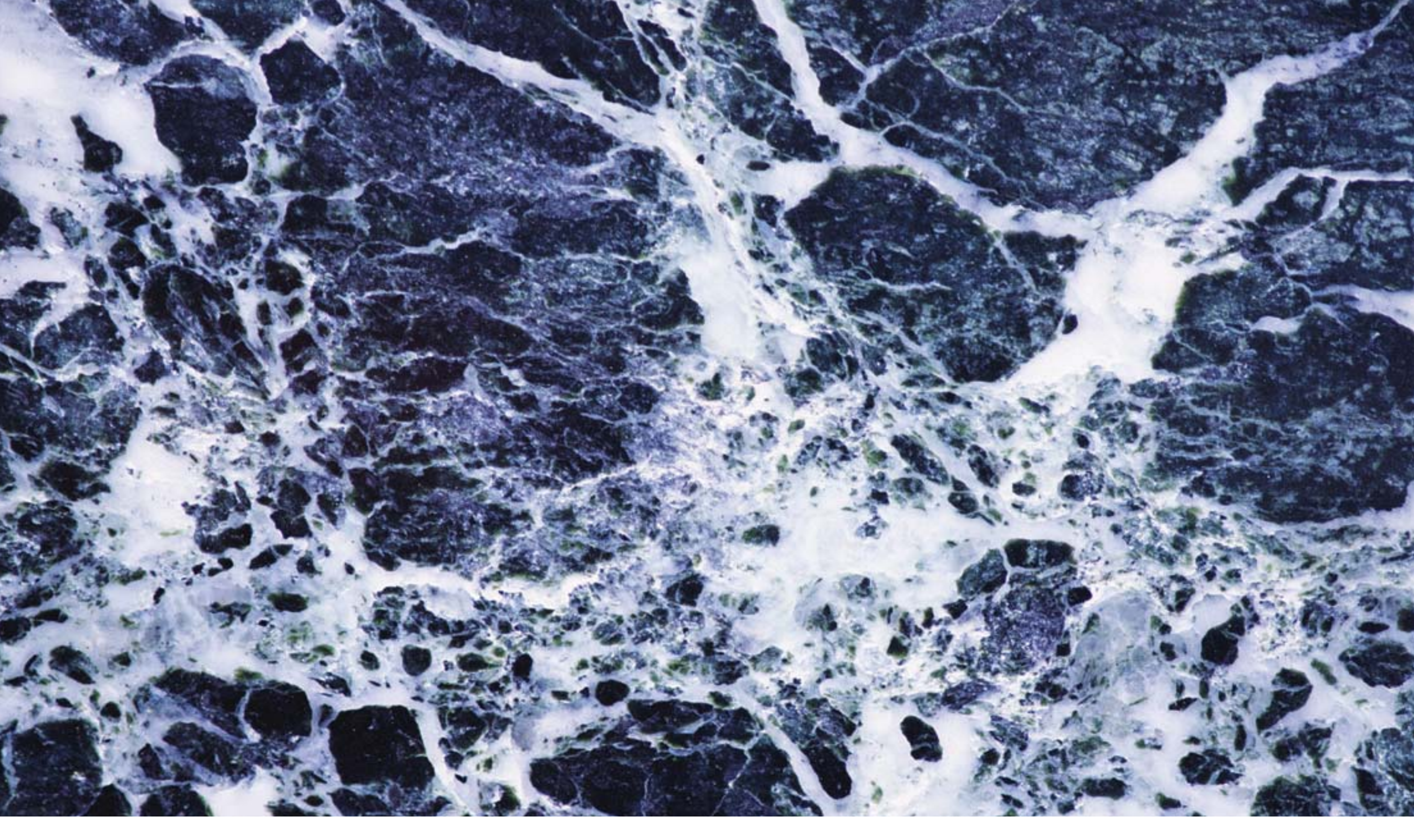
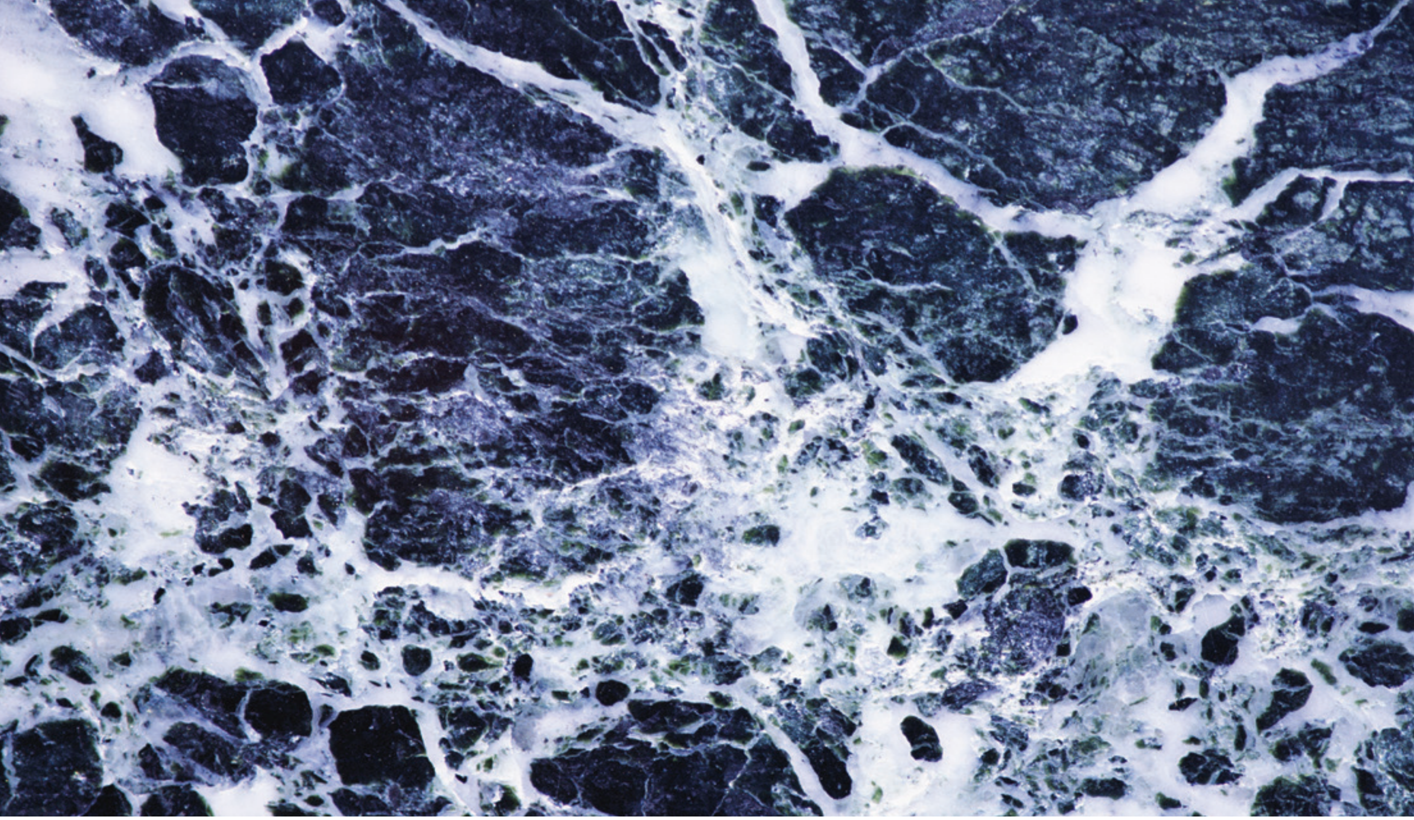


pbEncom™

Discover™ 3D 2013 User Guide



pbEncom™

Discover™ 3D 2013 User Guide

Discover 3D is developed and supported by
Pitney Bowes Software Pty Ltd

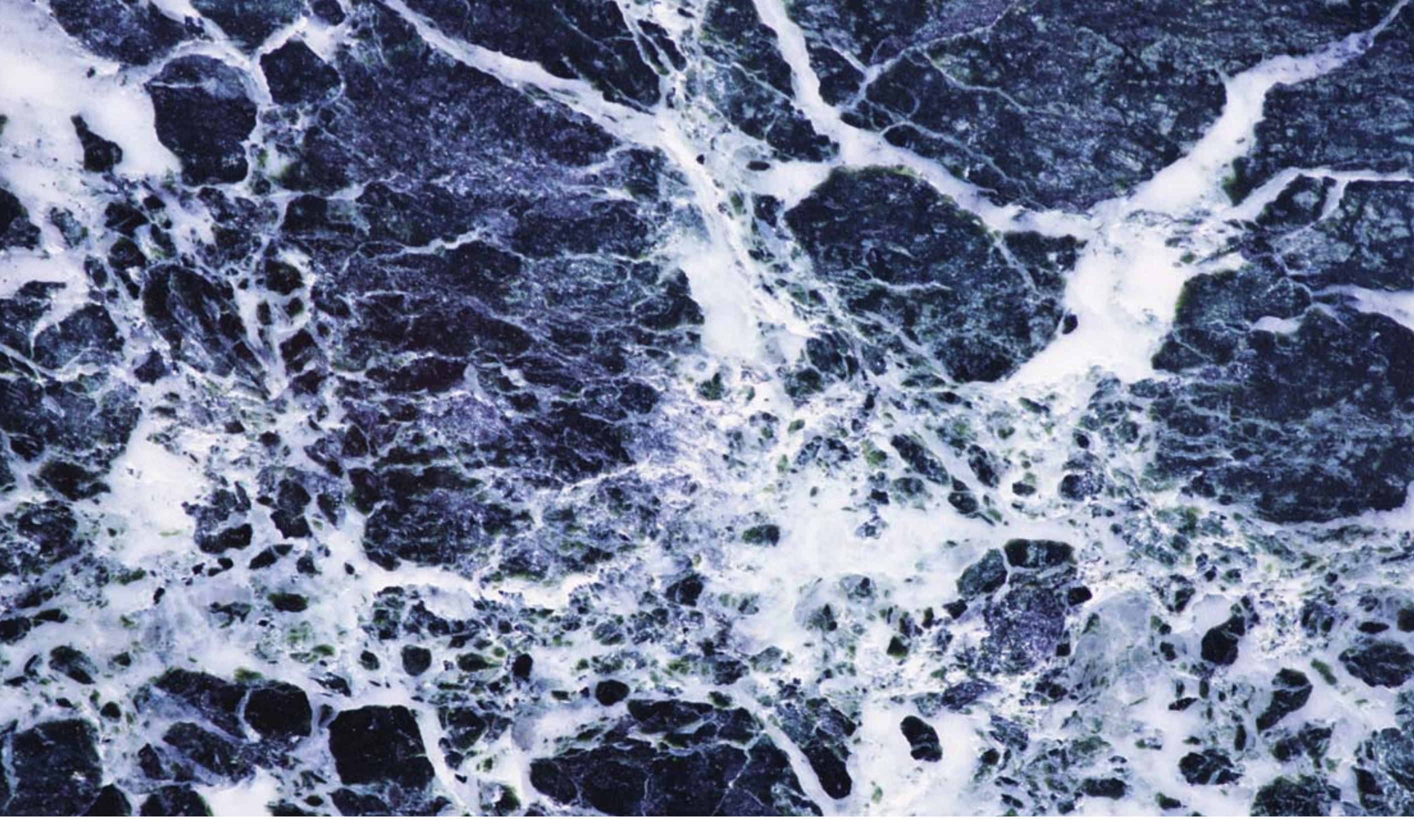
ASIA PACIFIC / AUSTRALIA
Level 7, 1 Elizabeth Plaza
North Sydney NSW 2060 Australia

SUPPORT
T: +61.2.9437.6255

software.support@pb.com
www.pbencom.com

DISCOVER 3D RELEASE HISTORY

v1.0	April 2004
v2.0	June 2005
v2.1	February 2006
v2.5	February 2007
v3.0	July 2007
v4.0	July 2008
v5.0	July 2009
v6.0	June 2010
2011	June 2011
2012	June 2012
2013	June 2013



pbEncom™

Discover™ 3D 2013 User Guide

Discover 3D is developed and supported by
Pitney Bowes Software Pty Ltd

ASIA PACIFIC / AUSTRALIA
Level 7, 1 Elizabeth Plaza
North Sydney NSW 2060 Australia

SUPPORT
T: +61.2.9437.6255

pbbi.support@pb.com
www.pbencom.com

DISCOVER 3D RELEASE HISTORY

v1.0	April 2004
v2.0	June 2005
v2.1	February 2006
v2.5	February 2007
v3.0	July 2007
v4.0	July 2008
v5.0	July 2009
v6.0	June 2010
2011	June 2011
2012	June 2012
2013	June 2013

Table of Contents

1 Introduction ... 1

About this Guide ... 4
Conventions Used in this Guide ... 4
Obtaining Help ... 5
System Requirements ... 5
Licensing Encom Discover 3D ... 6
Access to Tutorials ... 7

2 Getting Started in 3D ... 9

3 What's New in Encom Discover 3D ... 13

4 Choosing a 3D Display Type ... 15

Raster Images ... 16
DXF Vector Data ... 16
Points and Lines ... 17
Gridded Surfaces ... 18
Drillholes ... 18
Voxel (Block) Models ... 18
Feature Database ... 19

5 Viewing Mapinfo/Discover Data in 3D ... 21

Discover 3D Menu (MapInfo/Discover) ... 21
- Open 3D Window Tool ... 23
- Open 3D Workspace Tool ... 23
- Save 3D Workspace Tool ... 24
- Refresh All 3D Data Tool ... 24
- View Map in 3D Tool ... 24
- View Objects in 3D Tool ... 24
- View Surface in 3D Tool ... 24
- Create 3D Points Tool ... 25
- Create 3D Lines Tool ... 25
- View Drillholes Tool ... 25
- View Trenches Tool ... 25
- View Sections Tool ... 25
- View Logs Tool ... 26
- View Section Layers Tool ... 26
- View Intervals as 3D Points ... 26
- View Seismic Sections Tool ... 26
- 3D Display Wizard ... 27

3D Extrusion Wizard 27
3D Utilities Menu 27
Multi Section Creator Tool 27
Image on Grid Creator Tool 28
Object Manager Tool 28
Toggle Auto 3D Selection Tool 29
Options Tool 29
Managing the 3D Workspace 30
Opening the 3D Window 30
Opening a Saved 3D Workspace 30
Saving the 3D Workspace 30
Using the 3D Display Wizard 31
Entire Map Window 33
Objects from a Table 35
Refreshing Data from MapInfo/Discover 39
Displaying Map Window Views as 3D Images 39
Maximising Image Resolution 41
Displaying Map Objects 42
Selecting Map Objects to Display 42
Displaying Map Objects as 3D Vectors 43
Displaying Surfaces in 3D 45
Displaying Points in 3D 46
Displaying Lines in 3D 48
Displaying Drillhole and Trench Data in 3D 50
Viewing Drillholes in 3D 50
Viewing Drillhole Intervals as 3D Points 53
Viewing Drillhole and Trench Sections 57
Maximising Image Resolution 59
Viewing Drillhole Logs 59
Viewing Section Layers 61
Viewing Trenches 64
Extruding Objects in 3D 66
Displaying Other Types of Sections 67
Draping Images Over a Grid 69
Adding Landscape and Ornamental Features 70

6 Understanding the Discover 3D Interface 75

Discover 3D Window 75
Discover 3D Menus 77
File Menu 78
View Menu 81
Display Menu 84
Features Menu 86
Tools Menu 90
Grids Menu 91

Utilities Menu 92
Help Menu 93
Discover 3D Toolbars 94
Showing and Hiding Toolbars 95
Main Toolbar 96
Data Objects Toolbar 97
Zoom Controls Toolbar 98
3D View Manager Tool 100
Cursor Plane Toolbar 101
Features Toolbar 102
Feature Editing Toolbar 104
Discover 3D Session Files 106
Customising the 3D Window 107
Docking and Undocking Toolbars and Windows 107
Auto-hide Windows 108
Customising Toolbars and Window Views 109
Workspace Tree 109
Working with the Workspace Tree 111
Workspace Branch Properties 114
3D Map Properties 115
Title Tab 115
Appearance Tab 116
Lighting Tab 117
Scale Tab 119
Projections in 3D 120
Navigating in 3D 120
3D Navigation Modes 121
3D Navigation Controls 121
Using the 3DConnexion SpaceNavigator™ 125
3D Display Modes 126
Normal View Mode 127
Page Layout Mode 127
Full Screen Mode 127
3D Stereo Visualisation 128
HDMI 3D 128
Dual Projectors 129
Anaglyph Mode 130
Cursor Plane 130
Controlling the Cursor Plane Display 131
Controlling the Cursor Plane Position and Orientation 132
Navigating on the Cursor Plane 133
Changing Cursor Plane Properties 134
Plane 135
Appearance 137
Floating Colour Bar 138
Sky Map 140

7 Working with Raster Images in 3D 143

Encom Georeferenced Bitmap (EGB) Format 145
Converting a Map Window into a 3D Image 145
Maximising Image Resolution 146
Opening an Existing Georeferenced Bitmap Image 146
Using the Georeferenced Image File Creation Wizard 147
Interactive Image Registration 150
Directly Importing Data as EGB Images 152
Changing Image Display Properties 153
Display Tab 155
Surface Tab 156
DTM Tab 158
Transparency Tab 159
Transform Tab 160
Image Tab 161
Grid Tab 162

8 Working with Vector Data in 3D 165

Displaying 3D Vectors 166
Changing 3D Vector Display Properties 168
Displaying 2D Vectors in 3D 169
Manipulating, Transforming and Reprojecting Vectors 171
Creating Orientated 3D Vector Symbols 171
Extruding 2D Vectors into 3D 172
Generating 3D Models from Vectors 172

9 Working with 3D Points 173

Interrogating Point Information 174
Changing Point Display Properties 174
Saving or Loading Point Display Settings 175
Data Selection 175
Offsetting and Scaling Data 176
Symbol Style 178
Labels 181
Colour Modulation 183
Size Modulation 184
Rotating Point Symbols 185
Decimation 187

10 Working with 3D Lines 189

Interrogating Line Information 189

Displaying Lines as 3D Tubes 190
TubeShape Manager 191
Changing Line Display Properties 193
Saving or Loading Line Display Settings 193
Line Selection 194
Offsetting and Scaling Data 195
Line and 3D Tube Style 198
Colour Modulation 199
Thickness Modulation 200
Colour Filling Line Profiles 201
Decimation 202

11 Working with Directional Vectors 203

Changing Directional Vector Display Properties 204
Vector Tab 204
Lines Tab 206
Origin Tab 207
Components Tab 208
Tail Tab 210
Head Tab 211
Colour Tab 213

12 Displaying Gridded Surfaces in 3D 215

Supported Grid Formats 216
Loading a Grid into 3D 216
Grid Flipper 217
Creating a Grid Legend 218
Grid Groups and Surfaces Branches 219
Transparency and Colour-Intensity Balance 222
Texture Mapping 223
Viewing Grid Information 224
Changing Surface Display Properties 225
Scaling Grids 225
Grid Compression 227
Constant Compression 227
Variable Compression 228
Offsetting a Grid 229
Rendering Surfaces 231
Pseudocolour Surfaces 232
Controlling the Colour Layer 233
Controlling the Intensity Layer 234
RGB Image Surfaces 236
Draping Located Images 238
Surface Contours 239

Controlling the Contour Layer 240
Exporting Contours 244

13 Creating Gridded Surfaces 245

Interpolating a Gridded Surface from Digitized Drillhole Intercepts 247
Using the Surface Gridding Tool 250
Gridding Tool Preview Display 252
Gridding Tool Input Tab 253
Breaklines 253
Coincident points 254
Statistics Explorer 254
Data Conditioning 255
Field Data Conditioning 256
Gridding Tool Method Tab 256
Minimum Curvature 257
Kriging 257
Inverse Distance Weighting 261
Search Tab 264
Spatial Neighbour 267
Triangulation 268
Density Grid 268
Distance Grid 269
Bi-Directional Spline 270
Gridding Tool Geometry Tab 271
Gridding Tool Output Tab 272

14 Working with Drillholes and Trenches in 3D 275

Displaying Drillholes and Trenches in 3D 277
Drillhole Dataset Format 278
Discover Drillhole Project 279
Geosoft WHOLEPLOT Database 280
Desurvey Methods 281
Back Calculation 281
Segments 282
Minimum Curvature 282
Akima Spline 283
Bezier Spline 283
XYZ (trenches only) 283
Interrogating Downhole Information 283
Changing Drillhole Display Properties 285
Saving and Applying Display Settings 286
Selecting Drillholes 287
Trace Style 289
Labelling 292

Symbols ... 295
Colour Modulation ... 297
Drillhole Colour Legends ... 300
Thickness Modulation ... 300
Display Drillhole Logs Images ... 305
Data Compression ... 310
Changing Trench Display Properties ... 310
Saving and Applying Display Settings ... 311
Selecting Trenches ... 312
Trace Style ... 312
Colour Modulation ... 312
Thickness Modulation ... 312
Labelling ... 312
Symbols ... 313
Drillhole Planner ... 313
Planning Drillholes in 3D ... 314
Collar RL/Z Value Assignment ... 318
Drillhole Types ... 320
Targeting Example ... 322
Drillhole Planner Projects ... 326

15 Viewing Seismic Sections ... 329

Seismic Workflow ... 329
Seismic Project Files ... 335
Coordinates ... 336
Managing Views and Appearance ... 338
Digitizing Horizons ... 339
Managing Horizons ... 340
Viewing Profiles ... 341
Dialog Help ... 341
Project Settings Dialog Box ... 342
Section Properties Dialog Box ... 343
Plan View Dialog Box ... 345
Generate 3D Dialog Box ... 345

16 Digitizing and Managing 3D Features ... 347

Feature Object Types ... 349
Opening an Existing Feature Database ... 350
Interacting with Features ... 351
Creating a New Feature Database ... 352
Feature Database Creation Wizard ... 353
Importing Data into a Feature Database ... 355
Supported Import Formats ... 356
Using the Import Feature Database Wizard ... 356

Saving Cosmetic Features to a Feature Database 360
Modifying the Structure of a Feature Database 362
Exporting Feature Databases 364
Changing Feature Display Properties 364
Feature Appearance 365
Labelling Features 365
Selecting Features 367
Graphical Selection 368
Select by SQL Query 368
Browser Selection 369
Creating and Editing Features 370
Creating Features 370
Snapping Features to Other Data 372
Editing Features 375
Advanced Editing Functions 379
Triangulate 380
Combine 381
Intersect 382
Cut 383
Aggregate 388
Disaggregate 388
Break Mode 389
Consolidate 391
Feature Attributes 391
Data Window 392
Feature Database Window 392
Feature Selection 393
Attribute Editing 393
Node Editing 393
Feature Options 394

17 Displaying Voxel Models 397

What are Voxel Models? 397
Supported Voxel Model Formats 398
UBC Model Format 400
Mesh File 400
Property File 401
CEMI Models 401
Noddy Models 403
ASCII XYZ Models 405
Creating a 3D Voxel Model Display 405
Importing and Exporting Voxel Models 406
Using the Load Model Wizard 406
Using the Export Model Wizard 408
Import and Export Voxel Models with the Grid Management Tool .. 410

Viewing Voxel Model Information 410
Changing Voxel Model Display Properties 411
Turning Model Rendering Options On and Off 413
Offsetting and Exaggerating 414
Clipping Along Principal Axes 415
Chair Clipping 418
Colour and Transparency 419
Lithology Control 422
Thresholding 424
Interactive Mode 424
Lithology Mode 426
Isosurfaces 429
Lithology Mode 429
Interactive Mode 430
Using Wireframes 433
Slice View 434
Slice Orientation 435
Slice Display Settings 437
Viewing Slices as Georeferenced Images (EGB) 438
Viewing Slices on 2D Discover Drillhole Cross-Sections 439
Displaying a Floating Colour Legend 439
Displaying Multiple Voxel Models 440
Integrating Voxel Model Displays with Other Objects 440

18 Creating and Manipulating Voxel Models 443

Creating a Voxel Grid 446
3D Gridding Methods 449
Continuously Variable Gridding Methods 449
Inverse Distance Weighting 450
Kriging 450
Discretised Gridding Method 453
Distance Gridding Method 454
Density Gridding Method 455
3D Gridding Parameters 458
Input Data 458
Data Conditioning 460
Classification 462
Grid Size 463
Search Parameters 465
Searching 466
Anisotropic Search 467
Preview 467
Sample Selection 469
Gridding Rule 469
Method 469

Weighting Models 470
Kriging Estimation Method 473
Computing a Sample Variogram 473
Defining a Model Variogram 476
Advanced Gridding Properties 481
Convolution Filtering 481
Voxel Utilities 482
Assign Values to Features 483
Clip to Surface 485
Clip to Vector Volume 487
Convert Nulls 488
Convert Vector to Voxel 488
Encom ModelVision TKM models 489
Create Mesh 490
Merge 490
Resample 490
Volume Selection Method 490
Test Rule 491
Acceptance Test 493
Voxel Calculator 494
Advanced Calculator Properties 496
Display Assistant 497
Grid Management 500
Importing ASCII Voxel Models 501

19 Modelling Triangulated Surfaces and Solids 505

Modelling Methods and Tools 505
Extruding Models from Points, Lines and Polygons 508
Wireframing Models from Polylines, Polygons and Surfaces 520
Digitizing Section Vectors 523
Digitizing Layers in 2D 523
Digitizing Sections in 3D 523
Creating a Wireframe Model 524
Output Options 528
Using Algorithm Parameters to Control Wireframing 528
Compression 529
Refinement 529
Structure Analysis 530
Shape Matching 531
Using Tie Lines to Control Wireframing 533
Creating Tie Lines 536
Selecting Points Within a 3D Wireframe Model 540
Viewing Models in 2D MapInfo/Discover 541

20 3D Utilities 543

Fly Through Wizard 544
3D Solid Generator 544
Topology Checker 544
    Using the Topology Checker 545
    Topology Checker Controls 547
Transform Vector File 549
Batch DXF Transform 554
Combine DXFs 556
Extrusion Wizard 556
3D Point Symbols 557
Drillhole Planner 559

21 Creating Images and Movies 561

Capturing the 3D View as an Image 561
Adding the 3D View to a 2D Map Window 562
Printing 563
    Customising the Page Layout 565
    Default and Offline Printer Configuration 565
Creating a Geolocated Image 566
    Creating an Image Using the Cursor Plane 567
    Creating a Drillhole Cross-section Image 568
Creating a Fly-Through Animation 571
    How Fly-Through Animation Works 573
    The Fly-Through Creation Wizard 574
        Static View Mode 585
    Codecs and Movie File Format 586
    Playing an Existing Fly-Through File 588

22 Working with Colour Tables and Legends 589

Using the Colour Look-Up Table Editor 590
Using the Legend Editor 592
    Legend Editor Controls 595
    Legend Location 598
Advanced Colour Mapping 599

23 Customising Discover 3D 603

Customising 2D Interface Settings 603
    System Tab 603
    Display Tab 605
Customising 3D Interface Settings 605

File Tab ... 606
Size Tab ... 607
View Tab ... 608
File Locations Tab ... 609
Page Setup Tab ... 610
Fullscreen Tab ... 611
Customizing Toolbars ... 611
Adding and Removing Tools ... 612
Restoring Toolbar Settings ... 613
Toolbar Button Size ... 613
Show and Hide Tool Labels ... 613
Creating New Toolbars ... 613
Other Customizations ... 614
Assigning Custom Keyboard Shortcuts ... 614

Appendices ... 617

A 3D Cursor Keyboard Shortcuts ... 619

B Statistics Explorer Tool ... 621

Spreadsheet ... 622
Zoom, Selection and Display Controls ... 624
Display Modulation ... 625
Subsetting ... 625
Univariate ... 625
Bivariate ... 630
Spatial ... 632
Variogram ... 634

C Field Data Conditioning Tool ... 641

D Voxel Calculator Syntax ... 647

Logical Operators ... 647
AND Operator ... 647
NOT Operator ... 648
OR Operator ... 648
XOR Operator ... 649
IF Operator ... 650
Arithmetic Operators ... 651
Anti-Log Functions ... 651
Comparison Operators ... 652
Boolean Operators ... 652

Trigonometric Operators 653

Index 655

1 Introduction

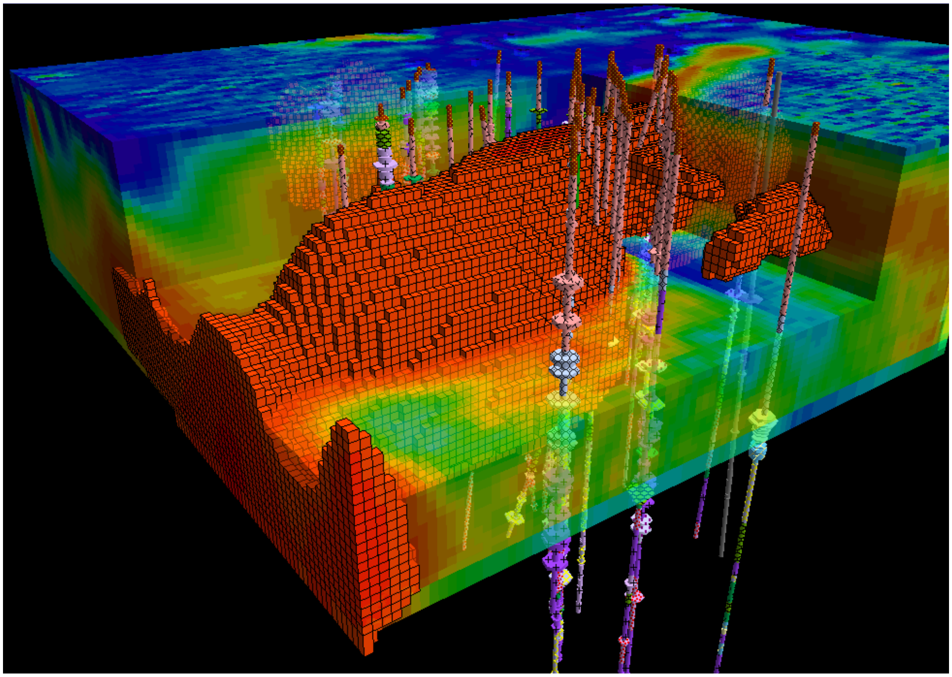
Encom Discover 3D 2013

Encom® Discover 3D is designed to work seamlessly with Encom Discover, allowing you to rapidly visualize, model and analyze your drillhole and related datasets in 3D, and then dynamically plan follow-up drilling to test your theories. Encom Discover 3D is installed with Encom Discover and requires an upgraded licence to activate its full functionality.

