data values whose corresponding mask channel value is not a dummy will be plotted or exported.
4. Specify the *Symbol* type.
5. Define symbol parameters: *Size*, *Weight*, *Line* and *Fill colour*.
6. Select the *Grouping*: *Group by line/group* or *Individual symbols*.
7. Click **OK**. The system plots and displays the symbols on your map.

To Post Data Values on a Map

Assay data values can be posted next to sample locations on your map. By defining different Posting locations, it is possible to post the values of numerous elements for each sample location (for example, Au, As, Cu, and Pb). Repeat the steps below for each value to post. An example is shown below.



1. On the *Map Tools* menu, select **Posting**.
The *Post data values on a map* dialog appears.



Post data values on a map

Post channel:

Posting size (mm): 1.8

Posting color:

Decimal places:

Post angle (deg CCW): 0.0

Angle relative to: Line direction

Posting location: Above centered

Horizontal offset (mm): 0

Vertical offset (mm): 0

Plot Dummy values?: No

Base to remove: 0.0

Detection limit:

Alternate +/-: No

Reference channel:

Reference increment: 1

OK Cancel

2. Select the *Posting channel* from the drop-down list.
3. Define the posting parameters.

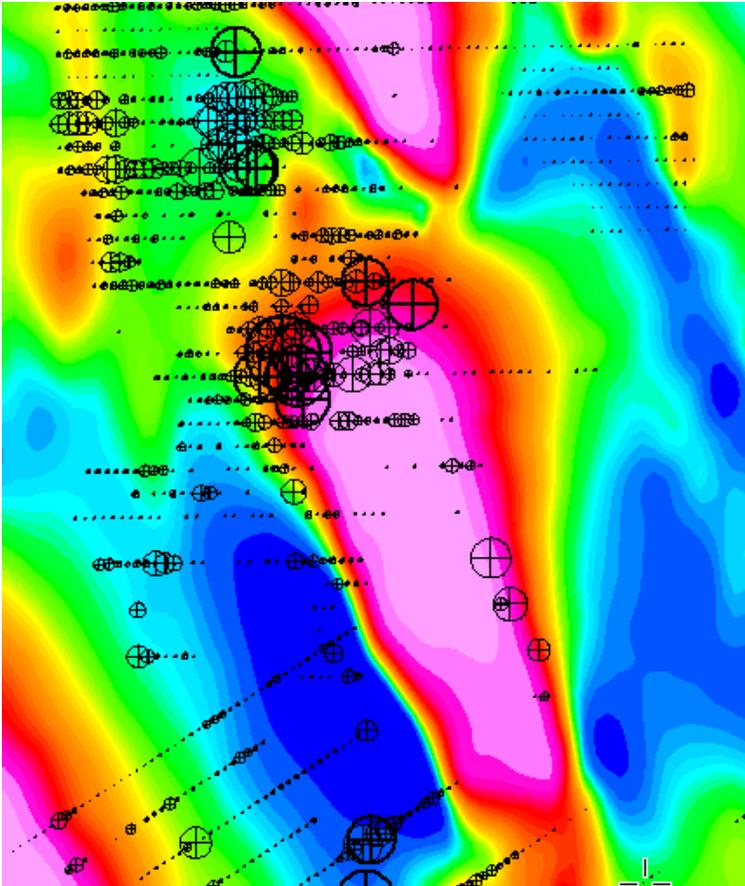


*For more information on the dialog selections, click the **Help** button on the dialog.*

4. Click **OK**. The map is redrawn with the values posted.

To Plot Proportional Sized Symbols

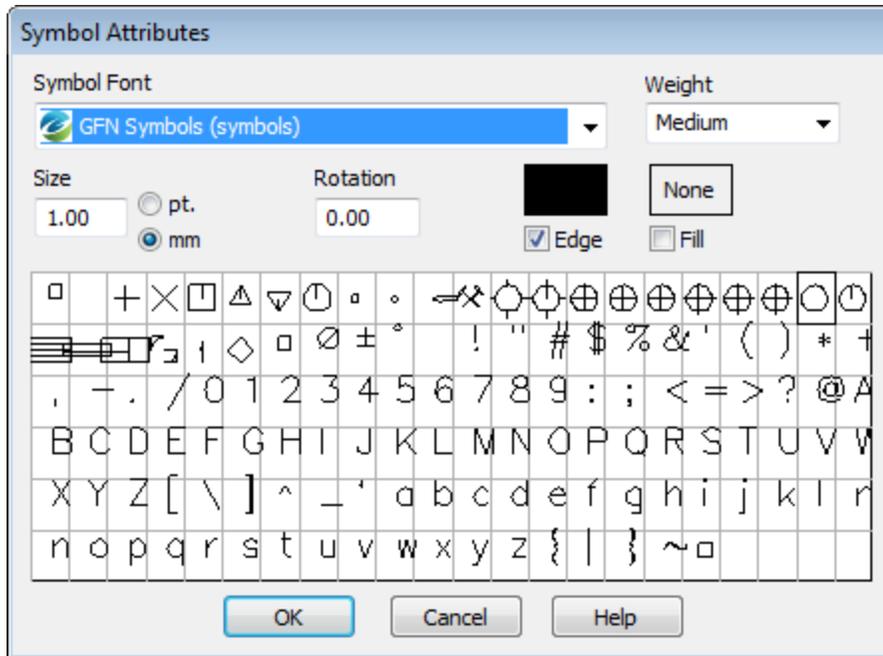
You can use the *Proportional size symbols* dialog to place proportionally scaled symbols on your map. An example is shown below.



If you specify a base value, the value is subtracted from the data before calculating the symbol sizes. Any negative values that are left after subtracting the base are not plotted.

1. On the *Map Tools* menu, select *Symbols* and then select **Proportional Size**. The *Proportional size symbols* dialog appears..

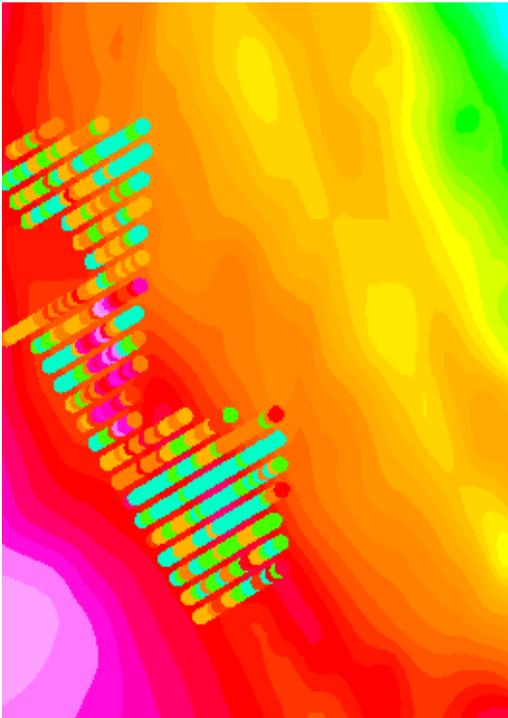
2. Select the *Data channel* from the dropdown list.
3. Define the symbol parameters. The symbol *Scale factor* value indicates how many data units are represented by a 1mm wide symbol. For example, if you specify that the scale is 1000 then a symbol, which has a data value of 20000, is plotted 20 mm wide.
4. Click **Symbol**.
The *Symbol Attributes* dialog appears.



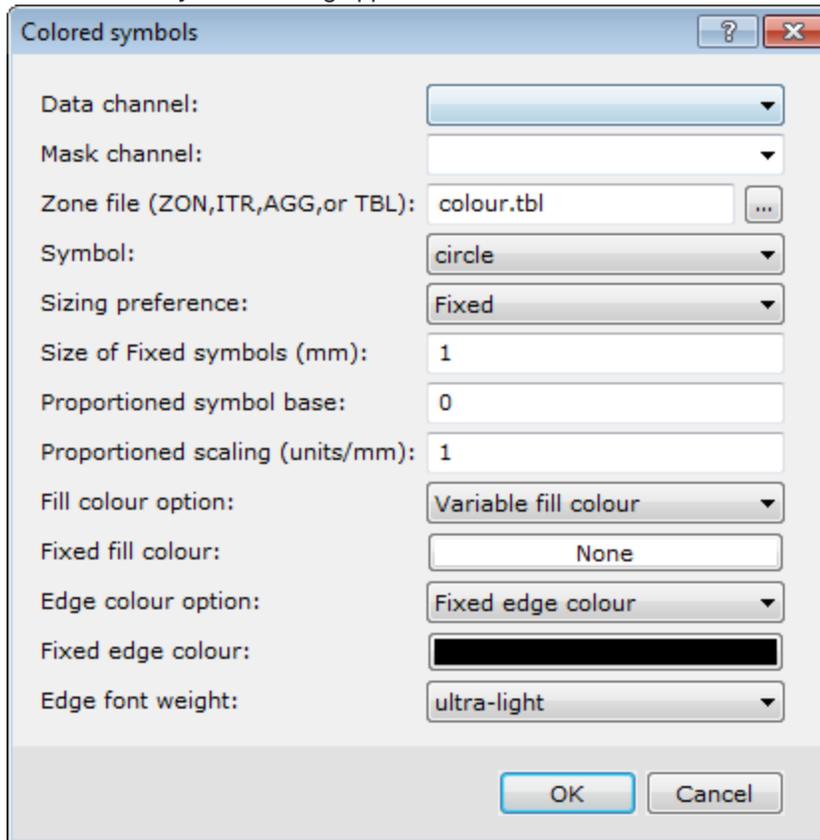
5. Define the symbol attributes.
6. When you are satisfied with your symbol attributes, click **OK**. The *Proportional size symbols* dialog reappears.
7. Click **OK** to display the symbols on your map.

To Plot Zone Coloured Symbols

Use the *Colored symbols* dialog to place coloured symbols on your map. Symbols can be fixed size, or sized in proportion to data values. An example is shown below.

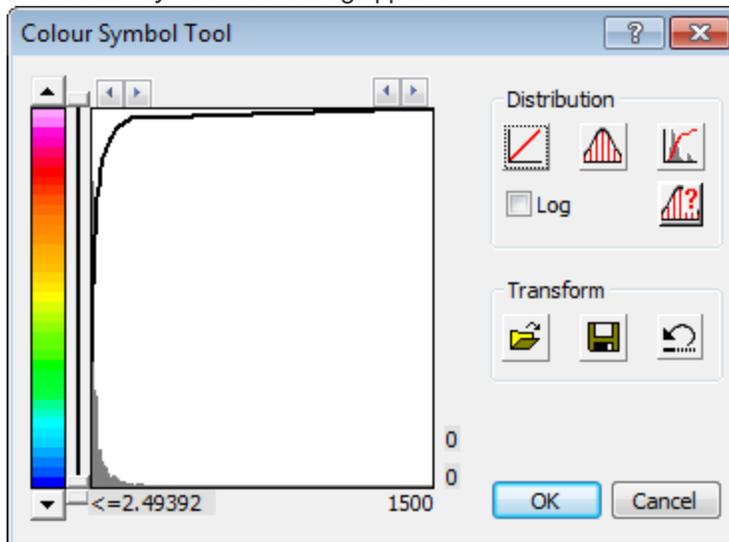


1. On the *Map Tools* menu, select *Symbols* and then select **Zone Coloured**. The *Coloured symbols* dialog appears.



2. Select the *Data channel* from the drop-down list.
3. Specify the rest of the plotting parameters as desired.
4. Click **OK**.

The *Colour Symbol Tool* dialog appears.



5. Define the Symbol parameters.

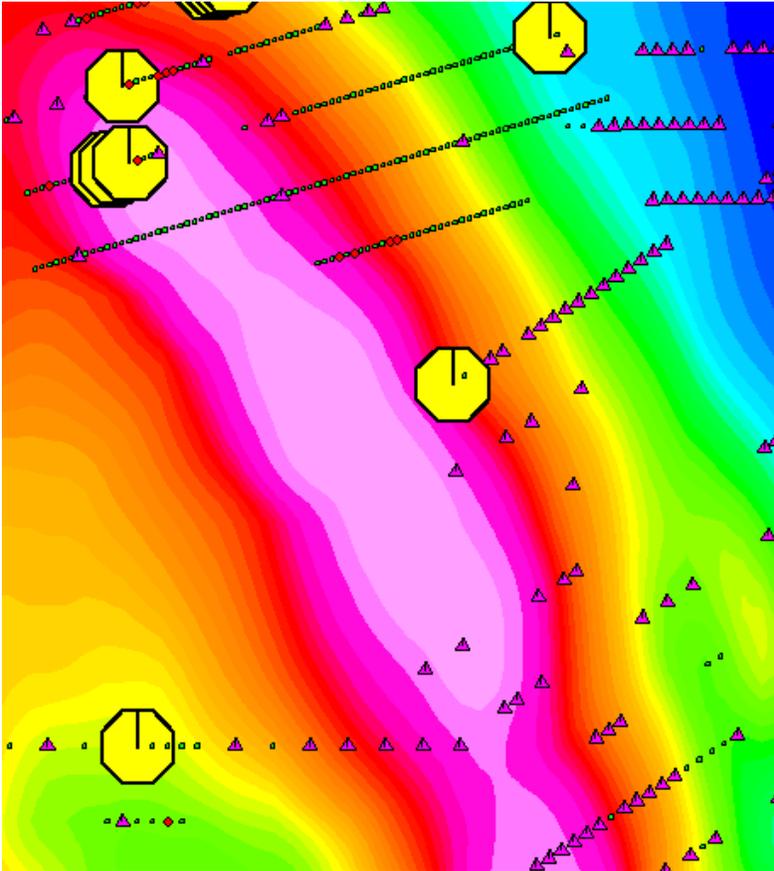


For more information on the *Colour Symbol Tool*, click the **Help** button on the dialog.

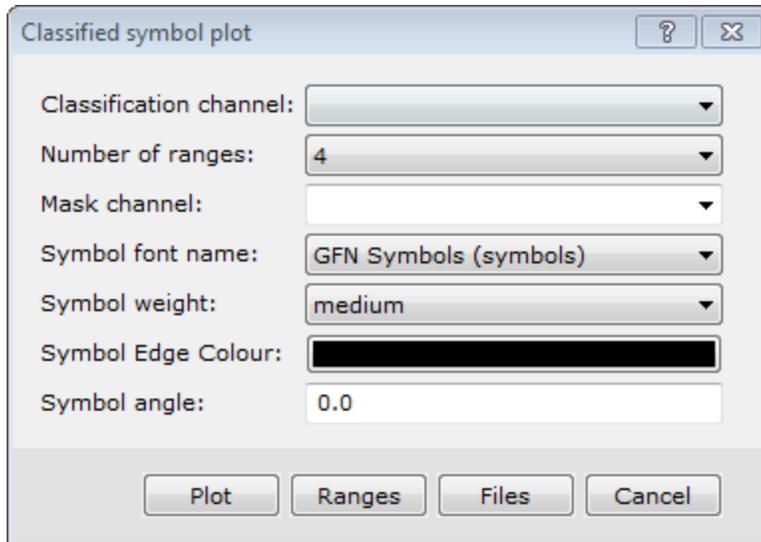
6. Click **OK**. The system plots the zone coloured symbols on the map.

To Plot Colour Range Symbols

The Colour Range Symbols are the most flexible of the entire symbol tools collection. The dialog allows for different ranges of data to be plotted using different symbols, colours and sizes. An example is shown below.

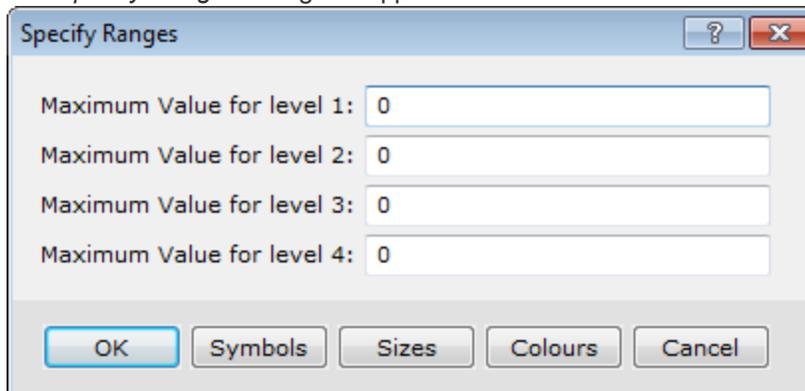


1. On the *Map Tools* menu, select *Symbols* and then select **Colour Range Symbols**. The *Classified symbol plot* dialog box appears.



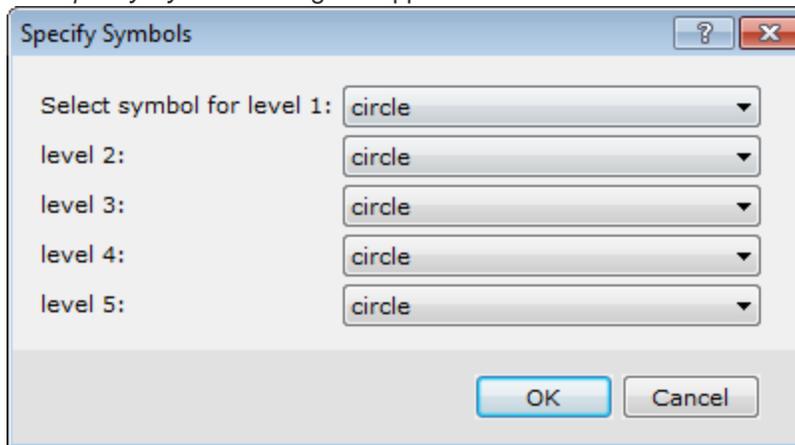
2. Specify *Classification channel* from the dropdown list.
3. Define the symbol parameters as desired.
4. Click **Ranges**.

The *Specify Ranges* dialog box appears.



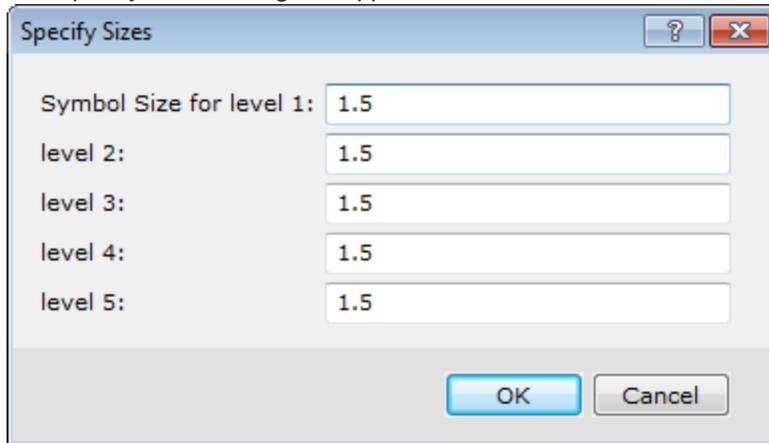
5. Specify the extent of each range by entering the maximum value for each range.
6. Click **Symbols**.

The *Specify Symbols* dialog box appears.



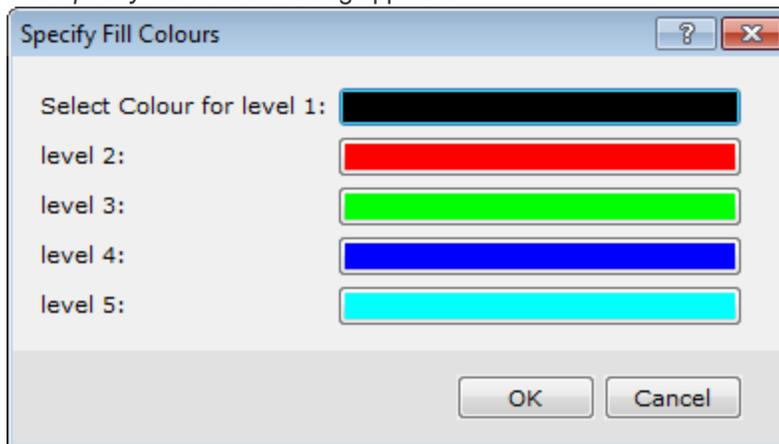
7. Specify the desired symbol shape for each range. In this example, circles were chosen for all of the ranges.
8. Click **OK** to return to the *Specify Ranges* dialog box.
9. Click **Sizes**.

The *Specify Sizes* dialog box appears.



10. Specify the *Symbol Size* for each level.
11. Click **OK** to return to the *Specify Ranges* dialog box.
12. Click **Colours**.

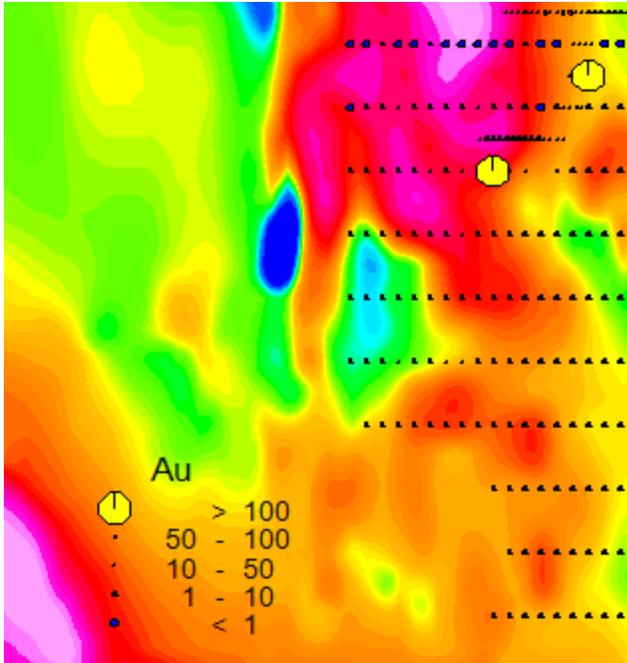
The *Specify Fill Colours* dialog appears.



13. Choose the colours for the 5 levels.
14. Click **OK** to return to the *Specify Ranges* dialog box.
15. Click **OK** to return to the *Classified symbol plot* dialog box.
16. Click **Plot**. The classified symbols are plotted for all 5 ranges on the map.

To Plot a Colour-Range Symbol Legend

When you use classified symbols, you will most likely create a legend showing the symbol colours and size ranges used on the map. An example creating a legend for gold (Au) is shown below.



This option is also used for plotting legends for the zoned colour symbols where a (*.ITR) file must be specified. A (*.ITR) file is created when the zoned colour symbols are plotted. The default name is the channel-name.itr, e.g. Au.itr.



Legends can be created for several symbol types. Please refer to the Application Help files for details on creating other legends.

1. On the *Map Tools* menu, select *Symbols* and then select **Colour Range Symbol Legend**. The *Classified Symbol Legend* dialog appears.