All Discover users can access the viewer mode of this application; map windows views can be draped over gridded surfaces in 3D, and existing 3D sessions from other users can be viewed.

A full Encom Discover 3D licence allows gridded surfaces representing topography, geochemistry and geophysics, drillholes containing assays or lithological data or any other mapped data combinations created using MapInfo® Professional and Encom Discover to be transferred to an interactive three-dimensional environment. Additional objects such as 3D DXFs, grid surfaces and raster images plus point and line data can also be added to the 3D displays from MapInfo Professional or independently.

The display below is typical of the type of displays derived from various data types.

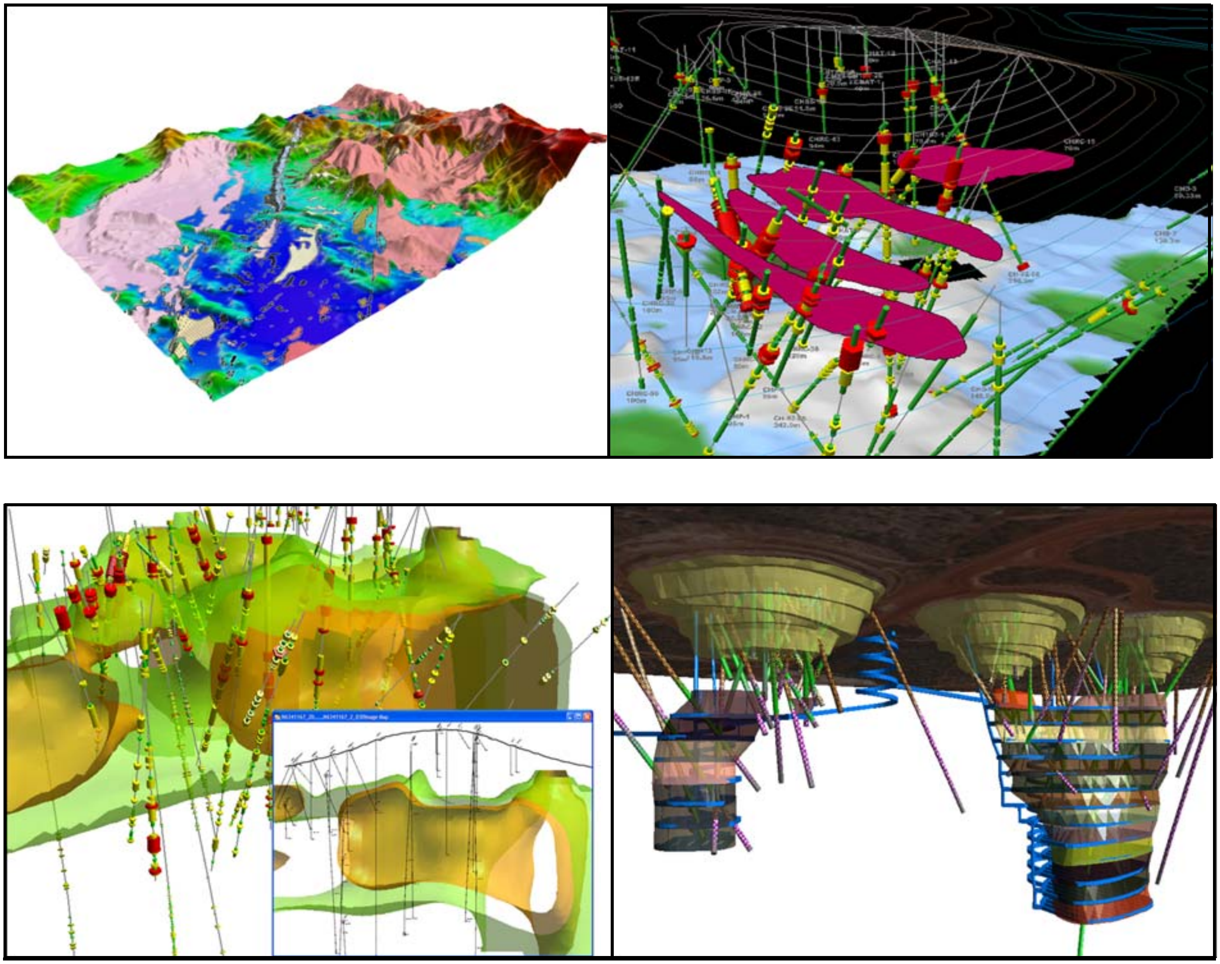


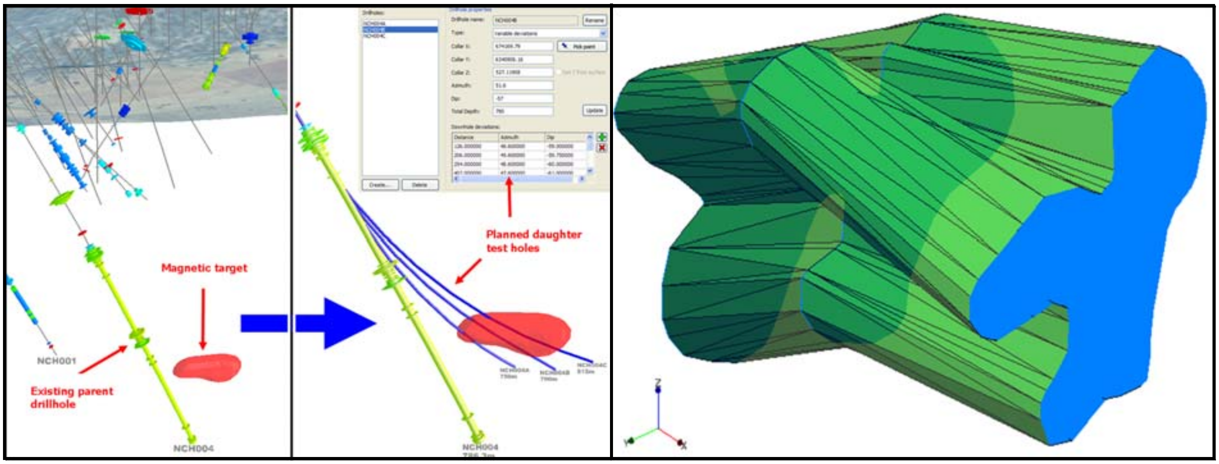
A complex 3D view incorporating an airphoto image draped over a DEM, colour and thickness modulated drillholes, ore body and development drive vector models, and a voxel model of magnetic susceptibility displayed as an isosurface.

Encom Discover 3D enables data to be viewed interactively with zooming, panning and fly-through capability controlled by either a regular mouse or the intuitive SpaceNavigator 3D mouse. 3D views can be captured as images and either displayed in MapInfo Professional in a layout for presentation purposes, or used in other applications (such as PowerPoint). Or, for a more powerful and effective presentation, create dynamic 3D movies for shareholder and management presentations.

Encom Discover 3D puts a powerful suite of modelling tools at your fingertips, whether you are developing mineralization, structural or geological models. Not only can 3D objects be created from your 2D datasets; points, polylines and polygons can be digitized directly into the 3D environment, accurately snapping to drillhole intervals or outlining trends. These 3D objects can then be extruded and wireframed to create triangulated (TIN) surfaces and polyhedral solids. A toolbox of advanced editing functionality allows these TIN surfaces and solids to be further manipulated; for instance modelling fault truncation of an orezone.

Encom Discover 3D operates seamlessly with Encom Discover. If you know how to create a map, a drillhole project and sections in Encom Discover, this is all you need to operate Encom Discover 3D. No additional project specification or data setup is required since the displays within MapInfo Professional, created by Encom Discover, are passed directly to Encom Discover 3D for presentation and interactive use.





About this Guide

This guide describes existing features in this version of Encom Discover 3D. The manual is organised into chapters containing logically grouped functions that relate to the Discover 3D menu structure.

This guide applies to Encom Discover 3D version 2013 only. Pitney Bowes Software has taken care to ensure that the information presented here is accurate but reserves the right to make alterations to Encom Discover 3D at any time.

This is not an Encom Discover manual and knowledge of both Encom Discover and MapInfo Professional is required for Encom Discover 3D to be used to its best advantage. We recommend that you refer to the user guides available with Encom Discover and MapInfo Professional.

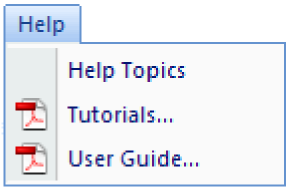
Conventions Used in this Guide

Certain conventions are used throughout this guide.

- Keys on the keyboard appear in small capital letters. For example, the Ctrl key appears as CTRL in the text.
- Menu options and dialog items are in normal text but bolded. For example, **choose File>Run MapBasic Program**.
- Buttons to be clicked are bolded. For example, click the **Remove Entry** button.

- References to other sections in the documentation are italicised. For example, see the *Display 3D Surfaces chapter.*

Obtaining Help



The Discover 3D Help menu provides access to the Discover 3D Help, Tutorials (PDF), and User Guide (PDF). The Help is an interactive version of the User Guide.

When contacting product support for Encom Discover 3D, please have the following details available:

- Licence Number (use **Discover Help>About Discover** tool in the 2D environment).
- Encom Discover version number.
- MapInfo Professional Version (use **Help>About this MapInfo Product**).
- A full description of the problem or query. This should include any system messages (from Encom Discover 3D, MapInfo Professional or the operating system) and other pertinent information detailing the circumstances.

Contact details are:

E-mail	*software.support@pb.com*
Web	*www.pbencom.com*

If you experience any problems with Encom Discover 3D, or have suggestions or comments, Pitney Bowes Software would be pleased to hear from you.

System Requirements

Encom Discover 3D is installed with the Encom Discover 2013 software. Encom Discover 2013 requires MapInfo Professional version 10.5 or later.

Encom Discover 3D is supported on Windows® XP Professional (Service Pack 3) and Windows 7 and 8 32-bit and 64-bit operating systems but not on earlier Windows systems.

Minimum requirements

- Core 2 series or equivalent CPU processor
- 2 gigabyte of RAM memory
- A monitor of at least XGA capability (1024 x 768 resolution)

Recommended requirements

- Core 2 series or equivalent CPU processor (note that Discover 3D does not utilise multiple CPU cores)
- 4 gigabytes of RAM memory
- Dedicated graphics card with 256 MB dedicated VRAM memory, and with OpenGL optimised drivers, such as Nvidia Quadro or ATI FireGL series.
- A monitor of at least SXGA capability (1280 x 1024 resolution)

Licensing Encom Discover 3D

Encom Discover 3D is accessible to any Encom Discover user in a viewer-only mode. To access all disabled or unavailable options, an Encom Discover 3D licence is required to be added to the existing Encom Discover licence.

For details on activating, upgrading or transferring Encom Discover 3D on your Encom Discover licence, refer to the licensing pocedure in the *Encom Discover User Guide*.

Note that it is not possible to transfer a Encom Discover 3D licence between computers independently; it must be transferred in conjunction with the associated Encom Discover licence.

Access to Tutorials

To assist you in learning how to use Encom Discover 3D, a set of tutorials is provided as part of the installation. The tutorials comprise of a descriptive document that explains the operation and steps to follow, plus data that are used in the tutorial exercises. All tutorials are based on real exploration situations in which Encom Discover 3D can be used to display, enhance and visualise field data.

The tutorial is available as a PDF document. The PDF file can be accessed from the Help menu within the Discover 3D window.

Discover 3D tutorials are installed from the Encom DVD-ROM. A total of 80 Mb of disk space is required to install the tutorial data. For information on viewing and printing PDF documentation, see the *Encom Discover User Guide*.

2 Getting Started in 3D

If you new to Discover 3D, the following steps will guide you through the essential sections of this User Guide to quickly get you displaying and visualising your data in the 3D environment.

Note It is recommended to work through the Discover 3D Tutorials before accessing this more detailed User Guide.

What Data Formats are Supported by Discover 3D?

The various formats for displaying data within the 3D environment are summarised in *Choosing a 3D Display Type*. Each format (e.g. raster image, vector, 3D points, etc) has benefits and limitations, and understanding these is essential to working efficiently and effectively in Discover 3D.

How Do I Display My MapInfo Data in the 3D Environment

Once you have decided how you would like to display your data in 3D (i.e. what format as discussed above), use the appropriate menu option on the Discover 3D menu (*Viewing Mapinfo/Discover Data in 3D*).

Discover 3D has been designed so that if you can display your data within the 2D MapInfo window, you can quickly and painlessly display it in 3D using the options on this menu (e.g. using *Displaying Map Window Views as 3D Images*, *Displaying Surfaces in 3D* or *Displaying Drillhole and Trench Data in 3D*).

Certain data formats can also be drag and dropped into the 3D window, or opened from directly within Discover 3D – these options are discussed fully in each data format's particular chapter.

Navigating the 3D Environment

With your data displayed in 3D, use the navigation controls (*Understanding the Discover 3D Interface*) to zoom, rotate and navigate around the 3D environment to start getting a better feel for the spatial relationships between your datasets. Perhaps toggle the Orthographic View mode to remove the distance bias associated with the default Perspective View.

Discover 3D also supports the 3DConnexion SpaceNavigator (see *Using the 3DConnexion SpaceNavigator™*) for smooth intuitive "fly-through"-style navigation.

The *Cursor Plane* allows you to view just portion of your data (clipping the data view), for example only the data behind the cursor plane, or a preset distance either side the cursor plane (envelope).

Modifying Data Displays in 3D

The display properties of the various 3D data formats are discussed in the sections devoted to each data format—for example, *Displaying Gridded Surfaces in 3D* and *Displaying Voxel Models*—and include general purpose controls for:

- **Transparency**.
- **Scaling** – for instance, to allow various geophysical grids of differing magnitudes to be sensibly displayed in the same 3D environment.
- **Z offset** – absolute positioning of datasets in the vertical dimension with respect to each other.

Other data-specific controls include options such as colour (and/or colour modulation by a data field), size (and/or colour modulation by a data field), labelling, width, decimation, etc.

Interrogating Data

The data tab of the Information window allows attribute information for data sets that are linked to MapInfo TAB files (i.e. 3D Points and 3D Lines, Drillhole downhole data) to be examined dynamically. The dataset needs to be set as both **Selectable** and **Browsable** in the *Workspace Tree*: moving the cursor over an object will highlight it's attributes.

Interpolating and Modelling Data in 3D

Discover 3D not only allows the user to visualise and modify the way 3D data is displayed, but allows interpretations to be made from this information in the 3D environment.

Points, polylines and polygons can be drawn in any orientation using the *Cursor Plane*, and saved into a Feature database (see *Digitizing and Managing 3D Features*). These feature objects can be edited, resized, moved and attributed. They can also be exported to a number of common 3D vector formats (such as .DXF). Discover 3D can also import vector linework into a Feature Database, such as interpreted section boundaries in a Discover Drillhole Project.

Accurate 3D digitization is aided with options such as:

- Snap mode (*Features Toolbar*) – feature object nodes can be snapped to an existing object's nodes.

- Orthographic View mode (*Zoom Controls Toolbar*) - all objects are the same size regardless of their distance from the viewer.

- Perpendicular View mode (*Cursor Plane*) - shifts the view perpendicular to the cursor plane.

Examples of feature objects digitized in 3D include mineralization intercepts or zones deduced from downhole assays, fault trends interpreted by spatially comparing downhole geological logging with magnetics grids and voxel models, or proposed drillholes/drilling targets.

Discover 3D can then wireframe between these feature objects, creating complex solids using a range of powerful modelling tools (see *Modelling Triangulated Surfaces and Solids*). Alternatively, Discover 3D can smoothly interpolate between your feature objects with the Surface Gridding tool (see *Creating Gridded Surfaces*), using methods such as Minimum Curvature and Inverse Distance Weighting. This powerful tool includes tie-line controls (see *Using Tie Lines to Control Wireframing*) as well as multi-body support. Your interpretations can thus be used to build 3D complex fault surfaces, mineralization grade shells, alteration halos, etc.

Discover 3D can also create voxel (block) models (see *Creating and Manipulating Voxel Models*), from your drillhole project downhole data using powerful 3D gridding methods such as Kriging and Inverse Distance Weighting. These mathematically interpolated models can then be visualised with a host of options, such as isosurfaces (i.e. gold grade shells), slices, thresholding, etc.

Producing Output from Discover 3D

Your Discover 3D view can be exported with a range of options:

- In *Page Layout Mode*, you can use the standard printing controls (File>Print) to produce hardcopy of the 3D view.

- A simple screenshot of the current 3D view can be created using the *File>Save View As* menu option (.JPG, .BMP, etc.)

- The current view can be sent to MapInfo as a mapper window using the *View>Send to MapInfo* menu option. This map window could therefore be added to a Layout window as a Frame Object: for instance a 2D cross section could have a small 3D view of the drillhole project added to give a 3D spatial perspective.

- All data in the envelope of a drillhole project’s cross-section can be captured as a Georeferenced Image and exported to that section in 2D using *Tools>Georeferenced Image Export*. This allows 3D data such as voxel models and solids interpolated from digitization to be displayed in 2D cross-sections.

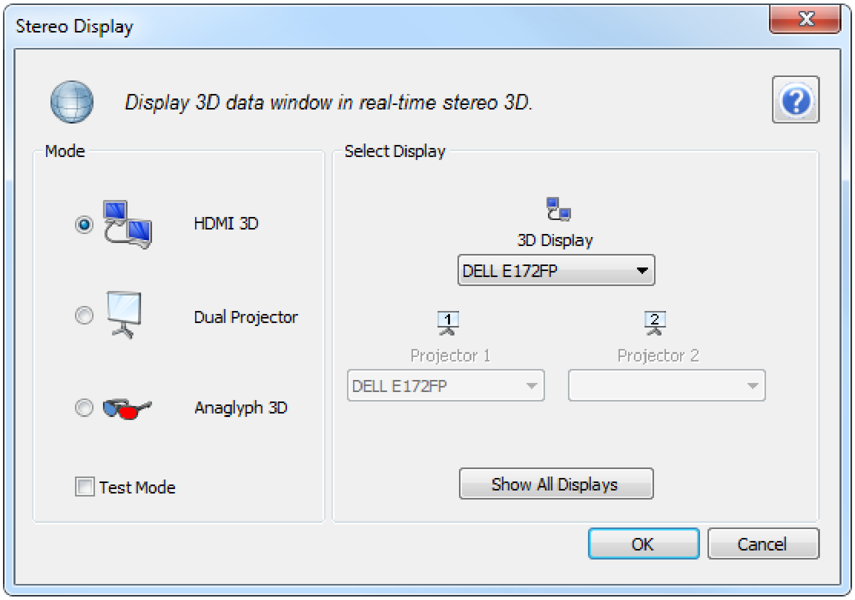
- Fly-through movies of your 3D environment (see *Creating a Fly-Through Animation*) can be created, with full control over the data display as the movie progresses (i.e. data visibility can be turned on/off and properties altered such as transparency, colour, etc throughout the movie).

3 What's New in Encom Discover 3D

New features available in Encom Discover 3D 2013 include:

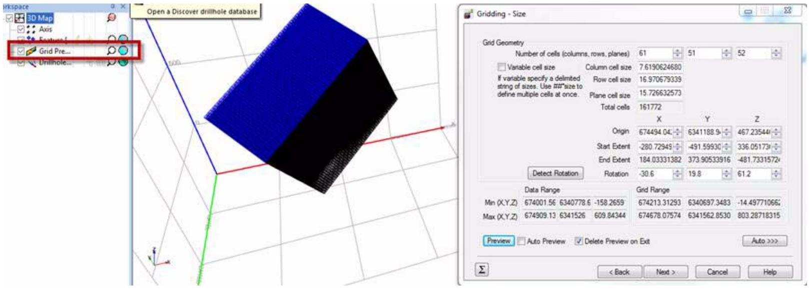
General

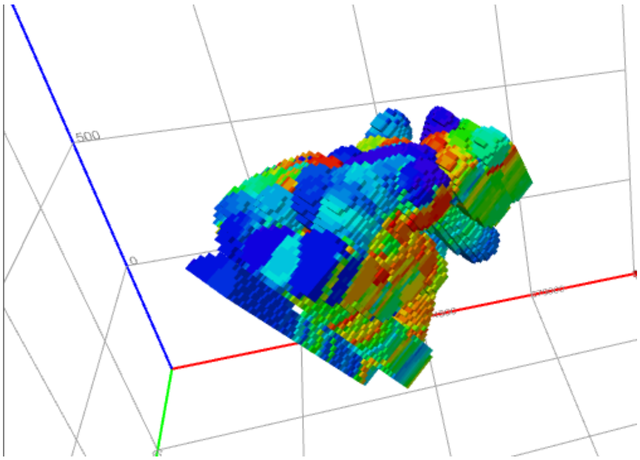
- **3D display support** – With support for 3D televisions, your project and management teams can now visualise and fully appreciate your interpretations, models and complex projects in stunning and immersive full 3D (via HDMI graphics output).



Modelling

- **New rotated voxel model support** – Interactively create, render and export rotated voxel/block models for far faster and more flexible block modelling (as well as much smaller file sizes), particularly when mineralisation is at a significant angle to the principal axial planes.





- **New 3D Batch Transform DXF tool** – Bring your Discover 3D presentations to life by easily populating the 3D environment with multiple instances of a DXF model (such as a vehicle, tree or house) using a TAB file of XYZ, dip, azimuth and scale parameters.

4 Choosing a 3D Display Type

The Discover 3D module comprises both a Discover 3D menu within the MapInfo 2D interface, as well as its own separate 3D interface/window. These allow the 3D environment to be populated both from datasets already open within 2D MapInfo/Discover interface, and/or by opening applicable data directly into the 3D window. This allows a large range of data formats to be created and supported by Discover 3D.

When choosing the data format that best suits your data, the following should be considered:

3D rendering and graphics memory

Certain formats are very efficient for displaying in 3D compared with others. For instance, multiple isosurfaces of a voxel model (e.g. grade shells) are very memory intensive and will slow navigation in 3D, whereas displaying the same isosurfaces as DXF vector files will have a negligible impact on 3D memory consumption

3D display properties available for the format

Generally the formats with more powerful visualisation options are less efficient to display in 3D. Displaying a .TAB file as 3D Points provides a myriad of display options such as colour and size modulation, labelling, etc, but is much slower to render than displaying the same dataset as a vector .DXF file (but this option has very limited display options).

Often the same dataset can be displayed in Discover 3D in a number of different formats; for example, point data in a MapInfo table can be displayed in 3D as either an image, a vector DXF file, 3D Points, 3D Symbols or a Feature Database. Understanding which of these formats can be applieed to your data can improve grapics performance in the 3D environment.

The benefits and limitations of each format supported by Discover 3D are discussed in the following topics:

- *Raster Images*
- *DXF Vector Data*
- *Points and Lines*
- *Gridded Surfaces*
- *Drillholes*

- *Voxel (Block) Models*
- *Feature Database*

Raster Images

Raster Images are a quick and simple way to display large amounts of data in 3D. The two primary image creation/registration tools are:

Capture all visible content of any MapInfo map window and display within the 3D environment as a geo-located image, using the **View Map in 3D** menu option (see *Displaying Map Window Views as 3D Images*). This can be draped over a gridded surface (e.g. topographic surface) or assigned a constant RL value.

From within Discover 3D, register an existing image (e.g. a scanned cross section) using the **Georeferencing Image File Creation Wizard** (see *Using the Georeferenced Image File Creation Wizard*).

Considerations:

- A single image can be used to display multiple datasets.
- Quick and easy to create an image from a mapper window.
- High resolution images (and therefore large file sizes) are required to preserve vector linework and labels.
- Data cannot be interrogated for attribute information; display options are therefore also limited.
- Multiple high resolution images may degrade rendering efficiency in 3D.

DXF Vector Data

The .DXF vector file format (AutoCAD) is a very efficient way of displaying vector data in 3D. This can include:

- MapInfo vector data such as points, lines and polygons displayed with the **View Objects in 3D** menu option (see *Displaying Map Objects as 3D Vectors*). For example infrastructure (roads), contour lines or orebody boundaries interpreted on drillhole cross sections (display in 3D using the *Viewing Section Layers* option).

- Extruding objects to create .DXF solids using the Extrusion Wizard (see *Extruding Models from Points, Lines and Polygons*). For example, surface traces of faults or polygons of building outlines extruded to create the objects in 3D.

- Existing DXF files (e.g. the output from modelling programs) can be opened directly into Discover 3D via a number of menu options, or by simply dragging and dropping the DXF from Windows Explorer.

- Visualising point data as 3D vector symbols (see *3D Point Symbols*).

Considerations:

- DXFs are a very efficient way of displaying a large amount of vector data, particularly points and polylines or complex bodies such as orebody models.

- Linework is honoured (without the pixelation seen in images).

- They have limited display options .

- Vector DXFs cannot be interrogated for attribute information.

Points and Lines

Point data in MapInfo tables can be displayed in Discover 3D as 3D points (see *Working with 3D Points*) or 3D lines (see *Working with 3D Lines*). These display types directly reference the source MapInfo TAB file, and therefore provide powerful display controls. These include point labelling, colour, size and rotation modulation by data values, offset controls and support for True Type fonts. Examples of these display formats:

- Colour modulating regional geochemical point datasets.

- Displaying prospect scale geochemical data as lines offset by an assay value (e.g. Cu) above a DEM surface.

- Displaying flight lines coloured by date.

Considerations:

- 3D Points and Lines provide powerful display and control options.

- Applying these options to larger datasets can severely affect 3D rendering performance.

Gridded Surfaces

Grids created using either the **Discover>Surfaces** menu or other products (e.g. topographic or geochemical surfaces in formats such as Geosoft, ER Mapper, Surfer etc.) can displayed in Discover 3D either directly or via the **View Surface in 3D** menu option (see *Displaying Surfaces in 3D*). A large range of grid types are supported. Support is also provided for the draping of ER Mapper ECW and .ALG files.

Considerations:

- Powerful display and control options.
- Multiple large grids can severely affect 3D rendering performance. Perhaps consider displaying some grids as .DXF contour lines.

Drillholes

Drillholes present in an open Discover Drillhole Project can be displayed in Discover 3D using the **View Drillholes** menu option (see *Displaying Drillhole and Trench Data in 3D*). The display of 3D drillholes can be modified by colour and thickness modulation, as well as labelling and annotation options.

Considerations:

- Powerful display and control options.
- Drillhole downhole information can be dynamically interrogated.
- Applying some of these options to large numbers of drillholes can severely affect 3D rendering performance.

Voxel (Block) Models

Discover 3D can create, display and manipulate Voxel block models (e.g. IP resistivity), supporting formats such as UBC, CEMI, and Noddy, as well as preliminary support for Datamine, Surpac, Vulcan and Gocad Voxet.

Considerations:

- Powerful display and analysis options
- Graphics memory intensive.

Feature Database

Points, polylines and polygons can be digitized and attributed directly into the 3D environment as part of a Feature Database. Additionally, dxf vector or MapInfo TAB files can be imported as feature database objects, and then edited from within the 3D environment. This allows the geoscientist to create interpretations and build complex models in the 3D environment using all relevant datasets (e.g. drillholes, geochemical and/or geophysical grids, voxel models, etc) without the being limited to traditional 2D cross-sections.

5 Viewing Mapinfo/Discover Data in 3D

The Discover 3D menu in the MapInfo/Discover interface provides the tools you need to display 2D data from a map window or MapInfo table in the Discover 3D window. The tools available from the Discover 3D menu are described in *Discover 3D Menu (MapInfo/Discover)*.

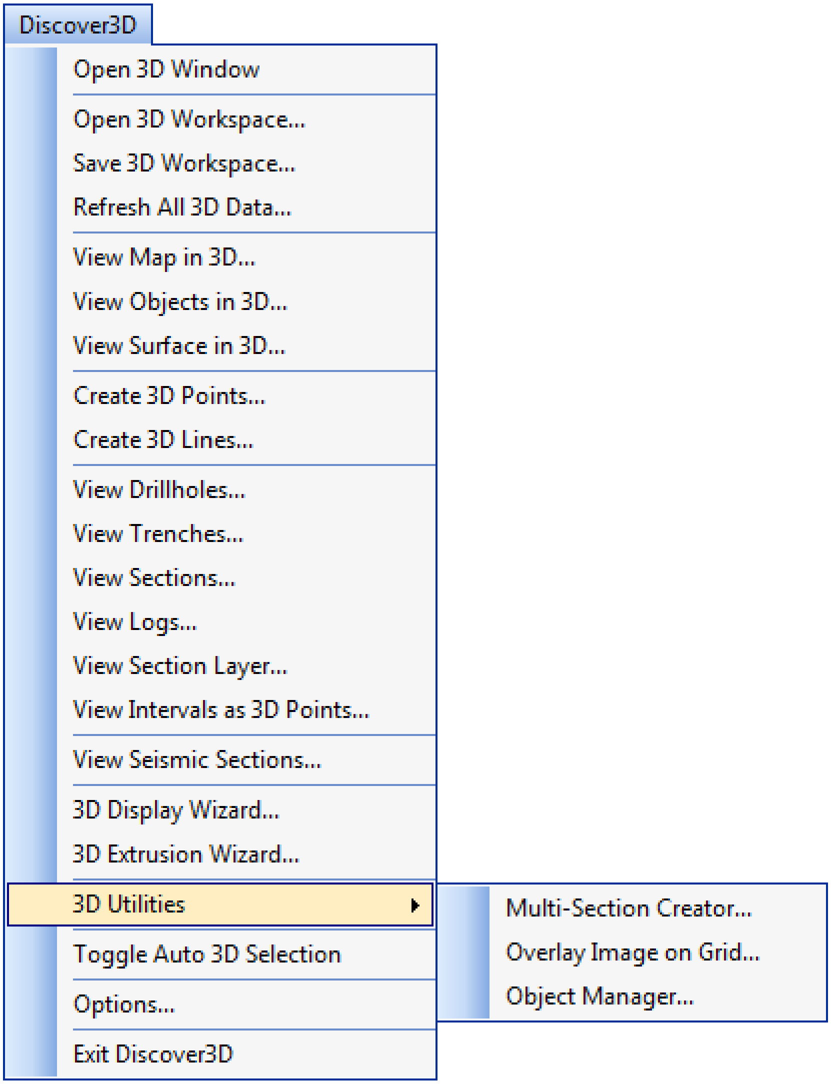
For information on how to use these tools to manage the 3D workspace and view data in 3D, see the following sections:

- *Managing the 3D Workspace*
- *Using the 3D Display Wizard*
- *Refreshing Data from MapInfo/Discover*
- *Displaying Map Window Views as 3D Images*
- *Displaying Map Objects*
- *Displaying Drillhole and Trench Data in 3D*
- *Extruding Objects in 3D*
- *Displaying Other Types of Sections*
- *Draping Images Over a Grid*
- *Adding Landscape and Ornamental Features*

Discover 3D Menu (MapInfo/Discover)

To show the Discover 3D Menu:

- In MapInfo, select **Discover>Discover 3D Menu** to add the Discover 3D menu to the MapInfo Professional menu bar.



The following tools are available from the Discover 3D menu:

- *Open 3D Window Tool*
- *Open 3D Workspace Tool*
- *Save 3D Workspace Tool*
- *Refresh All 3D Data Tool*
- *View Map in 3D Tool*

- *View Objects in 3D Tool*
- *View Surface in 3D Tool*
- *Create 3D Points Tool*
- *Create 3D Lines Tool*
- *View Drillholes Tool*
- *View Trenches Tool*
- *View Sections Tool*
- *View Logs Tool*
- *View Section Layers Tool*
- *View Intervals as 3D Points*
- *View Seismic Sections Tool*
- *3D Display Wizard*
- *3D Extrusion Wizard*
- *3D Utilities Menu*
- *Toggle Auto 3D Selection Tool*
- *Options Tool*

Open 3D Window Tool

Opens the Discover 3D Window. See *Understanding the Discover 3D Interface* for information on using the 3D window.

See also

...*Opening the 3D Window*

Open 3D Workspace Tool

Opens a saved 3D session including data in the 3D window and associated tables and map windows in MapInfo Professional. Navigate to the folder where the Discover 3D workspace was saved, and select the .D3D workspace file.

See also

... *Opening a Saved 3D Workspace*

Save 3D Workspace Tool

Saves the current workspace in both MapInfo Professional and Discover 3D. In the **Save 3D Workspace** dialog, type a name for the 3D Workspace file and browse to a folder location.

See also

... *Saving the 3D Workspace*

Refresh All 3D Data Tool

Updates the 3D Window with changes made to associated tables that are open in MapInfo/Discover (a drillhole project, for example).

See also

... *Refreshing Data from MapInfo/Discover*

View Map in 3D Tool

Displays data visible in a MapInfo map window as a georeferenced bitmap image in Discover 3D. These raster images may include surface layers such as DEM or topography grids or other raster files such as aerial photography or satellite imagery, vector data or annotation layers.

See also

... *Displaying Map Window Views as 3D Images*

View Objects in 3D Tool

Displays point, line or polygon map objects in a Discover 3D as 3D DXF vectors. A number of different options are available for assigning the Z values. The target objects can be either a selection or an entire open MapInfo table.

See also

... *Displaying Map Objects as 3D Vectors*

View Surface in 3D Tool

Displays selected grid files in Discover 3D. Only grid files that are currently open in MapInfo Professional can be displayed.

See also

...*Displaying Surfaces in 3D*

Create 3D Points Tool

Displays point data tables in Discover 3D. Point data tables require each record to have an X (Easting), Y (Northing) and Z (RL) field; a line identifier field is optional. A geochemical soil sampling program is an example of a point data file that can be viewed in Discover 3D.

See also

...*Displaying Points in 3D*

Create 3D Lines Tool

Displays line data tables in Discover 3D. Line data tables require each record to have an X (Easting), Y (Northing) and Z (RL) field along with a line identifier field. Geophysical survey line profiles are an example of a line data file that can be viewed in Discover 3D.

See also

...*Displaying Lines in 3D*

View Drillholes Tool

Displays selected drillholes from an open Discover drillhole project in Discover 3D. Only drillholes that are part of a drillhole project can be displayed. See the *Discover User Guide* for information on Discover drillhole projects.

See also

...*Viewing Drillholes in 3D*

View Trenches Tool

Trenches/Costeans in a open drillhole project can be migrated into 3D. See the *Discover User Guide* for information on Discover drillhole projects.

View Sections Tool

Displays drillhole sections created in a Discover drillhole project as georeferenced bitmap images in Discover 3D. Section images can include downhole display attributes or ore body boundary polygons along with annotation data.

See also

... *Viewing Drillhole and Trench Sections*

View Logs Tool

Displays drillhole logs created in a Discover drillhole project as georeferenced bitmap images in Discover 3D. Drillhole log images can include up to 16 columns of downhole lithological information, assay values or geophysical readings displayed as text, linegraphs or histograms.

See also

... *Viewing Drillhole Logs*

View Section Layers Tool

Displays drillhole section boundaries created in a Discover drillhole project as either 3D vectors or as 3D features in Discover 3D. Section layer files can represent lithological or ore body boundaries, or other geological interpretations.

See also

... *Viewing Section Layers*

View Intervals as 3D Points

Converts a 2D downhole drillhole dataset (such as a lithology or assay table) directly into a 3D Point dataset displayed in Discover 3D.

The resulting mappable MapInfo Professional table can also be queried (e.g. with **Select by Group** or **SQL Select**) to select a subset such as only fault intercepts or copper values above 10,000 ppm); these can then be viewed in 3D with the *Create 3D Points Tool* , for modelling/interpolation with the 3D Gridding or Surface Gridding tools.

See also

... *Viewing Drillhole Intervals as 3D Points*

View Seismic Sections Tool

Interpret velocity profiles on seismic SEGY sections in the 2D environment, then generate depth profiles from these for viewing in Discover 3D.

See also

... *Viewing Seismic Sections*

3D Display Wizard

The 3D Display Wizard provides a user-friendly step-by-step guide to displaying either the entire current map view or individual map objects in Discover 3D. If the entire map window is selected it is displayed as a georeferenced raster image, whereas map objects are displayed as a 3D vector file. The wizard displays a series of dialogs. Use the **Next** and **Previous** buttons to navigate through the dialogs and make your selections. Click **Finish** to display your selections.

See also

...*Using the 3D Display Wizard*

3D Extrusion Wizard

The 3D Extrusion Wizard allows you to extend (extrude) the shape of a 2D or 3D feature (a polygon or polyline, for example) from a base surface to a second upper or lower surface. This allows visualisation of various bodies such as fault surfaces, mine shafts and workings, vein systems and buildings in three dimensions. This tool is fully documented in .

See also

...*Extruding Objects in 3D*
...*Extruding Models from Points, Lines and Polygons*

3D Utilities Menu

- *Multi Section Creator Tool*
- *Image on Grid Creator Tool*
- *Object Manager Tool*

Multi Section Creator Tool

This tool is specifically designed for viewing cross-section images in 3D that are not part of a drillhole project. For example, geological sections which have been scanned from geological map sheets or company reports or images of vertical slices cut through a geophysical model can be displayed in the correct 3D geographical space using this utility.

The cross-ections must be in PNG, BMP or JPG image format. A MapInfo Professional table containing polylines representing the position of each cross-section in plan view is also required. Each polyline must be attributed with the cross-section image file name and the top and bottom elevations of the image.

When the **Multi Section Creator** tool is run, select the table containing the attributed cross-section polylines, and corresponding image name, top and bottom fields. An optional group field allows images to be grouped into sub-layers in the output EGB file. Then choose an output EGB file name and location (defaults to the same directory as the source polyline table) and the format of the cross-sectional images. If images of differing formats are involved, specify the image file name extension in the section name field, and select the **Section name includes extension** format option. A single EGB file can be generated for all sections, or separate EGB file for each section.

See also

... *Displaying Other Types of Sections*

Image on Grid Creator Tool

This tool creates an EGB file for an image draped over a grid, similar to the **View in 3D/View Map In 3D** options. This EGB file can then be displayed in 3D. However unlike the **View in 3D/View Map In 3D** options, the EGB file output by **Image on Grid Creator** references the input image file, rather than creating a new associated PNG file. In this way, the resolution of the image is maintained, without having to maximise the size and alter the shape of the image map window to fit the image.

See also

... *Draping Images Over a Grid*

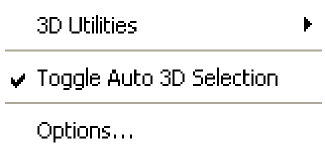
Object Manager Tool

This tool allows the 3D environment to be quickly populated with multiple image objects. Examples of use include displaying multiple images of trees and other vegetation, signposts and people in a landscape, texturing walls and simple structures (e.g. buildings) and placing signboards on buildings. This adds realism to the 3D environment and is particularly useful for creating animations.

See also

... *Adding Landscape and Ornamental Features*

Toggle Auto 3D Selection Tool



After the initial display of objects in a layer, selecting this option will automatically display subsequent map object selections from the same layer in Discover 3D without having to reselect a Z value.

Selections created in Auto 3D Selection mode are temporary and will be overwritten each time a new selection is made. To save a selection to a 3D DXF file, use **Discover 3D>Show Current Selection**.

See also

...*Selecting Map Objects to Display*

Options Tool

The Options dialog provides various controls for converting 2D data into the 3D window.

Some of the available controls include:

- Output image resolution
- Default grid compression
- Temporary directory location

See also

...*Customising 2D Interface Settings*

Managing the 3D Workspace

- *Opening the 3D Window*
- *Opening a Saved 3D Workspace*
- *Saving the 3D Workspace*

Opening the 3D Window

From the MapInfo menu bar, on the **Discover 3D** menu, click **Open 3D Window** to open the Discover 3D Window.

Note Most Discover 3D menu tools will open the Discover 3D window if it is not already open. However, for very large datasets, this may cause memory problems. It is recommended to open the Discover 3D window with the **Open 3D Window** tool before loading large datasets with any of the other Discover 3D menu options.

Opening a Saved 3D Workspace

From the MapInfo menu bar, on the **Discover 3D** menu, click **Open 3D Workspace** to open a previously saved 3D workspace. Navigate to the folder where the Discover 3D workspace was saved, and select the .D3D workspace file. This opens data in the 3D window and associated tables and map windows in MapInfo Professional. For more information about the files saved in the 3D workspace, see *Saving the 3D Workspace.*

Saving the 3D Workspace

From the MapInfo menu bar, on the **Discover 3D** menu, click **Save 3D Workspace** to save the current workspace in both MapInfo Professional and Discover 3D. In the **Save 3D Workspace** dialog, type a name for the 3D Workspace file and browse to a folder location.

When a 3D Workspace is saved, a number of files are created in the designated folder:

WOR file

The .WOR file is a standard MapInfo Professional workspace file. The workspace lists all the tables open in the current MapInfo Professional session and their associated file paths, the map window dimensions, layer order, graphical displays, labelling and any printer information. The workspace created when saving a Discover 3D Workspace can be opened using MapInfo Professional **File>Open Workspace**.

EGS file

For more details on EGS Discover 3D session files, see *Discover 3D Session Files*.

D3D file

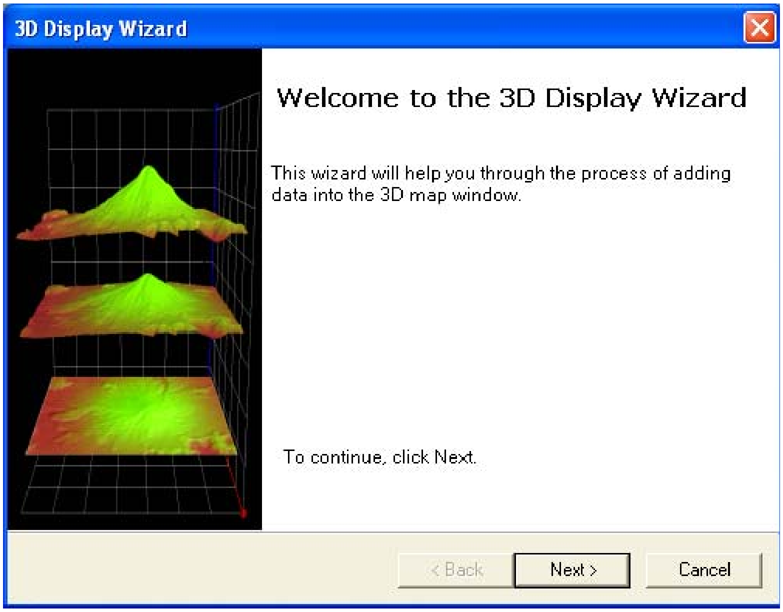
The .D3D file is a text file that lists the pathways to the associated .WOR and .EGS files. The .D3D file is opened through the **Discover 3D>** **Open 3D Workspace** menu option and results in the saved work session being opened up in both MapInfo Professional and Discover 3D. To edit a .D3D file simply modify the tables and views in either MapInfo Professional or Discover 3D and re-save the 3D Workspace.

See also

...*Opening a Saved 3D Workspace*

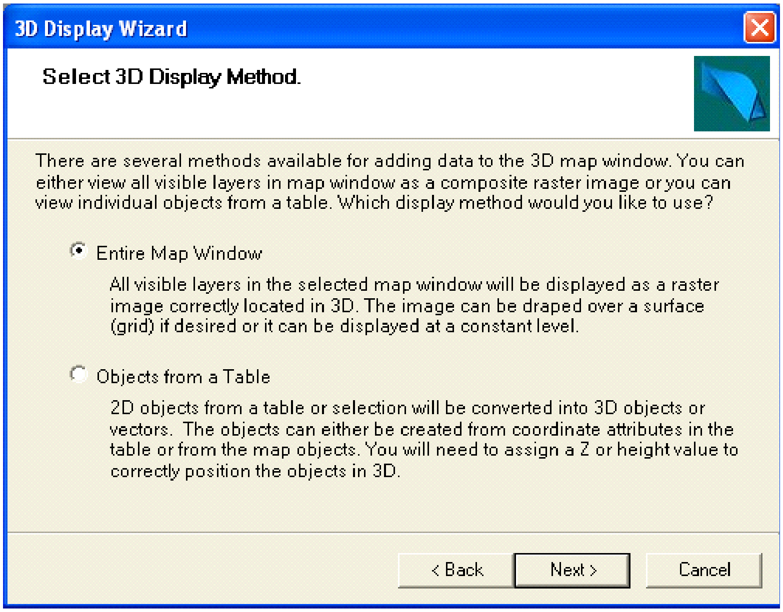
Using the 3D Display Wizard

The **3D Display Wizard** provides a user-friendly step-by-step guide to displaying either the entire current map view or individual map objects in Discover 3D. If the entire map window is selected it is displayed as a georeferenced raster image; map objects are displayed as a 3D Vector file. The menu option **Discover 3D>3D Display Wizard** opens the **3D Display Wizard**.



3D Display Wizard

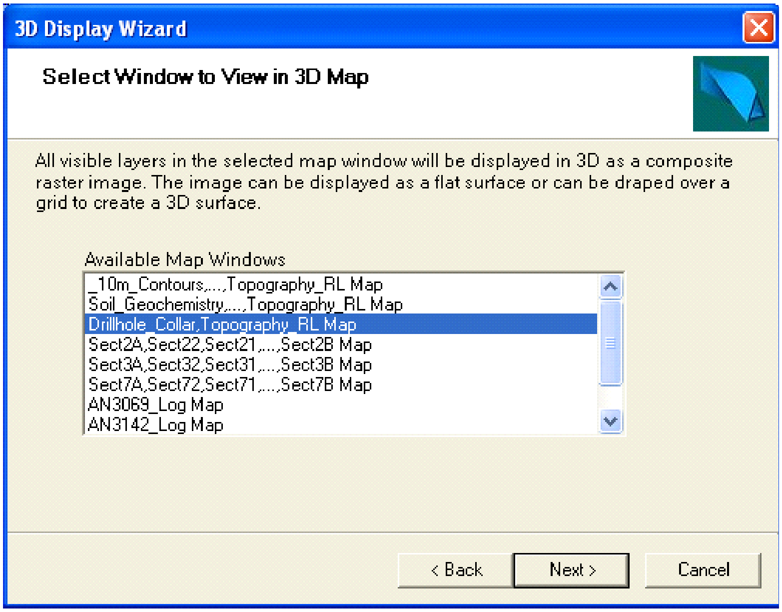
The wizard displays a series of screens with **Next** and **Back** buttons to progress through the various choices. Use these buttons to navigate forward and backwards through the entry screens if modifications need to be made to initial selections.



Initial 3D Display Wizard screen.

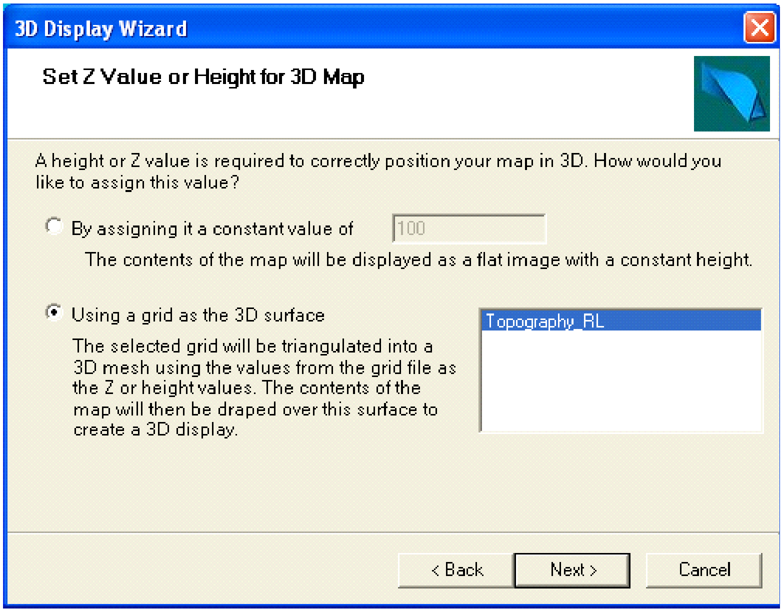
Select the type of data to view in 3D. To display an entire map window as a raster mapper image (**Georeferenced Bitmap Image**) choose *Entire Map Window*. To display individual map objects as 3D vectors select *Objects from a Table*. Click **Next** to continue.

Entire Map Window



Select map window to display in 3D

If the **Entire Map Window** 3D data view is selected then a list of available map windows for transfer to 3D is displayed. Only one mapper can be selected at a time. Click **Next** to continue.



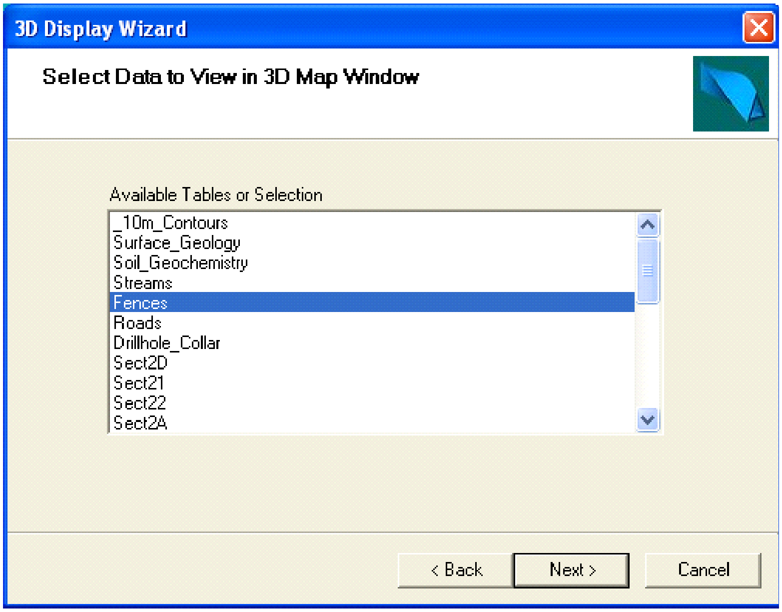
Select the Z value or height

In order to view the mapper in the 3D window it must be assigned a Z value or height. To display the mapper image as a flat image at a constant height select the **By assigning it a constant value** option and enter a value.

Alternatively, the mapper image Z value or height may be derived from a gridded surface such as a Digital Terrain Model (DTM) or other gridded surface which covers the mapper image extents. The gridded surface must be open in MapInfo Professional to be available for selection. Choose **Using a Grid as the 3D surface** and highlight the gridded surface in the available surface window. The mapper image displays in the Discover 3D map window as a variable Z value or height image which appears to be “draped” over the gridded surface.

Objects from a Table

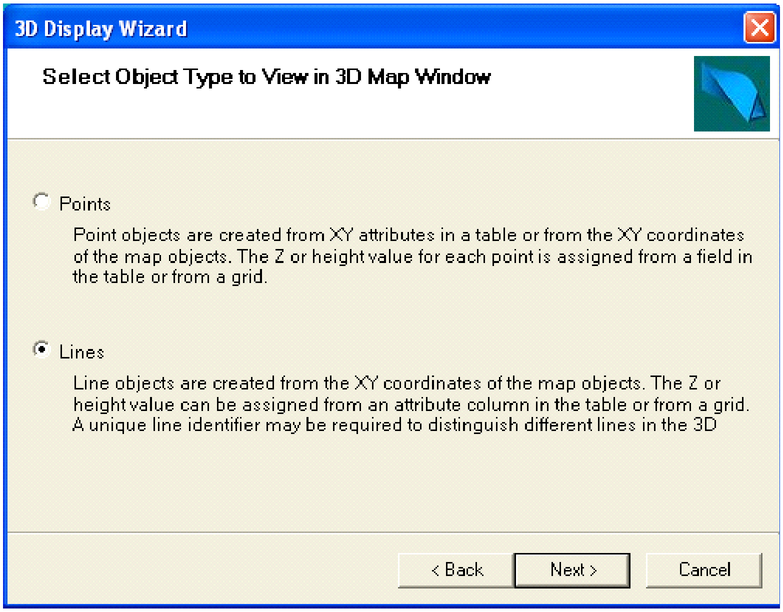
If the **Objects from a Table** 3D data view is selected the next dialog containing a list of the currently open tables is displayed.



Select Data to View dialog

Choose the table that contains the data objects to display as vector objects in the Discover 3D map window. To display only a subset of data from a table use any of the standard MapInfo Professional or Discover Selection or Query tools and choose *Selection* from the list of available tables.

Once the table selection has been made the **Select Object Type** dialog is displayed:



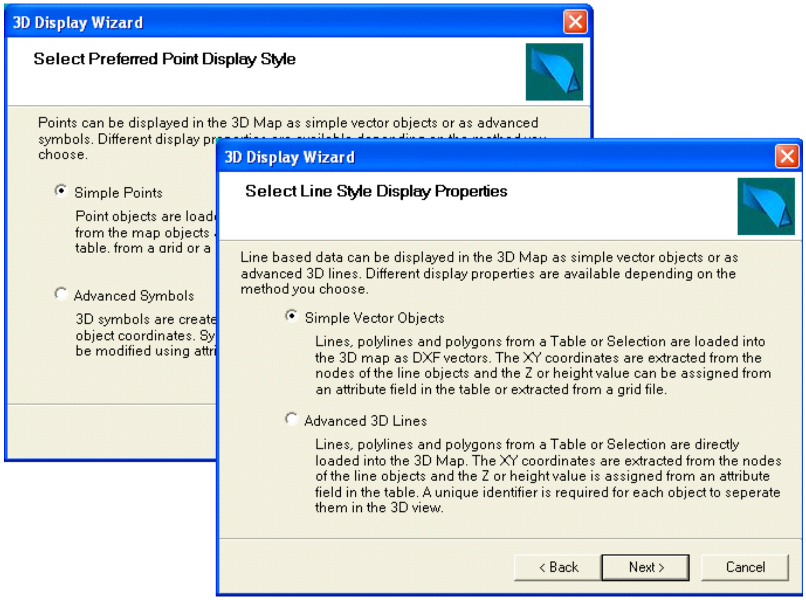
Select Object Type dialog

The **Select Object Type** dialog contains two options to display data objects in the Discover 3D map window. Data objects may be displayed as one of the following:

- **Point Data** – Individual objects or data locations are displayed in Discover 3D as isolated points. The points can have their size, shape, and colour modified based on the data in a selected data table.

- **Line Data** – Data reading points can be joined by a line. The colour and thickness of the line can be controlled depending on the data in a selected data table.

Select the appropriate **Object Type** and click on the **Next** button to continue.

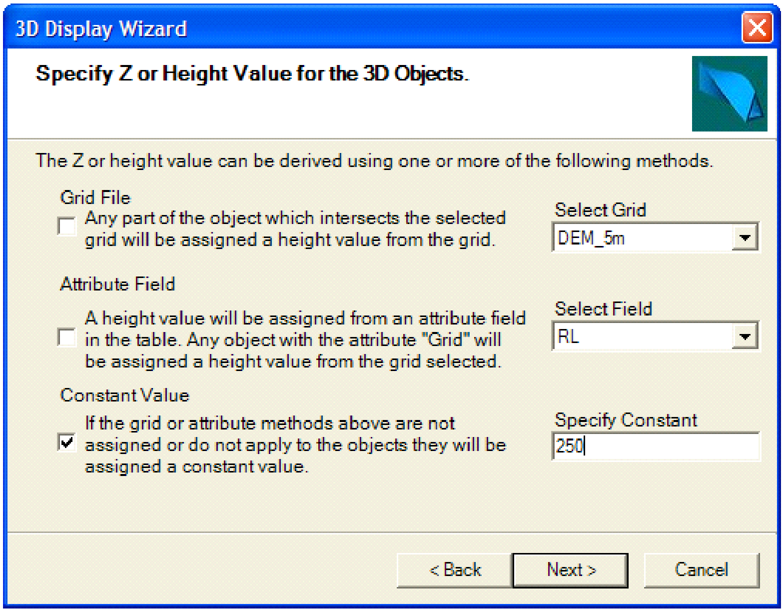


Select Point and Line Style property dialogs

Regardless of which **Object Type** is selected, the following **Select Style** property dialogs allow the choice of the method for determining the Z value or height for the data objects from the following options:

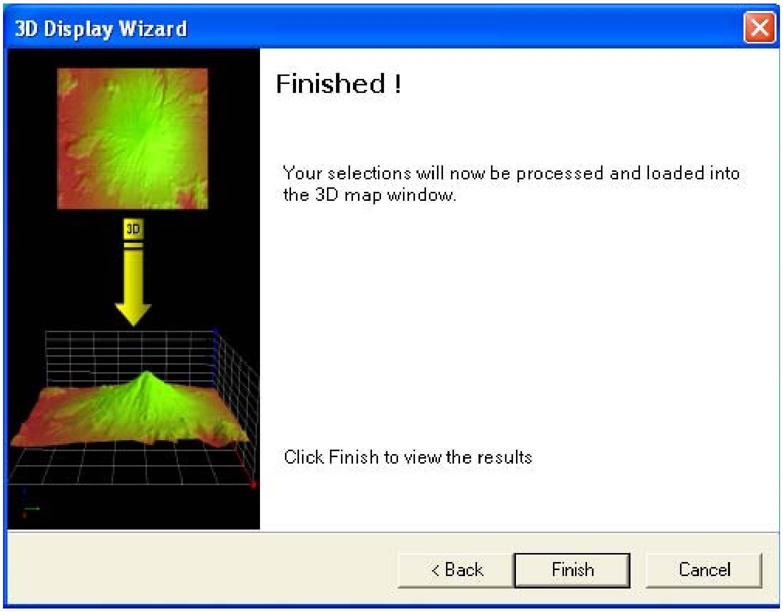
Simple – The Z value or height for the data objects may be derived from a gridded surface, a column in the data table or from a specified constant value.