Classified Symbol Legend ? X

Title for Legend (Optional):

Subtitle for Legend (Optional):

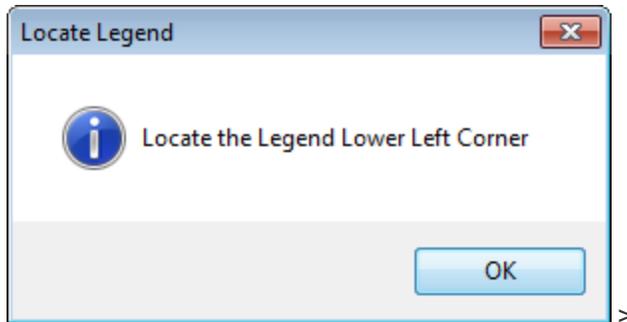
X Position (mm):

Y Position (mm):

Font Size (mm):

Scale Factor for Symbols:

2. Specify the Legend title and Subtitle, if desired.
3. Click **Locate**. The *Locate Legend* dialog appears.



4. Click **OK**. Your mouse will turn into a cross-hair. Left-click where you want the lower left corner of the legend to be located.
5. The system redisplay the *Classified Symbol Legend* dialog box with the *X and Y Position* fields completed.
6. Click **Plot**. The system plots the legend on the map.



When you select this option, the system automatically records the colour, symbol and size settings you specified when creating the range classified symbols. This makes generating a legend easy.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Colour Imaging

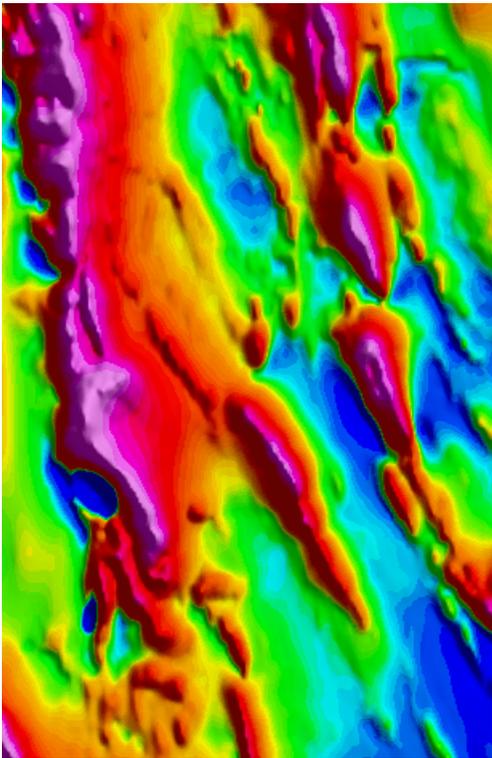
The system provides a variety of capabilities for colour imaging. This How-To Guide introduces you to these concepts.



This tutorial assumes you have a Geosoft project open with a database and map available.

Creating Colour Shaded Images

The shaded relief image is a common method used to present geophysical and other types of data. This presentation technique creates a 3-dimensional effect using 2-dimensional rendering or plotting capabilities. The result is useful for checking the quality of the data and for data interpretation. An example is shown below.



When you create a shaded relief map the system automatically creates a new grid file for that map with **_s** after the original grid name. For example, **mag.grd** will become **mag_s.grd**.

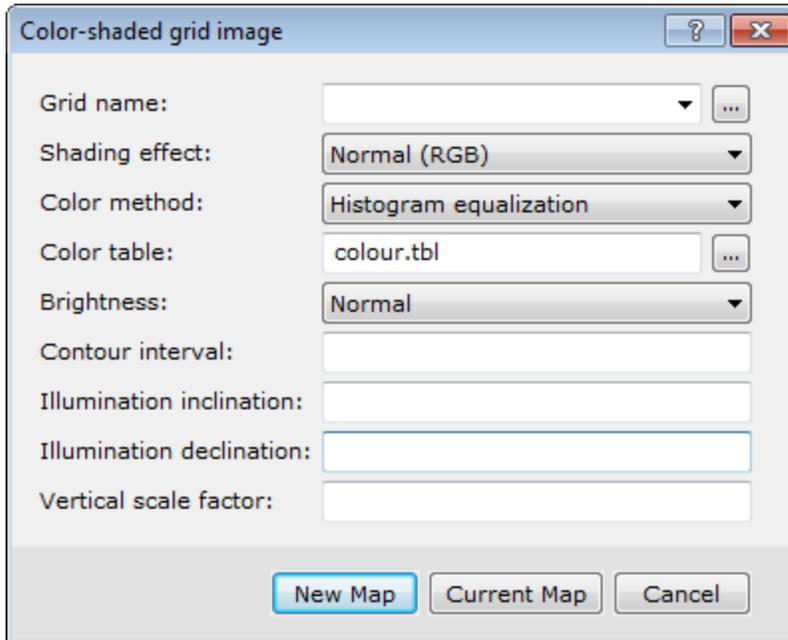
Shaded relief maps can be created interactively using **Run Dynamic Shadowing** from the *Colour Tool* dialog.



You can also manually create a shaded grid using the *Colour-Shaded Grid* option from the *Grid and Image* menu.

To Create a Colour Shaded Image

1. On the *Grid and Image* menu, select **Display** and then select **Colour-Shaded Grid**. The *Colour-shaded grid image* dialog appears.



2. Use the *Browse* button to select the *Grid name*.
3. From the dropdown list, select the *Shading effect*. The options are *Normal* or *Wet-look*.
4. Accept the default parameters for the *Illumination inclination* and *declination* angles. This simply means the sun illumination angles.
5. Click **New Map** or **Current Map**. Selecting *Current map* will overwrite your current map with colour shading.
6. If you select *New Map*, the *Map Exists* dialog appears. This is a safety feature built into the system to make sure that you that you do not overwrite your map.
7. Click *No* so that you do not overwrite your current map. The system prompts you for a *New map name*.
8. Type the name of the new map.
9. Click **OK**. The system will create a colour-shaded image from the grid file and place the image in a new map window.

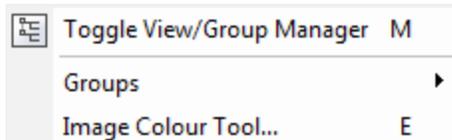
Interactive Colour Imaging and Shadowing

Oasis montaj provides a *Colour Tool* that enables you to modify colour grid zoning interactively on the screen, store custom colour configurations in a specialized colour palette file and apply these to all of your grid image products. The system supports the following transform methods for zoning the data.

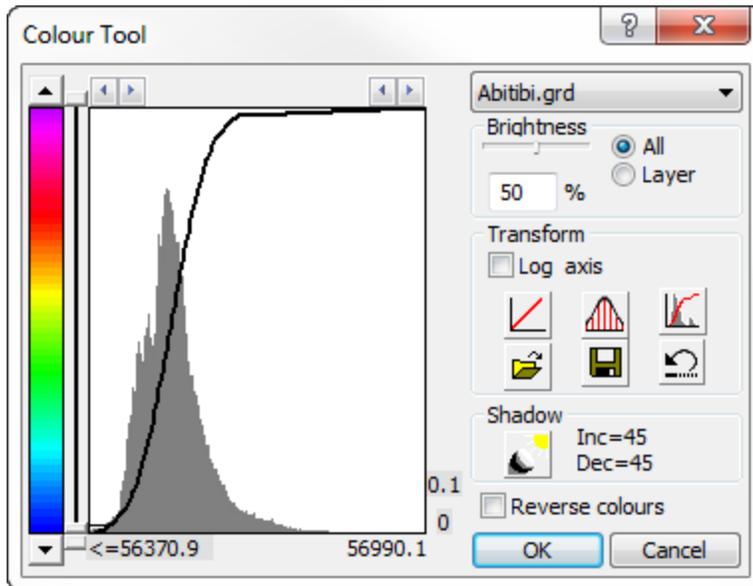
- **Linear** - Divides the range of grid values linearly into even increments
- **Normal** - Assumes your grid data are normally distributed and determines zones based on the distribution for a standard bell curve
- **Equal Area (Histogram)** - Evaluates your grid values and statistically determines the zones, such that each colour occupies an equal area of the map

To Activate the Colour Tool

1. Verify the map and grid are selected in your project.
2. Click *Map Group Mode*.
3. Right-click and select *Image Colour Tool* from the pop-up menu.



The *Colour Tool* dialog appears. This dialog shows the data distribution relative to the selected colour table.



4. From this dialog, you can select to *Transform* the data based on *Linear*, *Normal*, or *Equal-Area* distribution.

To Manipulate the Colour Spectrum / Percentile Groups

1. Press and hold your left mouse button on the left or right arrows at the top corners of the transform box. This will expand or contract your view of the transform in the window.
2. Click on the up or down arrows on the colour slide bar and notice how the colours compress and stretch through the data range. The percentile value is displayed on the right-hand side of the colour tool bar. Also notice how the colour values change at the same time on your map.

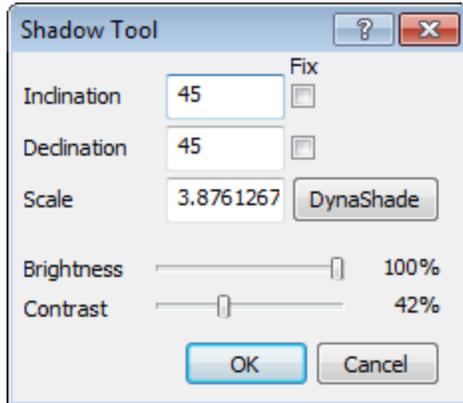


Click **Reset**  to set the colours back to their original settings.

To Activate the Interactive Colour Shading Tool

1. Ensure that the *Colour Tool* is active.
2. Select the image to be used by the *Colour Tool* from the dropdown list that displays the available grids. A new image transform is displayed.

3. Click the *Dynamic shadowing* icon (). The *Shadow Tool* dialog appears.



4. Click *DynaShade*. Your cursor will turn into a sun icon.
5. Click the mouse around the image to change the sun illumination relative to the centre of your map window. When you have adjusted the sun angle as desired, click **Stop**. The cursor returns to normal.



The illumination is relative to the centre of the map window and not the centre of the image.

6. You can also manually set the *Brightness* and *Contrast* settings.
7. When you have completed your shadow adjustments, click **OK**. You are returned to the *Colour Tool* dialog box.
8. Click **OK** to save your changes.

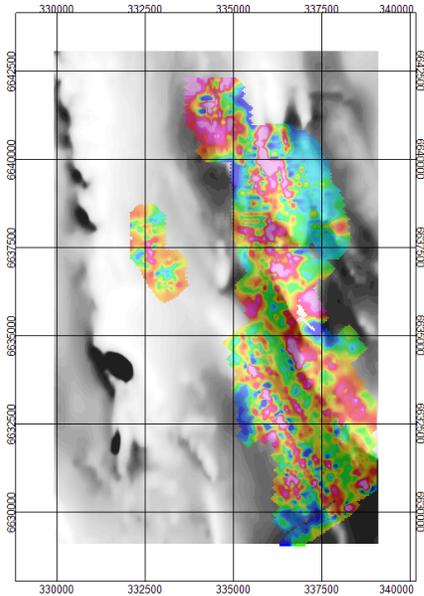
How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Oasis montaj Advanced Imaging

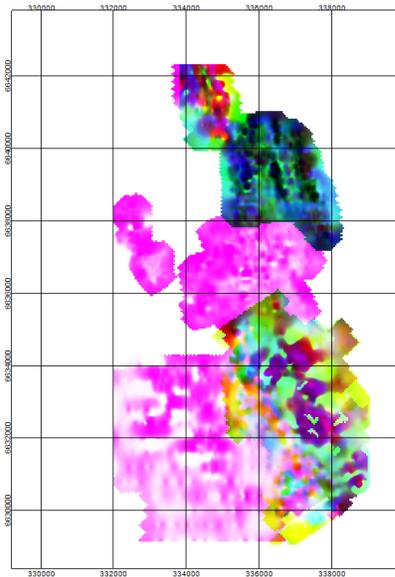
It is often helpful for interpretation to combine multiple types of data together using special colour schemes. It is possible to create maps and add images to them, including single colour grid images, colour-shaded grid images, and JPG (or other image format) files. In addition, the system also supports a variety of advanced options that you can access via the *Display* option from the *Grid and Image* menu.

Two common methods of combining grid images are:



2-Grid Composite Map

This map uses two grids with varying colours. In this example (shown above) we see a greyscale colour shaded grid of topographic data overlain by a colour grid of magnetic data.



Ternary Diagram Presentation

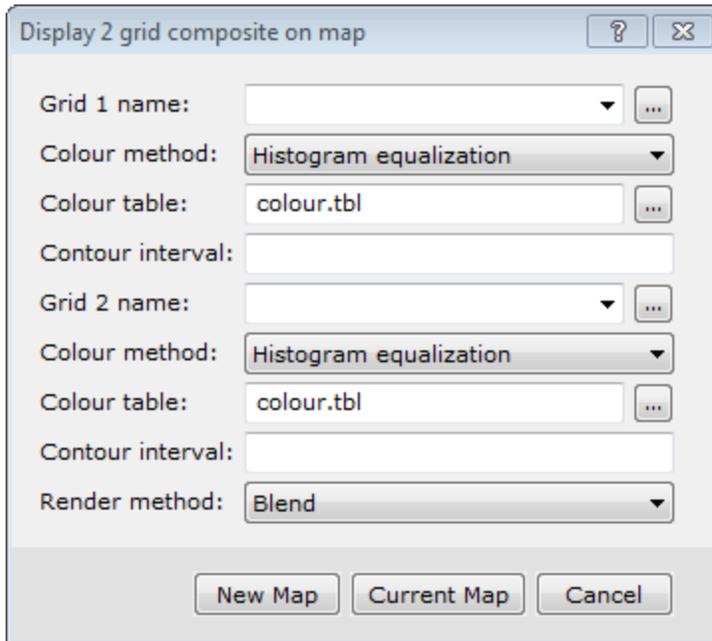
This map uses three grids with varying colours. The map (shown above), includes three separate data grids that are using either a cyan-magenta-yellow (CMY) or red-green-blue (RGB) colour model. This display method is commonly used with airborne radiometric data.

Create Advanced Images

The following workflow describes how to create 2 and 3 grid composite images, as shown in the images above.

2-Grid Composite Map

1. Open the database and the map.
2. From the *View/Group Manager Tool*, hide all *Data* groups except the coordinates group. If your map does not have all these groups (path, profile, etc.) you will not need to hide them.
3. On the *Grid and Image* menu, select *Display* and then select **2-Grid Composite**. The *Display 2 grid composite on map* dialog appears.



4. Select the first *Grid name* from the dropdown list (or use the "browse" button to load the desired grid into the project, if you haven't done so already).
5. Select the second *Grid name* from the dropdown list.
6. Specify *Colour* parameters for each grid. In order to differentiate between the grids, each grid should be displayed using different colour tables.



To replicate the example above, one grid would be mapped using the grey.tbl Colour table option. The other grid would use the default colour.tbl.

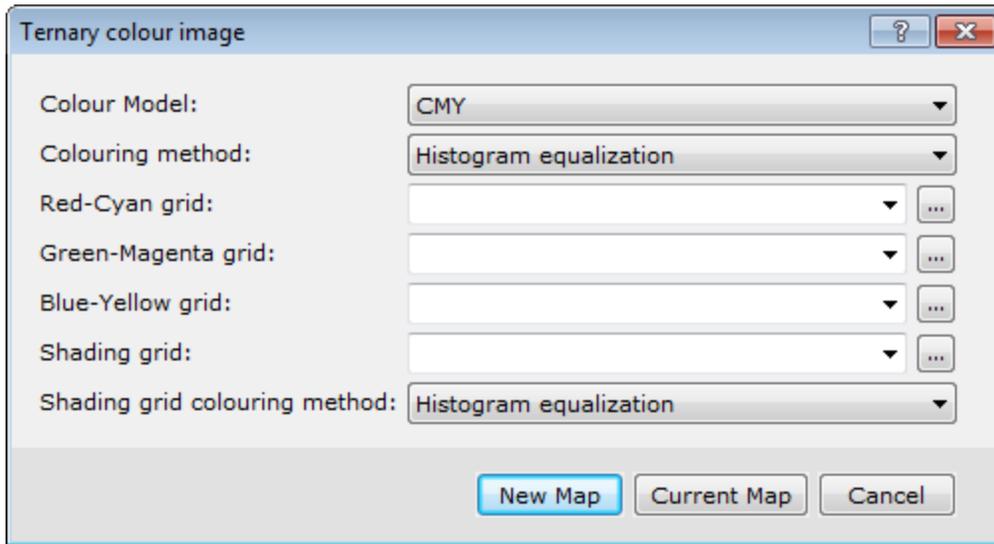
7. Select the *Render method* from the dropdown list. This determines how the colours interact on the map. The default is *Blend*.
8. Click **Current Map** to load the map into the currently open map or **New Map** to create a new map using the specified settings.

3-Grid Ternary Plot

Before you create a 3-grid ternary plot, you must have created grid files, (*.grd) for the 3 channels from the selected database.

1. Open the database and the map.
2. From the *View/Group Manager Tool*, hide all *Data* groups except the coordinates group. If your map does not have all these groups (path, profile, etc.) you will not need to hide them.

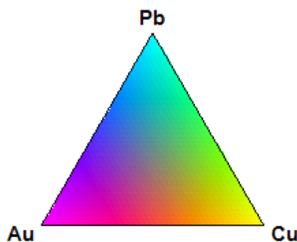
- On the *Grid and Image* menu, Select *Display* and then select **Ternary Image**. The *Ternary colour image* dialog appears.



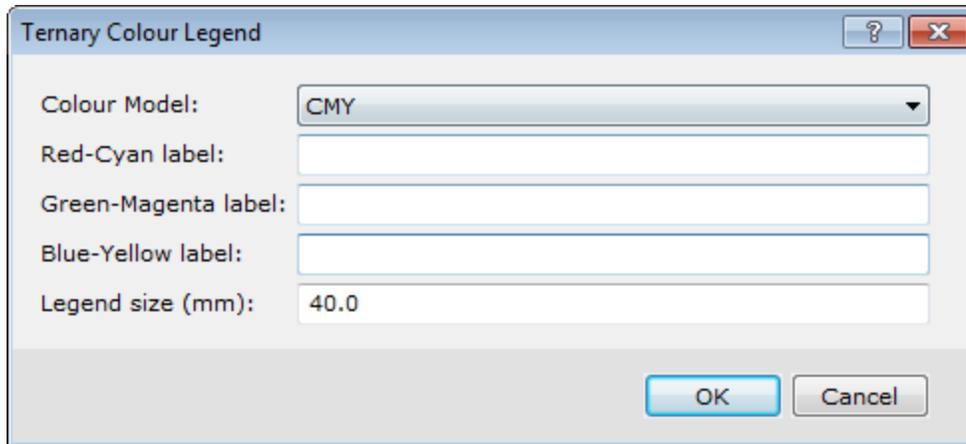
- Select the *Colour Model* from the dropdown list. This determines which colour tables will be used to draw the map. The default is *CMY*.
- Select the *Colouring method*.
- Select the grid to control the *Red-Cyan* colour from the dropdown list (or use the "browse" button to load the desired grid into the project, if you haven't done so already).
- Select the grid to control the *Green-Magenta* colour from the dropdown list.
- Select the grid to control the *Blue-Yellow* colour from the dropdown list.
- The *Shading grid* is an optional grid that controls the shading colour. Specify shading parameters, if desired.
- Click **Current Map** to load the map into the currently open map or **New Map** to create a new map using the specified settings.

To Add a Ternary Legend to a Ternary Map

A Ternary Legend is a triangle legend added to the map which allows you to designate a label for the map colours. An example is shown below.

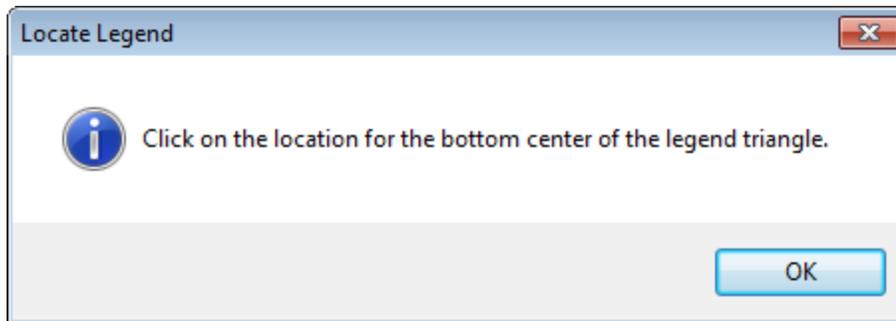


- On the *Grid and Image* menu, select *Display* and then select **Ternary Legend**. The *Ternary Colour Legend* dialog box appears .



2. Select the *Colour Model* from the drop-down list. This should match the colour model selected for the map.
3. Enter the *labels* for the colours. This should reflect the grids chosen for the specific colours.
4. Specify the *Legend size*.
5. Click **OK**.

The *Locate Legend* dialog appears.



6. Click **OK** and left-click the desired location on the map to display the legend. The system creates and displays the ternary legend on your map.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

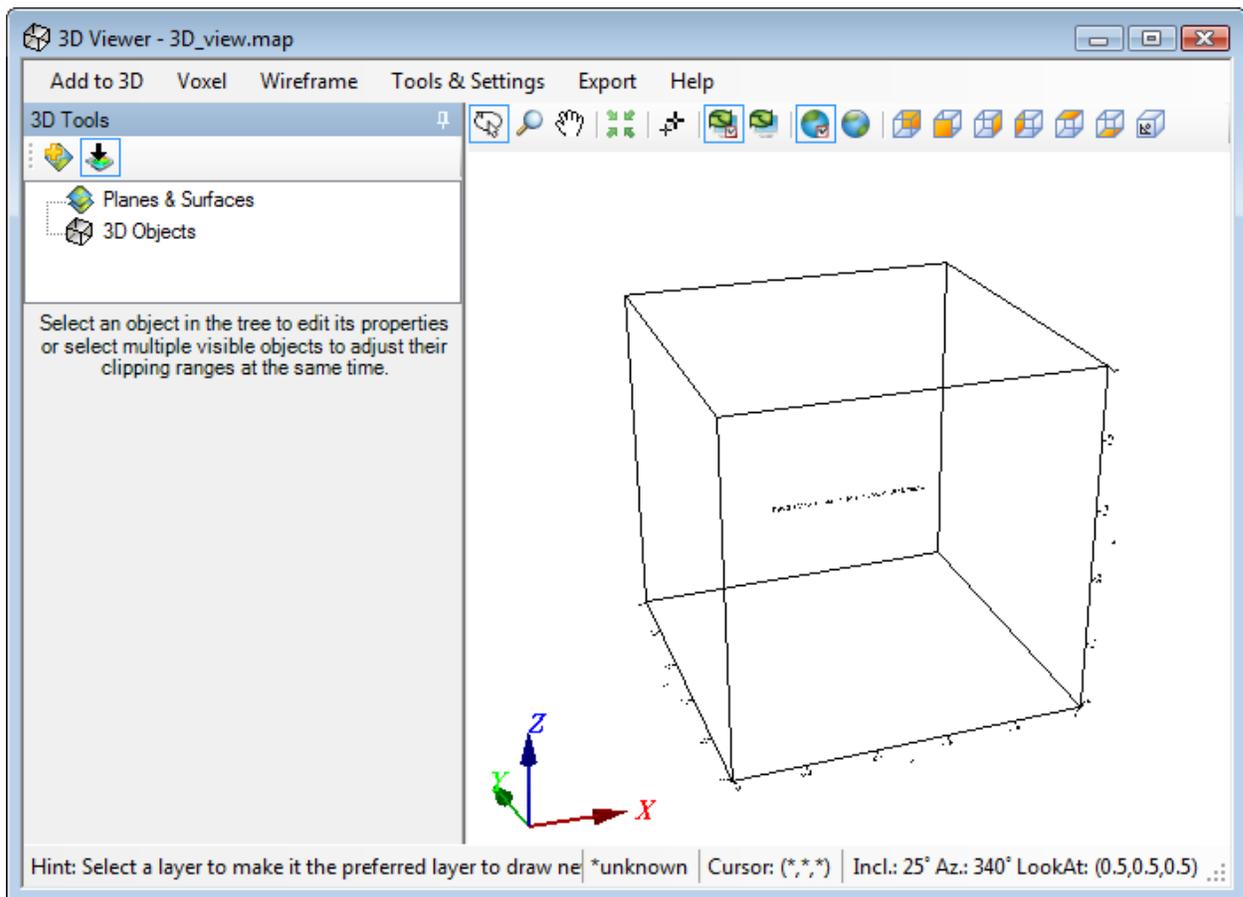
Oasis montaj 3D Viewer

The *3D Viewer* has been redesigned to streamline your workflow and provide enhanced usability. The *3D Viewer* provides all of your 3D mapping tools, functions and settings in one integrated 3D environment.

This How-To Guide introduces you to the *3D Viewer* environment and shows you how to work with the 3D tools, including creating 3D views, planes, symbols and voxels.

Opening the 3D Viewer

When you display a 3D view, either by using the *Open 3D Viewer* menu option, or by selecting (double-click) a 3D view in the *View/Group Manager Tool*, or by simply double-clicking a 3D view in an open map file, the 3D view will be displayed in the *3D Viewer*.



Navigating the 3D Viewer

The *3D Viewer* consists of an interactive dialog with five Menus (*Add to 3D*, *Utilities*, *Settings*, *Export* and *Help*), a *3D Viewing window*, a *3D Tools window* and the bottom *Status Bar*.

➤ Navigating the 3D Viewer

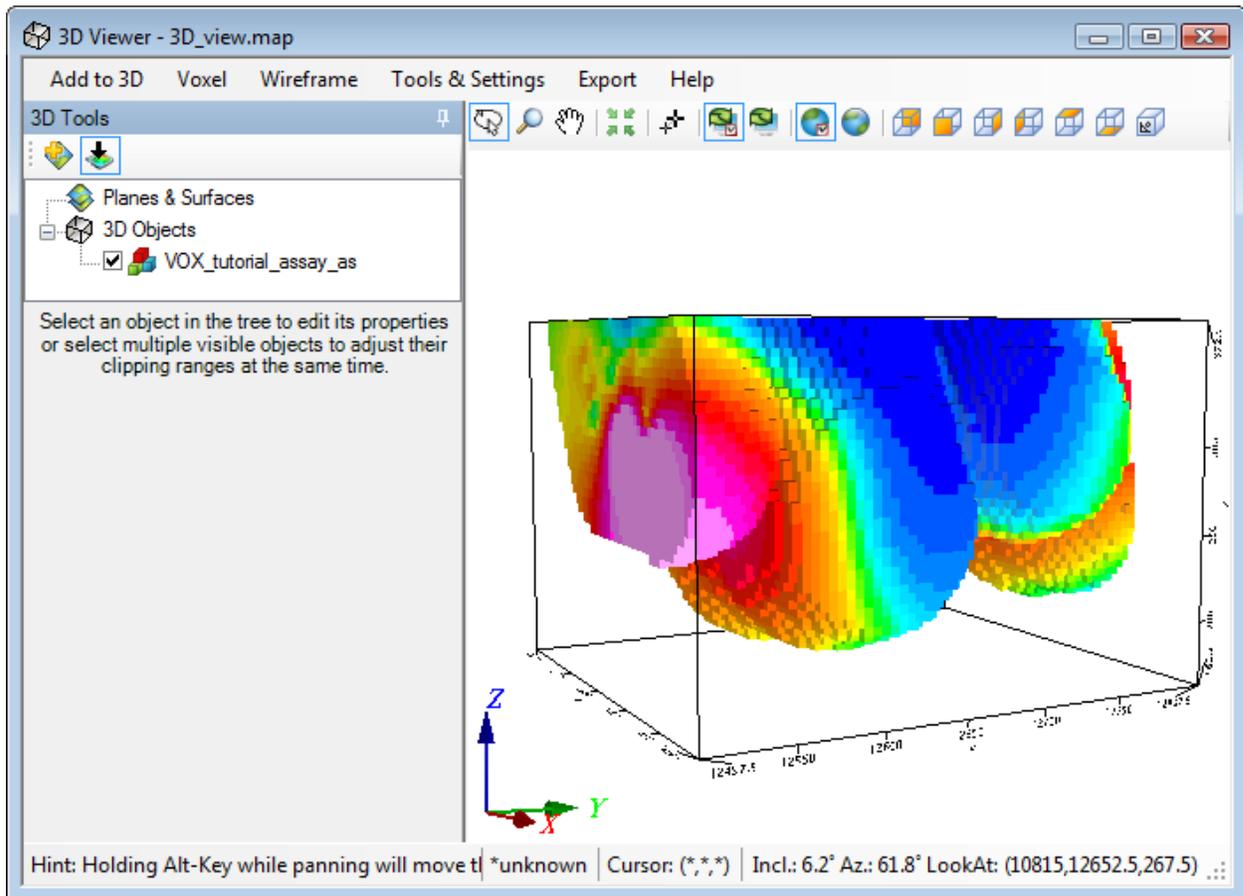


For more detailed information on the 3D Viewer select **Help** on the 3D Viewing window.

The *Geosoft 3D Viewer* provides an interactive 3D view of your data and is used along with the *3D Tools* and menu items to interactively control the appearance of your 3D view. When you are satisfied with the appearance of your 3D view in the *3D Viewer*, simply close the *3D Viewer* window (click the X button in the top-right corner) and the *3D Viewer* will be closed and the 3D view will be displayed on your current map.

Once a 3D view has been added to a map, the 3D view can be modified by opening the 3D view using the **Open 3D Viewer** button, which opens the 3D view for edit. When you are satisfied with your changes, closing the *3D Viewer* saves your changes and displays the updated view on the current map.

A sample 3D view is shown below.



3D Tools

The parameters available in the *3D Tools* window are dependent on the type of 3D group currently selected. Each group enables you to modify different aspects of the 3D view within the 3D Viewer. Select the *Attribute* and *Clipping* tabs to access the 3D display tools.

3D Viewing Window

Interactive viewing buttons are provided on the toolbar directly above the *3D Viewing window*.

Status Bar

The *Status Bar* at the bottom of the *3D Viewer* displays helpful navigation hints, along with the current 3D view's coordinate system, the shadow cursor location, the view's inclination, declination and look at point (centre of rotation).

Notes about 3D Views

3D Views are rendered coarsely when interactively changing the orientation or point of view. When you release the mouse in any of these modes, the objects will be refined in a way that is optimum for the given point of view. The time it takes to refine the picture depends on processor speed, your 3D graphics capabilities, the complexity of the 3D object being drawn (number of planes, number of groups, relief resolution, etc.), and the amount of RAM on your system.

If a plane has a relief view, the groups drawn on the plane will only appear where the relief view is defined. Groups drawn on flat planes (those without relief Views) will be drawn to the extents of the group.

3D Views may have clipping areas just like 2D Views. In 3D views, the clip area is applied vertically relative to the X, Y coordinate of each plane.

3D Views in Oasis montaj

Oasis montaj 3D views are a representation of a 3D drawing space on a standard Geosoft map. The 3D capabilities enable you to display multiple surfaces, each with its own relief and contents, and each with its own orientation in 3D space.

Creating 3D views requires the licensed processing engine, but 3D views on an existing map can be viewed and manipulated in the Oasis montaj free Viewer.

Understanding Views

Geosoft Maps use views to organize and display information. Most Geosoft maps contain both a Base view and a Data view, but some maps may have more views for different purposes. Views contain Groups, and these groups in turn contain drawing elements. All Groups within a 2D View are drawn in the coordinate system of that view. The Base view uses coordinates in millimetres relative to the final printed paper, and the Data view uses a coordinate system that can represent a projected or geographic coordinate system on the earth, or simply an arbitrary X, Y Cartesian coordinate system. 2D Views contain one flat drawing surface, which is defined by the view's coordinate system.

Like 2D Views, a 3D view also represents coordinates, but in this case in X, Y and Z. 3D views may contain Voxel (3D Grids) and any number of drawing planes. Any Group or drawing object that can be drawn on a 2D view can be drawn on a plane in a 3D view. Each plane in a 3D view can be oriented independently in the 3D coordinate system. A plane in a 3D view can be flat or it can have a relief surface defined by a grid file.

Planes will normally share the X, Y coordinates of the 3D view, and flat planes will have a constant Z value representing the vertical location of the plane. However, a plane coordinate system may be oriented in any direction in the 3D space for special applications. If a plane does have a Relief view, the axis of the relief is always in the Z direction of the plane.

3D views appear on a 2D map as a standard view, except that what you see is the 3D representation of the view from a fixed perspective. You can manipulate a 3D view using the *3D Viewer*. To do this, select the 3D view, right click and select **3D Viewer** from the pop-up menu, or simply click the **Open 3D Viewer** icon, and the 3D view will be displayed in the *3D Viewer*.



You must be in **Map View Mode** to select a view

The *3D Viewer* provides an integrated 3D environment that includes all of the tools, functions and settings to create a 3D map. Using the *3D Viewer* you can add any number of 3D and 2D views and groups to your map, including; planes, relief surfaces, grids, voxels, vector voxels, isosurfaces, drillholes, geostrings, geosurfaces, 3D symbols, 2D map groups, and 3D import file formats.

When viewing the 3D view in the *3D Viewer*, you can change the orientation of the view relative to your point of view by rotating the coordinate system horizontally and vertically, and you can zoom and pan into the view, and you can modify certain display parameters such as delete planes, and adjust plane transparency using the 3D Tools. The 3D Tools enable you to modify the 3D parameters, display characteristics and the appearance of your 3D view in the *3D Viewer*. The 3D Tools are available in the window on the left side of the *3D Viewer*.

Additional options are available using the *3D Viewer* menus:

- Add to 3D
- Voxel
- Wireframe
- Tools & Settings
- Export
- Help

When you close the *3D Viewer* window, the 3D view will be drawn on the 2D Map as it last appeared in the *3D Viewer*.

Creating a New 3D View in the 3D Viewer

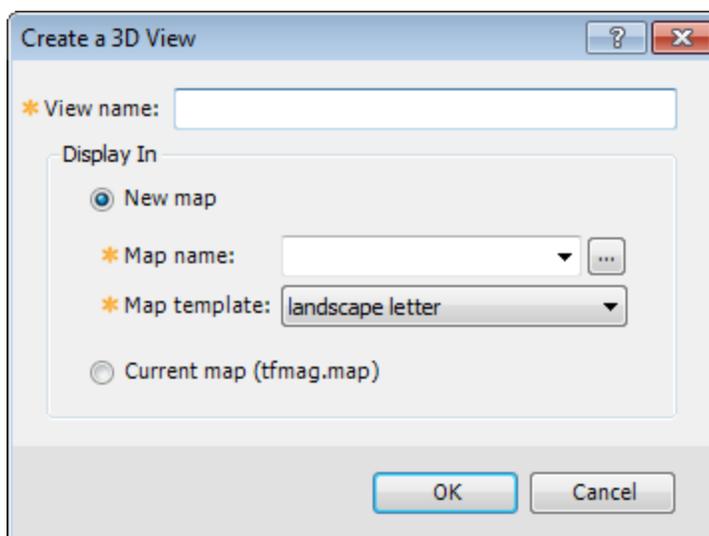
Using the *Create a New 3D View* dialog you can add a new 3D view to a new 3D map or a current map.

To Create a New 3D View

Newly created 3D views will open in the *3D Viewer*.

1. On the *3D* menu, select **Create a New 3D View**.

The *Create a 3D View* dialog appears.



2. In the *View name* text box, specify a name for your new 3D view. The default is *3D*.

- Using the radio buttons, select either **New map** or **Current map**.



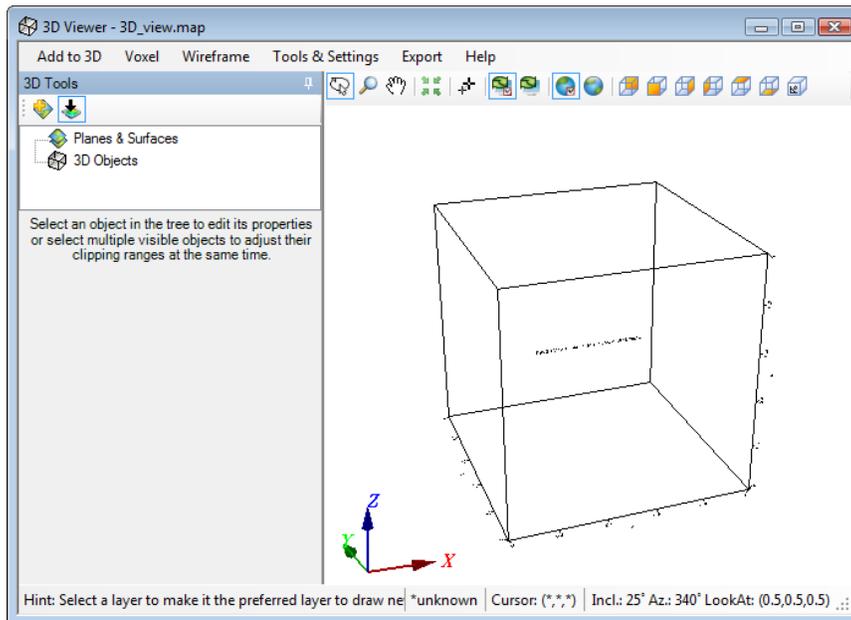
The current map name is displayed to the right of the **Current map** radio button.

- If you select **New map**, the *Map name* and *Map template* fields become enabled. Specify the parameters.



It is extremely important that when you create a new 3D view that you create a new name for the view rather than selecting a name from the dropdown list. The dropdown list is presented to enable you to see what view names you may already have. **Do not overwrite one of the existing 2D views, unless you want to replace it.**

- Click **OK**. A new empty 3D view is created and displayed in the *3D Viewer*.



The empty 3D view will display the following message in the 3D view area, "*This 3D View Contains No Surfaces or 3D Groups.*"

Working with Planes

A 3D view may contain any number of planes and new planes can be added to any 3D View.

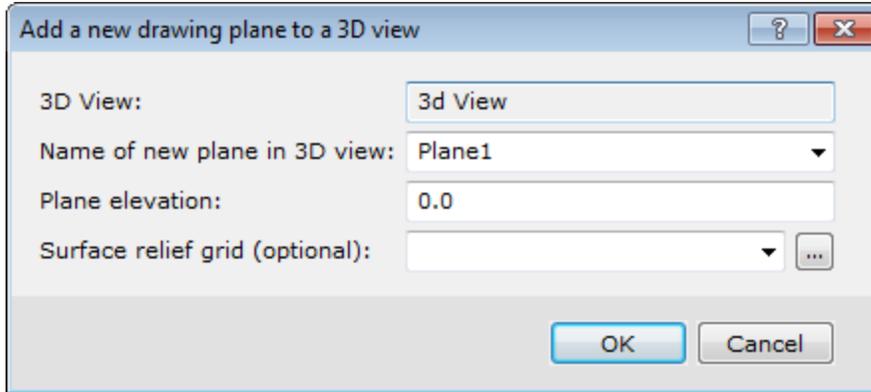
To add a Plane to an Existing 3D View

Before you begin, verify you have an existing 3D View open in the *3D Viewer*.

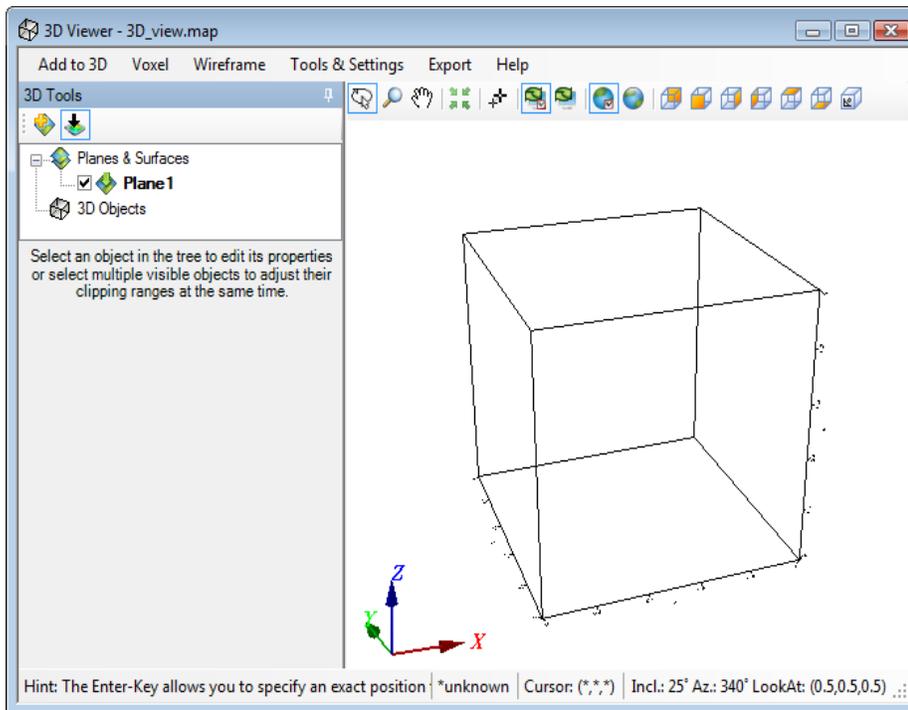
- On the *Add to 3D* menu, select **Plane**.
The *Add a new drawing plane to a 3D view* dialog appears.



You can also click the **Add Plane** button.



2. Specify the *Name of new plane in 3D view* . The default is *Plane1*.
If you enter the name of an existing plane, that plane will be **replaced!**
3. Specify the *Plane elevation*.
4. You can also select a *Surface Relief grid* if you would like the plane to have relief.
5. Click **OK**. The new plane is added to the current 3D view in the *3D Viewer*. An example is shown below.



Using the *Navigation tools* and the *3D Tools* in the *3D Viewer* window you can orientate the view to the desired perspective. When you are satisfied with the orientation close the *3D Viewer*.

To Set a Plane as the Default Drawing Plane

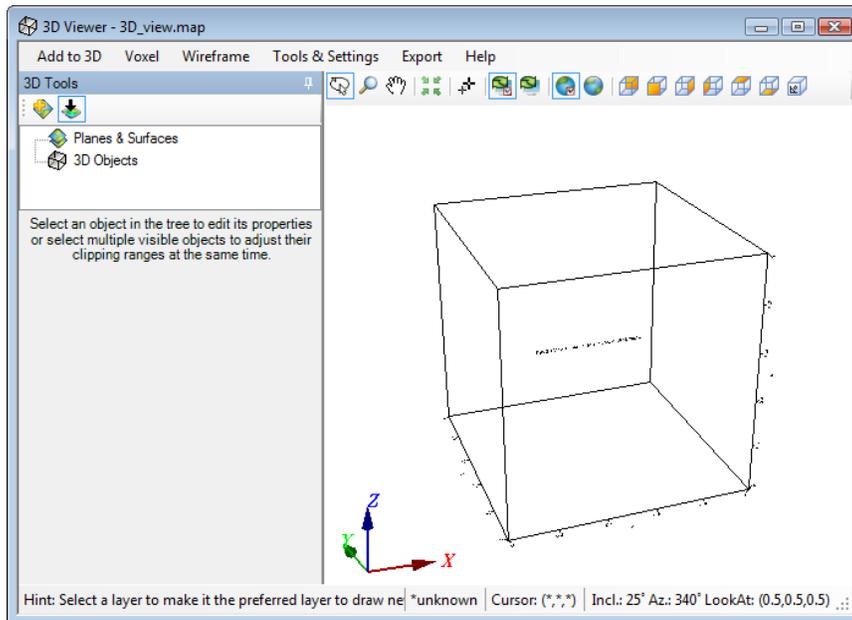
You can draw groups including grids, images, contours, and symbols to a plane in a 3D view just like drawing to a 2D view. After creating a 3D view, it becomes your *Default Drawing View* so that the next group you plot will be drawn on the *Default Drawing Plane* which is the most recent plane in the view, unless you have set it otherwise.



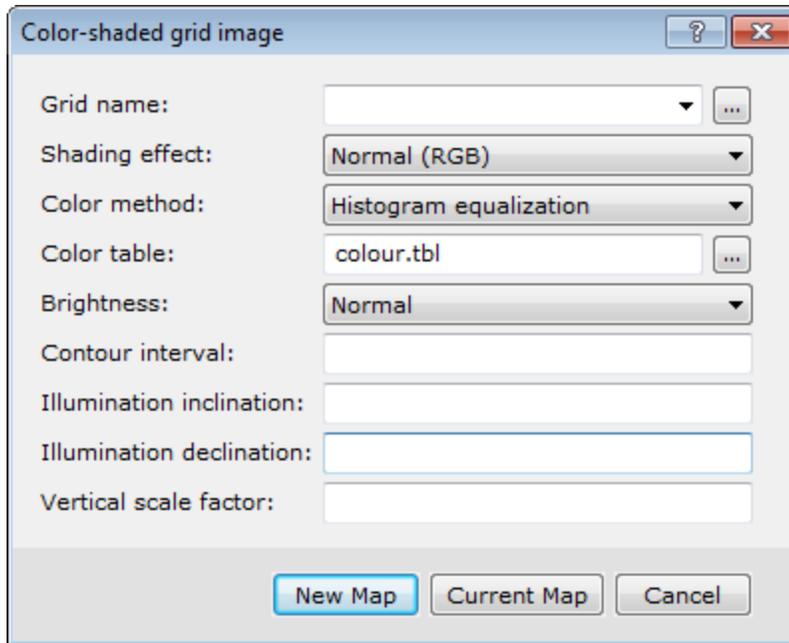
You can also use the clipboard to cut/copy groups from other maps to the default drawing plane of a 3D view.

Before you begin, verify that you are in **Map View Mode**. This example walks you through setting a Colour-shaded image grid as the default drawing plane. You may follow the same basic procedure for different types of grids.

1. From an empty 3D view, right-click the 3D view and select **3D Viewer** from the pop-up menu. The empty 3D view opens in the *3D Viewer*.



2. Select (highlight) the *Planes & Surfaces* item.
3. On the *Add to 3D* menu, select *2D Map Tools*, then select *Grid and Image Display* and then select **Colour-Shaded Grid**.
The *Colour-shaded grid image* dialog appears.