Clicking **Next** with this option selected the **Specify Z or Height Value options** dialog will open. This dialog allows selection of the appropriate methods for determining the Z value from those above. The Z value or height may also be derived from a combination these options. i.e. data objects that fall within a gridded surface will be assigned a Z value or height from the grid, data objects outside of a gridded surface will be assigned values from a column in the table otherwise any data objects which do not meet any of the above criteria will be assigned a specified constant Z value or height value.



Simple method Z value or height options

Advanced – The data objects using this option use a height determined by a data value or the location (Z value) of the objects.



Final screen of the 3D Display Wizard

The final screen of the wizard is displayed before the mapper raster image or data objects are transferred to Discover 3D. If you wish to change any of the selections made in the wizard select the **Back** button. To proceed with the 3D display operation, click the **Finish** button.

If there is no Discover 3D map window currently open a new 3D map window will be opened with the mapper raster image or data objects displayed. Otherwise the 3D data will be added to the current 3D map window display.

Refreshing Data from MapInfo/Discover

The **Discover 3D>Refresh All 3D Data** tool updates the 3D Window with changes made to datasets tables that are open in both Mapinfo/Discover and Discover3D (this includes Points, Lines, Drillholes, Trenchs in the 3D Window). Alternatively, you can use **Discover 3D>Create 3D Points** or **Discover 3D>Create 3D Lines** and **Discover 3D>View Drillholes** to reload the currently opened dataset in the 3D Window.

Displaying Map Window Views as 3D Images

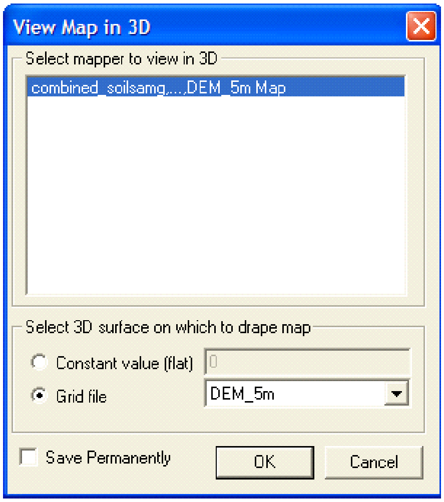
Datasets visible in a MapInfo mapper window views can be viewed in Discover 3D as georeferenced bitmap images. These raster images may include surface layers such as DEM or topography grids or other raster files such as aerial photography or satellite imagery, vector data or annotation layers.

To visualise data in 3D as a raster image:

1. Open all the tables to be displayed into one map window in MapInfo Professional. Re-order the layers as necessary (using the MapInfo Professional Layer Control) and apply annotations as desired.

2. Ensure that the mapper window view encompasses the area of interest (dimensions and scale).

3. Either right-click in the mapper window, and from the shortcut menu, choose **View in 3D** or select **Discover 3D>View Map in 3D** menu option. The dialog appears:



View Map in 3D dialog

4. Select the map window from the list in Select **mapper to view in 3D**. If the **View in 3D** menu option is selected, the map window from which this was activated is automatically selected as the map window and shows only the **3D surface** controls.

5. Two options are available to control the Z value of the 3D image:

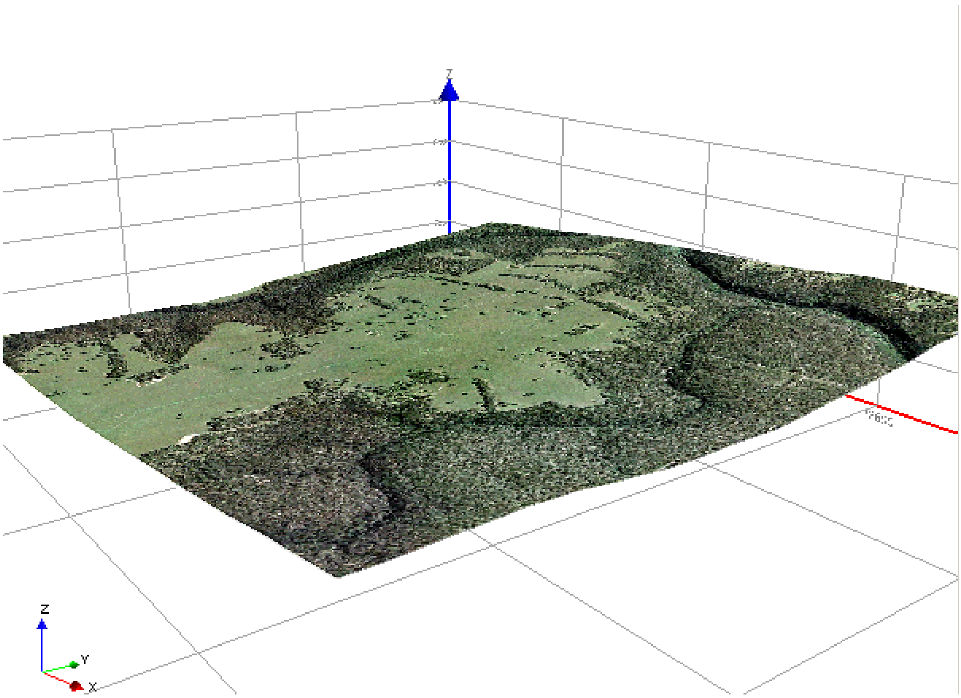
   - To display the entire image at a fixed elevation, check the **Constant value (flat)** control and enter a specific height.

   - Alternatively, the image can be 'draped' over a grid file such as an elevation grid. Check the **Grid file** control and select the appropriate grid file from the pull-down list. Only grid files that are currently open in MapInfo Professional will be listed.

6. Enabling the **Permanently Save** tick box at the bottom of the dialog will open the **Save 3D Information** as dialog upon pressing **OK**. This allows the output (a georeferenced image: see below) to be named and saved to a selected directory. Pressing **OK** without enabling this option will create an georeferenced image using the map window's name, and save this into the Discover 3D Temporary directory (set under **Discover 3D>Options**). See *Customising 2D Interface Settings*; these temporary files will be deleted on exiting 3D if the **Remove temporary files on exit** option is enabled).

The map window is now displayed in a 3D map window as a georeferenced bitmap image (if the Discover 3D window was not already open, it will automatically open prior to displaying the image). A georeferenced bitmap image is made up of two files:

- The first file is a raster image file of the map window with a .PNG (Portable Network Graphics) extension.
- The second file is a text header file with an .EGB (Encom Georeferenced Bitmap) extension. This header file stores information relating to image files such as the image name, format and projection details.



A map window displayed in Discover 3D draped over a topographic surface

To modify 3D image parameters such as transparency, scaling or offset refer to the *Changing Image Display Properties* section of the **Raster Images** chapter.

Maximising Image Resolution

The quality of the output 3D georeferenced image can be improved via:

- Utilising a larger MapInfo mapper window (but do not maximise it).
- Ensuring that your MapInfo mapper window is zoomed to the extents of the required data.

- Under **Discover 3D>Options**, increase the **Create Raster Images at …Screen Resolution** factor set in the **Display** tab of the dialog. This will increase the output image resolution. However it will also result in a larger output PNG image file size; displaying multiple high resolution images within Discover 3D may reduce the redraw speed and efficiency.

- If your input dataset in the map window is already an image (e.g. an air photo), and it will be draped over a grid (and no other data is involved), perhaps use the use the *Image on Grid Creator Tool* utility. This will create a .EGB header file referencing directly to the source image, thus preserving the source image's resolution.

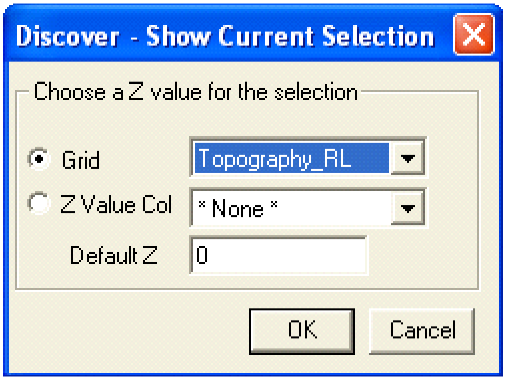
Displaying Map Objects

- *Selecting Map Objects to Display*
- *Displaying Map Objects as 3D Vectors*
- *Displaying Surfaces in 3D*
- *Displaying Points in 3D*
- *Displaying Lines in 3D*

Selecting Map Objects to Display

If a number of map object selections are to be viewed consecutively in Discover 3D, the **Auto 3D Selection** mode can be activated using the **Discover 3D>Toggle Auto 3D Selection** menu option. A tick next to this menu option denotes **Auto 3D Selection** mode has been activated.

This option will enable the **Discover - Show Current Selection** dialog to be displayed after the first map object selection from a layer is made and any subsequent map object selections from the same layer will automatically be displayed in Discover 3D without having to select a Z value option each time.



If a subsequent selection involves map objects from a different layer, the **Discover - Show Current Selection** dialog will displayed after the first selection to select the appropriate Z value option and then each subsequent selection in the new layer will be automatically displayed in Discover 3D.

Selections created in Auto 3D Selection mode are displayed as **Temporary** selections (Selection.dxf) in Discover 3D and will be overwritten each time a new selection is made. To save a **Permanent** Selection to a 3D DXF file use the **Discover 3D>Show Current Selection** menu option.

To de-activate **Auto 3D Selection** select the **Discover 3D>Toggle Auto 3D Selection** menu option again.

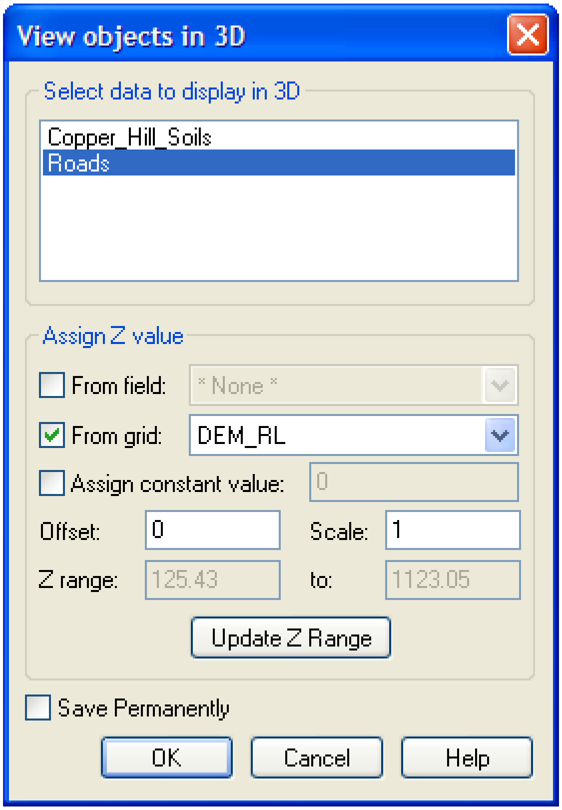
Displaying Map Objects as 3D Vectors

The **Discover 3D>View Objects in 3D** menu option enables point, line or polygon map objects to be displayed in a Discover 3D as 3D Vector files, with a number of different options available for assigning the Z values. The target objects can be either a selection or an entire open MapInfo table.

To display MapInfo data as vector objects in 3D:

1. If the input data is to be a selection, select the objects using any of the available MapInfo Professional or Discover selection tool

2. Select the **Discover 3D>View Objects in 3D** menu option. The **View Objects in 3D** dialog is displayed:



View Objects in 3D dialog

3. **Select data to display in 3D** from the list of layers available: either the *Selection* layer or a table.

4. **Assign the Z value** for the selected data by either:

    - Selecting the from **Field** to use for Z values in the pull-down list.

    - Selecting a **Surface grid** from Grid pull-down list. In order for a surface grid to be available for selection, the grid must be open in a map window.

    - Manually entering a **Constant Z value**.

If the Z value for a selection is to be taken from a column or a constant value, make sure that a surface grid has not already been selected from the pull-down list as a grid Z value entry will automatically override any other Z value options.

Note If multiple Z options are selected, they are processed sequentially. Thus if all options are ticked, the utility will first check for a Z value in the assigned field. If no value exists, it will then check the selected grid for a value. If no grid value exists, it will then use the specified constant value.

5. **Offset** and **Scale** parameters can also be specified for the 3D display. The **Update Z Range** button allows the Z range display for the selected data to be updated.

6. Enabling the **Save Permanently** option at the bottom of the dialog will open the **Save 3D DXF** as dialog upon pressing **OK**. This allows the output DXF vector file to be named and saved to a specified directory. This is of particular use when the selection layer is chosen for input; if the Save option is not enabled, the output DXF is named using the consecutive Query(n) (eg. Query1.dxf) from MapInfo Professional and is stored in the nominated Discover 3D temporary folder (set under **Discover 3D>Options**. See *Customising 2D Interface Settings*; these temporary files will be deleted on exiting 3D if the **Remove temporary files on exit** option is enabled).

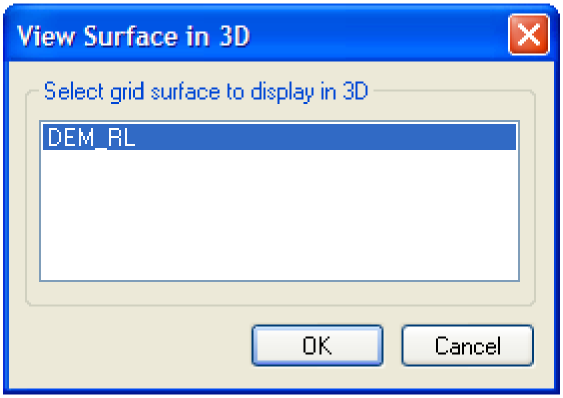
7. Pressing **OK** without enabling the Save option will create a DXF based on the input file name and save this into the same directory as the input file. The selected map objects are displayed in the Discover 3D map window in a new Vector branch.

To modify 3D Vector parameters such as transparency, scaling or offset refer to the *Changing 3D Vector Display Properties* section of the **Vector Objects in 3D** chapter.

Note Another way to generate 3D vector files from Mapinfo is using the 3D Extrusion Wizard. See *Extruding Models from Points, Lines and Polygons*. Also, the **Discover>Import or Export>Vector Export** provides a Z field option for 3D formats.

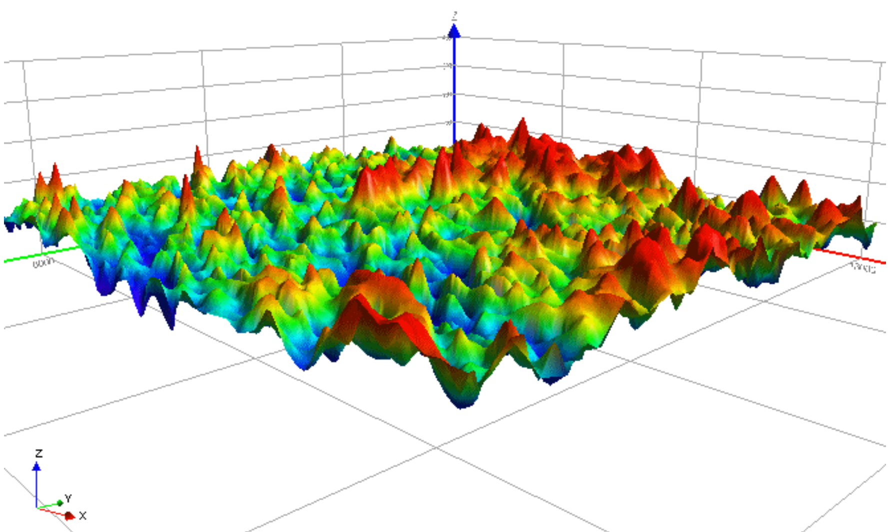
Displaying Surfaces in 3D

The **Discover 3D>View Surface in 3D** menu option allows the display of selected grid files in 3D. This feature will only allow display of grid files that are currently open within MapInfo Professional. Selecting this option opens the **View Surface in 3D** dialog:



View Surface in 3D dialog

Select the desired surface, and press **OK** to display in 3D.



3D gridded surface display

See the *Changing Surface Display Properties* section of the **Gridded Surfaces in 3D** chapter for further information of modifying and controlling surfaces in 3D.

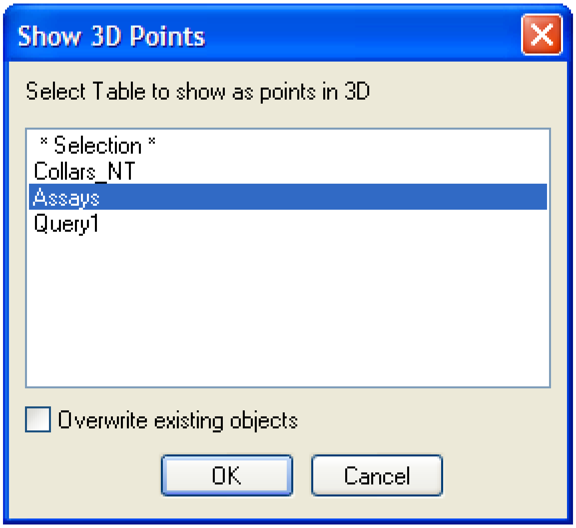
Displaying Points in 3D

Point data tables require each record to have an X (Easting), Y (Northing) and Z (RL) field; a line identifier field is optional. A geochemical soil sampling program is an example of a point data file that can be viewed in Discover 3D.

To view point data as 3D Points:

1. Open a point data table into a mapper window. To view only a subset of the point data table make a selection in the map window.

2. Select **Discover 3D>Create 3D Points**. The Discover **Show 3D Points** dialog is displayed:



Discover – Show 3D Points dialog

3. Select the **point data table** or **Selection** from the list of open tables. Select **Overwrite existing objects** to reset the current field mappings for the X, Y, Z and Line fields in Disover3D.

4. Click **OK** to load the data into Discover 3D. The point data table is displayed in the 3D map window. A dialog is displayed prompting for the X, Y, Z and optionally a line field.

   If a map window selection is used, for future reference a copy of a point data selection is stored in the same folder as the original point table and given a file name such as ~MAP0330.TAB.

To modify point symbol type, colour, size, rotation and labelling refer to *Changing Point Display Properties*.

Note When a point table is opened in Discover 3D for the first time, an .EHF (Encom Header File) is added to the directory where the table is located. This file contains the X, Y and Z fields to use in the 3D display.

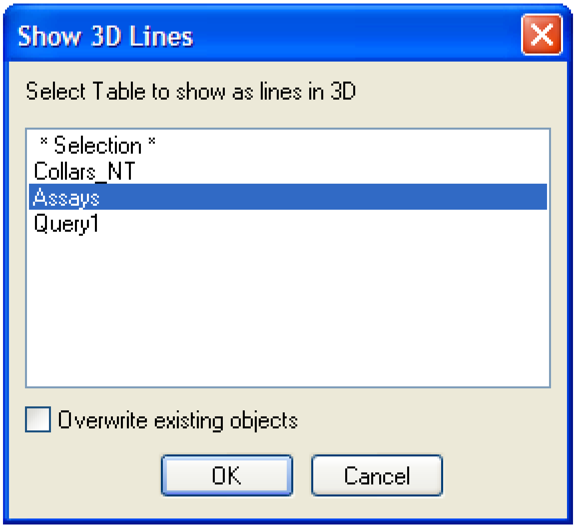
Displaying Lines in 3D

In Discover 3D, line data generally refers to data that has been collected systematically along linear traverses with samples or measurements taken at discrete intervals. The data format is exactly the same as a point data table except whereas point data is displayed as individual points in Discover 3D, line data is represented by a single linear feature for each sample line. Line data tables require each record to have an X (Easting), Y (Northing) and Z (RL) field along with a line identifier field. Geophysical survey line profiles are an example of a line data file that can be viewed in Discover 3D.

To view point data as 3D Lines:

1. Open a line data table into a map window. To view only a subset of the line data table make a selection in the map window.

2. Select **Discover 3D>Create 3D Lines**. . The Discover **Show 3D Lines** dialog is displayed:



Discover – Show 3D Lines dialog

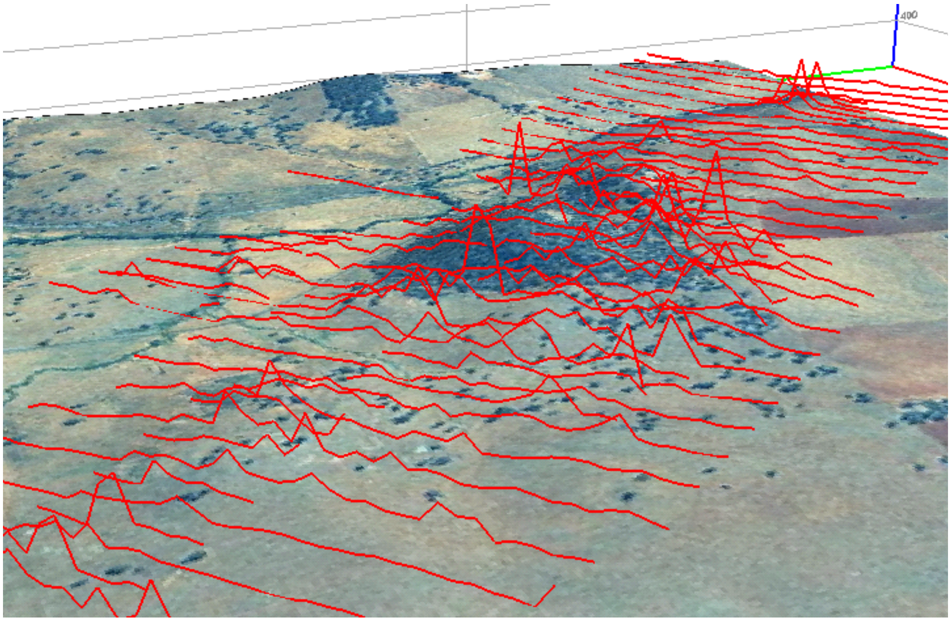
3. Select the line data table or *Selection* from the list of open tables. Select **Overwrite existing objects** to reset the current field mappings for the X, Y, Z and Line fields in Disover3D

4. Click **OK** to load the data into Discover 3D. The line data table is displayed in the 3D map window. A dialog is displayed prompting for the X, Y, Z and optionally a line field.

If a map window selection is used, for future reference a copy of a line data selection is stored in the same folder as the original lien table and given a file name such as ~MAP0386.TAB.

Note When a line table is opened in Discover 3D for the first time, an EHF (Encom Header File) is added to the directory where the table is located. This file contains the X, Y and Z fields to use in the 3D display.

To modify line parameters such as colour modulation, labelling, displaying reading or sample locations, applying fill colours above or below a nominated threshold or removing lines from the 3D map view, refer to *Changing Line Display Properties*.



Offset line data using the copper data channel

Displaying Drillhole and Trench Data in 3D

- *Viewing Drillholes in 3D*
- *Viewing Drillhole Intervals as 3D Points*
- *Viewing Drillhole and Trench Sections*
- *Viewing Drillhole Logs*
- *Viewing Section Layers*
- *Viewing Trenches*

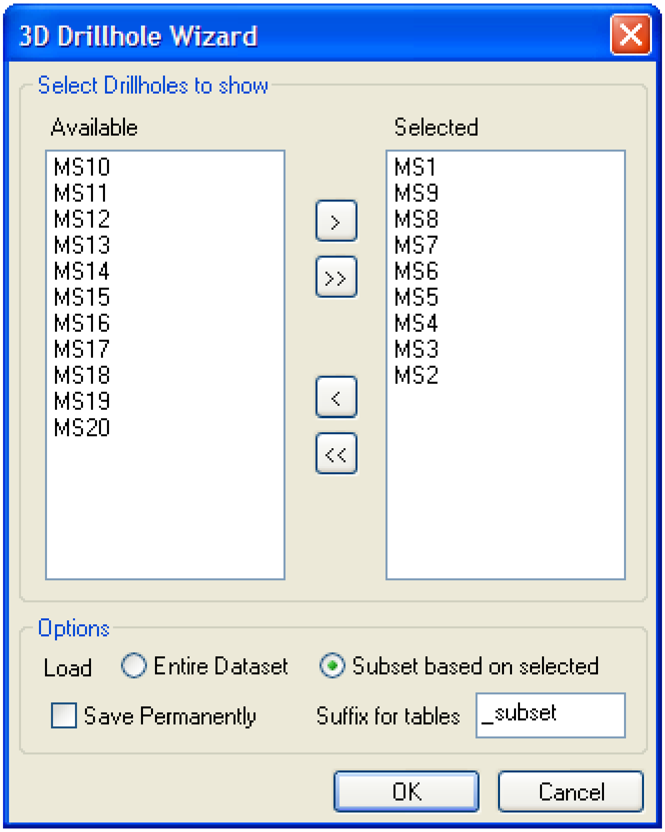
Viewing Drillholes in 3D

In order to view drillholes in Discover 3D from the Discover/MapInfo interface the drillholes must be part of a Drillhole Project. See the *Discover User Guide* for information on Creating a Drillhole Project.

Open a drillhole project using the **Drillholes>Project Manager** dialog and select the appropriate drillhole project from the pull-down list. Once the drillhole project is open, the drillholes to display in Discover 3D may be selected from a list of available drillholes or from a map window selection.

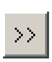
To view drillholes in 3D:

1. Select **Discover 3D>View Drillholes**. The **3D Drillhole Wizard** dialog appears:

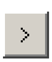


The Drillhole Wizard dialog with drillhole selection to be created as a new subset

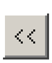
2. The **Available** box contains a list of all the drillholes in the collar table. Any drillholes listed in the **Selected** box will be automatically displayed in Discover 3D.



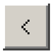
Use the **Select All** button to copy all the drillholes in the list to the **Selected** box.



Use the **Select** button to select individual drillholes. Hold down the left-mouse button and drag or hold down the SHIFT key to select consecutive drillholes.



Use the **Deselect All** button to remove **All** drillholes from the **Selected** box list.



Use the **Deselect** button to remove individual drillholes. Hold down the left-mouse button and drag or hold down the SHIFT key to select consecutive drillholes.

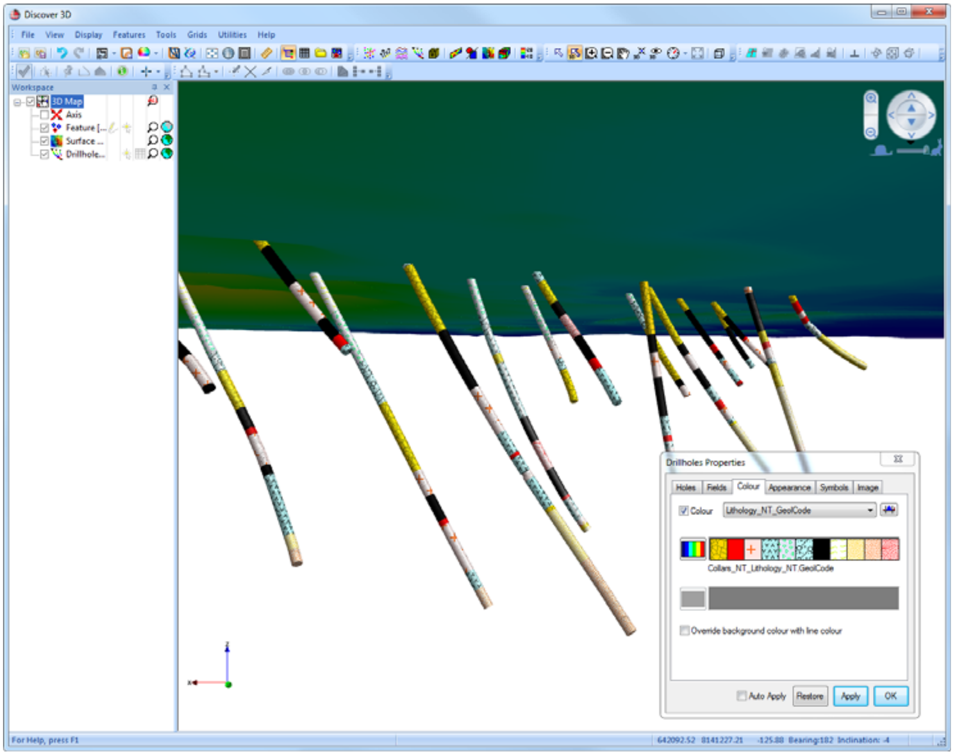
Alternatively, select the desired drillholes from a collar browser or map window and then select the **Discover 3D>View Drillholes** menu option. The selected drillholes are automatically listed in the **Selected** box.

3. The **Options** section at the bottom of this dialog controls the size of the drillhole dataset imported into Discover 3D (as opposed to what is displayed in 3D). This is of particular use when dealing with very large drillhole projects comprising thousands of holes. By default the entire drillhole project is imported into 3D, but for large datasets this may slow down rendering and navigation in 3D.

   Ticking the **Subset based on selection** option will import (and display) into 3D only the **Selected holes**. The relevant file subsets (collars, assays, etc) will be created with the selected **Suffix for tables** applied. Choosing the **Save Permanently** option will enable the data subsets to be saved to a specified directory, otherwise these files will be saved to the Discover 3D temporary folder (set under **Discover 3D>Options**. See *Customising 2D Interface Settings*; these temporary files will be deleted on exiting 3D if the **Remove temporary files on exit** option is enabled).

4. Click **OK** and view the selected drillholes in Discover 3D. The selected drillholes are displayed in the 3D map window. To display downhole attributes, colour patterns, labels, etc. refer to the *Changing Drillhole Display Properties* section of the **Drillholes in 3D** chapter. An example of a 3D drillhole display is shown below.

Note If the drillhole project or a subset is already open in 3D, a prompt to replace the data will be displayed. Accepting this will automatically reapply any existing 3D display settings (e.g. colour or thickness modulation) to the new data.



Display of drillholes beneath the air photo with lithology modulation

Viewing Drillhole Intervals as 3D Points

The **View Intervals as 3D Points** tool converts a 2D drillhole downhole table (such as a lithology or assay table) directly into a 3D Point dataset displayed in Discover 3D, using either the downhole table interval top, middle or bottom locations.

The resulting mappable MapInfo Professional table can also be queried (e.g. with **Select by Group** or **SQL Select**) to select a subset such as only fault intercepts or copper values above 10,000 ppm). These can then be viewed in 3D with the *Create 3D Points Tool*), for more advanced modelling including:

- Precise digitisation targets by snapping to the 3D Point downhole subset.
- Block model interpolation 3D Point downhole subset with the 3D Gridding tool (see *Interpolating a Gridded Surface from Digitized Drillhole Intercepts*).

- Surface interpolation (e.g. unconformity, water table, fault plane) of an appropriate 3D Point downhole subset using the **3D Solid Genertator** tool (see *Wireframing Models from Polylines, Polygons and Surfaces*).

Mapping downhole data in 3D:

1. Ensure a drillhole project is open in the 2D environment (**Drillholes>Project Manager**). It is also generally useful if the drillhole traces are already displayed in 3D (*View Drillholes Tool*).

2. Select **Discover 3D>View Intervals as 3D Points**.

3. Select the target Downhole Data table from the top pull down list.

4. Select the interval position on each downhole sample interval to calculate 3D coordinates, either:

    - Top

    - Midpoint (the midpoint of a straight line between the top and bottom positions; this should be used cautiously for longer intervals such as when logging by stratigraphic unit)

    - Bottom

5. The output file is automatically placed into the Drillhole Projects root directory, and includes a suffix indicating which interval position was selected in step 4.

6. By default the resulting mappable MapInfo table will be automatically displayed in 3D as 3D points with the **View in 3D** option (bottom left of dialog)enabled. If you wish to view a subset of the downhole table in 3D (see below) rather than the complete table, disable this option .

7. Press OK

8. The new table will be auto-opened into MapInfo Professional, with the following components:

    - XYZ coordinate fields will be created and populated within the table for each intervals:

      Top (FromX/FromY/FromZ)

      Midpoint (MidX/MidY/MidZ) and

      Bottom (ToX/ToY/ToZ) locations

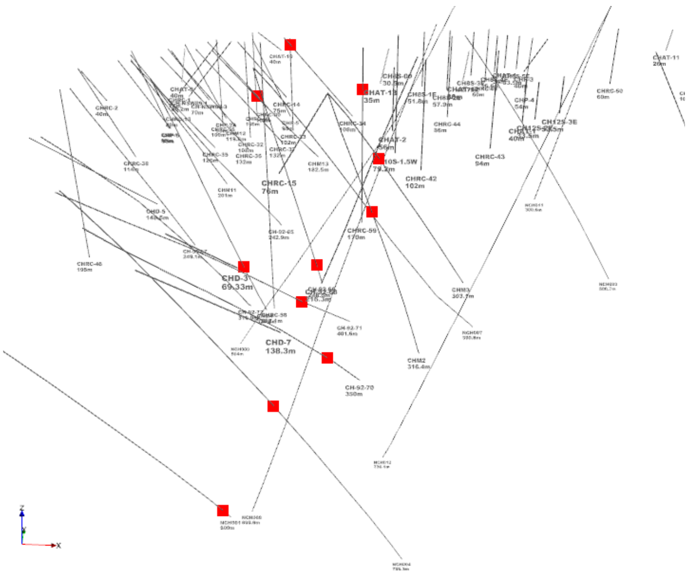
- The table will be automatically mapped on the selected interval positons coordinates
- Both the populated coordinates and the table mapping utilise the drillhole collar table's projection

9. If the View in 3D option was checked, the dataset will also load into Discover 3D, using the selected interval position's XYZ coordinates.

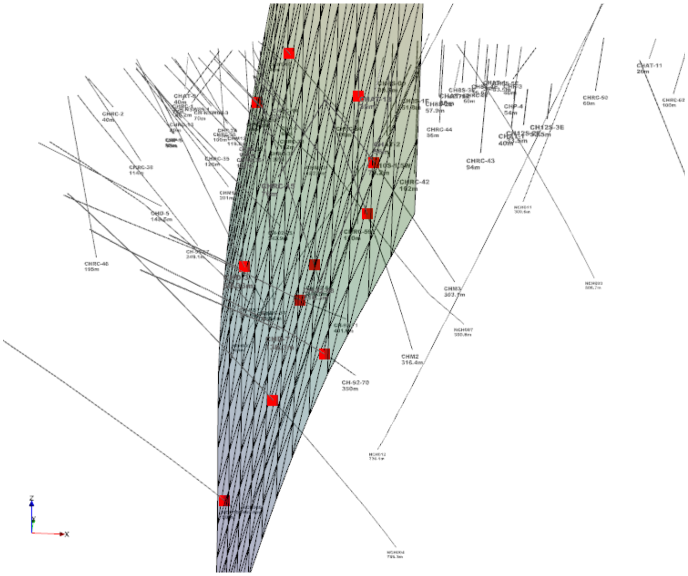
Displaying a subset of downhole data in 3D:

(follows on from previous procedure)

10. Perform a query on the mappable table created with the previous procedure (not the source downhole data table). The following query tools are recommended:
    - **Discover>Data Utilities>Select by Group** tool, particularly for geological/lithological selections (e.g. to display all occurrences of limestone).
    - **Query>SQL Select** tool, particularly for selections involving numeric data (eg Au assays > 50 ppm, or quartz vein intensity > 25%).
11. Run **Discover 3D>Create 3D Points**, and select the query generated in step 10. Ensure the Z field used (FromZ/MidZ/ToZ) is the same as selected in step 4 (inherent in the target table name)



Display of a drillhole data subset (fault intercepts) in 3D using View Intervals as 3D Points



Applying the Surface Gridding tool in 3D to interpolate the same 3D Point dataset as a gridded surface.

Note Drillhole selections can also be displayed dynamically in the 3D environment by filtering the drillhole dataset using the *Field Data Conditioning Tool*) in tandem with *Colour Modulation* or *Thickness Modulation*.

Viewing Drillhole and Trench Sections

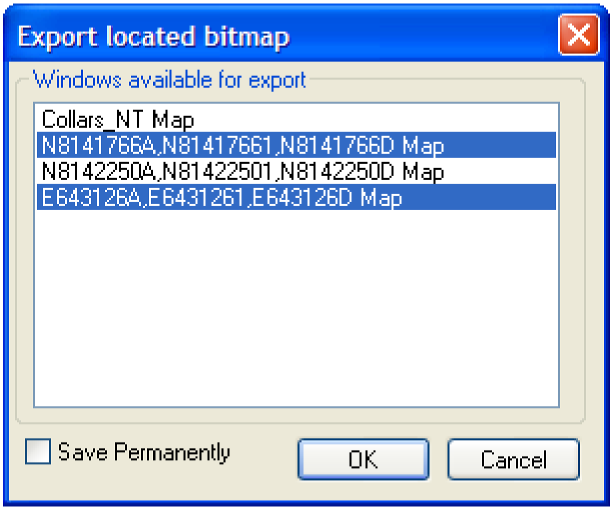
Drillhole and trench sections created in Discover can be viewed in Discover 3D as georeferenced bitmap images. These section images may include downhole display attributes or ore body boundary polygons along with annotation data. See the *Discover User Guide* for information on creating drillhole and trench sections.

Create a drillhole or trench section in Discover using the **Drillholes>Define New Section or Plan** menu option. Or you can open an existing section into a map window via the **Drillholes>Section Manager** menu option. Alternatively, open all of the associated section files into a map window using the **File>Open** command or from a previously saved workspace.

Note This option takes a screenshot image of the section map window. To view the section as 3D vectors, see *Viewing Section Layers*.

To view a drillhole or trench section as a georeferenced bitmap image in Discover 3D:

1. Either right-mouse click in the drillhole or trench section map window and choose **View in 3D** from the shortcut menu or select **Discover 3D>View Sections**. The **Discover – Export Located Bitmap** dialog appears:

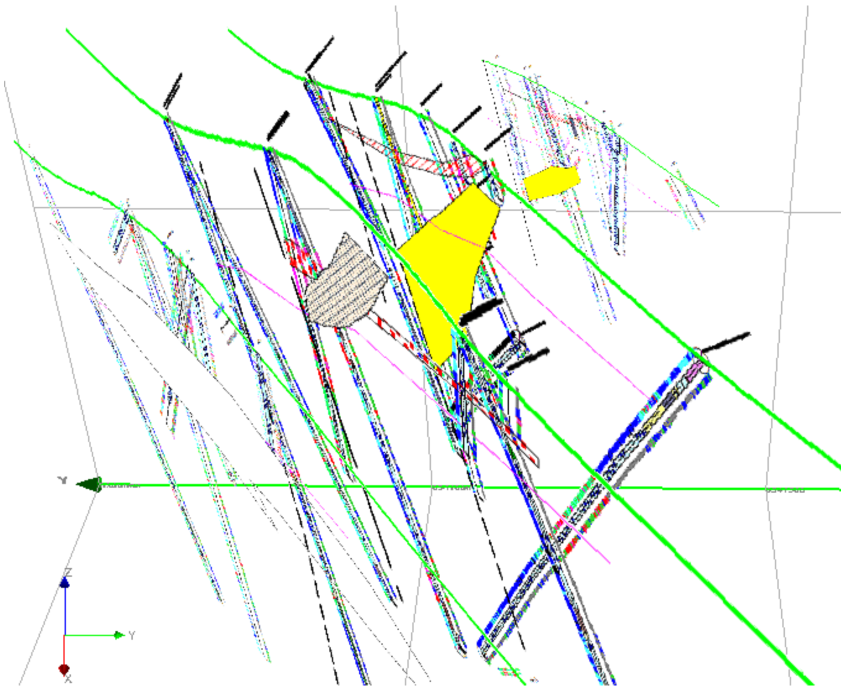


Discover – Export located bitmap dialog

2. Select one or more section map windows from the list of available map windows.

3. Enabling the **Permanently Save** tick box at the bottom of the dialog will open the **Save Sections for 3D display** dialog upon pressing **OK**. This allows the output georeferenced image to be named and saved to a selected directory. Pressing **OK** without enabling this option will create an georeferenced image using the map window's name, and save this into the Discover 3D Temporary directory (set under **Discover 3D>Options**. See *Customising 2D Interface Settings*; these temporary files will be deleted on exiting 3D if the **Remove temporary files on exit** option is enabled).

4. The sections are displayed in the 3D map window. As the section is created as a georeferenced bitmap image, two files are created - a .PNG (Portable Network Graphics) raster image file and a text header file with an .EGB (Encom Georeferenced bitmap) extension.



Drillhole section images displayed in 3D

To modify section parameters such as transparency, image processing refer to the *Changing Image Display Properties* section of the **Raster Images** chapter.

Maximising Image Resolution

The quality of the output 3D georeferenced image can be improved via:

- Utilising a larger MapInfo mapper window (but do not maximise it).
- Ensuring that your MapInfo mapper window is zoomed to the extents of the required data.
- Using **Discover 3D>Options**, increase the **Create Raster Images at …Screen Resolution** factor set in the **Display** tab of the dialog. This will increase the output image resolution. However it will also result in a larger output PNG image file size; displaying multiple high resolution images within Discover 3D may reduce the redraw speed and efficiency.

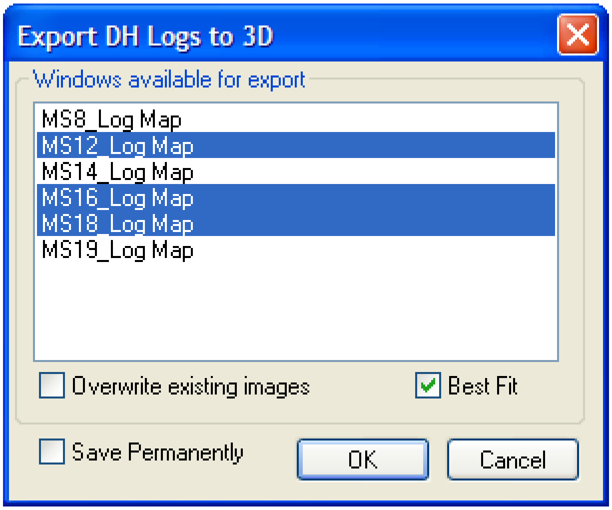
Viewing Drillhole Logs

Drillhole Log Profiles created in Discover can be viewed in Discover 3D as georeferenced bitmap images. These drillhole log images may include up to 16 columns of downhole lithological information, assay values or geophysical readings displayed as text, linegraphs or histograms. See the *Discover User Guide* for information on *Creating a Drillhole Log*.

Create a drillhole log in Discover using the **Drillholes>Log Display** menu option or open an existing drillhole log in MapInfo Professional.

To view a drillhole log as a georeferenced bitmap image in Discover 3D:

1. Select the **Discover 3D>View Logs** menu option. The **Discover – Export DH Logs to 3D** dialog appears:



Discover – Export DH Logs to 3D dialog

2. Highlight the logs to display in 3D from the available list. If drillhole log images are already displayed within Discover 3D, enabling the **Overwrite existing images** option will replace them with these newly selected logs. Otherwise, the newly selected logs will simply be added to the 3D display in addition to those already displayed.

3. Selecting **Best Fit** will increase the resolution and accuracy of the log image displayed in 3D. This option is on by default, and is recommended.

4. Enabling the **Save Permanently** option allows the user to specify the names and output directory (this defaults to the drillhole project directory) for these files and retain these images at the end of the 3D session. Pressing **OK** without enabling this option will create georeferenced images using each map window's name, and save these into the Discover 3D Temporary directory (set under **Discover 3D>Options**. See *Customising 2D Interface Settings*; these temporary files will be deleted on exiting 3D if the **Remove temporary files on exit** option is enabled). Press **OK** to display the logs in Discover 3D.

5. Alternatively, place the cursor in the log display window and right click to select the **View in 3D** menu item. This will add just the selected log display to the 3D window, in a cumulative fashion.

To modify log image parameters such as transparency, image processing or orientation refer to the *Display Drillhole Logs Images* section.

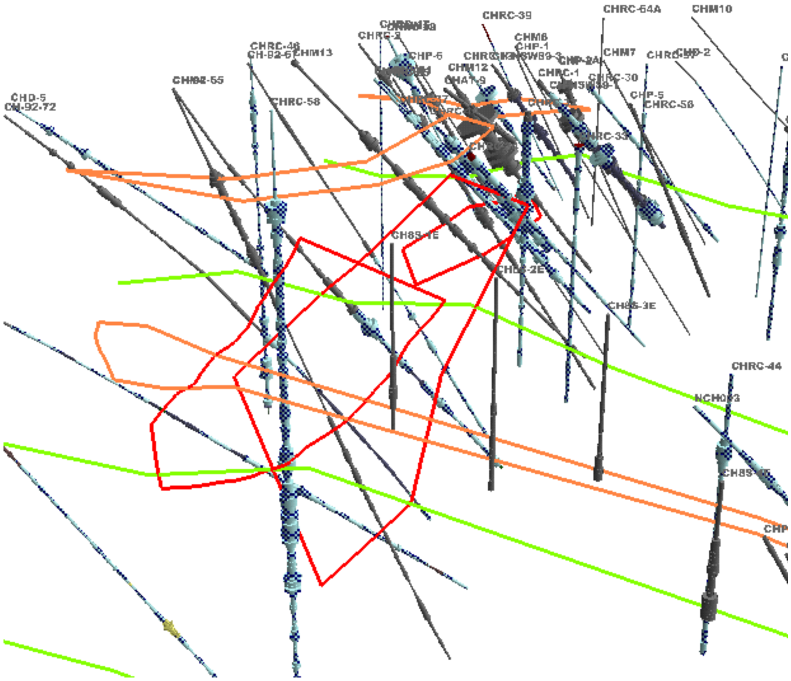
Note To increase the output resolution of an image created within Discover, increase the **Create Raster Images at …Screen Resolution** factor set in the **Display** tab of the **Discover 3D>Options** dialog. This will however result in a larger output PNG image file size; displaying multiple high resolution images within Discover 3D will reduce redraw speed and efficiency.

Viewing Section Layers

Note Polyline sections are not supported by this tool.

Drillhole section layers created in a Discover drillhole project can be viewed in Discover 3D as either 3D vectors (see *Displaying 3D Vectors*) or 3D features (see *Digitizing and Managing 3D Features*). These section layer files may contain lithological or ore body boundaries or polylines, or other geological interpretations such as faults and structural interpretations.

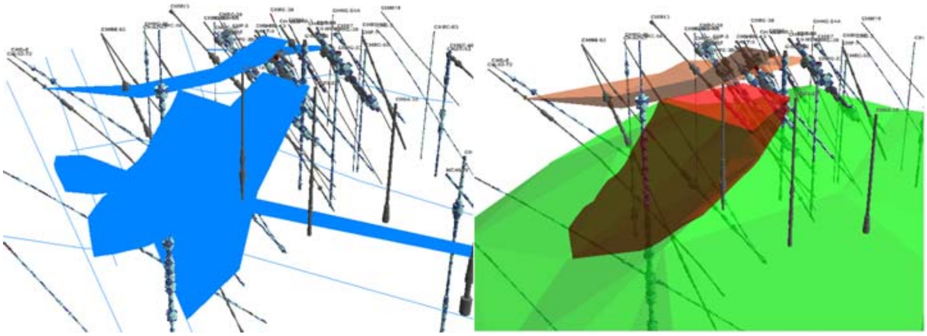
Create a drillhole section layer in Discover using the **Drillholes>Section Manager>Advanced>Add section layer** menu option or open an existing drillhole section that contains custom layers. See the *Discover User Guide* for more information on creating a section layer.



Section layers (red, orange and green linework) displayed as 3D vector files over modulated drillholes in 3D

Displaying section layers as 3D vectors (see *Displaying 3D Vectors*) in 3D allows attributed interpretations to be coloured individually (e.g. red ore body, blue supergene zone, green south_fault), etc). Vector files are also a far more efficient way of displaying this information than as an image (eg. via the *Viewing Drillhole and Trench Sections* menu option).

To create a 3D vector file from a drillhole section layer, use the Discover **Drillholes>External Data Formats>Export Section Layer as 3D DXF** menu option. If the 3D vector file already exists, open up the 3D vector file directly from within the Discover 3D application.



Section vector layers displayed in 3D as Feature Objects (left), and then converted into 3D solids using the Solid Generator (right: the colour scheme is identical to the previous image)

Displaying section vector layers as 3D features (see *Digitizing and Managing 3D Features*) allows 3D Solids to be built from the source data vector layers; the **3D Solid Generator** (see *Wireframing Models from Polylines, Polygons and Surfaces*) is used to wireframe between the existing vector layers. This is a powerful way to convert 2D interpretations into truly 3D geological bodies.

To view a drillhole section vector layers table as 3D Feature Objects:

1. Open the drillhole project, and the necessary cross-sections.
2. Use the **Discover 3D>View Section Layer** menu option. The **Export Layer to 3D Feature** dialog is displayed.
3. Select one or more drillhole section layers from the list.
4. By default the output Feature database file is named FeatureExport.fdb and is saved to the drillhole project root directory although this location can be altered if required. Attribute information (e.g. the Feature Code field) will be preserved in the .FDB file.
5. Select the output **File format**; the “Multiple files based on Feature Code field” option is recommended, as it will generate one feature database for each unique attribute in all selected sections.

   For example, if multiple 'shale' and 'limestone' bodies have been digitised into the custom layers of 20 sections, and attributed as such in the inherent 'Feature Code' field, this will generate two feature databases, one containing all shale bodies in the 20 sections, the other the limestone bodies. this is invaluable for 3D solid generation.

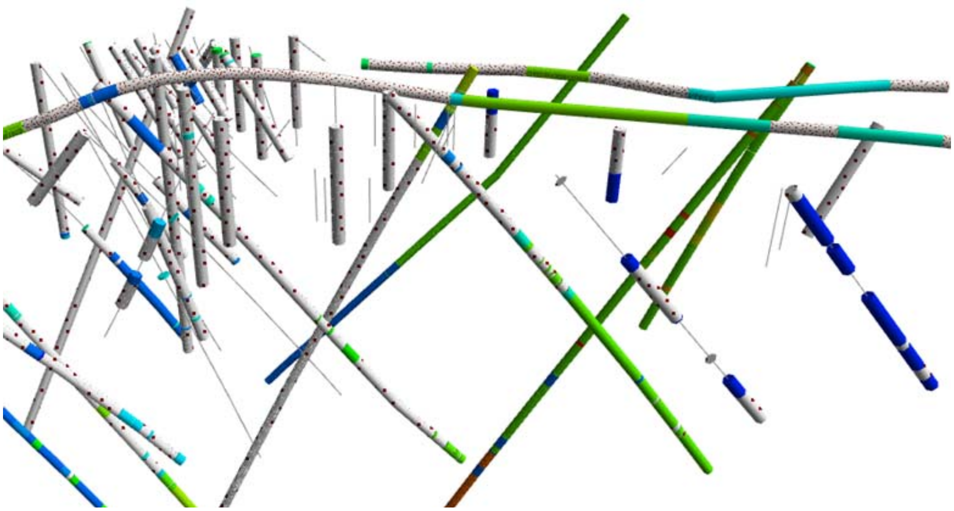
   The single File option will simply place all custom layer objects from all selected sections into the one (fully attributed) feature database.

6. Press **OK**. The new Feature Database is displayed in the 3D window.

See *Digitizing and Managing 3D Features* for further information on exporting drillhole section layers as **Feature Objects**.

Viewing Trenches

To view trenches in Discover 3D from the Discover/MapInfo interface, the trenches must be part of a Drillhole Project. See the *Discover User Guide* for information on creating a drillhole project with trenches.



Open a drillhole project using the Drillholes>Project Setup dialog and select the appropriate drillhole project from the pull-down list. Once the trenches project is open, the trenches to display in Discover 3D may be selected from a list of available trenches.

Note Trenches in 3D does not support Bearing and Distance Trench project with Segment distances. It also does not support Bearing and Distance Survey (either cumulative or segments) when a Topographic DEM surface grid is associated with the project.

To view trenches in 3D:

1. Select **Discover 3D>View Trenches**. The 3D Trench Wizard dialog is displayed:

2. The **Available** box contains a list of all the trenches in the collar table. Any trenches listed in the **Selected** box will be automatically displayed in Discover 3D.

   - Use the **Select All** button to copy all the trenches in the list to the **Selected** box. Use the **Select** button to select individual trenches . Hold down the left-mouse button and drag or hold down the SHIFT key to select consecutive trenches .

   - Use the **Deselect All** button to remove all trenches from the **Selected** box list. Use the **Deselect** button to remove individual trenches . Hold down the left-mouse button and drag or hold down the SHIFT key to select consecutive trenches .

3. The Options section at the bottom of this dialog controls the size of the trenches dataset imported into Discover 3D (as opposed to what is displayed in 3D). This is of particular use when dealing with very large trenches projects comprising thousands of holes. By default the entire trenches project is imported into 3D, but for large datasets this may slow down rendering and navigation in 3D.

   Selecting the **Subset based on selection** option will import (and display) into 3D only the selected holes. The relevant file subsets (collars, assays, etc) will be created with the selected suffix for tables applied.

   Choosing the **Save Permanently** option will enable the data subsets to be saved to a specified directory, otherwise these files will be saved to the Discover 3D temporary folder (set under **Discover 3D>Options**, see *Customising 2D Interface Settings*). These temporary files will be deleted on exiting 3D if the **Remove temporary files on exit** option is enabled.

4. Click OK and view the selected trenches in Discover 3D. The selected trenches are displayed in the 3D map window. To display trench attributes, colour patterns, labels, etc. see *Changing Trench Display Properties*.

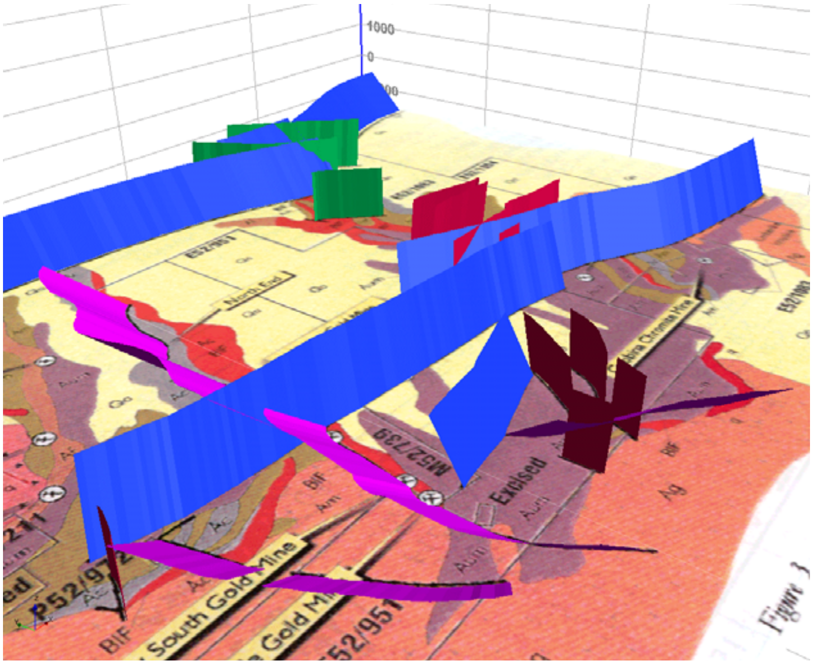
Note If the trenches project or a subset is already open in 3D, a prompt to replace the data will be displayed. Accepting this will automatically reapply any existing 3D display settings (e.g. colour or thickness modulation) to the new data.

Extruding Objects in 3D

The **3D Extrusion Wizard** allows you to extend (extrude) the shape of a 2D or 3D object (e.g. a polygon or polyline) from a base surface to a second or upper/ lower surface. This allows meaningful and useful visualisation of various bodies such as fault surfaces, mine shafts and workings, vein systems and buildings in three dimensions.

This tool is fully documented in *Extruding Models from Points, Lines and Polygons*.

The **Extrusion Wizard** can be accessed from both the 2D and 3D windows. It can operate on objects either within a mapped MapInfo Professional table (or a map window selection) or a Feature database (see *Digitizing and Managing 3D Features*). Height information can be specified from fields within the MapInfo Professional table/Feature dataset or manually set by the user. The sides of the extruded shape can be created as polygonal walls that give the impression of an enclosed volume within the extruded outline of the source object(s).

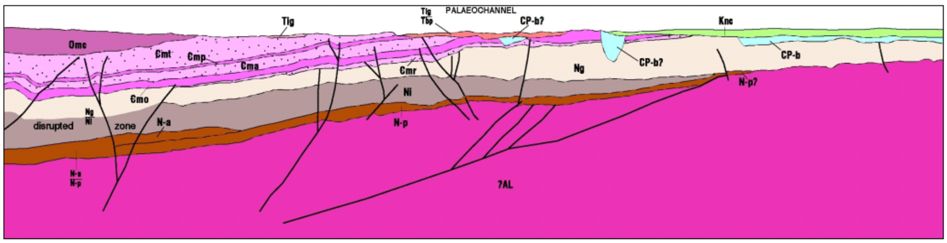


Fault visualisation using the 3D Extrusion Wizard

Displaying Other Types of Sections

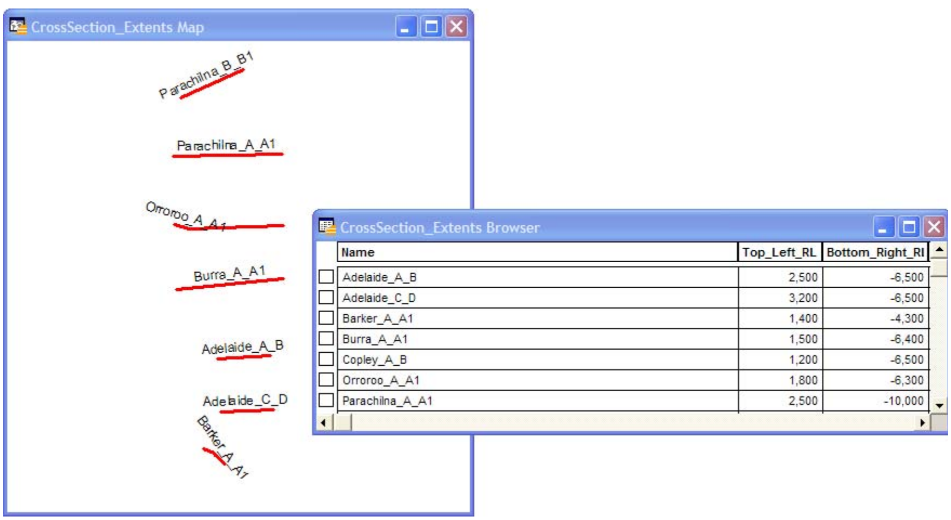
Multiple geological or geophysical cross-sectional profiles can be easily georeferenced and displayed in 3D as an EGB file using the **Multi Section Creator**.