4. Use the *Browse* to locate the *Grid name* from your working directory.
5. Set the desired grid parameters.

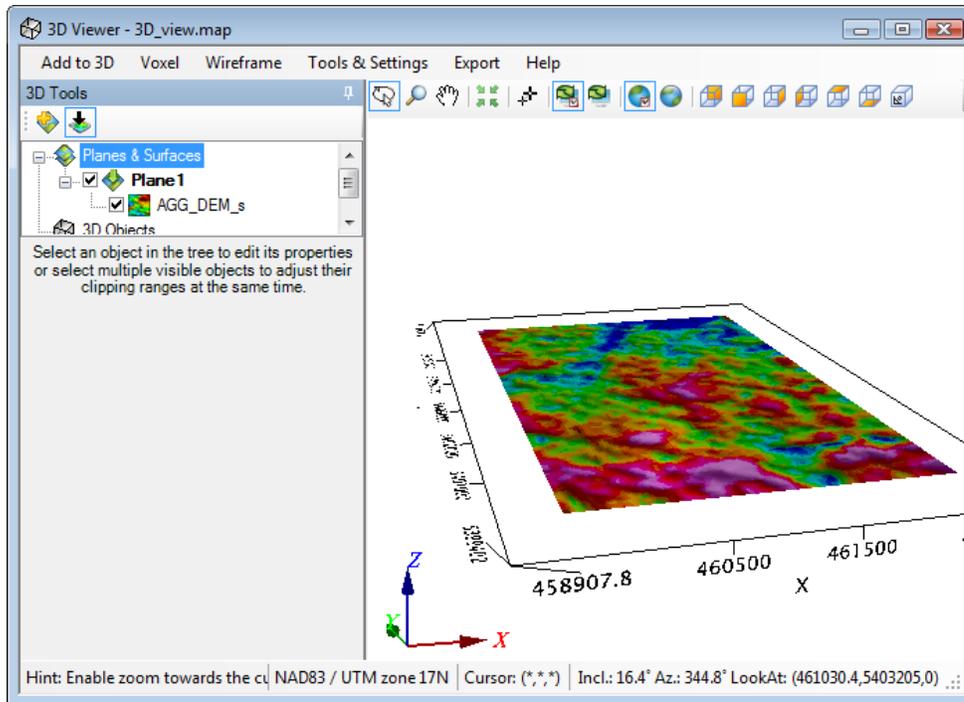


*For more information on Color-shaded grid image settings, click the **Help** button.*

6. As we are adding this plane to our open empty view, click **Current Map**. The colour-shaded grid will be displayed on the current drawing plane in the *3D Viewer*. An example is shown below. When you close the *3D Viewer* your 3D view will be displayed in the centre of the map file.



*To create a new map, click **New Map** and follow the steps to create a new map.*



Plotting 3D Colour Range Symbols

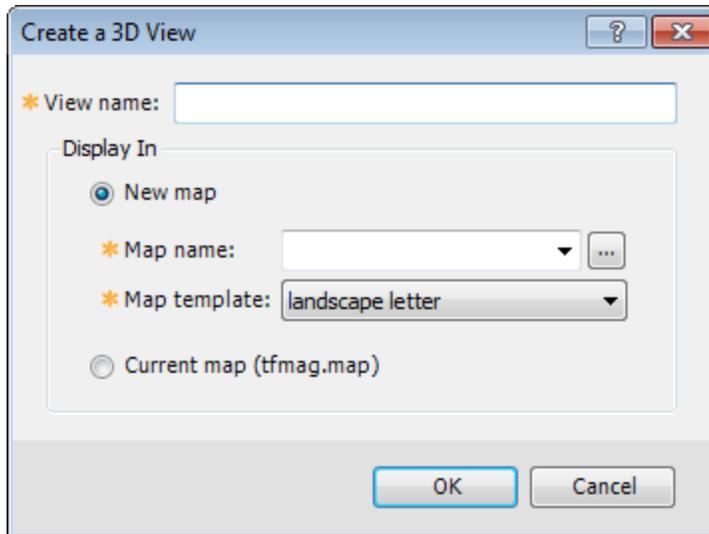
Oasis montaj provides a range of options for visualizing your data in three dimensions including displaying multiple surfaces, each with own relief and contents, and each with its own orientation in 3D space.

The 3D colour range symbol plotting option enables you to plot 3D symbols with fixed or variable colours to a 3D map. The colours can be varied based on the values from another data channel.

To Plot 3D Colour Range Symbols

Before you begin, verify the database is open.

1. On the **3D** menu, select **Create a New 3D View**.
The *Create a 3D View* dialog appears.

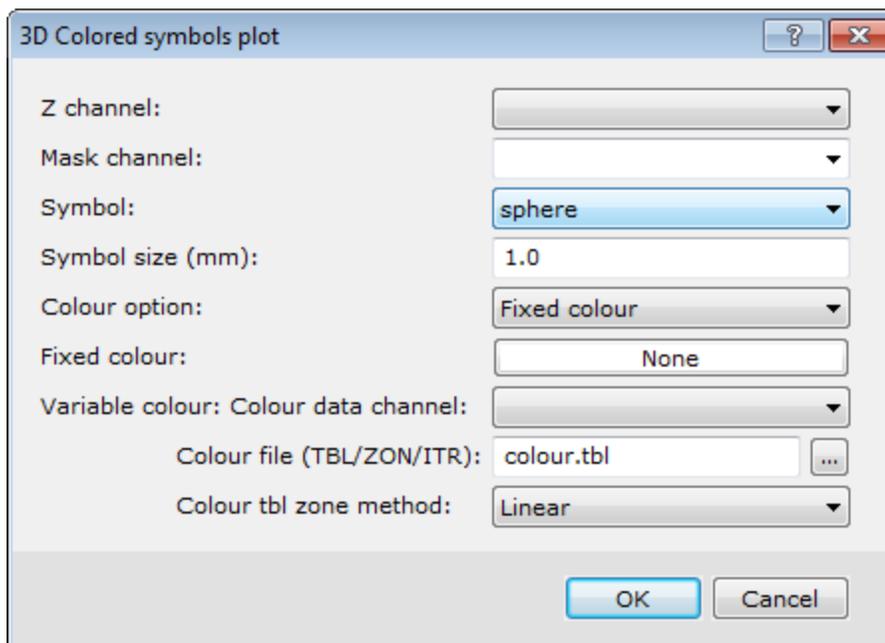


2. In the *View name* text box, specify a name for your new 3D view. The default is *3D*.
3. Using the radio buttons, select either **New map** or **Current map**.



*The current map name is displayed to the right of the **Current map** radio button.*

4. If you select **New map**, the *Map name* and *Map template* fields become enabled. Specify the parameters.
5. Click **OK**. A new empty 3D view is created and displayed in the *3D Viewer*.
6. On the *Add to 3D* menu, select **3D Symbols**.
The *3D Colored symbols plot* dialog appears.



7. Using the dropdown list, select the *Z channel*.
8. Select the *Mask channel*, if desired. If the use of the mask channel is enabled, and set to this

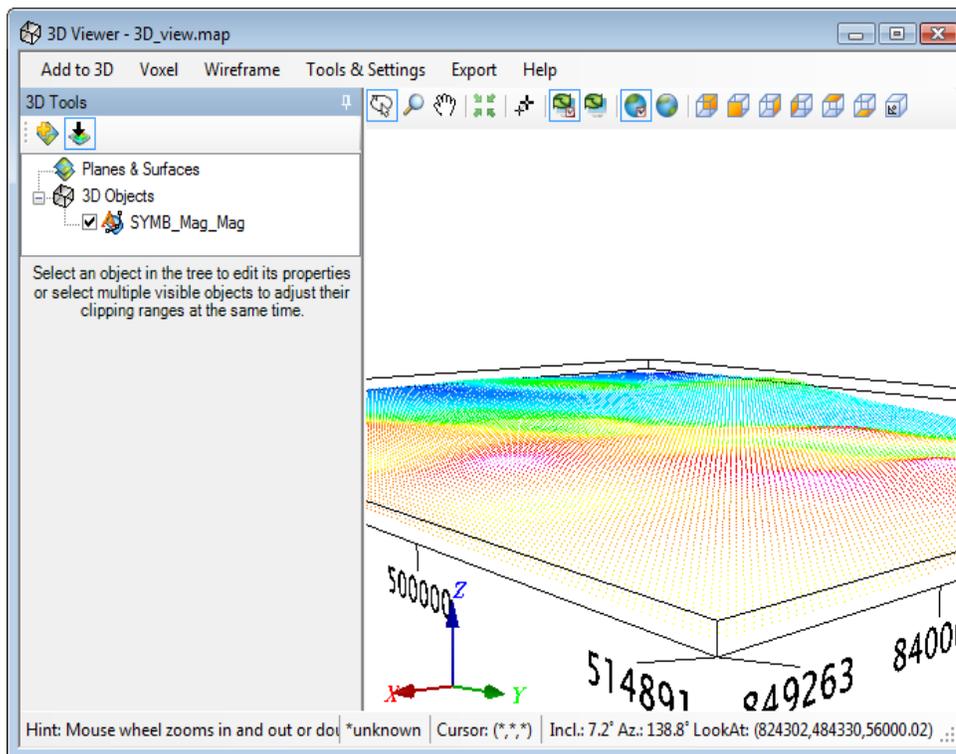
channel, only those data values whose corresponding mask channel value is not a dummy will be plotted or exported.

9. Specify the *Symbol* and **Symbol size** (in mm).
10. The *Colour option* can be set to either *Fixed* or *Variable*



If you choose Fixed colour, set the colour to be the value specified under Fixed colour.

11. If you would like to colour your data using the values from a secondary channel, select the *Variable colour* option and then specify the *Variable colour: Colour data channel*.
12. To change the *Colour table* from the default colour.tbl file, specify the desired TBL, ZON or ITR file.
13. Select the *Colour method*: *Histogram equalization*, *Normal distribution*, *Linear* or *Log-linear*.
14. Click **OK** to plot the 3D colour range symbols to the *3D Viewer*. An example is shown below. When you close the *3D Viewer* the view will appear on the associated 2D map just as it last appeared in the *3D Viewer*.



Working with Voxels

Use the **Voxel** option on the *Add to 3D* menu to place a Voxel in a 3D View.

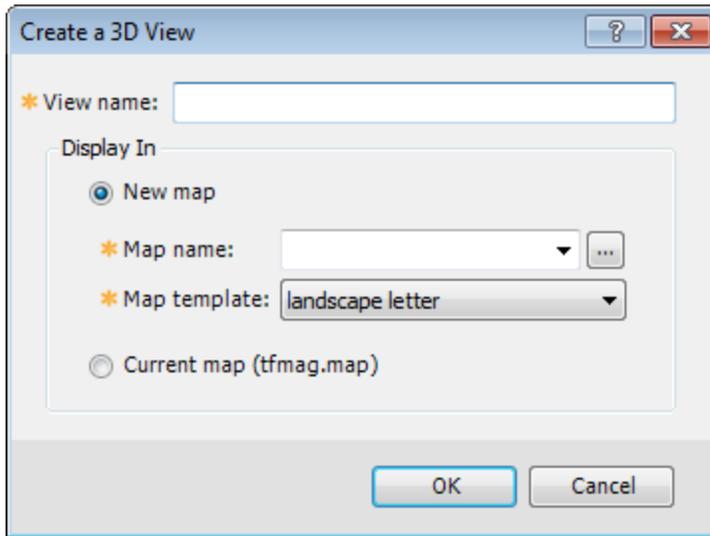


Voxel is just another word for a 3 dimensional grid.

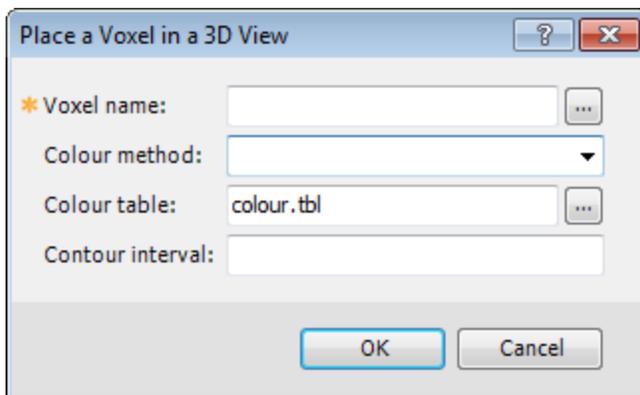
To Display a Voxel in a 3D View

This exercise assumes you have an available voxel to display. You can create a Voxel using the *3D Gridding* option on the *3D* menu.

1. On the **3D** menu, select **Create a New 3D View**.
The *Create a 3D View* dialog appears.

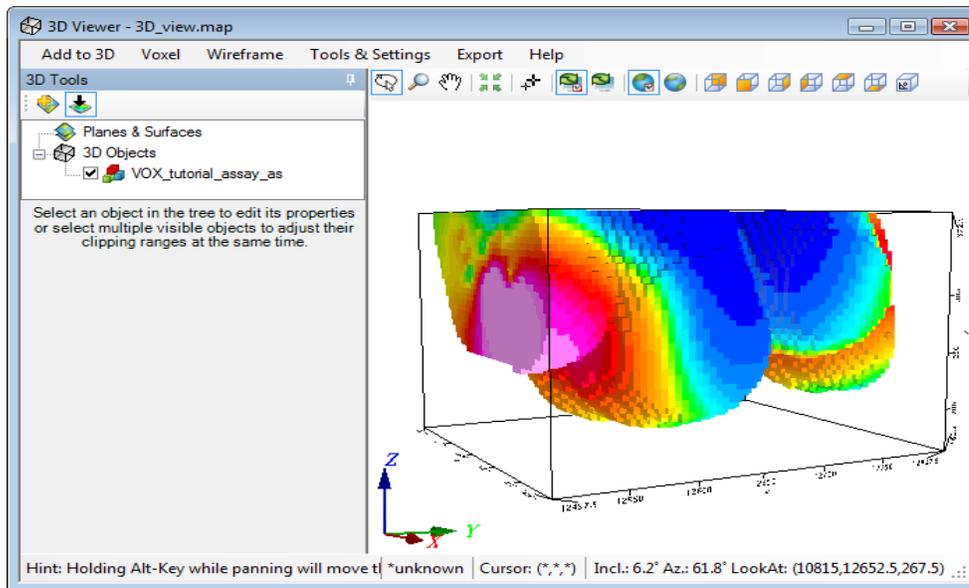


2. In the *View name* text box, specify a name for your new 3D view. The default is *3D*.
3. Using the radio buttons, select either **New map** or **Current map**.
 *The current map name is displayed to the right of the **Current map** radio button.*
4. If you select **New map**, the *Map name* and *Map template* become enabled. Specify the parameters.
5. Click **OK**. A new empty 3D view will be created and displayed in the new map in the background and in the *3D Viewer* in the foreground of your project.
6. On the *Add to 3D* menu, select **Voxel**.
The *Place a Voxel in a 3D View* dialog appears.



7. Use the *Browse* button to locate the *Voxel name*.
8. Select the *Colour method*: *Default*, *Histogram equalization*, *Normal distribution*, *Linear* or *Log-linear*. If you leave the field blank, the default will be applied.
9. To change the *Colour table* from the default *colour.tbl* file, specify the desired TBL, ZON, AGG or ITR file.

10. Specify the *Contour interval*, if required.
11. Click **OK**. The voxel file appears in the 3D viewer. An example is shown below. When you close the 3D Viewer the Voxel will appear on the 2D map just as it last appeared in the 3D Viewer.



Select (highlight) the voxel to enable the 3D tools. These controls enable you to change the different attributes that make up the 3D View. For example, the Transparency of the voxel can be set here.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

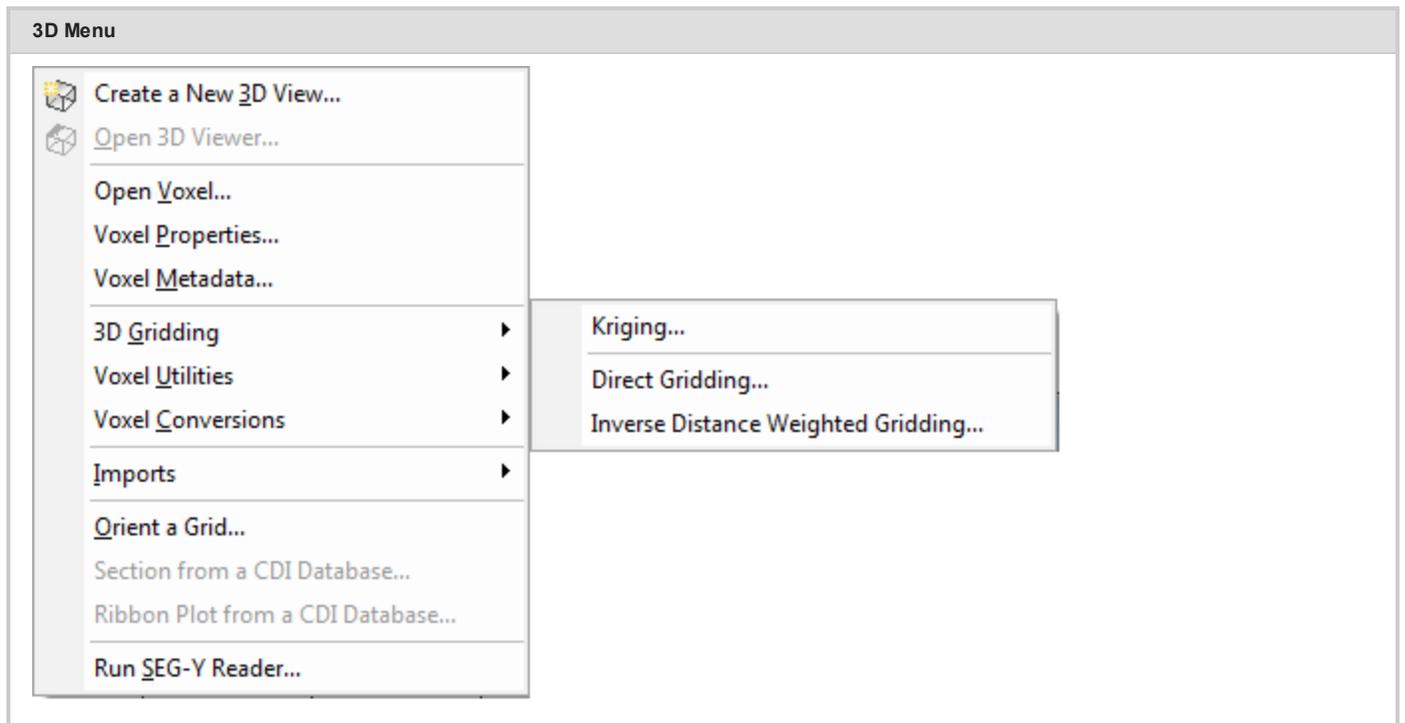
3D Gridding Methods in Oasis montaj

A "Voxel" is short for "volume pixel", the smallest distinguishable box-shaped element of a three-dimensional image. A Voxel is the 3D conceptual counterpart of the 2D pixel.

You can create a Voxel using the *3D Gridding* submenu, located on the *3D* menu or in the *3D Viewer* from the *Voxel|3D Gridding* menu.

The *3D Gridding* submenu offers 3 different gridding algorithms, each with a different focus:

- ▶ The *Kriging* method best suits clustered data occurring in a region with otherwise sparse data distribution, such as geochemistry data.
- ▶ The *Direct Gridding* method on the other hand should be used to grid data uniformly and densely sampled throughout the region, such as GPR or seismic data.
- ▶ The *Inverse Distance Weighted Gridding* method is most suitable for gridding data collected along specific directions - such as borehole data.



The data to be gridded must have defined X, Y and Z coordinates. To define the coordinates use the menu, *Coordinates|Set Current X,Y,Z Coordinates*. The 3 dimensional coordinate system will locate the data in 3D space. The Z axis is interpreted as positive up. The **3D** workflow includes tools for creating, displaying and processing voxels. The *3D Viewer* enables visual inspection of voxels and comes with an extensive set of 3D viewing tools.

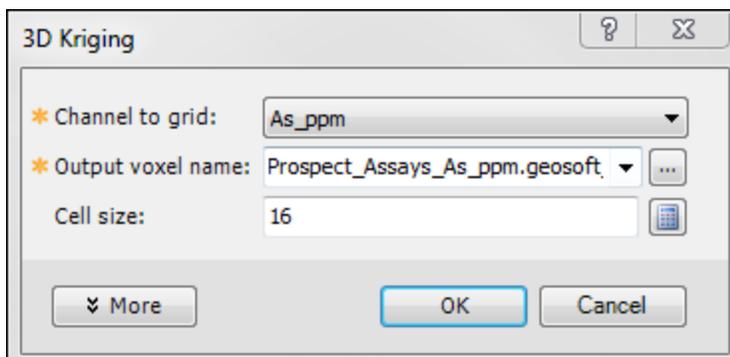
-  When applying 3D gridding, halving the cell size increases both the file size and processing time by a factor of 8 (2³).

Creating a Voxel using the Kriging Gridding method

From the *3D Gridding* menu, select the **Kriging** option to perform a statistical-based 3D gridding. The input data must be supplied in a Geosoft database (GDB file). The output will be a voxel model within the confines of the data extent.

To Create a 3D Voxel model using Kriging:

1. Open a Geosoft database containing X, Y, Z and the data channel you would like to grid. If the coordinate system is not completely defined, a message will pop up and you will not be able to continue until the corrective action is taken.
2. In the 3D Viewer, select *Voxel*, then select *3D Gridding* and then select **Kriging** (or on the main Oasis montaj *3D* menu, select *3D Gridding* and then select *Kriging*. The *3D Kriging* dialog appears.



-  If a database is not already open you will be prompted to open one.