This utility is specifically designed for viewing cross-section images in 3D that are not part of a drillhole project. For example, geological sections which have been scanned from geological map sheets or company reports or images of vertical slices cut through a geophysical model can be displayed in the correct 3D geographical space using this utility.



Example of a 2D geological cross-section (courtesy PIRSA, Australia)

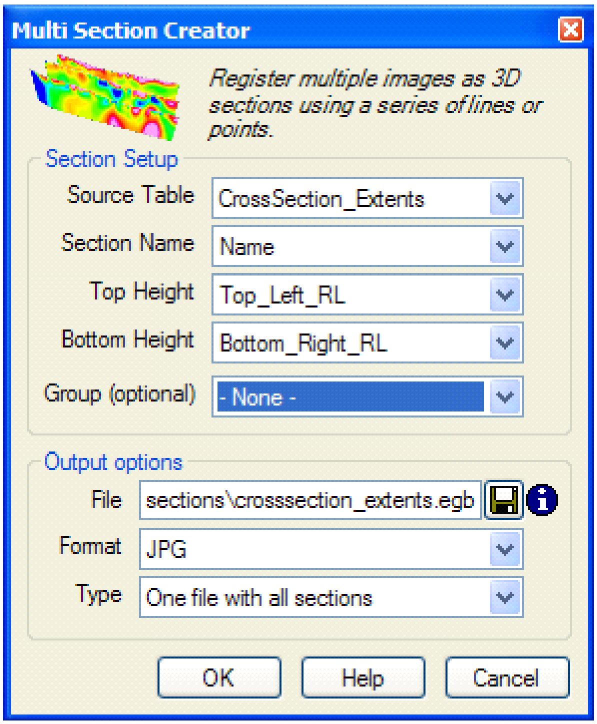
A MapInfo Professional table containing polylines representing the position of each cross-section in plan view is also required. Each polyline must be attributed with the cross-section image file name and the **Top** and **Bottom** elevations of the image. An example is displayed below.



Example of attributed polyline table required for georeferencing of multiple geological cross-sections (courtesy PIRSA Australia)

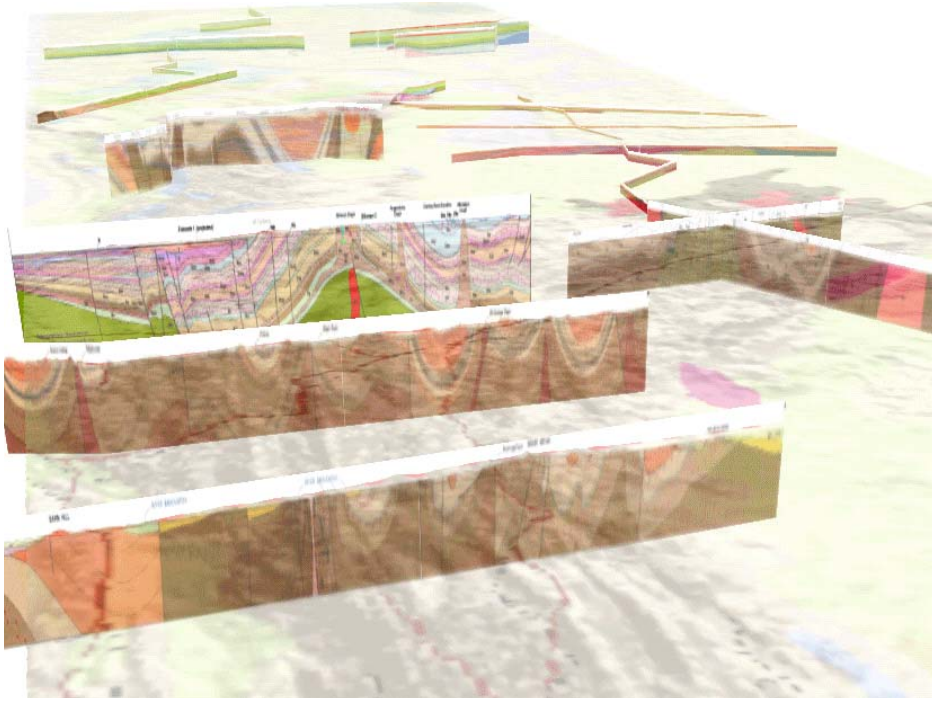
Run the **Discover 3D>3D Utilities>Multi Section Creator**. Specify the **Table** containing the attributed cross-section polylines, and then specify the appropriate image **Name**, **Top** and **Bottom Height** fields. An optional **Group** field can also be specified, grouping images with a shared attribute in this field into sub-layers in the output EGB file.

Choose an **Output** EGB file name and location (this defaults to the same directory as the source polyline table), and the **Format** of the cross-sectional images. If images of differing formats are involved, the image format suffix can be specified (in addition to the image name) in the **Section Name** field, and the **Format** set to **Section Name Includes Extension**. The Output **Type** pull-down list allows the choice of either one EGB file for all the sections, or separate EGB file for each section. Select **OK**.



Multi Section Creator dialog

A dialog indicating the percentage completion status will appear. At the completion of this process, and the EGB file will be opened and displayed in the 3D window as a new image branch.



3D view of multiple geological sections with a transparent geological map (courtesy PIRSA Australia)

Draping Images Over a Grid

This utility creates an EGB file for an image draped over a grid, in a similar fashion to using the **View in 3D/View Map In 3D** options. This EGB file can then be displayed in 3D. However unlike the **View in 3D/View Map In 3D** options, the **Image on Grid Creator** output EGB directly references the input image file, rather than creating a new associated PNG image file. Thus the resolution of the image is maintained, without having to maximise the size and alter the shape of the image map window to fit the image.

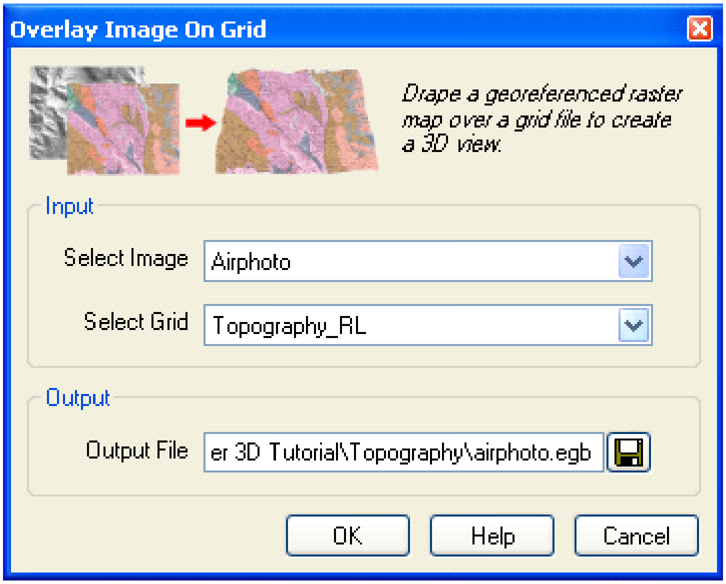


Image on Grid Creator dialog

To drape an image over a grid:

1. Ensure both the image and grid files are open within MapInfo Professional.

2. Select **Discover 3D>3D Utilities>Overlay Image on Grid**.

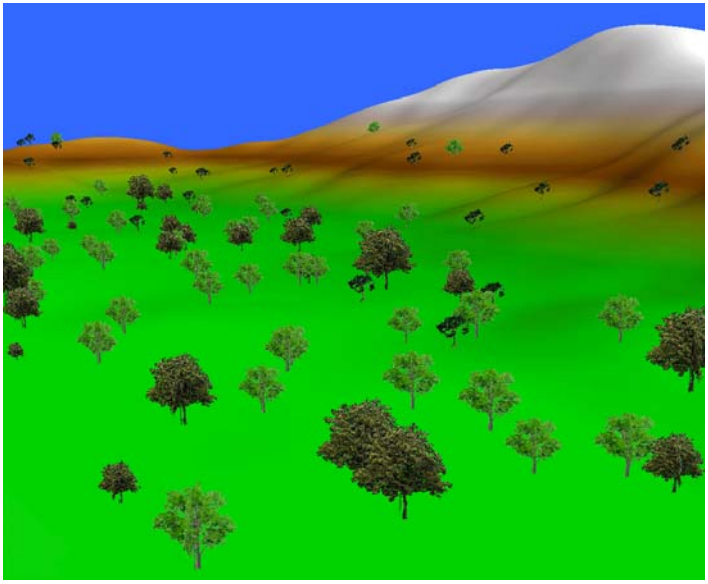
3. In the **Overlay Image on Grid** dialog, select these files in the appropriate pull-down lists.

4. Select an output file name and directory, and press **OK**.

5. The EGB file is opened and displayed in the 3D window as a new image branch.

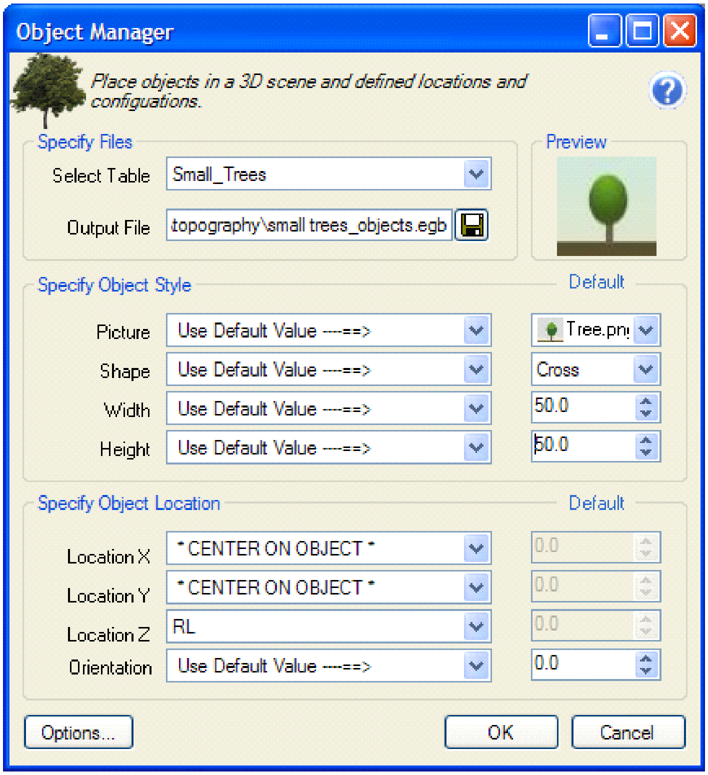
Adding Landscape and Ornamental Features

The **Object Manager** allows the 3D environment to be easily and rapidly populated with multiple image objects. Examples of use include displaying multiple images of trees and other vegetation, signposts and people in a landscape, texturing walls and simple structures (e.g. buildings) and placing signboards on buildings. These can be a useful way of giving a 3D environment a sense of scale (particularly useful for *Creating a Fly-Through Animation*)



Example of a 3D environment populated with image objects

The Object Manager is accessed via the **Discover 3D>3D Utilities>Object Manager** menu option. Use the **Select Table** pull-down list to select either a point or polyline TAB file representing the desired image locations. Specify an **Output File** name and location (this will be an EGB file).



Object Manager dialog

The following parameters can be specified:

- The **Object Style** parameters can be specified via either appropriate attribute fields within the source file, or using default values assigned within the Object Manager dialog.

- The **Picture** parameter controls the image used to represent the object. Using the Default Value allows image selection from all available images within the Image Location directory specified using the **Options** button at the bottom left of the dialog. A field can also be selected to specify the image; for each object, this must contain either a full image filename (if the image is in the same directory as the object location file), or a relative or full directory path as well as the image file name.

- The **Shape** parameter provides 3 default value options for the shape of the displayed image: Plane (a single image presented as a vertical plane), Cross (two perpendicular images crossing centrally, creating an illusion of a 3D image object from a distance) or Box (four images placed in a square around the object location). Alternatively, a field containing these parameters as text attributes can be specified.

- The **Width** of the image object can be set as a Default Value, assigned from a specified field, or for polyline objects, set as the Line Segment Width.

- The **Height** of the image object can also be set either as a Default Value, or assigned from a specified field.

- The **Object Location** parameters control the positioning and orientation of the image objects.

- The **Location X** & **Y** parameters provide options to Centre On Object (recommended option for point objects), Centre On Line Segments (recommended option for polyline objects), Use a Default Value or specify a field from which to read location data.

- The **Location Z** parameter can be specified as a field or set as a default value. If the image objects are to be placed on a Surface (e.g. a DEM or topographic surface), set the Default Value as 0, and once the images are displayed in the 3D window, use the Offset tab of the Images Properties dialog to Add a DTM offset using the appropriate surface.

- The **Orientation** of the output image objects can be specified as a Default Value, selected from a defined field or for polyline objects, calculated using the Line Segment Angles.

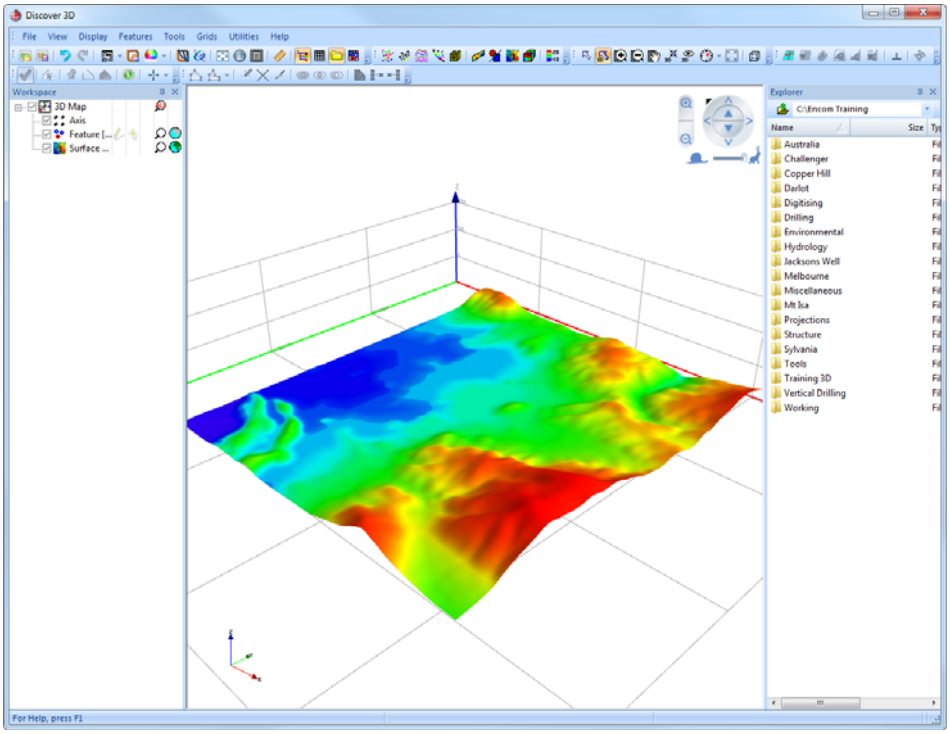
When these parameters are set, press **OK** to generate the EGB file. The EGB file will be opened and displayed in the 3D window as a new image branch.

6 Understanding the Discover 3D Interface

- *Discover 3D Window*
- *Discover 3D Menus*
- *Discover 3D Toolbars*
- *Customising the 3D Window*
- *Workspace Tree*
- *Projections in 3D*
- *Navigating in 3D*
- *3D Display Modes*
- *Cursor Plane*
- *Floating Colour Bar*
- *Sky Map*

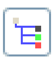
Discover 3D Window

For information on opening the 3D window, see *Opening the 3D Window*.



Discover 3D window

The 3D window is divided into six main regions:

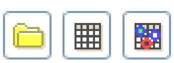


- The **Workspace Tree** (see *Workspace Tree*) provides a hierarchal view of all 3D data objects, data sources and controls. The Workspace Tree is the primary window for annotating, editing, ordering objects and general 3D window defaults.
- The **Display Window** is the main region for displaying 3D data, navigating and digitising.
- Dockable **Toolbars** are grouped into common operation commands used in Discover 3D. The toolbars are fully customisable and can be displayed as either floating or docked. For more information about the tools available, see *Discover 3D Toolbars*.

- The **Main** menu contains a more detailed list of Discover 3D tools than displayed on the dockable toolbars, the following menus are available:
  - **File** menu: Control the opening and saving of session files, exporting imagery, data packaging and printing.
  - **View** menu: Options to view/hide Information Windows, Toolbars, view modes, and status bar.
  - **Display** menu: Select vector or raster object types to insert into 3D window display.
  - **Features** menu: Controls for creating, editing, saving and querying Feature Databases.
  - **Tools** menu: Additional tools for vector and raster datasets and options for 3D global preferences.
  - **Grids** menu: Tools for creating, importing and querying Voxel models and surface grids.
  - **Utilities** menu: Additional productivity tools for creating AVI's, solid generating, extruding etc.
  - **Help** menu: Help and reference material.

For more information, see *Discover 3D Menus*.

- A **Status Bar** at the base of the screen displays 3D navigation and cursor coordinates bearing and inclination information and tool tips.



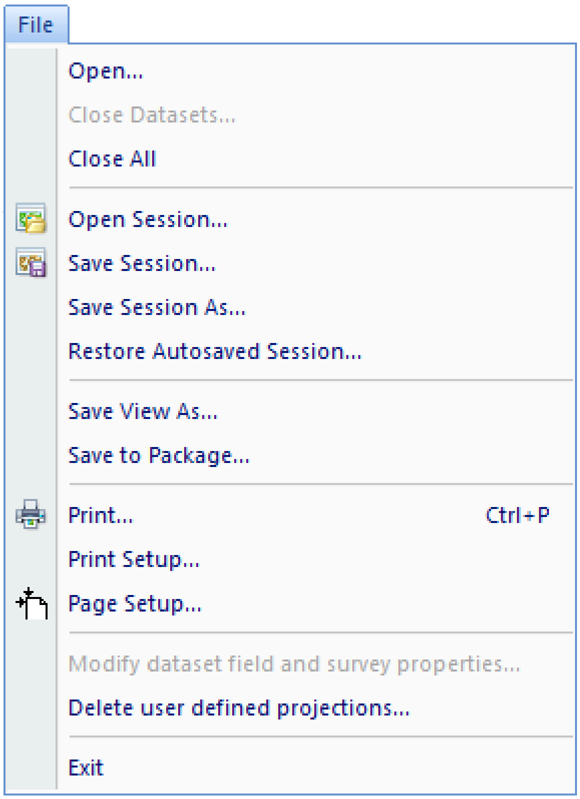
- The Information Windows are composed of the **Explorer**, **Data**, and **Feature Data** windows. These windows display dynamic information for data contained in the Display Window..

Discover 3D Menus

- *File Menu*
- *View Menu*
- *Display Menu*
- *Features Menu*

- *Tools Menu*
- *Grids Menu*
- *Utilities Menu*
- *Help Menu*

File Menu



File>Open

Open supported data types into Discover 3D.

Supported file types include:

- Discover Downhole files
- Geosoft Downhole files (before OASIS v 4.2)

- Geosoft Downhole files (after OASIS v 4.2)
- All supported Surface files
- All supported Image files
- All supported 3D Vector files
- All supported 2D Vector files
- All supported Session files
- All supported Fly-Through files
- Encom Voxel Model files

File>Close Datasets

Close selected drillhole datasets in Discover 3D

File>Close All

Close all open datasets in Discover 3D

File>Open Session

Open a Discover 3D Session file.

Session files do not contain data, objects or bitmaps. Consequently, if a session is opened but the data or object access are not available, Discover 3D will not be able to restore the session completely. A message is displayed in this circumstance indicating which source file is unreadable.

Session file restoration has a lost file recovery mechanism. If Discover 3D cannot locate some data or other necessary files, it alerts you and offers the option of browsing. This applies to all component file types that may have been saved with the session.

In the event that the requested file is not available, from the Open File dialog, select the Cancel button and the session will continue through its list of files to restore what it can.

File>Save Session

Save currently open dataset and **Display Window** views into a Discover 3D session file, or save edits to a currently open session. A 3D session file is a 3D equivalent of a MapInfo Professional Workspace file.

File>Save Session As

Save currently open dataset and **Display Window** views into a Discover 3D session file.

Note Sessions created in Discover 3D are not backward compatible. For example, a session created in Discover 3D 2012 will not open in Discover 3D 6.0.

File>Save View As

Save current **Display Window** contents into an image file for presentation purposes.

File>Save to Package

Save currently open dataset and **Display Window** views into a Discover 3D session file, and also save all currently open datasets into a designated folder. Packaging enables the easy transfer or sessions files and data between worksites or computers.

Note A packaged session file is equivalent to packing a workspace in MapInfo Professional using the tool located under **Discover>Table Utilities>Save Tables and Workspace**.

File>Print

Print the current **Display Window** to hardcopy or PDF format.

File>Page Setup

Define page size for hardcopy prints and page size for the page layout mode.

File>Modify Dataset Field and Survey Properties

Modify the field properties for all open drillhole datasets or modify the desurvey methods for downhole trace construction. Survey method to choose from include; Segments, Akima Spline, Bezier Spline, Back Calculation and Minimum Curvature. For changes to take effect, any existing drillhole branch containing the drillhole project must be deleted first and a new branch opened.

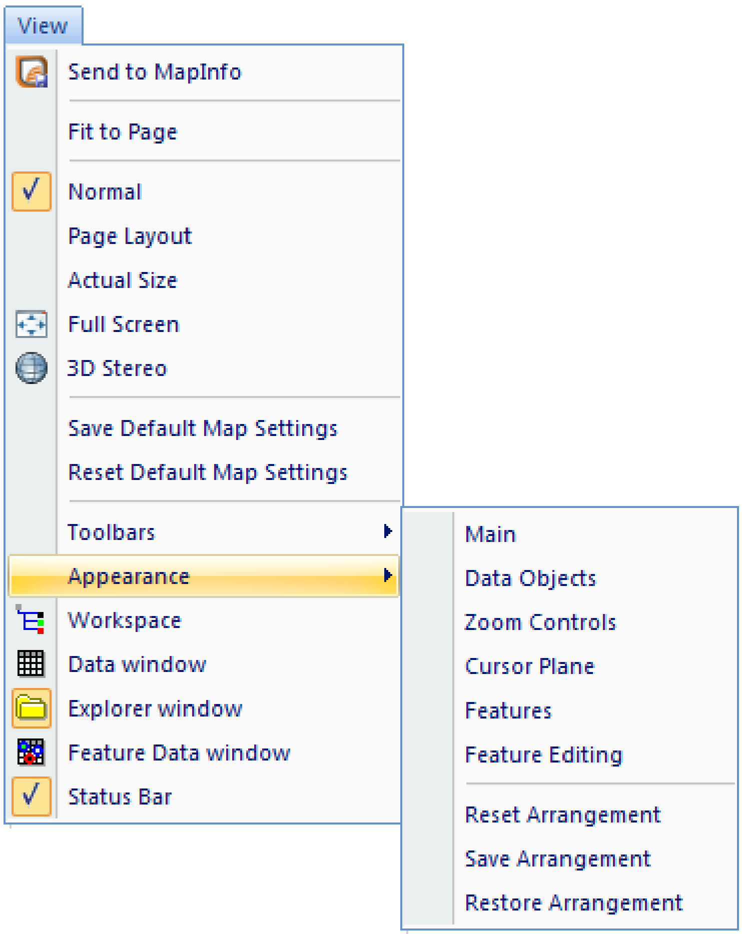
File>Delete User Defined Projections

Delete user-selected projections for individual files that have previously been opened in Discover3D. You can also clear the list of most recently used (selected) projections.

File>Exit

Exit Discover 3D application.

View Menu



View>Send to MapInfo

Export the current 3D **Display Window** to MapInfo Professional.

View>Fit to Page

Restore 3D **Display Window** size in page layout mode to fit the page.

View>Normal

Display 3D **Display Window** in normal mode for navigating, digitising and analysis.

View>Page Layout

Display 3D **Display Window** in page layout mode for hardcopy printouts and export.

View>Actual Size

Display the 3D **Display Window** in page layout mode at the actual page on screen.

View>Full Screen

Display the 3D **Display Window** in full screen mode.

View>3D Stereo

Open the 3D Stereo wizard.

View>Save Default Map Settings

Save modified properties for the **3D Map** and **Axis Workspace Tree** items, to load for subsequent Discover 3D sessions.

View>Reset Default Map Settings

Restore factory defaults for the **3D Map** and **Axis Workspace Tree** items.

View>Toolbars

Display or hide Discover 3D toolbars.

View>Toolbars>Main

Display or hide main toolbar.

View>Toolbars>Data Objects

Display or hide data objects toolbar.

View>Toolbars>Zoom Controls

Display or hide zoom controls toolbar.

View>Toolbars>Cursor Plane

Display or hide cursor plane toolbar.

View>Toolbars>Features

Display or hide features toolbar.

View>Toolbars>Feature Editing

Display or hide feature editing toolbar.

View>Toolbars>Reset Arrangement

Restore toolbar and window placement to factory settings.

View>Toolbars>Save Arrangement

Save toolbar and window placement, to load for subsequent Discover 3D sessions.

View>Toolbars>Restore Arrangement

Restore toolbar and information window placement to last saved arrangement settings.

View>Appearance

Select skin to display Discover 3D, settings include; Office 2000, Office XP, Office 2003, Windows XP, Office 2007.

View>Workspace

Display or hide **Workspace Tree** window.

View>Data Window

Display or hide **Data Window**.

View>Explorer Window

Display or hide **Explorer Window**.

View>Feature Data Window

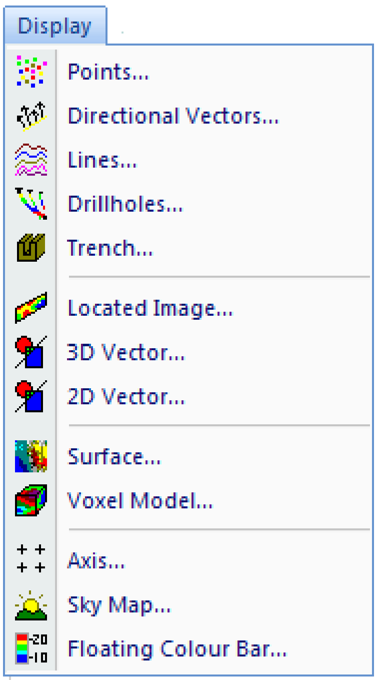
Display or hide **Feature Data Window**.

View>Status Bar

Display or hide **Status Bar**.

Display Menu

Discover 3D allows various external objects to be added to an existing 3D **Display Window**.



Display>Points

Add a Points display object to the **Workspace Tree** to display and annotate 3D point data.

Display>Directional Vectors

Add a Directional Vector object to the **Workspace Tree** to display and annotate 3D directional vector data. Directional vector data can typically be sourced from a drillhole or survey dataset, which contains a directional magnetic component.

Display>Lines

Add a Lines display object to the **Workspace Tree** to display and annotate 3D line data.

Display>Drillholes

Add a Drillhole display object to the **Workspace Tree** to display and annotate 3D drillhole data.

Display>Trench

Add a Trench display object to the **Workspace Tree** to display.

Display>Located Image

Add a georeferenced Image object (*.egb) to the **Workspace Tree** to display and annotate a georeferenced image. An example of a located image is an aerial photograph draped over a terrain grid

Display>3D Vector

Add a 3D vector layer to the **Workspace Tree** to display and annotate 3D vector data. 3D vector datasets can range from a 3D ore-body model to a contour dataset.

Display>2D Vector

Add a 2D vector layer to the **Workspace Tree** to display and annotate 2D vector data.

Display>Surface

Add a Surface layer to the **Workspace Tree** to display and annotate 3D surface data, such as a topographic surface.

Display>Voxel Model

Add a Voxel Model layer to the **Workspace Tree** to display and annotate 3D voxel data, such as ore block model derived from modelling packages.

Display>Axis

Add additional Axis layer to the **Workspace Tree**.