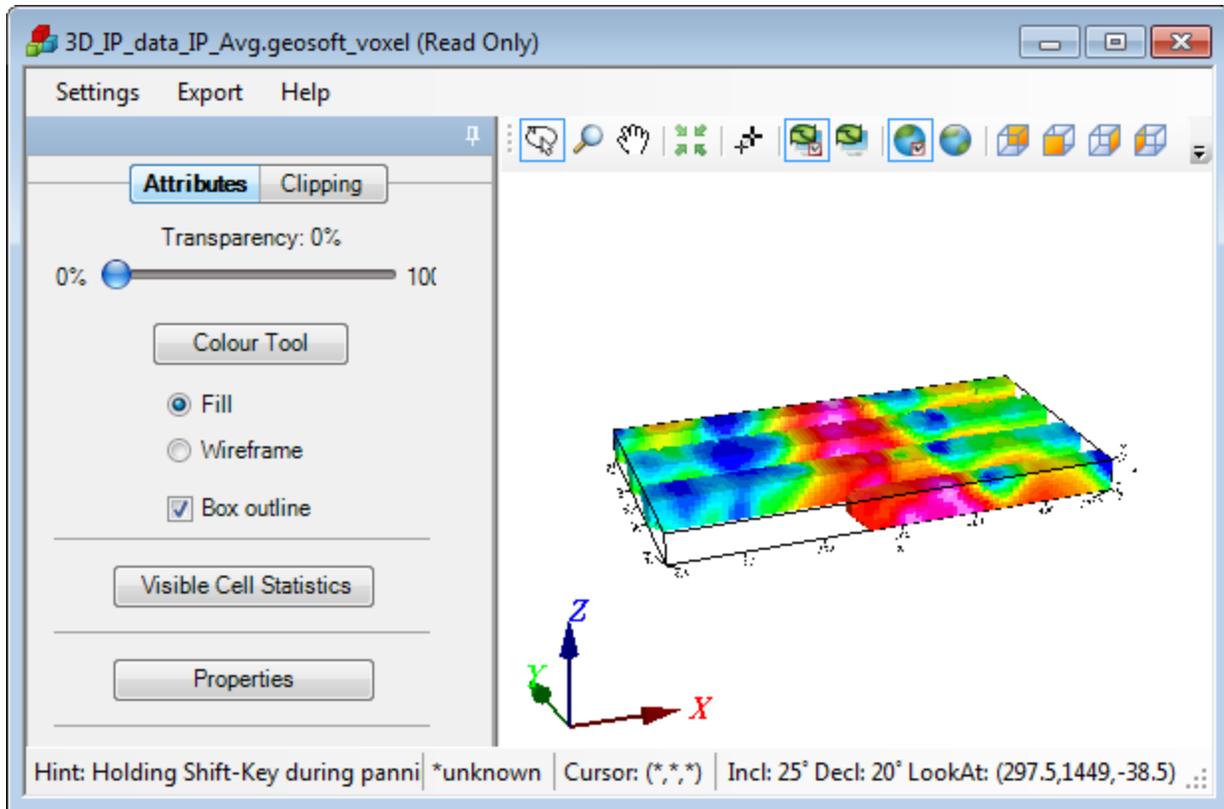
3. From the dropdown list, select the *Channel to grid*. The first time through the process, the default *Output voxel file* name will default to the provided channel name. The subsequent invocations of this tool will remember the user specified file name. You can rename the output voxel file or accept the default name.
4. Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button to obtain a reasonable cell size based on the extents of your data. The cell size is calculated for a single cube, assuming the volume is divided into 10000 individual cubes. The value is then rounded down to give a more satisfactory value.

 We recommend that your first attempt at gridding be using the defaults. After inspecting the outcome however you may decide to modify defaults such as the *Blanking Distance* to fill in the gaps, modify the *Z cell increment*, or generate the output on a logarithmic scale.

 Additionally you can customize the variogram to best suit your data.

 For more information, click the *Help* button on the *3D Kriging* dialog.

5. When you are satisfied with your selections click **OK**. The Voxel is created and displayed in the *Voxel Viewer*. An example is shown below:

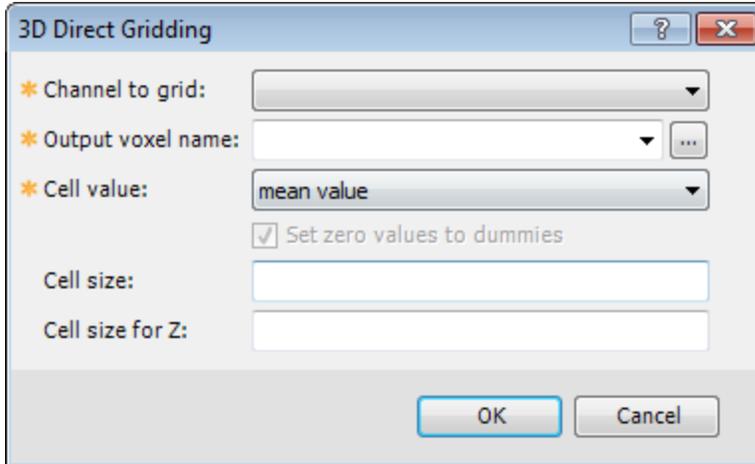


The *Voxel Viewer* provides additional viewing tools.

Creating a Voxel Model using the Direct Gridding method

To Create a 3D Voxel model using Direct gridding:

1. Open a Geosoft data database containing X, Y, Z and the array data channel to grid. If the coordinate system is not completely defined, a message will pop up and you will not be able to continue.
2. In the 3D Viewer, select *Voxel*, then select *3D Gridding* and then select **Direct Gridding** (or on the main Oasis montaj *3D* menu, select *3D Gridding* and then select *Direct Gridding*. The *3D Direct Gridding* dialog appears.



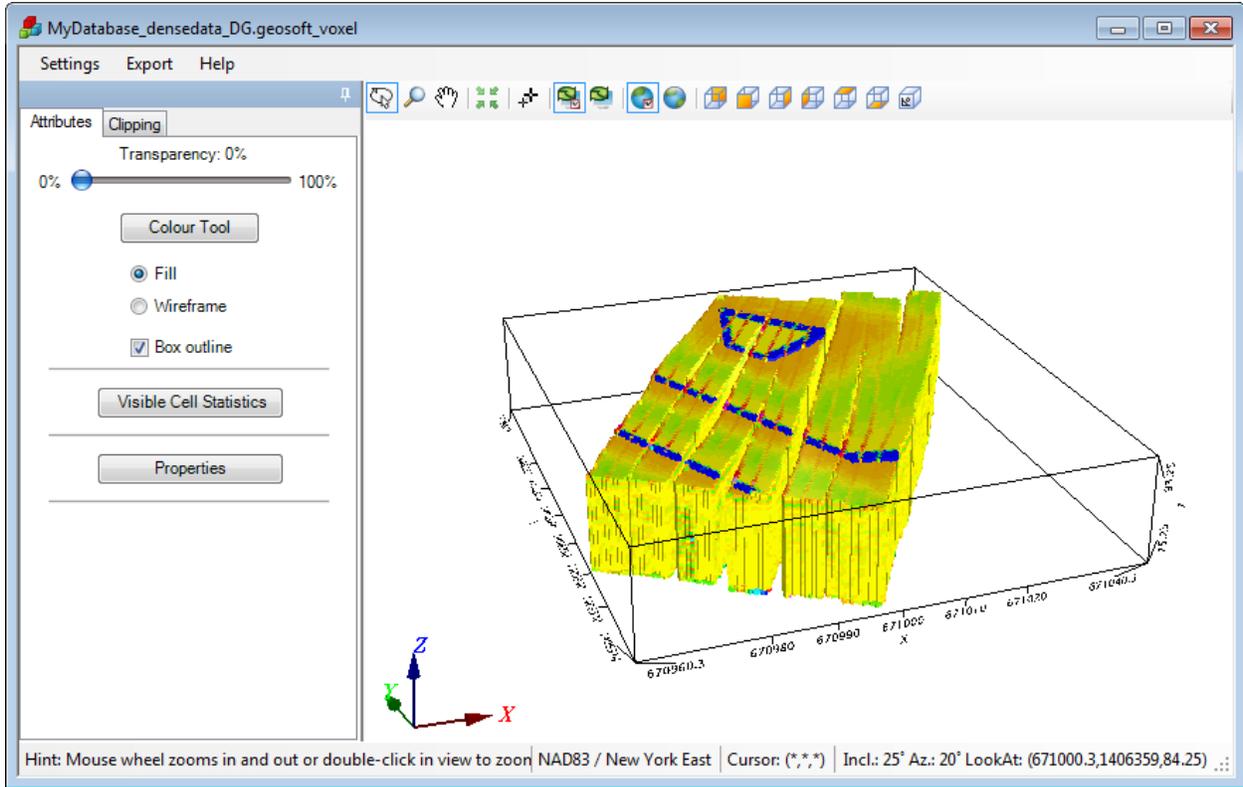
If a database is not already open you will be prompted to open one.

3. From the dropdown list, select the *Channel to grid*. Only array channels are eligible to be gridded. The first time through the process, the default *Output voxel file* name will be set to a combination of the current database and provided channel name. The subsequent invocations of this tool will remember the user specified file name. You can rename the output voxel file or accept the default name.
4. Specify the *Cell value*. You can choose the *minimum value* ; *maximum value* *mean value* or *number of items in a cell*
5. Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button, to obtain a reasonable cell size based on the extents of your data. The cell size is calculated for a single cube, assuming the volume is divided into 10000 individual cubes. The value is then rounded down to give a more satisfactory value.



For more information, click the Help button on the 3D Direct Gridding Help topic.

6. The default vertical Z cell size is set to the same value as the horizontal cell. If this value is not reflecting the desired resolution, modify it accordingly.
7. When you are satisfied with your selections click **OK**. The Voxel is created and displayed in the *Voxel Viewer*. An example is shown below.

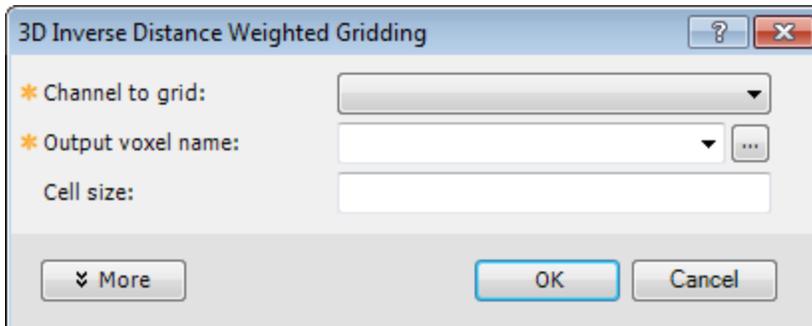


Direct Gridding of GPR data. Data courtesy of UIT

Creating a Voxel Model using the Inverse Distance Weighted (IDW) Gridding method

To Create a 3D Voxel model using Inverse Distance Weighted Gridding:

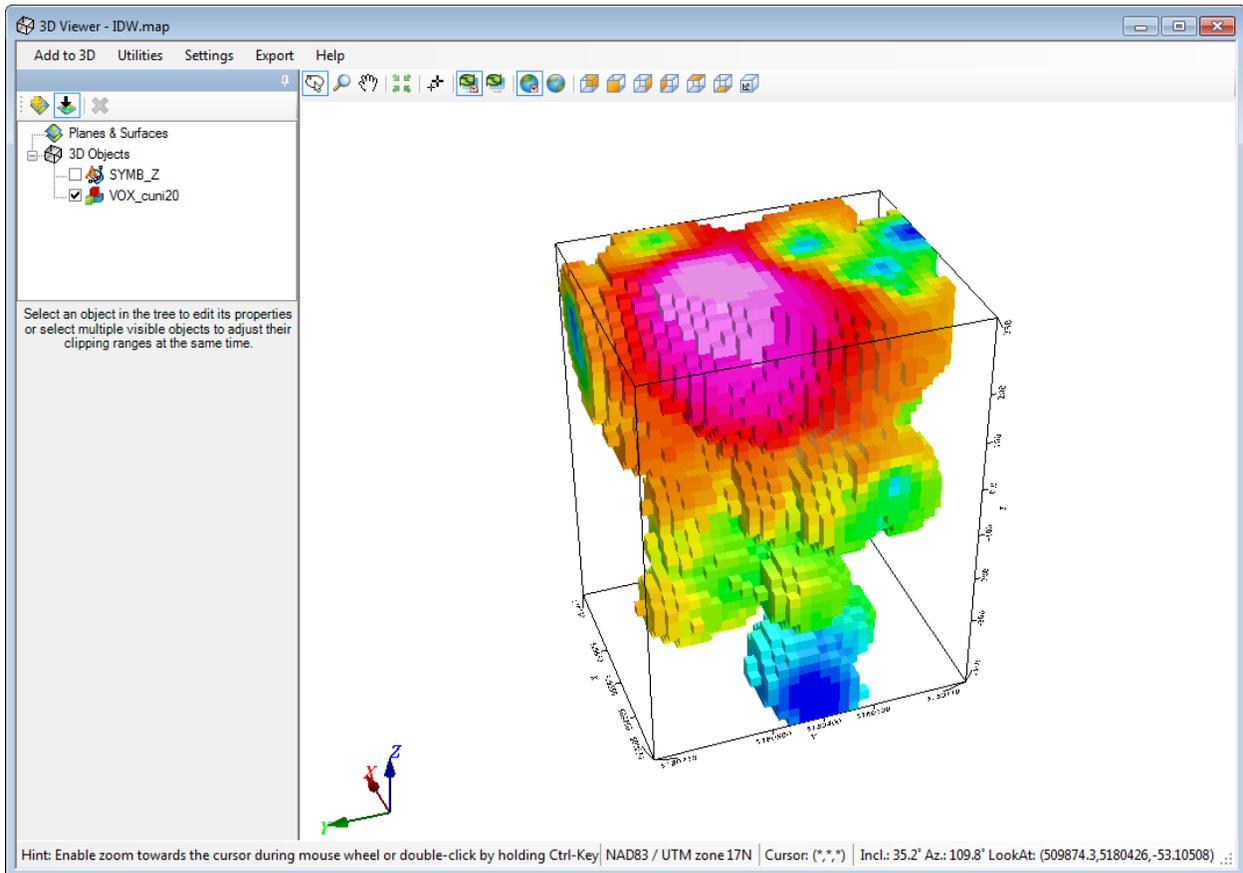
1. Open a Geosoft database containing X, Y, Z and the array data channel to grid. If the coordinate system is not completely defined, a message will pop up and you will not be able to continue.
2. In the 3D Viewer, select *Voxel*, then select *3D Gridding* and then select **Inverse Distance Weighted Gridding** (or on the main Oasis montaj *3D* menu, select *3D Gridding* and then select, *Inverse Distance Weighted Gridding*). The *3D Inverse Distance Weighted Gridding* dialog appears.





If a database is not already open you will be prompted to open one.

3. From the dropdown list, select the *Channel to grid*. The first time through the process, the default *Output voxel file* name will default to a combination of the current database and provided channel name. The subsequent invocations of this tool will remember the user specified file name. You can rename the output voxel file or accept the default name.
4. Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button, to obtain a reasonable cell size based on the extents of your data.
5. When you are satisfied with your selections click **OK**. The Voxel is created and displayed in the *Voxel Viewer*. An example is shown below.



Inverse Distance Weighted Gridding of Borehole data



*We recommend that your first attempt at gridding be using the defaults. After inspecting the outcome however you may decide to modify defaults such as the *Weighting power & slope* to alter the degree of influence of contributing data points relative to their distance, or the *Blanking Distance* to fill in the gaps, or generate the output on logarithmic scale.*

- Specify the *Cell size*. If you do not have a preset cell size, click the **Calculator**  button, to obtain a reasonable cell size based on the extents of your data. The cell size is calculated for a single cube, assuming the volume is divided into 10000 individual cubes. The value is then rounded down to give a more satisfactory value.



For more information, click the Help button on the 3D Inverse Distance Weighted Gridding Help topic.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

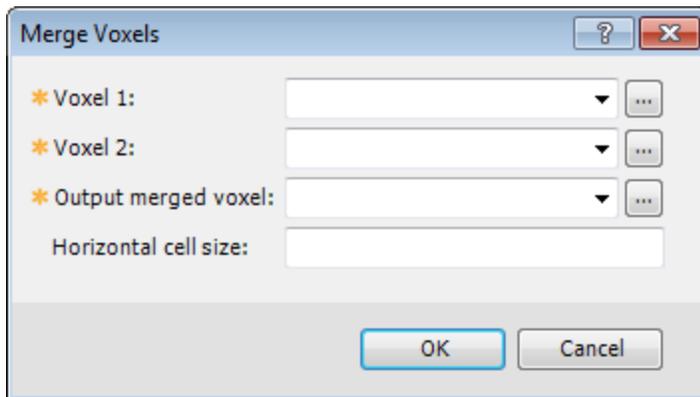
How to Merge Voxels in Oasis montaj

If two voxels overlap, they can be merged independent of their respective projections or the alignment and orientation of their voxel points. Prior to merging, when necessary, the second voxel is re-projected and re-gridded to align with the first voxel. If merging more than two voxels, select the primary voxel according to the desired projection and orientation and proceed with merging two voxels at a time.

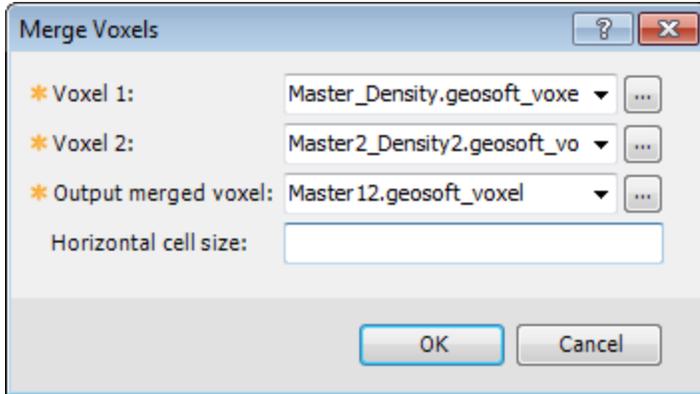
To Invoke the Merge Voxels Tool

1. From the **3D** menu, select *Voxel Utilities* and then select **Merge Voxels**.

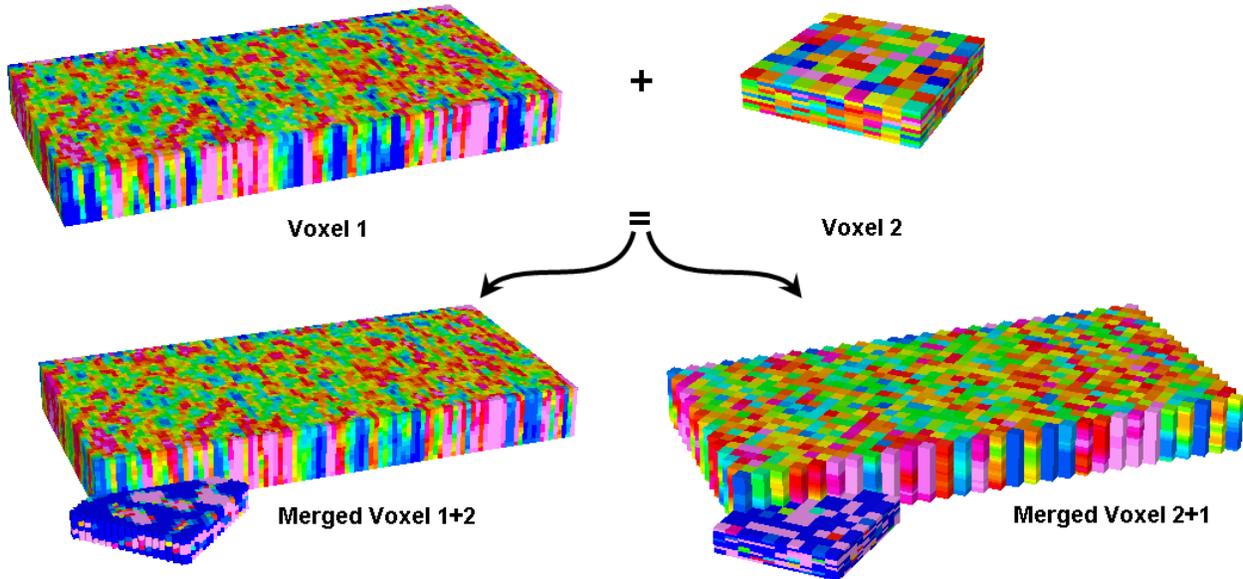
The *Merge Voxels* dialog opens.



2. Use the  button next to the *Voxel 1* field to select the first voxel. This voxel defines the projection and orientation of the merged output voxel.
3. Select *Voxel 2* in the same manner. This voxel will be aligned with the first voxel prior to merging.
4. Provide the name of the *Output merged voxel*. If the selected file already exists, you will be prompted to either enter a different name or overwrite the existing file.
5. You can optionally provide the output *Horizontal cell size*. If left blank, the minimum of the X and Y increment of the first voxel will be assumed on output.



6. Click **OK**. The merged output voxel will be displayed.



-  *If the second voxel extends beyond the first one and the first voxel has a variable increment in Z, the increment of the top/bottom two points of Voxel 1 is propagated out.*
-  *The Blend method uses a blending function over the volume of overlap so that the transition across the overlap from one voxel to the other is smooth. Points that do not overlap are not altered.*
-  *Where the edges of Voxel 1 and Voxel 2 cross at a single point, the average of the two point values is used. If there is a large shift between the two voxels, an overlap of one point will not allow for a smooth transition.*

How-To Guide Publication Date: 29/04/2014
 Copyright 2014 Geosoft Inc. All rights reserved.

How to Filter a Voxel in Oasis montaj

Spatial convolution filters can be applied to an existing voxel. Three 3x3x3 convolution filters are supplied. You can also save a custom filter in a file and apply it to a voxel.

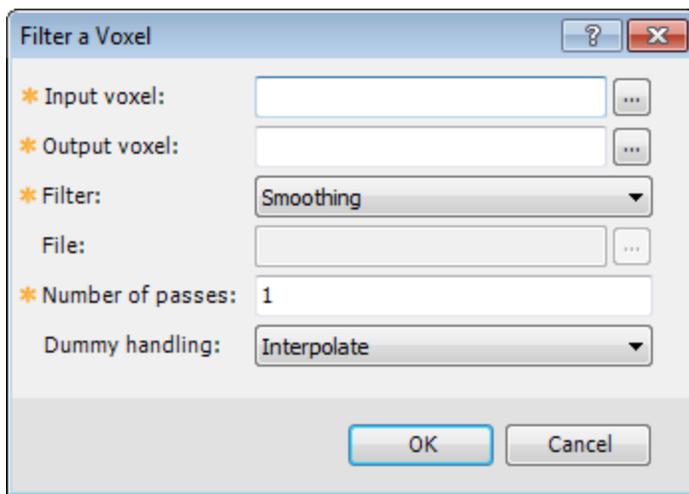
The supplied filters are:

- ▶ The **Smoothing** filter, useful for spatially smoothing 3D measured or calculated data.
- ▶ The **Laplace** filter, useful for emphasizing areas of rapid intensity changes.
- ▶ The **Total Gradient** filter, useful for delineating rock unit/contact boundaries.

To Smooth a Voxel

1. From the **3D** menu, select **Voxel Utilities | Filter a Voxel**.

The *Filter a Voxel* dialog opens.

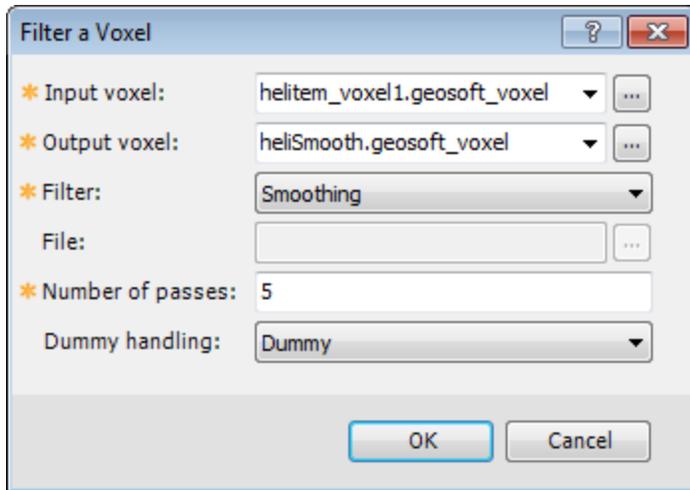


2. Use the  button next to the *Input Voxel* field to select the voxel to filter.
3. Provide the name of the *Output Voxel*. If the selected file already exists, you will be prompted to either enter a different name or overwrite the existing file.
4. From the drop down list of filters, select the **Smoothing** filter.
5. Enter the desired number of passes of the smoothing filter.

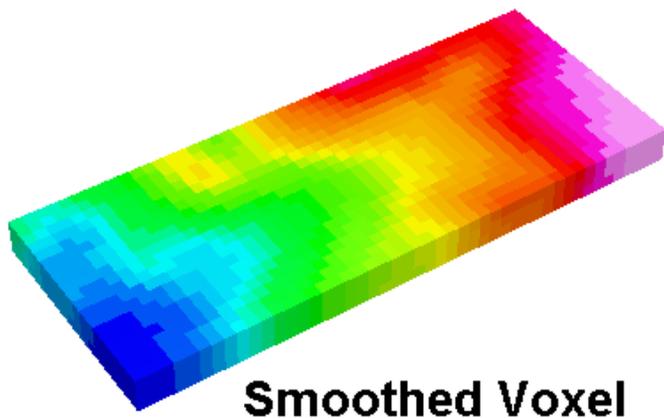
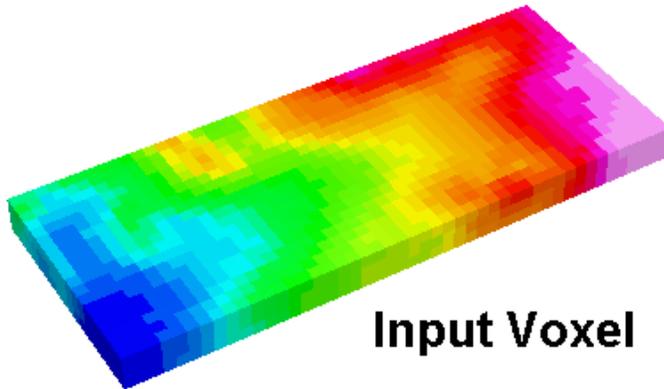


The larger the number of passes, the smoother the outcome.

6. Select **Dummy** for the handling of the edges of the voxel.



7. Click **OK** and the filtered voxel will appear.



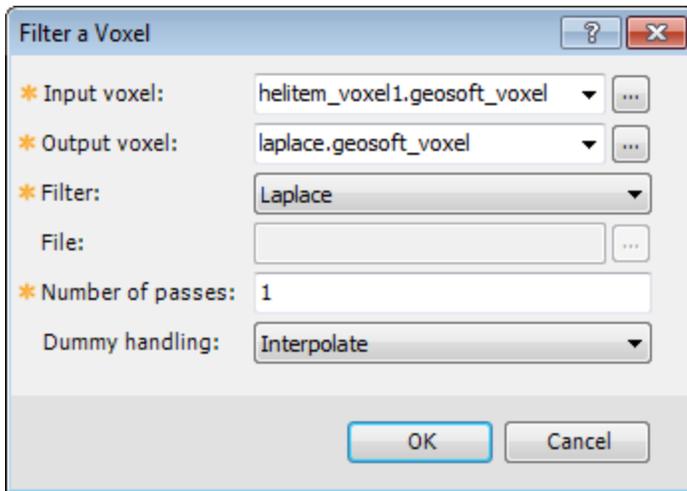
-  In each direction, the convolution operation requires 3 consecutive data points to apply the filter. The outcome of the operation will be located on the central point. Thus, the filtering of the edge points can be approached in 2 different ways:
- on output, the edge points are truncated because in at least one direction, they are missing a valid data point,
 - the next point adjacent to each end point is first extrapolated using valid data, after which the filter is applied, and the output voxel will be of the same size as the input voxel. A linear interpolation is performed for the inside dummy points and the nearest valid point is assigned for exterior extrapolation.
-  The supplied Geosoft filters have the extension `flt3d` and are installed in the `Geosoft\Oasis montaj\etc` directory. The convolution filter coefficients are supplied in the increasing order of X, then Y, and finally Z.

 In analogy to the supplied filter format, you can create your own custom filter. This filter can reside in any directory, however if you intend to use it in other projects, we recommend to save it in the directory `Geosoft\Oasis montaj\user\etc.`

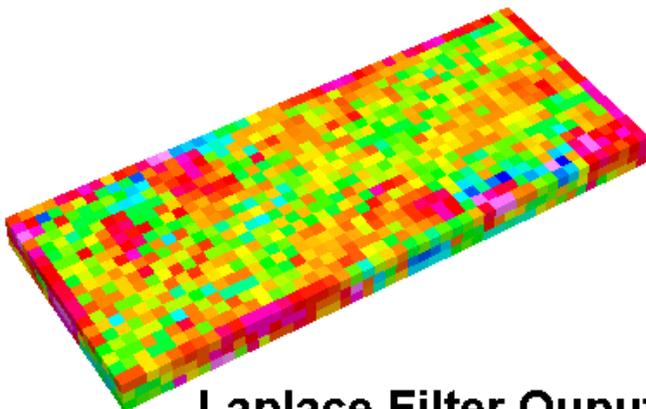
To Emphasize the Areas of Rapid Change

1. Follow steps 1,2,3 from the *smooth a voxel* procedure.
2. From the drop down list of filters, select the **Laplace** filter.

 Although you can enter up to 3 passes of the Laplace filter - since this filter accentuates the local variations - it is recommended not to go above 1 pass unless you have a compelling reason to do so.



3. Select **Interpolate** for the handling of the edges of the voxel.
4. Click **OK** and the filtered voxel will appear.



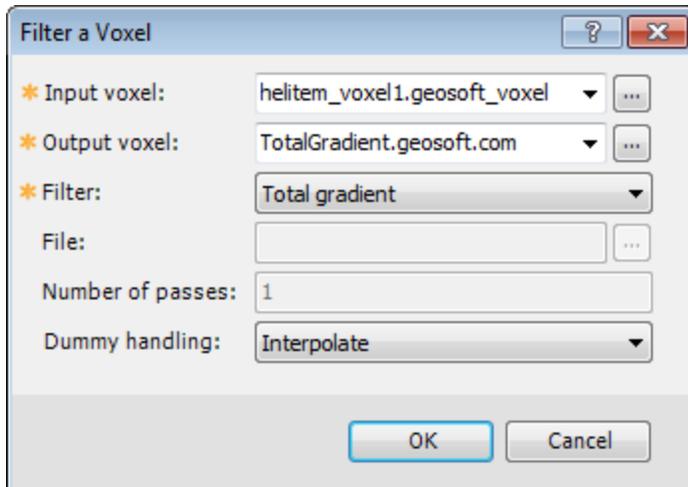
Laplace Filter Output

 The visible edge effect of the Laplace Voxel is due to the fact that on the outside, the nearest neighbour value is assigned to extend the voxel.

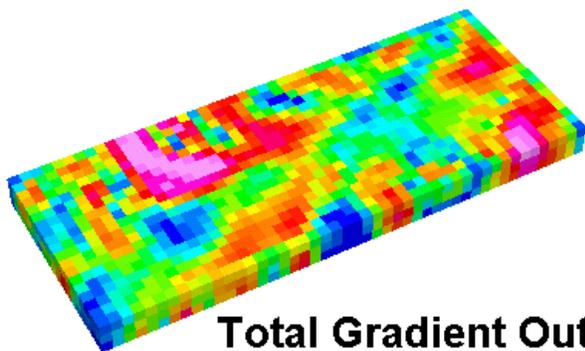
 If you find excessive noise in the Laplace voxel, you can apply a smoothing filter to the Laplace Voxel to smooth out the noise .

To Apply a Gradient Filter

1. Follow steps 1,2,3 from the *smooth a voxel* procedure.
2. From the drop down list of filters, select the **Total Gradient** filter.
3. You will notice that only one pass of the Total Gradient convolution filter can be applied.
4. Select **Interpolate** for the handling of the edges of the voxel.



5. Click **OK** and the filtered voxel will appear.



Total Gradient Output

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Wireframing Geological Interpretations in 3D

Once you have digitized a geological feature on at least two section maps, you can create a wireframe from the interpretations. This document provides a description of the workflow to join polylines or polygons in a geostrung file to form open surfaces or closed bodies that will be stored in a geosurface file. After saving the wireframe to a geosurface file, total surface area is reported for open surfaces and total volume is reported for closed bodies. Many open and closed surfaces can be saved to the same geosurface file. This single file solution makes it easy to manage the wireframed results and to share your work with others.

This document will guide you through:

- Starting a wireframing session
- Joining polylines to form an open surface
- Joining polygons to form a closed surface
- Viewing information for surfaces in a geosurface file

To start a wireframing session:

1. In the 3D Viewer, from the *Wireframe* menu, select **Start Wireframing**.
2. Select the **Geostrung to wireframe** and the **Feature to wireframe**.
3. Click **OK** on the *Start Wireframing* dialog.

The *Wireframing Tools* will display along the side of the 3D Viewer.

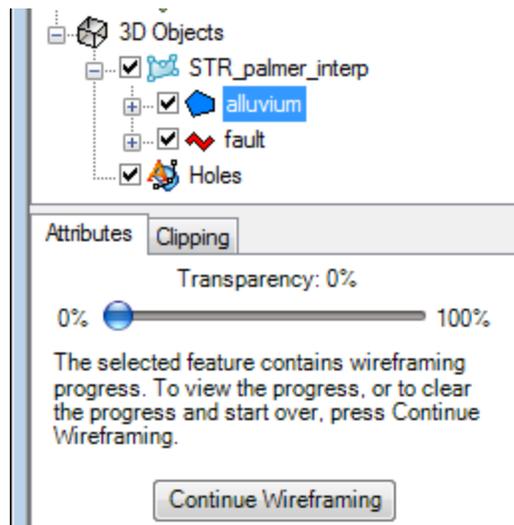
If the Wireframing Tools are not pinned, they will be hidden when the cursor moves into the 3D model.



Note:

*The wireframing sequence you build is stored in the geostrung file. To return to a partially completed wireframe for a feature, you can either use the "start a wireframing session" steps listed above, or, select the feature below the geostrung name in the 3D Objects tree and click **Continue Wireframing** in the Attributes tab. If the feature does not contain a saved wireframe sequence, the option will be **Start Wireframing** instead of*

Continue Wireframing.



To join polylines into an open surface wireframe:

1. In the *Wireframing Tools*, change the **Active Feature** to a polyline feature if you previously chose a polygon feature in the *Start Wireframing* dialog.
The polylines will be bounded in yellow.
2. Click on a polyline interpretation in the 3D model.
The selected polyline will be bounded in cyan.
3. Click an adjacent polyline to wireframe to the first polyline you selected.
You now have the beginning of a wireframe surface.
4. Continue selecting adjacent polylines in the 3D model to add to the wireframe surface.



Note:

While wireframing, you can use the Undo/Redo options (↶ / ↷), or use the Ctrl+Z or Ctrl+Y shortcut keys, respectively.

Geosurface Files

Geosurfaces are 3D vector files that store the results of wireframing polygon or polyline interpretations to form a closed body or an open surface, respectively.

Geosurfaces store the following information:

- XYZ locations
- Coordinate system
- Attributes including feature names and colours



Note:

The coordinate system for the geosurface is automatically set based on the geostring's coordinate system used to create the wireframe.

Closed surfaces are the result of wireframing polygons. Closed surfaces can be used in voxel math expressions. Open surfaces are the result of wireframing polylines. Many closed and open surfaces from a wireframing session can be saved to the same geosurface. Additionally, the surface names in a geosurface are set based on the feature names from the geostring used in the wireframing session.

To save the wireframe to a geosurface:

1. Under the *Wireframing Tools*, click **Save Wireframes as Geosurface File** .
2. Enter a **File name** and click **Save**.



Note:

Once you have saved wireframes for an active feature, you can add other wireframes for different active features to the same geosurface file.

To join polygons into a closed body wireframe:

1. In the *Wireframing Tools*, change the **Active Feature** to a polygon feature. The interpretations for the Active Feature will be highlighted in yellow.
2. Click on a polygon interpretation in the 3D model. The selected polygon will be bounded by a cyan border.
3. Click an adjacent polygon to wireframe to the first polygon you selected. You now have the beginning of a wireframe body. Both ends are bounded in cyan meaning you could continue adding to the wireframe from either side.
4. Continue selecting polygons in the 3D model to define a larger wireframe body.

Wireframing Options

Number of triangles

Select **Auto** to automatically determine the optimum number of triangles to join two shapes together while maintaining an even distribution of triangle edge sizes.

This option can be overridden by selecting the **radio button** below *Auto*. Then specify the number of triangles to create for each shape edge. You can type in a number or use the up/down arrows to set the number.

Extrude off-section vertices

Specify the Extrude off-section vertices option if this error message is returned while wireframing:

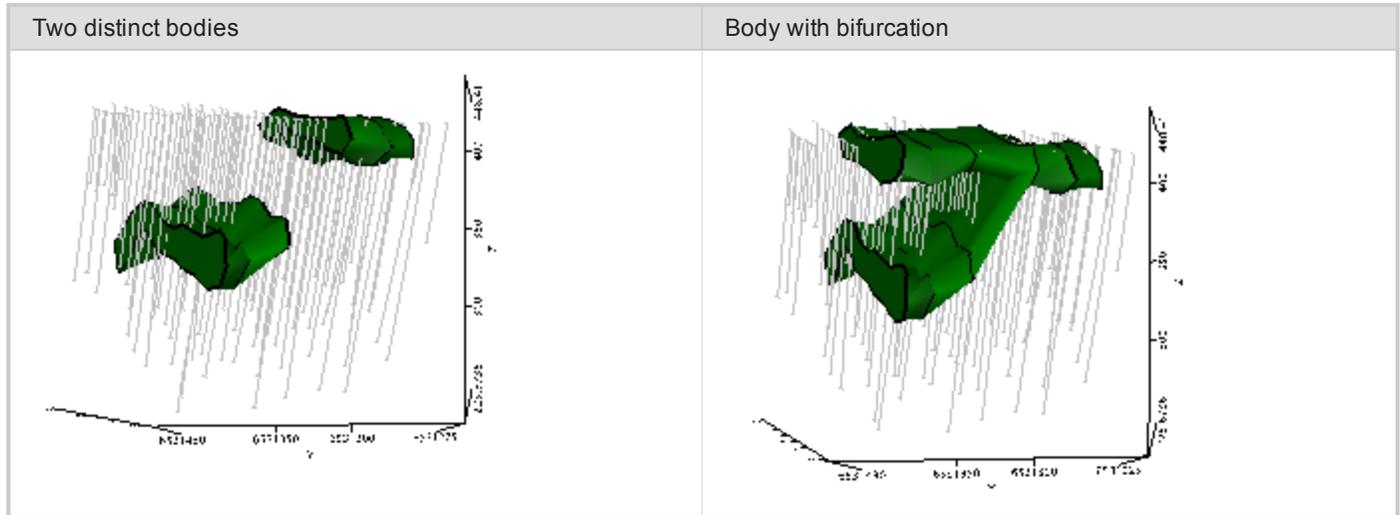
"Wireframing caused an invalid surface due to self-intersection. This can be caused by the angle between the sections being too large or by the use of snapped vertices.

If the latter is true, this may be resolved by checking the 'Extrude off-section vertices' checkbox during wireframing."

This option will extrude the off-plane section points of a shape to include all the digitized points in a body segment so that the ends of the segment will form flat planes. This will reduce the chance of generating a self-intersecting surface when joining shapes. Self-intersecting surfaces have triangles that “cut into each other”. They cannot be used to form closed bodies suitable for wireframing and geosurface generation.

New Segments

If you have interpretations that belong to the same active feature that need to be connected as separate wireframes, you can add new segments to the wireframe body. For instance, the same rock type may occur in two separate areas and may or may not be joined.



To add a new segment to the wireframe body:

1. From the *Wireframing Tools*, click **Add New Wireframe Body** .
All of the interpretations for the Active Feature are bounded in yellow.
2. Click on an interpretation in the 3D model.
The selected interpretations will be bounded by a cyan border.
3. Click an adjacent interpretation to wireframe to the first shape you selected.
4. Continue clicking interpretations to add to the wireframe.

Remove Wireframes for Active Feature

If you simply want to start over for the Active Feature, you can remove all wireframing steps and begin again.

To remove all wireframes for active feature:

1. From the *Wireframing Tools*, click **Remove All Wireframes for Active Feature** .
This only removes the wireframes for the active feature, not for all the features in the geosiring.
2. Click **Yes** to clear the current wireframe.
Wireframing Mode is active and all the interpretations for the active feature are bounded in yellow again

Wireframe Ends for Closed Surfaces

Polygons that have been joined into a wireframe must have closed end to be saved as a geosurface. If you do not add ends to the wireframe before saving, you will be asked if you would like to auto-close the wireframe with flat ends

To add ends to the wireframe body:

1. In the *Wireframing Tools*, click **Add Ends** .

The open areas of the wireframe are bounded in yellow.
2. Select a radio button (**Flat**, **Conical** or **Rounded**) under *Ends Options*.
3. Click one of the yellow open areas to create an end for it using the selected option.
4. Continue adding ends to the open areas of the body until no yellow areas remain.

To save the wireframe to a geosurface:

1. Under *Wireframing Tools*, click **Save Wireframes as Geosurface File** .
2. Browse to and select an existing geosurface or enter a **File name** to create a new geosurface.

 Remember that the wireframe sequence is saved in the geostring file. You can Start Wireframing again to revisit a wireframing session undoing and redoing segments and then saving the changes to a new or existing geosurface.

Viewing Total Volume and Total Surface Area

Total Volume can be seen for closed surfaces in a geosurface file.

To view the Total Volume for a closed surface:

1. In the *3D Objects tree*, select a closed surface.

Information about this surface, including *Total Volume* is given under the *Attributes* tab.
The Transparency, rendering and colour can be changed to alter how the surface displays.

Total Surface Area can be seen for open surfaces in a geosurface file.

To view the Total Surface Area for an open surface:

1. In the *3D Objects tree*, select an open surface.

Information about this surface, including *Total Surface Area* is given under the *Attributes* tab.
The Transparency, rendering and colour can be changed to alter how the surface displays.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Creating Isosurfaces from a Geosoft Voxel

A 3 dimensional surface that passes through points of equal value is called an isosurface. Isosurfaces can be thought of as 3D contours. Isosurfaces are written to a geosurface file. The geosurface file is able to save more than one isosurface. This allows you to easily share your results with colleagues to collaborate on projects.

In Geosoft, there are three different ways to create isosurfaces from a voxel:

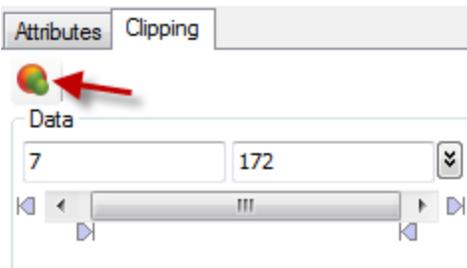
- Create a single isosurface at a specified value.
- Visually clip the minimum data values from a voxel and automatically extract an isosurface at the specified clip value.
- Automatically create multiple isosurfaces using a linear, log or user-specified distribution.

To create a single isosurface at a specified value:

1. In the 3D Viewer, click **Add to 3D menu | Isosurface | Create Isosurface**.
2. Browse to and select the **Input voxel**.
3. You can optionally change the default **Output geosurface name**.
4. Enter a **Surface value**.
This will be the 3D contour value.
The value you enter must be a data value in the voxel range.
5. You can optionally change the **Colour**.
6. If the Output geosurface file exists, you can append the isosurface to it by selecting **Append to geosurface**.
7. Click **OK** to create the isosurface.

To visually clip a voxel to create an isosurface:

1. Add the voxel to the 3D Viewer (**Add to 3D | Voxel**).
2. Select the **voxel** in the *3D Viewer tree list*.
3. Click the **Clipping** tab below the tree list.
4. In the *Minimum data value field*, specify a value larger than the minimum. You can type in a value or use the data slider.
5. Click **Define isosurface using minimum data clip value**.



The *Create Isosurface from Voxel* dialog will open automatically populated with the Input voxel, Output geosurface name, Surface value and Colour.

6. Click **OK** to create the isosurface.



Notes:

You can optionally specify a different minimum data value on the Clipping tab. Then use the Define isosurface using minimum data clip value again to append another isosurface to the geosurface file.

For the *Create Isosurface from Voxel* dialog:

- Isosurfaces can be appended to a geosurface as long as the Input voxel used to create the geosurface is the same.
- Appending the same surface value to the geosurface will overwrite the surface value and will update the colour if it was changed.
- If Append to geosurface is disabled and the Output geosurface exists, you will be prompted if you want to overwrite the output file.

To create multiple isosurfaces from a voxel:

1. In the 3D Viewer, click **Add to 3D | Isosurface | Create Multiple Isosurfaces**.
2. Browse to and select the **Input voxel**.
If the options under *More* are not changed, 5 isosurfaces will be extracted at linearly-spaced intervals between the input voxel's minimum and maximum values.
3. Click **More**.
4. You can optionally change the **Number of isosurfaces** to extract.
5. Distribution - after clicking **Use a linear distribution** or **Use a log distribution**, you can optionally change the *Minimum* and *Maximum* used to define the range from which to calculate the isosurface values.
6. You can also manually change the **Isosurface Value** and **Colour** by clicking a cell to edit.
7. Click **OK**.



Notes:

Colour tables can be loaded and saved.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Voxel Math Expressions in Oasis montaj

The Voxel Math Expression Builder is used to create, save, load and execute math expressions for your voxel data. This easy-to-use standardised dialog provides all the math expression options on a single tabbed dialog.

Use Closed Surfaces in Voxel Math Expressions

Closed bodies created by wireframing polygons together can be used as surfaces to clip voxels in the Voxel Math Expression builder. This allows you to clip a hole out of a voxel or to clip the extents of a voxel to the inside of the surface. New statistics can then be viewed for the output voxel.

To clip a voxel to the outside of a closed surface:

1. From the *Voxel* menu in the *3D Viewer*, select **Voxel Math**. Voxel Math is also available from the *3D | Voxel Utilities* menu.
2. Enter the **Expression**: $V0 = out(V1, S0)$.
3. In the *Assign voxels and grids* area, specify the output voxel, input voxel and input surface.
V0: this will be the new output voxel.
V1: this is the input voxel to clip.
S0: this is the input surface used to clip the voxel.
4. Click **OK**.

The output voxel is added to the 3D Viewer.

If the closed surface is fully contained by the input voxel, you may have to visually clip the X, Y or Z axes to see the results of the clipping in the output voxel.

To clip a voxel to the inside of a closed surface:

1. From the *Voxel* menu in the *3D Viewer*, select **Voxel Math**. Voxel Math is also available from the *3D | Voxel Utilities* menu.
2. Enter the **Expression**: $V0 = in(V1, S0)$.
3. In the *Assign voxels and grids* area, specify the output voxel, input voxel and input surface.
V0: this will be the new output voxel.
V1: this is the input voxel to clip.
S0: this is the input surface used to clip the voxel.
4. Click **OK**.

The output voxel is added to the 3D Viewer.