The **Axis Properties** dialog controls all aspects of the displayed axes. Manual and automatic options are available for:

- Axis origins (as related to axis locations) and styles.
- Axis extents and bounding box settings
- Axis tick display and intervals
- Axis title and tick labelling

Display>Sky Map

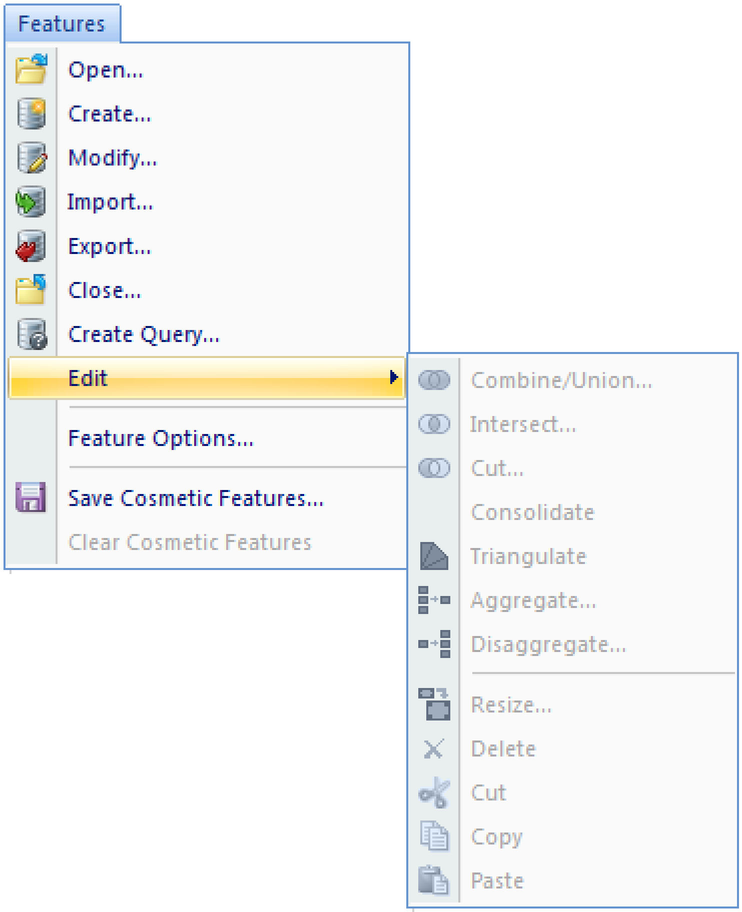
Add a Sky Map image layer to the **Workspace Tree** to display a photo-realistic sky view backdrop to your 3D environment. This can be particularly valuable for producing impressive image/video output.

Display>Floating Colour Bar

Displays a colour bar (or legend) for any open drillhole project, grid surface or voxel model.

Features Menu

Features are point, line, polygon or surface objects imported or created in the 3D environment which represent various entities such as geological or ore zone boundaries or structural interpretations.



Features>Open

Open 3D Feature database file (*.fdb).

Features>Create

Create a 3D Feature database file (*.fdb).

Features>Modify

Modify an existing Feature database name or field structure.

Features>Import

Import Vector or grid files into a Feature database.

Features>Export

Export objects from a Feature database into numerous vector formats.

Features>Close

Close selected Feature database.

Features>Create Query

Select objects from a Feature database by attribute query.

Features>Edit

Numerous editing tool for Feature Objects.

Features>Edit>Combine/Union

Combines or fuses multiple selected features into a single feature; the original geometries of the features are lost.

Features>Edit>Intersect

Outputs the intersection of any selected features (except points).

Features>Edit>Consolidate

Recombines a selected multi-part feature object.

Features>Edit>Triangulate

Primarily used to convert selected polygons into triangulated surfaces (3D polygon mesh or TINs). Can also be used to convert any object type into a Surface (TIN), such as a elevation point cloud or contours.

Features>Edit>Aggregate

Combines multiple selected features into one feature, but preserves the spatial geometry of each feature.

Features>Edit>Disaggregate

Ungroups or explodes aggregated features into individual features. Also detects any disconnected parts of a feature (created using the Break tool) and creates individual feature objects for each part.

Features>Edit>Resize

Resize selected Feature Objects by scale factor.

Features>Edit>Delete

Delete selected Feature Object.

Features>Edit>Cut

Create new feature by cutting the first selected feature along intersections with other selected features. Cut can be used for any of the following operations.

Selected objects in the same plane:

- Cut (or remove) part of a polygon feature where it overlaps other polygon feature(s).
- Cut a polygon feature along the intersection with other polyline feature(s).

Selected objects in different planes:

- Cut a polygon feature along intersections with other polygon feature(s).
- Cut a feature surface along intersections with other feature surface(s).

Features>Edit>Copy

Copy selected Feature Object onto clipboard.

Features>Edit>Paste

Paste selected Feature Object from clipboard into Feature database.

Features>Feature Options

Define default options for the behaviour and appearance of Feature databases.

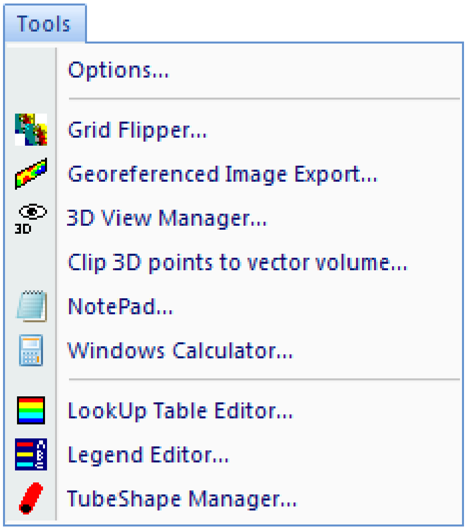
Features>Save Cosmetic Features

Save digitized objects from the Cosmetic Feature database into a new or existing Feature database.

Features>Clear Cosmetic Features

Clear digitized objects from the Cosmetic Feature database.

Tools Menu



Tools>Options

Display options for Discover 3D defaults.

Tools>Grid Flipper

Open Grid Flipper to quickly toggle between grids surfaces with a single surface group.

Tools>Georeferenced Image Export

Open Georeferenced Image Exporter to capture and display 3D image renders from the Cursor Plane or Drillhole section.

Tools>3D View Manager

Open 3D View Manager to save and restore 3D image view points.

Tools>Clip 3D Points to Vector Volume

Clip and select point objects from a Feature Database or points dataset using a vector volume in a Feature database or 3D Vector file.

Tools>NotePad

Open Windows Notepad.

Tools>Windows Calculator

Open Windows Calculator.

Tools>LookUp Table Editor

Open Colour Table Editor to create and modify lookup colour schemes for surface or drillhole rendering.

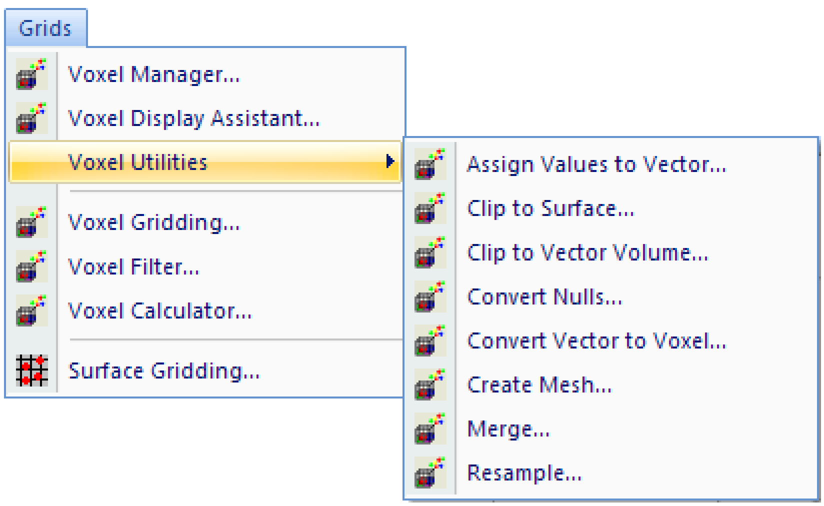
Tools>Legend Editor

Open Legend Editor to create and modify drillhole legend pattern and colour schemes.

Tools>TubeShape Manager

Open Tube Shape Manager to modify 3D data line styles.

Grids Menu



Grids>Voxel Manager

Open Voxel Manager to load, import, save and convert grids.

Grids>Voxel Display Assistant

Open Voxel Display Assistant to help display a voxel model thresholded, clipped or with an isosurface applied.

Grids>Voxel Utilities

A powerful range of utilities for advanced vector/voxel analytics, voxel resampling and merging and clipping voxels to vector surfaces/volumes.

Grids>Voxel Gridding

Open Voxel Gridding to create 2D (Planar) and 3D (Block Model) grids.

Grids>Voxel Filter

Open Voxel Filtering to apply filters to simply grids.

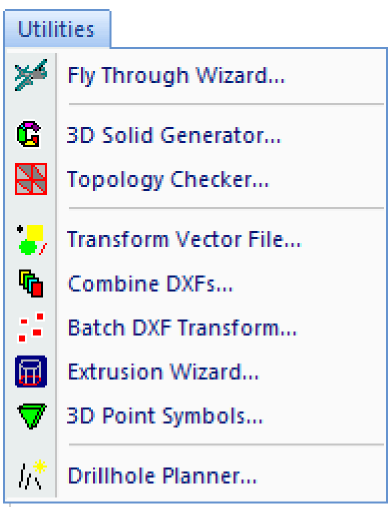
Grids>Voxel Calculator

Open Voxel Calculator to modify or process a grid.

Grids>Surface Gridding

Open Interactive Gridding to create a 3D surface grid.

Utilities Menu



Utilities>Fly Through Wizard

Open Fly Through Wizard to create 3D animations and movies.

Utilities>3D Solid Generator

Open 3D Solid Generator tool to wireframe models and calculate the volume of generated solids.

Utilities>Topology Checker

Open Topology Checker tool to analyse and correct the integrity of 3D vector models.

Utilities>Transform Vector File

Open the Transform Vector File tool to convert vector formats along with transforms for scaling, offset and rotation.

Utilities>Combine DXFs

Open Combine DXF tool to combine two or more DXF files.

Utilities>Batch DXF Transform

Multiple external DXF files can be positioned, scaled and rotated using a control table, and combined into one output file.

Utilities>Extrusion Wizard

Open Extrusion Wizard to extrude models.

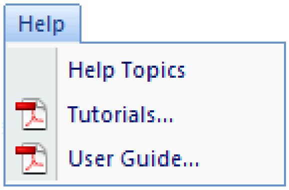
Utilities>3D Point Symbols

Open 3D Point Symbols tool to create 3D point objects.

Utilities>Drillhole Planner

Open Drillhole Planner tool to plan and position new drillholes directly in 3D.

Help Menu



Help>Help Topics

View Discover 3D interactive help topics.

Help>Tutorials

Open Discover 3D tutorial documentation (PDF format).

Help>User Guide

Open Discover 3D User Guide documentation (PDF format).

Discover 3D Toolbars

Discover 3D toolbars are fully customisable and can be displayed in either docked or floating mode.

- *Main Toolbar*: commonly used 3D tools.
- *Data Objects Toolbar*: display 3D objects.
- *Zoom Controls Toolbar*: 3D navigational tools.
- *Cursor Plane Toolbar*: tools for digitising and clipping objects.
- *Features Toolbar*:digitising and selecting in Feature Databases.
- *Feature Editing Toolbar*: advanced Feature Database object editing.

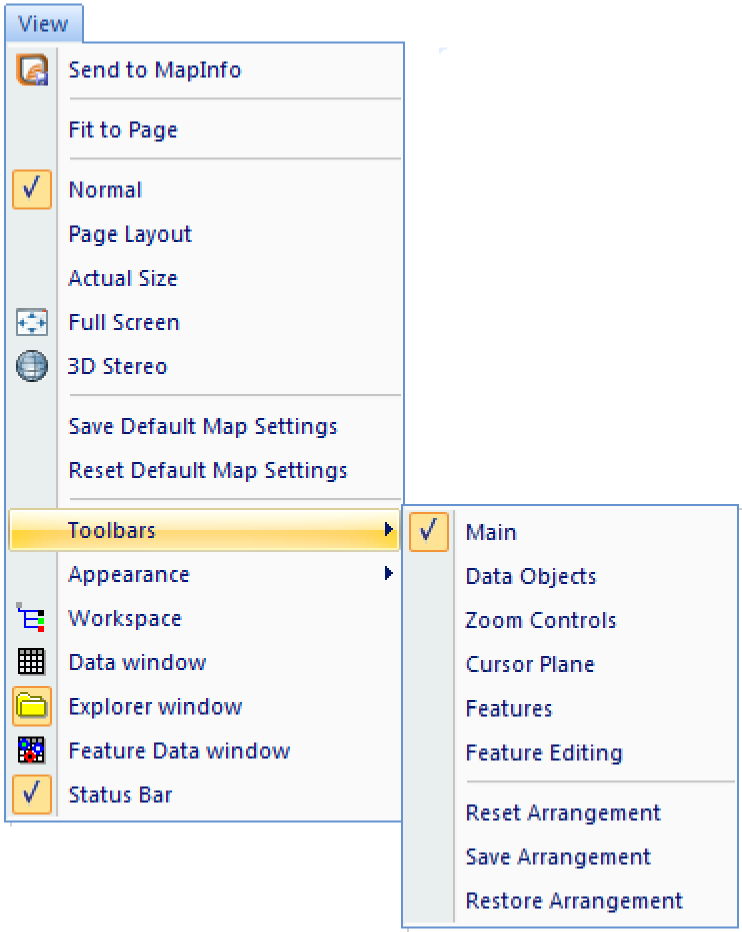
Note A brief description of the toolbar tool is displayed when hovering the mouse over a tool button, Alternatively, a description of the tool is displayed in the **Status Bar**.

For information on arranging and customising toolbars, see:

- *Showing and Hiding Toolbars*
- *Docking and Undocking Toolbars and Windows*
- *Customising Toolbars and Window Views*
- *Customizing Toolbars*

Showing and Hiding Toolbars

To display or hide a toolbar navigate to **View>Toolbars** and select the toolbar to either display or hide.



Display toolbars from Main menu.

Alternatively, right-click in the Main menu bar to display the toolbar shortcut menu.

Main Toolbar

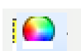
Open Session - Open a Discover 3D Session file (*.egs). A session file is an equivalent 3D version of MapInfo Workspace.

Save Session - Save a session file (*.egs). A session file is an equivalent 3D version of MapInfo Workspace which enables **Display Window** and object states to be preserved for future reference.

Undo/Redo - Undo or redo the last change committed in the **3D Display Window**.

Show 3D Properties - Display **Properties** dialogs for all object, axis and 3D Map items in the Workspace Tree.

Send to MapInfo - Export the current view displayed in the 3D Display Window as an image for presentation or analytical purposes into MapInfo Professional.



Background colour - Modify the default background colour in the 3D Display Window.

Open Discover drillholes - Open Discover drillhole data from MapInfo Professional TAB files.

Open Geosoft drillholes - Open Geosoft WHOLEPLOT drillhole Database.

Full Screen - Display the 3D Window in full screen mode.

3D Stereo View - Display 3D window in 3D stereo mode on HDMI enabled monitor.

Normal View/Page Layout - Toggle the **Display Window** between **Normal** and **Page Layout** views. The **Normal** view is used to perform most navigational, editing analytical operating in 3D. The **Page Layout** view is used for presentation of hardcopy outputs, however, standard navigational, editing analytical operations can be performed in this mode.

Distance/Bearing Measuring Tool - The ruler tool enables the measurement of distance, bearing, inclination and total distance for objects on the Cursor plane. Measurement of can be performed from one point to another by a single left mouse button click, or can be cumulative by left mouse button clicking along segments continuously. If the left mouse button is continuously held down whilst the cursor is moved, the measurements are read from the path of the mouse cursor. Enabling **Snapping** by pressing **S** on the keyboard can assist in measuring between vertices.

Workspace Tree - Display or hide the **Workspace Tree**.

Data Window - Display or hide the **Data Window** which interactively displays data from the following object types; Drillholes, Lines, Directional Vectors and Points.

Explorer Window - Display or hide the **Explorer Window** which is used to navigate and locate data to open in the 3D **Display Window**.

Feature Data Window - Display or hide the **Feature Data Window** which displays interactive information contained in a Feature Database.

Data Objects Toolbar

Display Points - Add a Points display object to the **Workspace Tree** to display and annotate 3D point data.

Display Directional Vectors - Add a Directional Vector object to the **Workspace Tree** to display and annotate 3D directional vector data. Directional vector data can typically be sourced from a drillhole or survey dataset, which contains a directional magnetic component.

Display Lines - Add a Lines display object to the **Workspace Tree** to display and annotate 3D line data.

Display Drillholes - Add a Drillhole display object to the **Workspace Tree** to display and annotate 3D drillhole data.

Display Trenches - Add a Trench display object to the **Workspace Tree** to display and annotate 3D trench data.

Display Located Image - Add a georeferenced Image object (*.egb) to the **Workspace Tree** to display and annotate a georeferenced image. An example of a located image is an aerial photograph draped over a terrain grid.

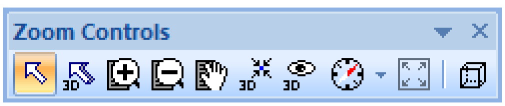
Display 3D Vectors - Add a 3D vector layer to the **Workspace Tree** to display and annotate 3D vector data. 3D vector datasets can range from a 3D ore-body model to a contour dataset.

Display Surface - Add a Surface layer to the **Workspace Tree** to display and annotate 3D surface data, such as a topographic surface.

Display Voxel Model - Add a Voxel Model layer to the **Workspace Tree** to display and annotate 3D voxel data, such as ore block model derived from modelling packages.

Display Floating Colour Bar - Add a Floating Colour Bar or data legend which can be link to the following object types; surface, voxel or drillhole.

Zoom Controls Toolbar



The Zoom Controls Toolbar controls navigation, zooming and custom view management in the 3D **Display Window**.

Select/Navigate - **Select/Navigate** is used as the primary control for selecting and navigating in the 3D environment. Holding the left mouse button down and moving the mouse cursor will pivot the view are the central display axes.

When a **Workspace Tree** object select option is enabled the **Select/Navigate** control can be used to display attribute information in the **Information Windows**.

3D Navigation - The 3D Navigation control is primarily used to navigate around the 3D **Display Window**. Holding the left mouse button down and moving the mouse cursor will pivot the view are the central display axes. Holding the right mouse button down and moving the mouse cursor will either zoom in or out from the central display axis. Refer to *Navigating in 3D* for more detailed operations.

Zoom in, Zoom out, Pan - The zoom controls either zoom in or out in the 3D **Display Window**, holding the **Shift** key will perform the opposite zoom function. With either the Zoom in or Zoom out control activated, when the left mouse button is held down and the mouse cursor is dragged, the **Display Window** will zoom in or out to the specified zoom field. The Pan control enables panning in the **Display Window.**

Reset View - Reset 3D View restores the 3D Display Window so objects are centred and displayed within the window.

View Manager - The 3D View Manager allows the user to save custom views within the 3D **Display Window**. See *3D View Manager Tool*.

Change View Direction - Rotate view direction of the 3D Display Window to the specified direction (North, South, East, West, Up and Down). For example, selecting **Look North** orientates the view to be looking to the north.

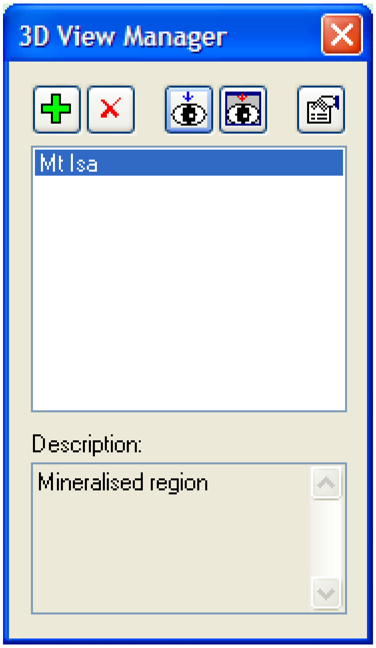
Fit to Page - Only available in **Page Layout** view; automatically resizes the page to fit to the extents of the view.

Perspective/Orthographic - 3D display modes

- **Perspective** – Objects that are further away from the viewer are scaled so they appear smaller than closer objects. Perspective view provides more information about depth and is often easier to work with because it simulates the real life view.

- **Orthographic** – A form of parallel projection (also known as an isometric or axonometric projection) where identically sized objects are displayed with the same size regardless of the distance they are positioned from the viewer. Orthographic view is best used when it is important to be able to judge proportion and size, such as, when digitizing or drawing features, and is particularly useful for measuring distances.

3D View Manager Tool



3D View Manager dialog

Add new 3D view to the **3D View Manager**.

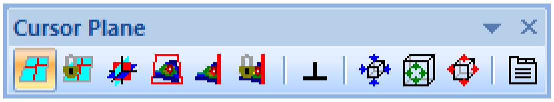
Delete the selected 3D view from the **3D View Manager**.

Display the selected 3D view from the **3D View Manager**.

Display the selected 3D view from the **3D View Manager** and restore the 3D data window to the select view extents and dimensions.

Rename the selected 3D view from the **3D View Manager**.

Cursor Plane Toolbar



The Cursor plane toolbar contains tools required for the operation of the Cursor plane. The Cursor plane operates as a drawing plane upon which Feature Objects are digitized in the 3D environment. The Cursor plane can also be used as a clipping plane to hide or obscure object data.

Show Cursor plane - The **Show Cursor plane** option displays or hides the Cursor plane in preparation for digitizing or clipping operations.

Lock Cursor plane - Disables all Cursor plane movement controlled by the keyboard shortcuts, preventing the Cursor plane from being accidentally moved during digitization. This feature locks the Cursor plane origin X, Y and Z coordinate.

Cursor plane orientation - Toggles the Cursor plane orientation around the X, Y and Z axes.

Bond - Bonds the Cursor plane to a selected georeferenced image or dataset (e.g. drillholes) in the Workspace Tree. This feature is useful for aligning the Cursor plane with images georeferenced in 3D for digitizing. It is also auto-enabled when selecting data in Spreadsheet mode—for interrogating drillholes dynamically, for example (see Browse tool in *Workspace Tree*).

Plane clipping - Plane clipping hides all 3D data objects in front of the Cursor plane. This is useful when a slice view is required of the 3D datasets when digitizing.

Defined clipping - Enables the supplementary clipping of a 3D dataset as defined by the current position of the Cursor plane. This option enables the current position of the Cursor plane to temporarily clip the 3D dataset whilst being able to move the Cursor plane to another position. This is useful when digitizing through a complex geometric object, or to reveal obscured features.

Perpendicular - This will orientate the view direction perpendicular to the Cursor plane. This is very useful when digitizing interpretations on adjacent sections, particular when used in tandem with the **Orthographic View** mode and **Envelope** or **Slice** clipping mode.

 Shrink

 Enlarge

 Fit

Shrink, Enlarge and Fit Cursor plane - The Shrink, Enlarge and Fit Cursor plane controls resize the Focus Box. The Focus Box is the bounding extents of the Cursor plane.

Cursor Plane Properties - Global properties for appearance and Cursor plane behaviour can be modified on the Cursor Plane Properties dialog.

Features Toolbar

The Features toolbar provides numerous controls for the creation and selection of Feature Objects on the Cursor plane. This enables the user to digitize points, polylines and polygons directly into the 3D environment.

A Feature database must be editable for the Feature toolbar to be enabled.

Example objects include geological interpretations, ore body outlines, faults and alteration zones.

Confirm feature - Option to display a dialog after each feature object is created to enable input of attribute data.

Polygonal Select - Create a selection region to select all Feature Objects that intersect or lie within the region. This selection region is independent of the Cursor Plane. It is drawn in the screen plane and will select all objects below it. The selected object attributes are listed in the **Data Window**.

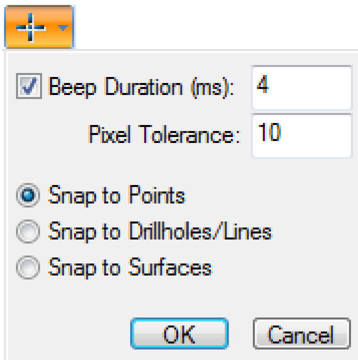
The **Select/Navigate** control can be used to select individual feature objects. When holding down the keyboard CTRL key multiple object selections can be made.

Create New Point Feature - Create a point object by clicking with left mouse button at the desired location on the Cursor plane.

Create New Line Feature - Create a polyline on the Cursor plane by clicking the left mouse button to create line inflections. Holding down the left mouse button and moving the mouse will create sinuous polyline. Double click to terminate the polyline creation.

Create New Polygon Feature - Create a polygon on the Cursor plane by clicking the left mouse button to create line inflections. Holding down the left mouse button and moving the mouse will create sinuous polygon boundary. Double click to terminate the polygon creation.

Feature Information - Display information for a Feature Object. Click the left mouse button to display attribute information in the **Data Window**.



Snapping - Snapping enables the precise interrogation, modification or digitization of points, polylines or polygons on a feature object or drillhole trace. Snapping will ensure the exact point, drillhole or surface is selected.

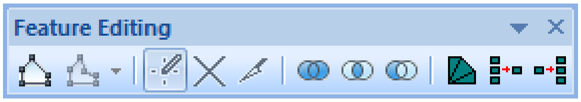
To assist snapping an audible beep is generated, the duration of the beep can be adjusted.

When snapping is enabled a circle will appear in the centre of the Cursor plane crosshairs, the radius of the circle can be controlled by the **Pixel Tolerance** value.

The **Snap to ...** options allow precise control on what is targeted:

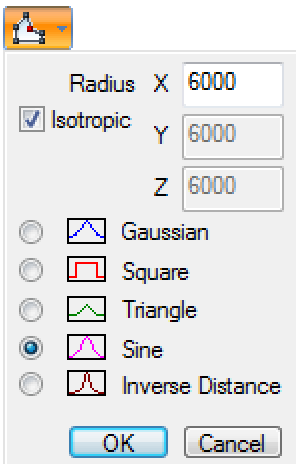
- **Snap to Points** - Snap to points or vertices on any feature object
- **Snap to Drillholes/Lines** - Snap to drillhole traces. Also snap to internal and external edges of feature polylines, polygons, surfaces and polyhedrons solids.
- **Snap to Surfaces** - Snap to the surfaces (triangulated faces) of feature polygon, surface or 3D solid polyhedrons.

Feature Editing Toolbar



The Feature Editing toolbar includes all tools required for advanced editing for Feature Objects.

Edit - Modify the selected Feature Object on the Cursor plane. The Edit control displays the object vertices; individual nodes can be selected and moved or deleted by pressing the keyboard Del key. Entire objects can be moved by holding the Shift keyboard key and moving to the intended location on the Cursor plane.



Elasticity - When moving a node on a feature object, the Elasticity option enables the automatic movement of adjacent object nodes. The area or region of nodes to be moved elastically is defined with the Radius X control. To define a variable region of elastic movement, uncheck the Isotropic option and set the Y and Z radii as desired. The type or shape of elastic movement can be set as Gaussian, Square, Triangle, Sine and Inverse Distance (Sine is a good starting point).

Add Mode - Add vertex to a polyline, polygon, surface or solid 3D Feature Object. Adding a vertex will allow the shape to be modified; after adding a vertex left mouse click and hold on the selected vertex to move.

Delete Mode - Delete vertex from a polyline, polygon, surface or solid 3D Feature Object. Deleting a vertex will allow the shape to be modified; to delete, select the vertex and press DEL on the keyboard.

Break Mode - Split polyline or surface Feature Objects.

Polylines are split at connecting vertices.

Surface feature objects are split along connected internal edges.

Combine - Combines or fuses multiple selected features into a single feature; the original geometries of the features are lost.

Intersect - Outputs the intersection of any selected features (except points).

Cut - Create new feature by cutting the first selected feature along intersections with other selected features. Cut can be used for any of the following operations.

Selected objects in the same plane:

- Cut (or remove) part of a polygon feature where it overlaps other polygon feature(s).
- Cut a polygon feature along the intersection with other polyline feature(s).

Selected objects in different planes:

- Cut a polygon feature along intersections with other polygon feature(s).
- Cut a feature surface along intersections with other feature surface(s).

Triangulate - Primarily used to convert selected polygons into triangulated surfaces. It can also be used for re-triangulating a modified feature surface.

Aggregate - Combines multiple selected features into one feature, but preserves the spatial geometry of each feature.

Disaggregate - Ungroups or explodes aggregated features into individual features. Also detects any disconnected parts of a feature (created using the Break tool) and creates individual feature objects for each part.

Discover 3D Session Files

A 3D session file is the 3D equivalent of a MapInfo Professional Workspace file. Its is stored as a binary .EGS file that is used to save and restore a Discover 3D session. Session files store the file path to all open drillhole, point and line data selections, surface grids, 3D vector files, georeferenced bitmap images and associated properties such as downhole attributes, colour tables, labels, translucency, 3D Axes, etc.

- Open session files using either:
  - **File>Open Session**
  - Drag-and-drop the .EGS file into the 3D window
- Create session files with the **File>Save Session As** tool
- Save (and update) session files with the **File>Save Session**
- A default backup 3D session is automatically saved at regular intervals, as specified by the **Autosave Session** option under the **Tools>Options>View** tab (5 minutes by default). This backup session can be reopened at any time via the **File>Restore Autosaved Session** menu option. If Discover 3D closes unexpected, upon restarting you will be prompted to open the last autosaved session.

  The autosaved session file is called Discover 3D_auto_save_tmp and is saved under `C:\Users\username\AppData\Local\Temp` (Windows 7 and 8) or `C:\Documents and Settings\username\Local Settings\Temp` (Windows XP)
- Export session files and all contained datasets to another Discover 3D user with the **File>Save to Package** menu option. This is similar in functionality to the Discover Save Tables and Workspace option; all datasets in the session will be copied into the specified package directory (creating a new empty directory is recommended) as well as the .EGS session file, and the dataset file paths are set to relative. This directory can then by sent to another user (e.g. zip up and email the directory) and the contained session file opened on another computer

For more information, see the Discover 3D *File Menu*.