To view voxel statistics:

1. In the *3D Viewer*, select the **Output voxel** from the *3D Objects tree*.
2. From the *Attributes* tab, click **Visible Cell Statistics**.

The *Voxel Statistics for Visible Cells* dialog opens and displays information such as, Volume, Minimum, Maximum Mean.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Import and Export 3DV Files in Oasis montaj

3D Views can be exported and imported using the geosoft 3DV file format. 3DV files contain all of the files listed in the 3D Viewer tree.

Export a 3D View as a 3DV File

Exporting a 3D View as a 3DV file creates a *.geosoft_3dv file which is a package of all the data that is listed in the 3D Viewer tree. The display options (i.e. zoom, rotation, which layers are turned on/off, transparency, colours, etc...) are also written to the 3DV file. This is an easy way to create a single file for all the data in the 3D View so you can move the View and datasets to another folder, or to share the View and data with a colleague to collaborate on a 3D project.

To export a 3D View as a 3DV file:

1. From the *Export* menu in the *3D Viewer*, select **Geosoft 3DV File**.
2. Enter an **Output 3DV name**.
3. Click **OK**.

Importing a 3DV file to a Map

Importing a 3DV file will extract the packed contents of the file (voxels, grids, geostings, geosurfaces) to the project folder. Before the data files are extracted, if the import detects that there are files with the same name in the project folder that are not identical to those contained in the 3DV, you will be prompted if you would like to overwrite the existing files. If you do not overwrite the files, the 3D View will still display, but the files will remain packed in the imported map, and are not extracted to the project folder.

To import a 3DV file:

1. From the **3D | Imports** menu, click **Geosoft 3DV File**.
2. Select the **3DV file** to import.
3. You can optionally change the **View name**.
This is the 3D View name that will appear in the View/Group Manager for the map.
4. Select if you would like the 3D View to **Display In a New map** or the **Current map**.
5. Click **OK**.

The 3D View is displayed in the map.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

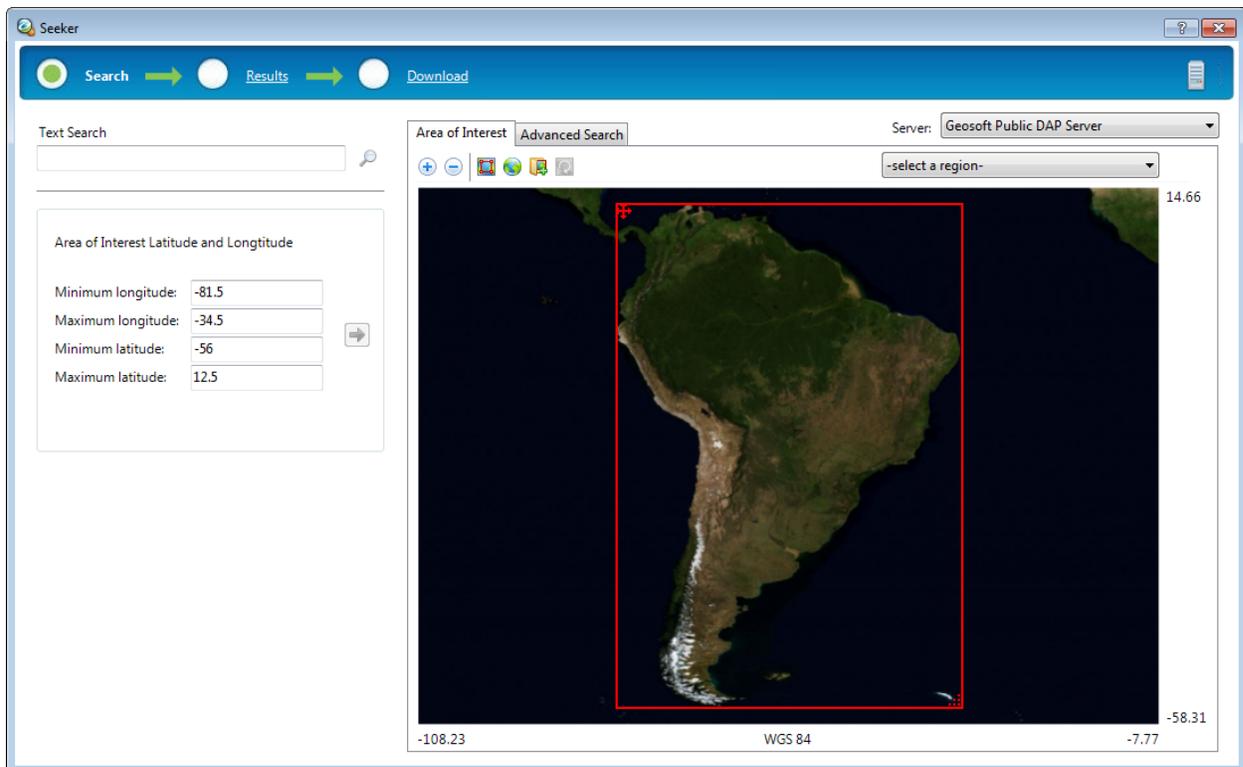
Geosoft Seeker Tool in Oasis montaj

The Geosoft Seeker tool is designed to help you search for and download spatial data stored on global and corporate spatial servers including Geosoft DAP servers and WMS servers, as well as data discovered by the Dapple Search data service and spatial data indexed locally by Desktop Cataloger.

From the **Seek Data** menu, select **Seeker** to open Seeker and find relevant exploration data.

The steps to using the *Seeker* Tool:

1. **Search** – Use the search tools to define spatial and text criteria for the data you are seeking.
2. **Results** – Preview a dataset as a quick thumbnail view or in geographical context using the Preview map, and review metadata.
3. **Download** – Download selected datasets directly into open map or project.



Searching for Data

Seeker enables you to find, view and download a dataset in 3 easy steps. The first step is to query your list of servers to determine what data are available that match your area of interest (AOI). Available data sources include DAP Servers, WMS servers, the Dapple Search data service, and local data indexed by Desktop Cataloger.

Searching for Data

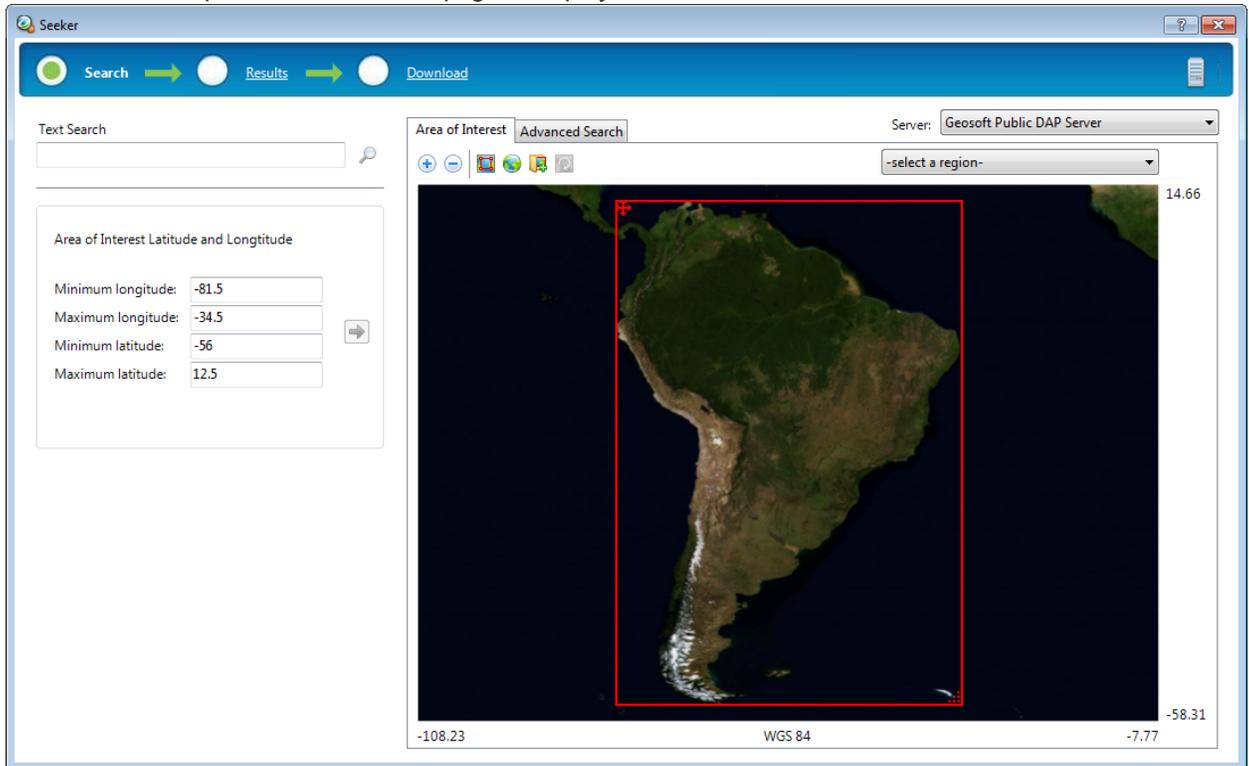
When you open *Seeker* the initial Search extents are based on your current open map. If you do not have an open map, the area of interest (AOI) is the world.

 To narrow results by metadata, click the **Advanced Search** tab.

The *Search tools* include the *Server list*, *Text Search*, *Area of Interest* and *Advanced Search*.

To Use Search Tools

1. On the *Seek Data* menu, select **Seeker**.
The *Seeker* tool opens and the *Search* page is displayed.



 If the AOI displayed in the window is not the same as your open map, click the *Update area of interest from current map extents* button ()

2. Narrow your search.
There are several methods to narrow the search to a specific region or data type:
 - Specify a keyword or phrase in the *Text Search* text box so that only datasets that intersect your AOI and/or contain text that matches a text string will be displayed. You can type in multiple words and use wildcards (*) and logical operators (AND, OR, AND NOT). Click Results, the magnifying glass (), or simply press **Enter** when finished filling in the Text Search box to move to the Results page.
 - Set the Area of Interest using the *select a region* dropdown list, which is provided through the selected DAP Server. This is a very handy list that can include countries, province/state boundaries, or mapsheet/quadrangle names. The *Area of Interest* map enables you to view and redefine your AOI. Use the *Navigational tools* or *Geographic (long, lat)* search to redefine your

AOI interactively while you view the changes in your map tab.

- » Click **Advanced Search** to select the intersection of the datasets with the AOI, as *Overlap area of interest* or *Inside area of interest*. The *Advanced Search* tab also provides tools that enable you to *Narrow results by metadata*, using Boolean operators and predefined dropdown lists made available through the selected *Server*. Your metadata search can be *Saved* for future use and then *Loaded* to reapply the same search criteria at a later time.

Include datasets: Overlap area of interest
 Inside area of interest

Narrow results by metadata:

-Select a metadata field- ▼
 -Select an operator- ▼
 -Type or select a value- ▼
 -None- ▼

Add

Example: Continent = "North America" AND Province/State = "Quebec"

-Saved search- ▼ ... Load Save



Some DAP Servers do not support metadata queries.

3. The selected Search results will appear on the *Results* page.

Read the How-To Guide

» [Geosoft Seeker Tool in Oasis montaj](#)

Viewing Results

The *Results* page enables you to view the results of the current search for all your data sources: DAP, WMS, the DappleSearch web service and My Data from your Desktop Cataloger.

The *Results Tools* include Text search tip, Server list, Results list, Preview Map tab and a Metadata tab.

➤ Viewing Results

- **Server List** - The server list expands the tree or displays the list of datasets from the selected server: *DAP* including Desktop Cataloger results under *My Data*, *Web*, and *WMS*. The number of valid results found per data source are presented. The number of results can be found to the right of the server name in *Tree* view and below the Results list in *List* view.
- **Results List** - The results pane enables you to view your search results in *Tree View* or *List View*. The view may depend on the selected server (*DAP*, *Web*, *My Data*, *WMS*) you are viewing. Not all data sources can be displayed in both views. Check a dataset, in any view, to display an image of the dataset in the *Preview Map* tab, display the dataset's metadata in the *Metadata* tab, and add the dataset to the *Download list*.
- **Preview Map** - Preview an image of a selected dataset from the Results list. The *Preview Map* includes *Navigation Tools* that enable you to zoom in, zoom out and zoom to full map and toggle the *Browser Map*.
- **Metadata** - Metadata (data about data) is captured by Oasis montaj from the first time the dataset is touched. The metadata information, which is based on ISO 19139 standards, is stored inside the data (if supported) or as a companion XML file. For more information click the **Help** button on the *Seeker* tool.

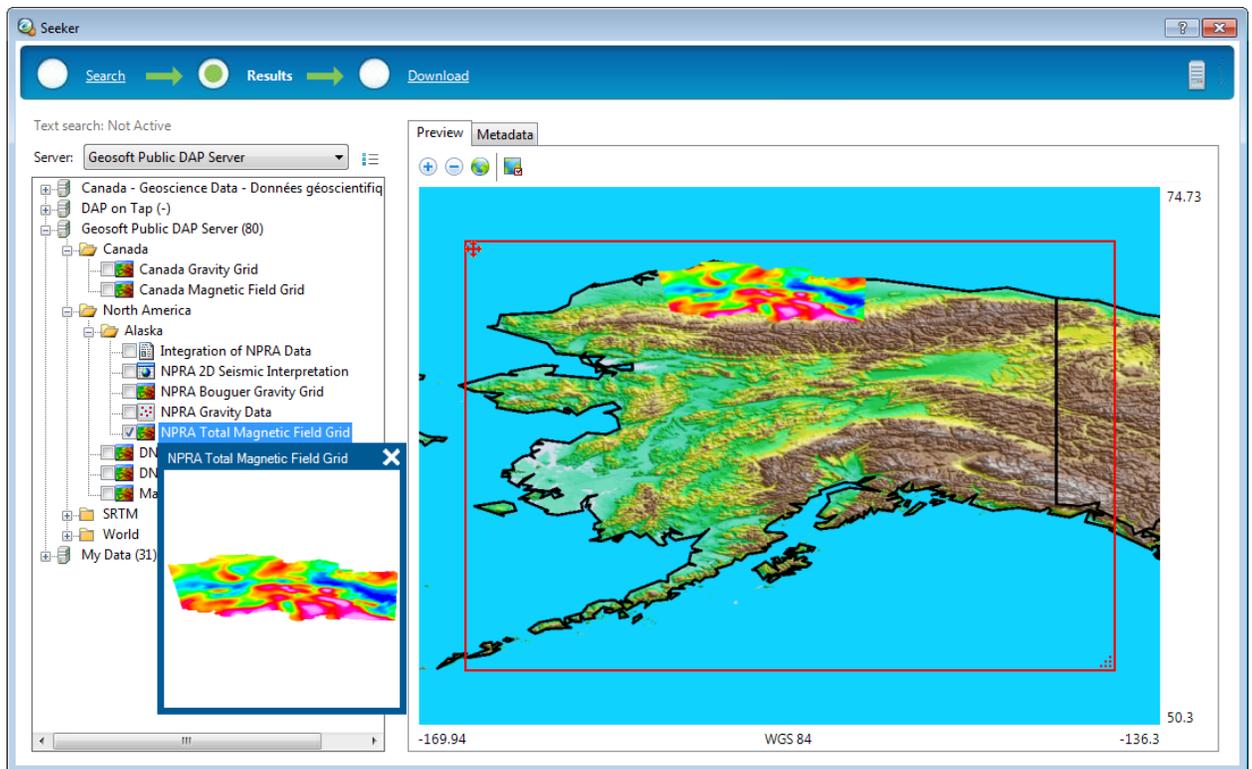
To Use the View Results Tools

1. Click **Results**. The *Results* page opens, displaying your current server search results.
2. If desired, you can view other data source results from the *Server list*. The data results are available in two views (*Tree* and *List*). Depending on the server, both views may not be available.



When viewing the results in tree or list view, if you hover your cursor over the data icons a thumbnail image will be displayed.

3. Select a dataset by placing a *Check* in the box in front of the dataset name.



4. Click the **Preview Map** tab, and the selected datasets are displayed in the preview map window. Click the Metadata tab to view the Metadata for the selected dataset.
5. When you are satisfied with your data selections, click the **Download**.

Read the How-To Guide

➤ [Geosoft Seeker Tool in Oasis montaj](#)

Downloading Selected Datasets

The *Download* page enables you to prepare a dataset for download. The *Download list* is created from all selected data from the *Results* page. Datasets from multiple servers (*DAP*, *WMS*, *Web* and *My Data*) will be displayed in the list of selected data.

From the *Download* page, you can take several actions:

- **Download Properties** - Enables you to specify download options for selected datasets. Select (highlight) a dataset in the *Download list* to enable the *Download properties* options.
- **Download All** - Downloads (or opens, if the data is in *My Data*), the selected datasets to a new or current map, if applicable.
- ⚠ **Datasets from *My Data* will be opened directly, not downloaded.**

To Download Selections

The *Download* page includes a *Download list*, *Download properties* area, and **Download All** button, as detailed below.

1. Select (highlight) the file to download.



To select multiple files for download use the **<Shift>** or **<Ctrl>** keys while selecting files.

2. When you are satisfied with the Download properties for each dataset, click **Download All**. Downloaded data may be displayed in your current open map, or if you do not have a map currently open, the downloaded data will be displayed in a new window as required.



You can remove datasets from the Download list by right clicking and selecting "Remove" or using the Delete key.

Read the How-To Guide

 [Geosoft Seeker Tool in Oasis montaj](#)

Desktop Cataloger

The Geosoft *Desktop Cataloger* is a free utility that helps you build a catalog of your spatial files from your local drives, and enables you easily find that data when using the Geosoft *Seeker* tool.

With Desktop Cataloger, you build a *My Data* catalog of spatial files by selecting folders on your computer or external drives (can not catalog network drives). You can also specify data types to be cataloged from a preset list of spatial dataset file types. The list of supported data types includes ECW, ERS, GDB, Geosoft VOXEL, Geosoft GRD, ArcGIS LYR, ArcGIS SHP, MapInfo TAB and GeoTIF Images.

For more information or to download Desktop Cataloger, visit <http://www.geosoft.com/pinfo/free/desktop-cataloger.asp>

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Working with Scripts in Oasis montaj

Scripting refers to the act of creating and running a file that will carry out many operations in a row. If you perform a task repeatedly in Oasis montaj, you can automate the task using a script. Geosoft Scripts are ASCII files (with a *.gs extension) that contain a series of actions or commands that are grouped together in a single file to accomplish a task automatically. This How-to Guide is intended to provide you with a basic introduction to Geosoft Scripts in Oasis montaj.

Some typical uses for scripts include:

- Automate complex or repetitive tasks
- Create standard map layouts
- Run Oasis montaj functions from the DOS command prompt
- Process large amounts of data overnight

Geosoft Scripts work by recording the actions you perform on a dataset. When you run a script, the commands you recorded are executed on data in a specified project. You can run scripts either interactively in Oasis montaj or from the command line using the Oasis montaj Scripts Processor (OMS.exe).



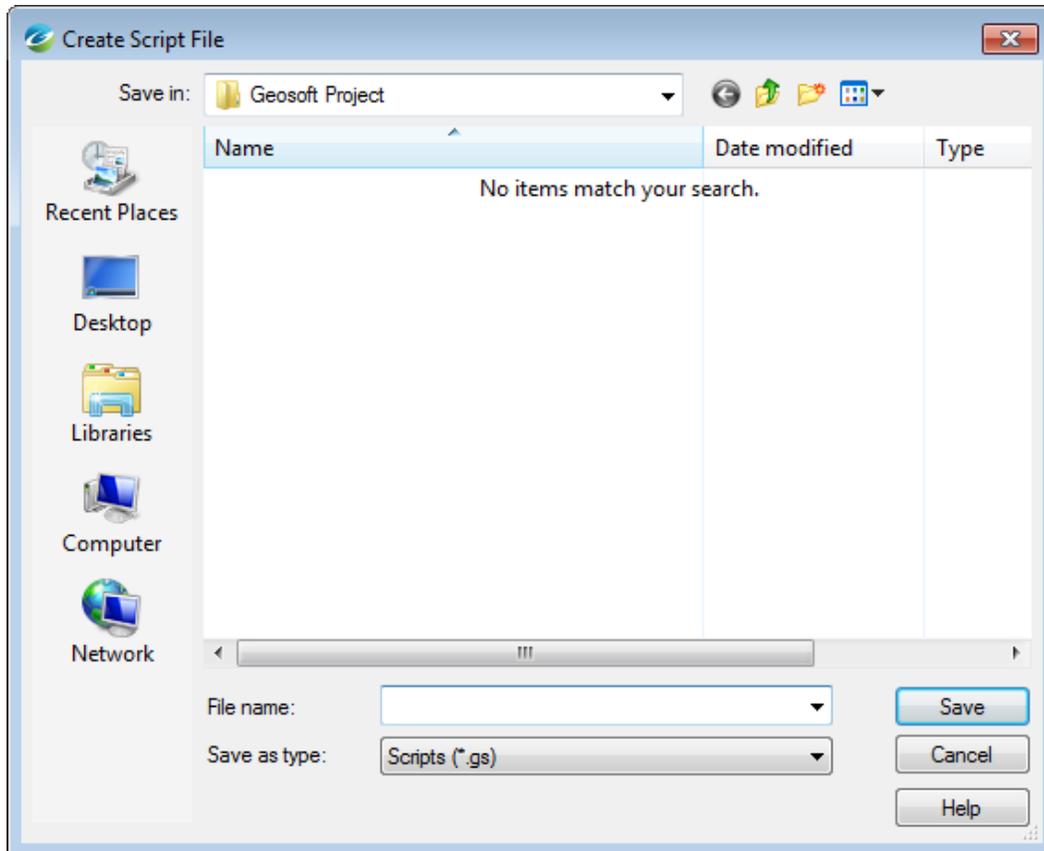
*For more information on using Scripts, see the *Creating Scripts In Oasis montaj Technical Note*, available online at: www.geosoft.com/resources/technotes/*

Batch Processing with Scripts

Oasis montaj Scripts are ASCII files that define a series of processing actions to be performed on selected datasets. You can run scripts interactively in Oasis montaj using the script toolbar buttons or the *Scripting* commands found under the GX menu.

To Create a Script File

1. On the GX menu, select *Scripting* and then select **Record Script**.
The *Create Script File* dialog appears



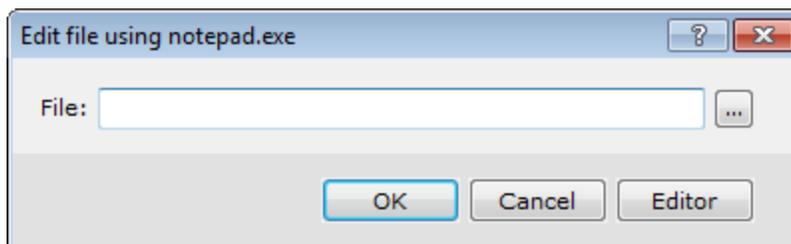
2. Specify a *File name*
3. Click **Save**.
4. Perform your scripting actions.
5. On the *GX* menu, click **Stop recording...**. The system stops recording your actions. All of your actions were recorded and saved to the specified script ***.gs** file.



Click the **Help** button for more information on creating a Script file.

To view a Script

1. On the *Edit* menu, click **Edit a File**.
The *Edit file...* dialog appears.



2. Using the *Browse* button, select the script file from your project folder.
3. Click **OK**. The system opens up the file in your default text editor.

Example

In the example below, we created a script file using the script file record option. This operation writes a script file (*.GS) by recording each GX that you run during the session until you stop script recording (from the GX menu). The resulting (*.GS) file contains sufficient parameter and database information to run the selected GXs in the same sequence. We recorded a script of two different GXs - added a line path and line profiles to a map.

Below is the recorded script file *test.GS*.

The two GXs that were recorded are shown beside the GX command: PATH and PROFILE.

```
/-----  
/ LOG OPENED : Mon Sep 10 15:37:30 2012  
/-----  
CURRENT Map,"e:\testing\Magnetics.map"  
CURRENT Grid,"e:\testing\mag.grd(GRD)"  
CURRENT Database,"e:\testing\Mag.gdb"  
SETINI PATH.LINE_COLOR="K"  
SETINI PATH.LINE_THICKNESS="0.15"  
SETINI PATH.TICKFLAG="0"  
SETINI PATH.LABEL_FORMAT="TL"  
SETINI PATH.LABEL_DIR="0"  
SETINI PATH.LABEL_COMPASS="0"  
SETINI PATH.LABEL_LOCATION="1"  
SETINI PATH.LABEL_SIZE="2.5"  
SETINI PATH.LABEL_COLOR="K"  
SETINI PATH.LABEL_WEIGHT="0"  
SETINI PATH.H_OFFSET="2"  
SETINI PATH.V_OFFSET="0"  
SETINI PATH.GAP="0"  
SETINI PATH.THINRES="0.02"  
GX path.gx  
CURRENT Map,"e:\testing\Magnetics.map"  
CURRENT Grid,"e:\testing\mag.grd(GRD)"  
CURRENT Database,"e:\testing\Mag.gdb"  
SETINI PROFILE.CHANNEL="mag"  
SETINI PROFILE.SCALE="100"  
SETINI PROFILE.BASE=""  
SETINI PROFILE.LOG="0"  
SETINI PROFILE.LOGBASE="1.0"  
SETINI PROFILE.SMOOTH="0"  
SETINI PROFILE.COLOR="MY"  
SETINI PROFILE.THICKNESS="0.15"  
SETINI PROFILE.STYLE="0"  
SETINI PROFILE.PITCH="5"  
SETINI PROFILE.PFILCOL="N"  
SETINI PROFILE.NFILCOL="N"  
SETINI PROFILE.GAP=""  
SETINI PROFILE.JOIN="1"
```

```
SETINI PROFILE.PLOTORDER="1"  
GX profile.gx  
/-----  
/ LOG CLOSED : Mon Sep 10 15:37:46 2012  
/-----
```



*For more information on using Scripts, see the *Creating Scripts In Oasis montaj Technical Note*, available online at: www.geosoft.com/resources/technotes/*

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.

Exporting and Printing Maps in Oasis montaj

Oasis montaj provides the ability to export interim processed data to another system for additional manipulation or to a GIS system for creating final map products and integrating with other sources of data. Required tools for exporting data to Geosoft XYZ and other formats are also provided.

Oasis montaj uses your installed Windows system drivers to create printer or plotter output. Depending on your installed driver, you may find it does not produce the highest quality output or that it takes a long time to produce a plot.

Geosoft provides a special printer configuration capability that enables you to initialize an internal raster engine that performs most of the print/plot processing and passes it to the driver for final output.

If you are not satisfied with your driver's performance, you can try selecting this option. Normally, with high-quality drivers, you should not have to use this option but it can help with problem drivers or PostScript printing.

Exporting Maps

Geosoft recognizes the need for the ability to move data, grids and images easily to other software systems. Supported export formats for maps include:

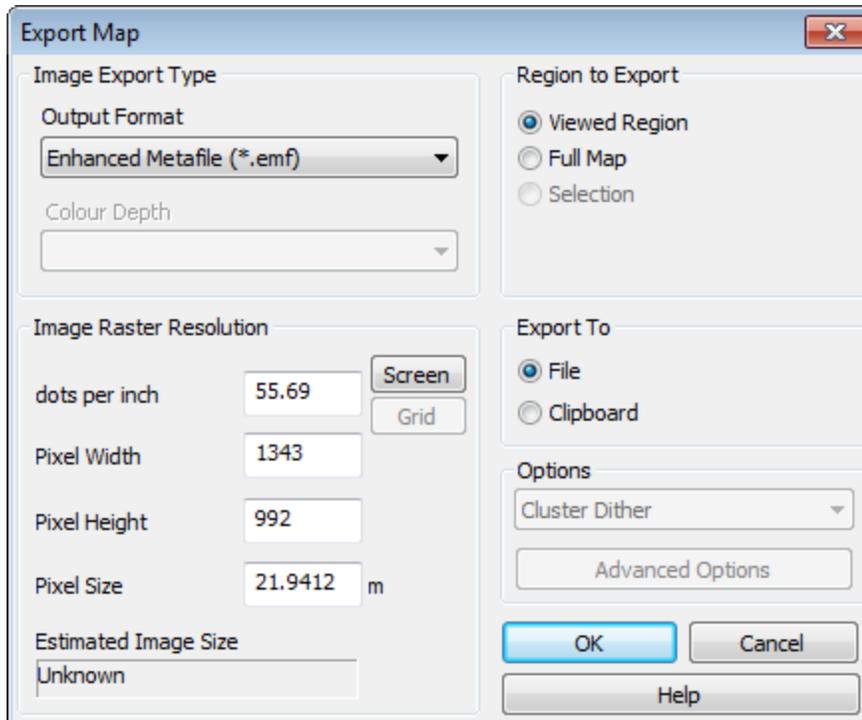
Enhanced Metafile	MapInfo TAB
ArcView Shapefile	Geosoft Plot file
CGM Plot	DXF AutoCAD
Bitmap	DXF AutoCAD v12
PCX	JPEG
PNG	JPEG High Quality
Encapsulated Post Script	TIFF
TIFF Compressed	GeoTIFF
Geosoft COLOUR Grid	MapInfo TIFF
ER Mapper RGB	ArcView TIFF
ER Mapper ECW Compressor	

Exporting Maps to a File

The Export Map option enables you to export your maps to a file or the clipboard. For more information on exporting data, refer to the *How-To Guide: Importing and Exporting Databases*.

To Export Maps to a File

1. Select an open map in your project.
2. On the *Map* menu, select **Export**.
The *Export Map* dialog appears.



3. Select the *Output Format*, such as *.EMF, *.TIF, *.PLT or as required.
4. Specify the *Region to Export* as *Viewed Region*, *Full Map* or *Selection*.
5. Set the *Image Raster Resolution*.
6. You can *Export To* to *File* or *Clipboard*.
7. From the *Options* dropdown list, select from *Cluster Dither*, *Error Diffusion* or *No Dither*.



For information on any of these parameters, click the **Help** button on the *Export Map* dialog.

9. Click **OK**. The system exports the map.



If you selected to export to a File the system will display the *Exported file name* dialog. Specify a name for the file and click **Save**.

Printing Maps

By default, the system is set up to use your current Windows driver. When you start printing maps for the first time, you will most likely accept the defaults. Depending on your driver's performance however, you may want to add a new configuration that uses a more advanced printing option, such as Geosoft bands, Geosoft bands and dither, or HP-RTL. Refer to the on-line help system for a complete discussion on the pros and cons of the different printing modes.

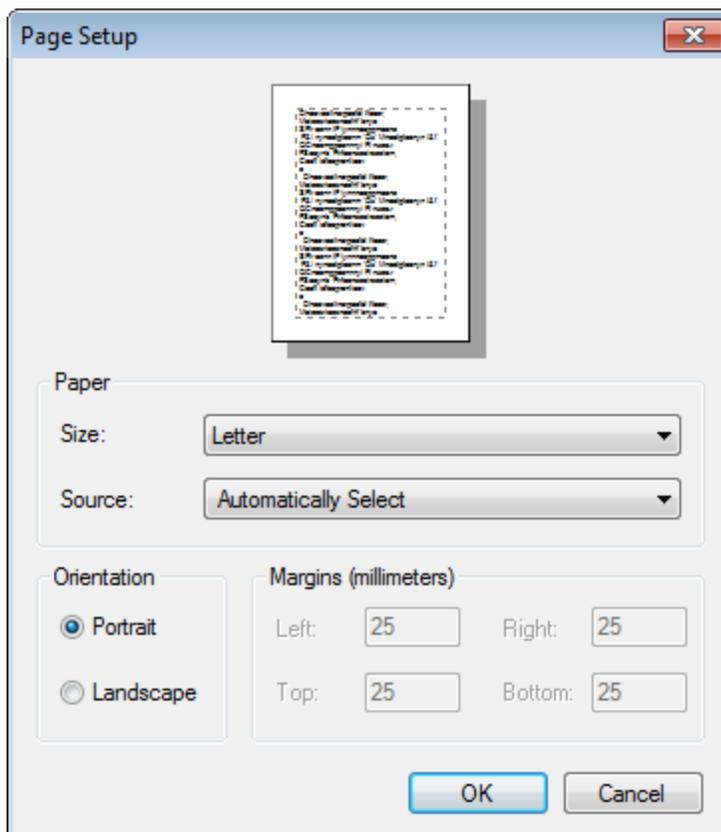
Page and Printer Setup

For optimal printing, you may have to experiment with printer settings and properties. Different drivers allow different selections (paper size, orientation, dithering patterns and colour controls). However, they can be very slow especially for large plots.

To Configure Your Page Setup

Before you begin, verify your project is open.

1. On the *File* menu, select **Page Setup**.
The *Page Setup* dialog will appear.

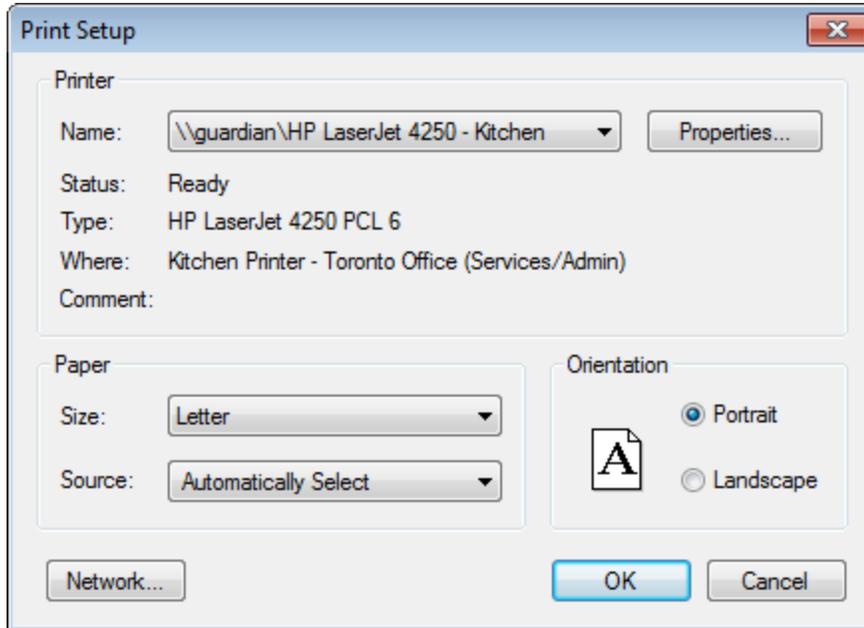


2. From the dropdown lists, select your *Paper Size* and *Paper Source*.
3. Specify the *Orientation: Portrait* or *Landscape*.
4. If required, adjust the *Margins*.
5. Click **OK** to confirm the settings.

To Configure Your Printer Setup

Before you begin, verify your project is open.

1. On the *File* menu, select **Printer Setup**.
The *Print Setup* dialog will appear.



2. From the dropdown list select the printer *Name*.



You can click *Network* to browse for a different network printer.

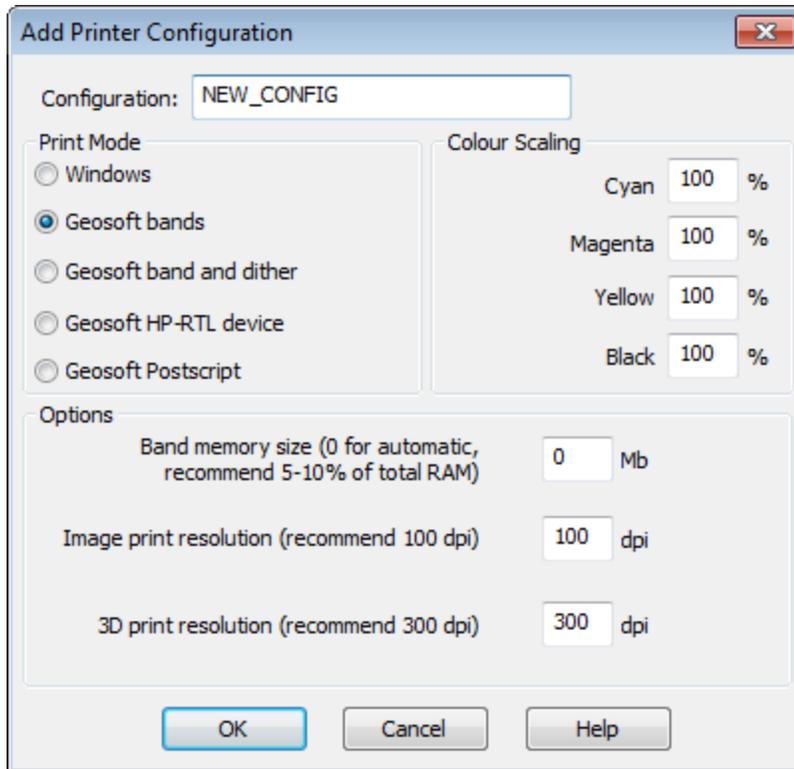
3. Adjust the *Paper* and *Orientation* settings as desired.
4. Click **OK** to confirm the settings.

Change or Add New Printer Configuration

Use the *Change Printer Configuration/Add Printer Configuration* dialog(s) to change the configuration of your printing mode, colour scaling and other print options.

To Add a New Printer Configuration

1. On the *Print* dialog, click the **Add Config** button.
The *Add Printer Configuration* dialog will appear.



2. Select your *Print Mode*.
 - If you want to use Windows drivers for printing, select *Windows*.
 - If you are having problems with your Windows drivers and want to optimize your printing speed, select *Geosoft bands* or *Geosoft bands and dither*
 - Select *Geosoft HP-RTL device* if you have this type of printing device
3. Adjust the *Colour Scaling* to change the CYMK colour values, if required.
4. If you want to adjust the *Image print resolution* change the default dpi value.
5. Click **OK**. The system saves your new configuration and initializes it for use.



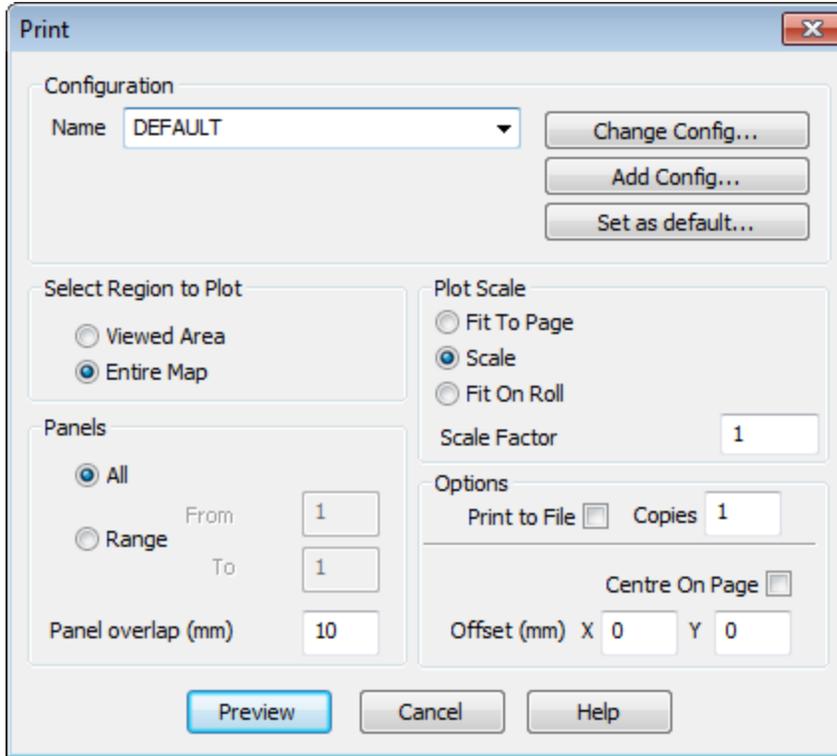
To change your printer configurations click the **Change Config** button from the *Printer Setup* dialog.

Print Preview

Use the Print Preview dialog to display the active map, as it would appear when printed.

To Preview a Map Before Printing

1. On the *File* menu, select **Print**.
The *Print* dialog will appear.



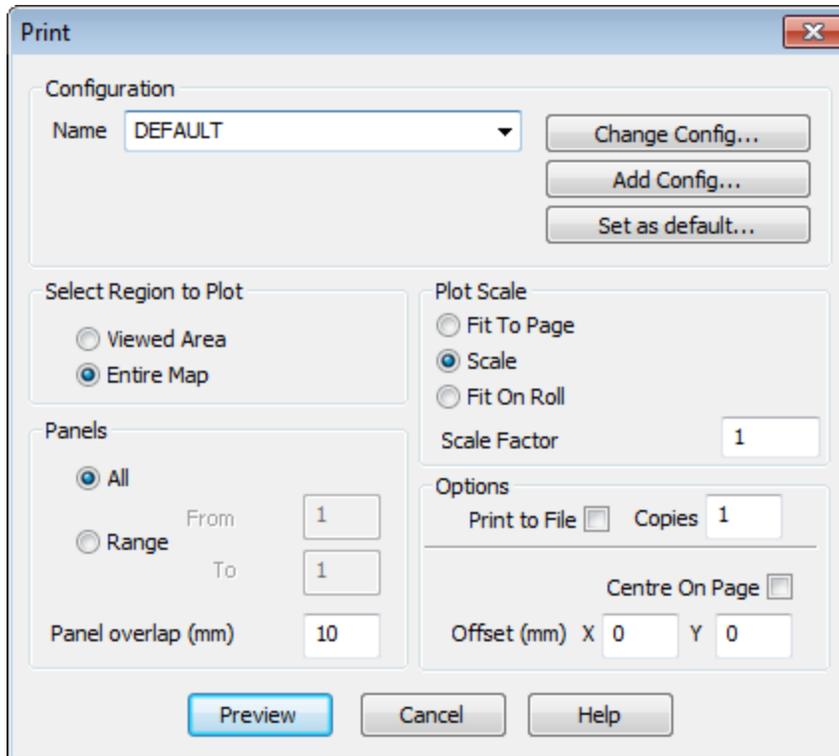
2. Specify the *Select Region to Plot* and the *Plot Scale*.
3. If your map uses *Panels*, specify that option.
4. Click **Preview**. The system displays a preview of how your map will look when printed on paper.
5. Click **Print** if the map is ready for printing or click **Close** to return to the *Print* dialog.

Print a Map

Use the *Print* dialog to specify how you want your print job to appear and to send a job to the printer.

To Print a Map

1. Select (highlight) the map that you would like to print.
2. On the *File* menu, select **Print**.
The *Print* dialog will appear.



3. Specify the *Region* you want to print: *Viewed Area* or the *Entire Map*.
4. Specify the *Plot Scale*.



To maintain the map scale, select the *Scale Factor* option and specify a value, (1 = 100 per cent). If you use this option, the map may require more than one page depending on the media to which you are printing or plotting.

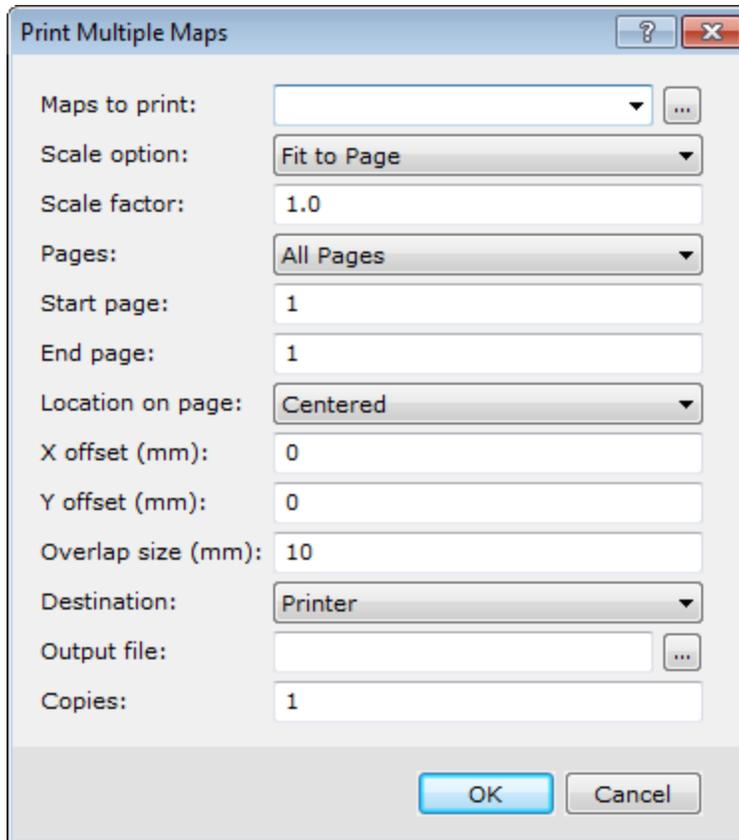
5. If the plot uses *Panels*, specify either *All panels* or a *Range* of panels. If you are printing multiple panels, you can specify an overlap in mm. This helps you to match panels later.
6. Specify the number of *Copies* to be printed.
7. If required, select the *Print to File* option box and specify a filename.
8. If required, select *Centre on Page* to centre the plot on the page. You can also centre the plot on a paper roll by selecting *Fit On Roll*.
9. To move the map from its origin (lower left corner of page or plot), specify an *Offset* distance in mm.
10. Click **Print** to send the map or plot to the currently configured printer or plotter.

Printing in Batch

For high efficiency automated printing and plotting, Geosoft provides a batch printing capability. This capability enables a number of plots to be printed sequentially without any hands-on intervention.

To Print in Batch

1. On the *File* menu, select **Batch printing**.
The *Print Multiple Maps* dialog will appear.



2. On the *Maps to Print* box, specify the maps you want to print. To specify more than one map, map names should be separated by a '|' (vertical bar) character.



*You can also select multiple files by clicking the **Browse** button. In the *Maps to Print* dialog press and hold the **Ctrl** key and while clicking the left mouse button to select the maps you wish to print.*

3. Specify the other parameters as required.



*For more information about Batch printing options click the **Help** button.*

4. Click **OK**. The system sends the maps to the installed printer or plotter.

How-To Guide Publication Date: 29/04/2014

Copyright 2014 Geosoft Inc. All rights reserved.