Note Sessions created in Discover 3D are not backwards compatible. For example a session created in Discover 3D 2012 will not open in Discover 3D 6.0.

Customising the 3D Window

Discover 3D can be customised to suit the requirements of each individual to improve the user experience.

- *Docking and Undocking Toolbars and Windows*
- *Auto-hide Windows*
- *Customising Toolbars and Window Views*

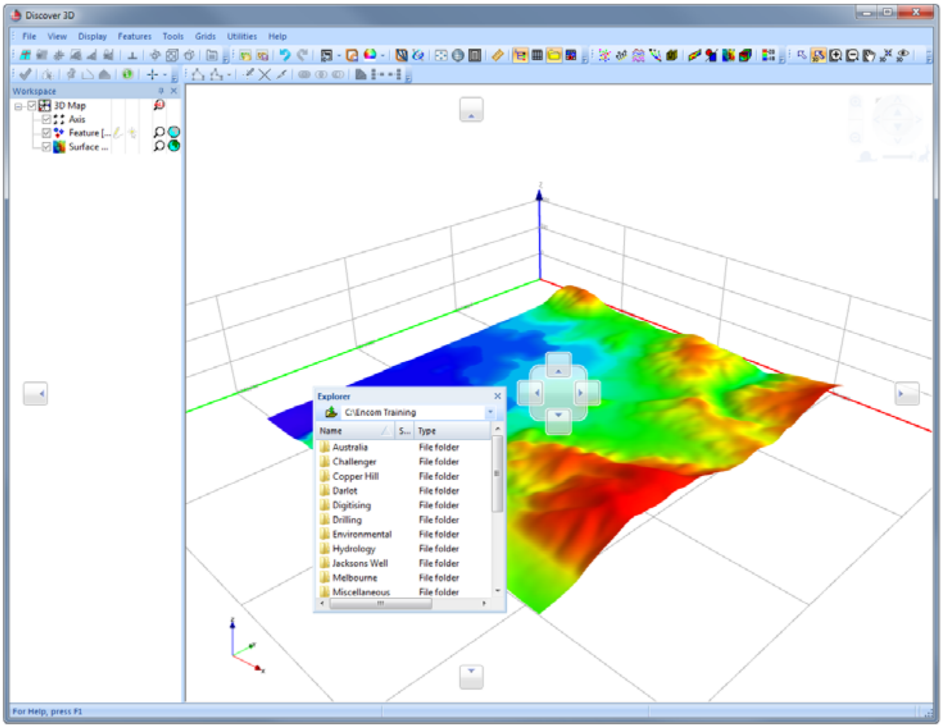
Other customisations can be found in *Customising Discover 3D*.

Docking and Undocking Toolbars and Windows

The toolbars are dockable and can be freely dragged around the Discover 3D application and displayed as either docked or floating.

The Information Windows and Workspace Tree can be displayed as docked or floating windows. To dock these windows around the 3D window click and hold down the left mouse button in the title bar region for any of the windows. When the window is moved a series of docking indicators will appear in the 3D window. To dock the window, simply hover the mouse over one of the docking indicators and release the mouse button.

Alternatively, the window can be dragged and placed into an appropriate location as a floating window. The positions of these windows will be save upon exiting Discover 3D.

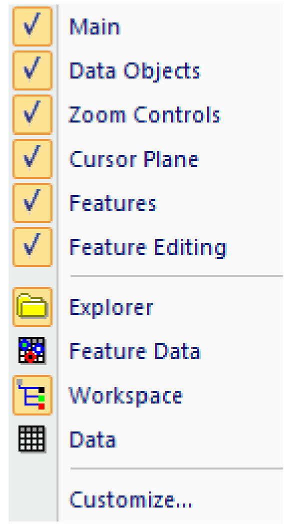


Discover 3D with window docking indicators.

Auto-hide Windows

- Located on the **Information Windows** and **Workspace Tree** is an **Auto-hide** control. Instead of displaying the windows as either floating or docked the windows can be temporarily hidden and displayed when the mouse cursor is hovered over. To enable or disable the auto-hide function, click the **Auto-hide** control (pincushion).

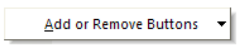
Customising Toolbars and Window Views



To control the display of Toolbars and Information Windows, right click in the toolbar region to display a list of items. Click the item to display or hide.

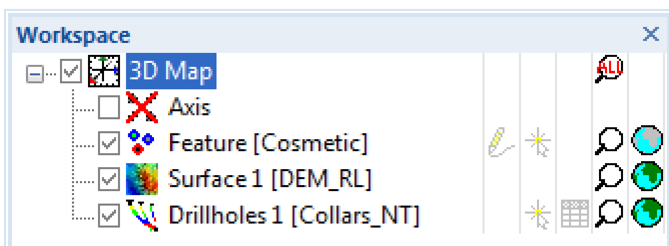
Click **Customize** on this shortcut menu to customize individual menus and toolbars (see *Customizing Toolbars*).

Located on the right of every toolbar is a dropdown control which enables the customisation of each toolbar. Left mouse click to access this option and select tools to display or hide.



Workspace Tree

The **Workspace Tree** is the primary control of data used in Discover 3D. The Workspace Tree operates by listing all objects shown in a display. These entities are listed as a hierarchy with various tree branches.



Workspace Tree with data objects

Each branch is subordinate to a higher branch, such that control properties of a higher branch overrule the same properties of a lower branch. The **Workspace Tree** grows incrementally as various objects are added or modified.

To the right of many branches are several selectable controls:

Edit – Enables objects to be added or modified in a selected *Digitizing and Managing 3D Features*. Only one Feature Database can be editable at a time. Making a feature database editable will automatically turn on the Cursor Plane for object digitization.

Select – Enables objects within the selected dataset to be selected for the purpose of querying information. For example information can be displayed for individual drillhole samples, vector linework, points or Feature Databases.

Browse – Attribute information for a selectable datasets can be dynamically viewed in the **Data Window**. Objects types which are browsable include drillholes, lines, directional vectors and points. The Cursor Plane is automatically enabled in this mode and bonded to the Browsable dataset (see Bond tool on *Cursor Plane Toolbar*).

Zoom Extents – Zoom the 3D view to the extents of the selected dataset.

Zoom All – Zoom the 3D view to the extents of all data in the 3D **Data Window**.

Projection Information – View the embedded projection of the branch's data file. For data files that do not support embedded projections (files that do not have an associated TAB file or EGB 3D image header), you can define a new projection for the objects coordinates in the branch.

There are three states of the Projection

- Projection defined – it matches the 3D Map's projection
- Projection defined – does not match the 3D Map's projection. Reprojection on the fly is active.
- Projection not defined – the branch is assumed to be in the same projection as the map window. Click to define a projection.

Each object branch in the **Workspace Tree** has a visibility check box. To turn off the visibility of an object branch in the 3D display window uncheck the visibility box. Turning off the visibility for an object branch will also turn off the visibility of any sub-branches.

To the left of each subordinate branch is a ⊞ or ⊟ control. These controls indicate lower levels exist for that branch. By clicking on these boxes you can expand or collapse the object contents.

For more information, see:

- *Working with the Workspace Tree*
- *Workspace Branch Properties*
- *3D Map Properties*

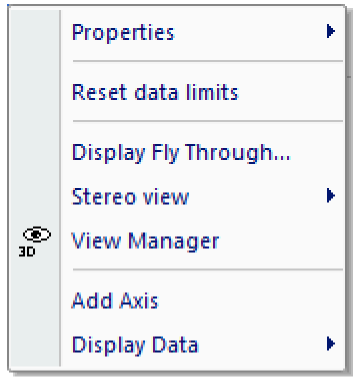
Working with the Workspace Tree

Selecting a tree branch and clicking the right mouse button displays a shortcut menu that has available operations applicable to that object type.

A complete list of all shortcut menu items for each branch is provided below.

Right click **To display shortcut menu**

3D Map



Axis

Feature

Voxel Model

Right click	To display shortcut menu
Surface	Properties Open Grid Flipper... Remove Surface Move Up Move Down
2D Vectors	Properties Remove 2D Vectors Move Up Move Down
Vectors	Properties Remove Vectors Move Up Move Down
Images	Properties Load Properties Save Properties Remove Images Move Up Move Down

Right click	To display shortcut menu
Drillholes	Properties Load Properties Save Properties Remove Drillholes Move Up Move Down
Lines	Properties Load Properties Save Properties Remove Lines Move Up Move Down
Directional Vectors	Properties Remove Directional Vectors Move Up Move Down
Points	Properties Load Properties Save Properties Remove Points Move Up Move Down

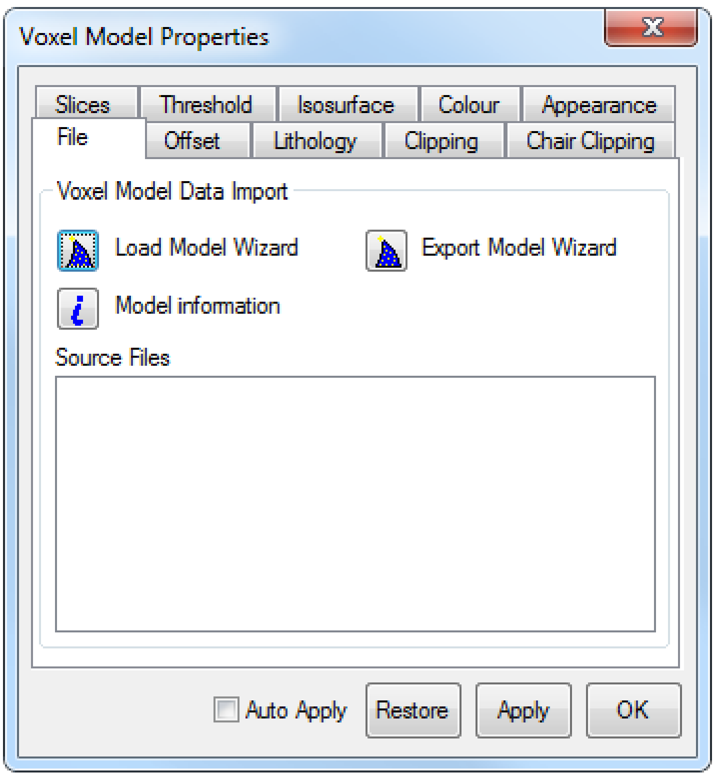
Workspace Branch Properties

The workspace branch property dialogs controls many of the options settings required for the workspace objects. Several methods are provided to display the properties of objects in the **Workspace Tree**, including:

- Right click the object in the **Workspace Tree**.
- Double click the object in the **Workspace Tree**.

- Click the **Show 3D Properties** button on the Main toolbar and select the object type.

In most cases, the **Properties** dialog has tabs displayed at the top. Each tab controls a separate aspect of the item selected.



Example Properties dialog

If the **Auto Apply** option is disabled, changes to properties are not made until the **Apply** button is clicked. This is a useful feature that enables changes in various tabs of a **Properties** dialog to be setup before they are applied (particularly for large datasets where redraw times may be significant).

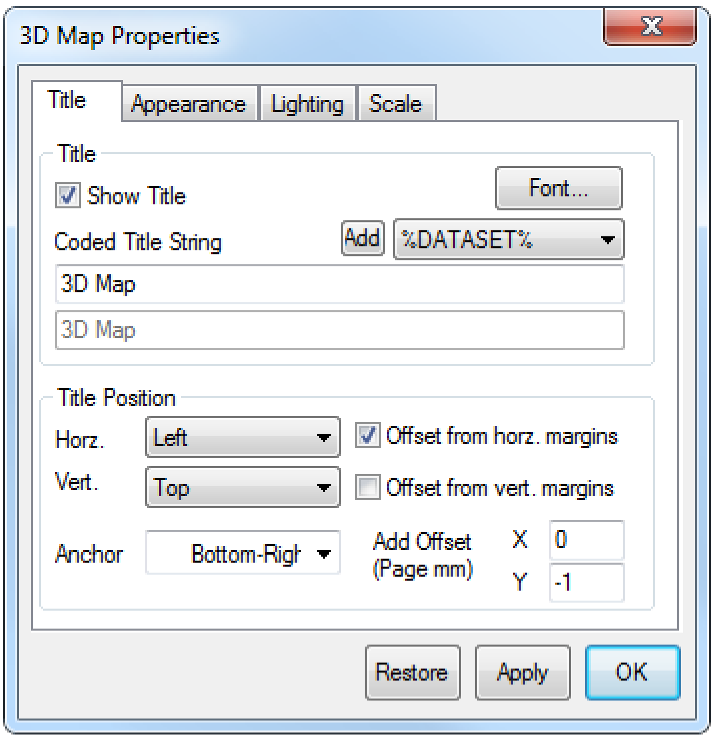
If a mistake is made while entering changes, select the **Restore** button to return the control settings to their original state. In contrast, enabling the **Auto Apply** option will apply any changes immediately. Property changes can also be applied by clicking the **OK** button, which will also close the dialog.

3D Map Properties

The options specified in the **3D Map Properties** dialog control the title, appearance, lighting and scaling of the 3D **Display Window**.

- *Title Tab*
- *Appearance Tab*
- *Lighting Tab*
- *Scale Tab*

Title Tab

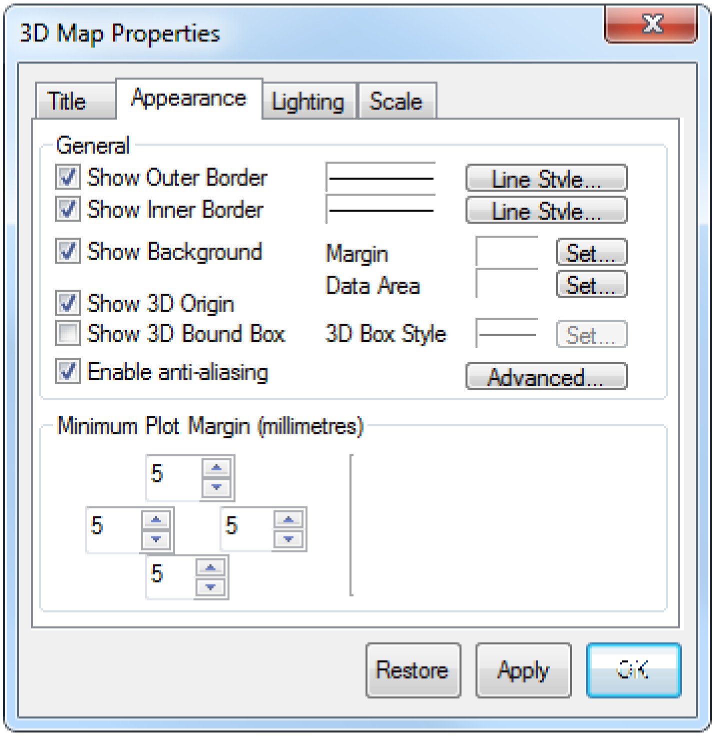


3D Map Properties Title Tab dialog

The **Title** tab of the **3D Map Properties** dialog controls the placement, content and font of the map title displayed within the **Page Layout** mode. The 3D window mode is toggled from the **View>Normal** or **View>Page Layout** menus. This can be particularly useful when generating titled images from **File>Save View As** or when printing hard-copies.

The **Coded Title String** provides variable title string syntax which automatically inserts values from the 3D dataset. Select the variable from the list and press the **Add** button to add the syntax to the title. Alternatively, a title can be manually entered into the title box.

Appearance Tab



3D Map Properties Appearance Tab dialog

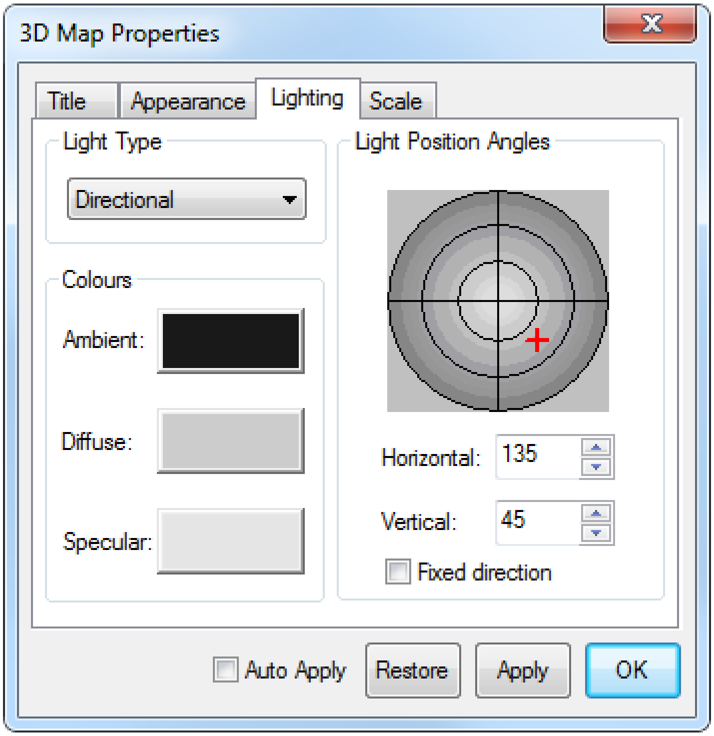
The **Appearance** tab of the **3D Map Properties** dialog controls the border styles, colours and margins for 3D displays in the **Page Layout** mode along with numerous other general appearance settings.

Options for toggling the background colour of the 3D **Display Window** are available, along with the display option for a 3D bounding data box. The bounding box displays a minimum bounding rectangle around the 3D dataset.

To smooth lines and edges of all objects in 3D, enable the anti-aliasing option. The **Advanced** button provides default colour options for vector objects in 3D.

Note The 3D window mode is toggled from the **View>Normal** or **View>Page Layout** menus.

Lighting Tab



3D Map Properties Lighting Tab dialog

The **Lighting** tab of the **3D Map Properties** dialog controls the lighting displays for the 3D **Display Window**. Two light modes exist, **Directional** and **Global Ambient**.

Directional lighting enables a choice of three lighting modes. Directional light enables the casting of shadows and can give the 3D display the appearance of depth, and highlight variability in terrain.

- **Ambient**: Select a colour for the general light or illumination of objects displayed in the 3D **Display Window**. For example, if white is selected the objects in 3D will appear very bright and illuminated; if black is selected objects in 3D will appear dark with a low brightness level, independent of the light direction.

- **Diffuse**: Select a colour which will be used to cast shadow effects on undulating objects or topography. Generally, shades of grey produce the best effects. Diffuse light gives the appearance of a unidirectional light source.

- **Specular**: Select a colour to be used for highlights in the 3D **Display Window**. Highlights are objects or surface facets perpendicular to the lighting direction.

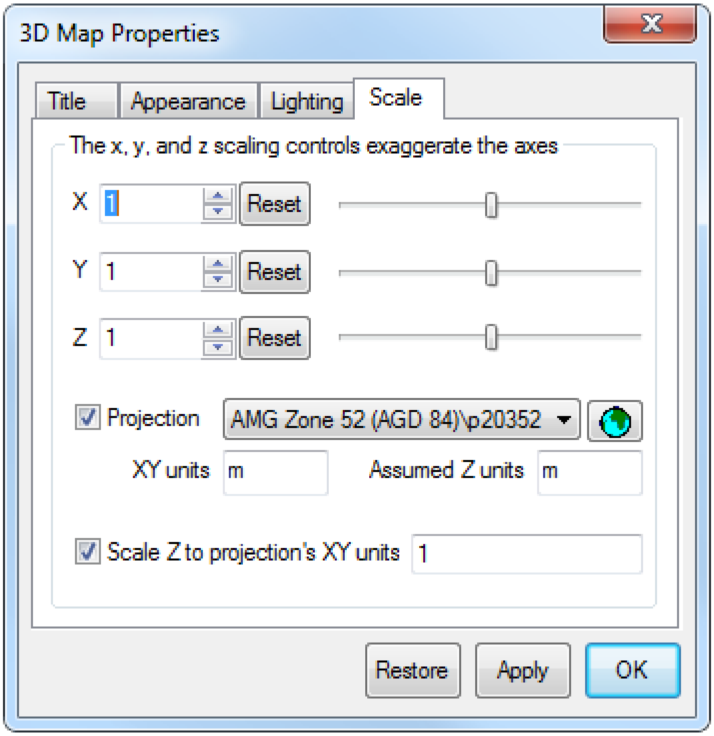
The **Light Position Angles** control contains a dynamic ellipse to define the **Horizontal** and **Vertical** positions of the directional light. The light source direction can be modified by pressing and holding down the left mouse cursor over the red cross on the ellipse and moving to an appropriate position; or by manually adjusting the **Horizontal** and **Vertical** values.

The **Fixed direction** option will fix the light source direction as specified on the ellipse independent of the position of the 3D **Display Window** axes. If the option is not selected the light position is defined by the axes position in the 3D **Display Window**.

Global Ambient lighting enables one lighting mode, and refers to the general light conditions. Global ambient light doesn't allow the generation of shadows or highlights.

- **Ambient**: Select a colour for the general light or illumination of objects displayed in the 3D **Display Window**. For example if white is selected the objects in 3D will appear very bright and illuminated; if black is selected objects in 3D will appear dark with a low brightness level.

Scale Tab



3D Map Properties Scale Tab dialog

The **Scale** tab of the **3D Map Properties** dialog allows the **X**, **Y** and **Z** axes along with the 3D objects to be scaled independently along each axis. This can be achieved by manually entering a scale factor or by adjusting the slider bar.

The **Scale Z to projection's XY units** enables automatic scaling of the Z units (as listed in **Assumed Z units**) to the **XY Units**. The XY units are automatically set from the map's **Projection**, however there is an assumption for what the Z units are. In general, these are the same as the XY units, but for some projections, such as lat/lon (which is degree units), a guess is made. If the **Assumed Z units** are incorrect, do not enable this option. Instead manually scale the Z axis.

The **Projection** list defines the coordinate system shown for all coordinates and the map axis in the 3D Map window.

Note The scaling factors will apply to all datasets within the 3D data window. To scale a dataset independent of the global scaling, use the appropriate scale options within specified object property dialog.

Projections in 3D

Discover3D supports coordinate systems defined by Mapinfo (TAB files) as well as ESRI .PRJ or Geosoft metadata.

Note Discover3D only supports the 2D coordinate system and does not recognise a vertical datum or Z shifts which may occur between different datums or ellipsoids. Only the X and Y coordinate will be reprojected.

The 3D window behaves similarly to Mapinfo Map windows. Once the first file is opened that has a known projection, the map window's coordinate system is set to this file's projection. All other files with known projections will be reprojected on-the-fly to this coordinate system. File with no known projection (such as DXF) will be displayed with the raw coordinate and assumed to be in the same coordinate system as the map window.

For certain data types, such as drillholes, points, and lines data series, you cannot override or define a projection. The same is true with images (EGB file) and Feature database layers. But all other data types, such as voxels and vectors, can override the embedded projection. Or the projection can be defined if the embedded projection is unknown.

Note that overriding the projection via the workspace tree will not reproject the file. This only defines a coordinate system for the units. It is not recommended to do this unless you are sure there is erroneous projection metadata for the file.

If known, the parameters for a coordinate system will be displayed in a readable description—if the coordinate system is a known system. A list of known systems with their descriptive name and parameters are stored in the `Encom.prj` file.

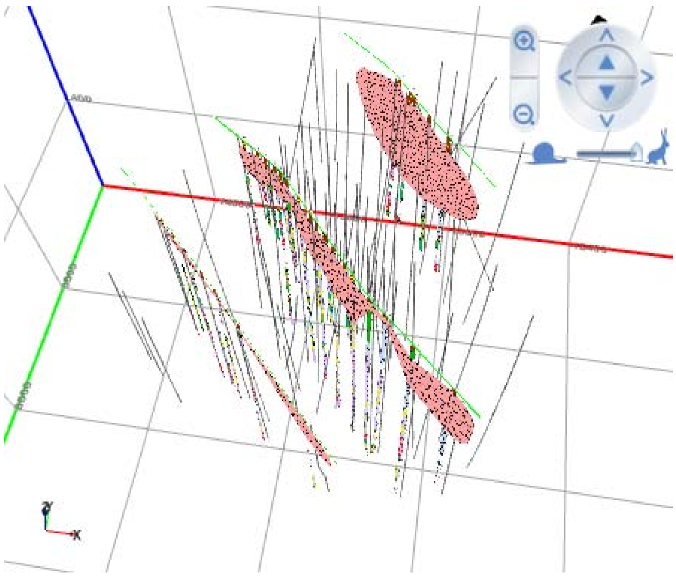
Navigating in 3D

- *3D Navigation Modes*
- *3D Navigation Controls*
- *Using the 3DConnexion SpaceNavigator™*
- *3D Display Modes*

3D Navigation Modes

There are two navigation modes, which can be selected from the Zoom Controls toolbar:

3D Navigate—This is the primary navigation control and provides the most precise control in 3D. When selected, the 3D Navigation tool is displayed in the top-right corner of the Display Window. See *3D Navigation Controls* for details.

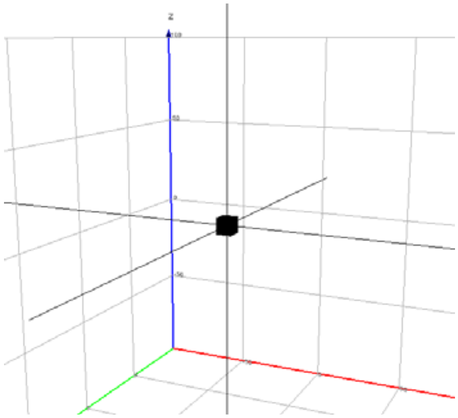


Select/Navigate—Use this mode to create, edit and interrogate objects in the 3D environment. When selected, position the cursor inside the Display Window, then hold the left mouse button down and drag the cursor to rotate pivot the view about the centre of the display. You can use the other controls on the *Cursor Plane Toolbar* (Zoom In, Zoom Out, Pan, Change View Direction, and such) in conjunction with the Select/Navigate tool.

3D Navigation Controls

To navigate in 3D, click the  **3D Navigation** button on the Zoom Controls toolbar.

All 3D navigation is orientated about the view point. The view point is the black cube at the intersection of the XYZ axis lines that appears when clicking in the Display Window.

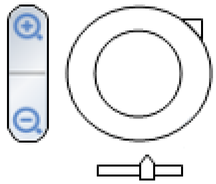
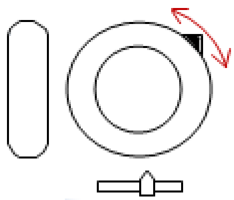


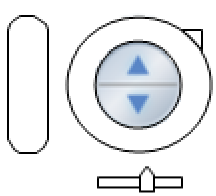
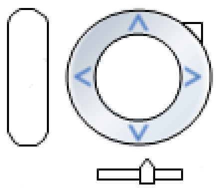
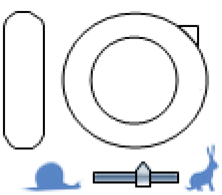
The 3D navigation view point

The **3D Navigation** tool is displayed in the top-right corner of the Display Window. This tool fades when there is no movement after approximately 5 seconds. To redisplay, move the cursor into the top-right corner. This tool controls the eye position and the view point (the rotation point). Similar (and some additional) functions can also be performed by combinations of mouse button and keyboard keys, as described below.

The 3D navigation tool is displayed in 3D Navigation mode.

The X, Y and Z coordinates of the view point and the bearing and inclination of the view direction (from the eye position) are displayed in the Status Bar.

Movement	Using the 3D Navigation Tool	Using the Mouse/Keyboard
Zoom in and out	 **Zoom**: Zooms in and out from the view point.	 **Zoom**: Hold down the right mouse button with the cursor positioned above (to zoom in) or below (to zoom out) the view point. Alternatively, roll the mouse wheel.
Rotate around view point	 **Z-rotate**: Rotates around the Z-axis.	 **Free rotate around view point**: Hold down the left mouse button. The view will rotate freely as if you are pressing on a basketball: whichever side of the view point you click, the view will rotate away in that direction.
Rotate around eye point		 **Free rotate around eye point**: Hold down the CTRL key and the left mouse button. The view rotates in the direction of the cursor about the current eye position.

Movement	Using the 3D Navigation Tool	Using the Mouse/Keyboard
Vertical pan	 **Z-pan**: Moves the view point vertically (in the Z-direction)	 **Z-pan**: Hold down the SHIFT key and the left mouse button. The view point moves along the vertical axis in the screen plane in the direction of the cursor.
Horizontal pan	 **XY-pan**: Moves the view point horizontally (in the XY-plane)	 **XY-pan**: Hold down both the left and right buttons. The view point moves within the XY-plane in the direction of the cursor.
Speed/ sensitivity	 **Sensitivity**: Use the slider control to adjust the speed of rotation, zooming, and movement. Clicking the bar selects that setting. Clicking the snail or rabbit changes the setting by one increment in either direction.	**Speed**: Speed is controlled by the distance of the cursor location from the view point: the further this distance, the faster the movement.  **Sensitivity**: The numeric keys (1-0) can be used for more precise speed control: 1 is the slowest speed and 9 and 0 are fastest.
		Refer to *3D Cursor Keyboard Shortcuts* for a complete list of mouse/keyboard controls.

The **Reset 3D View**, **View Manager** and **Change View Direction** can also be used when in 3D Navigation mode (see *Zoom Controls Toolbar*).

Using the 3DConnexion SpaceNavigator™

Discover 3D Viewer supports navigation using the 3DConnexion SpaceNavigator™ device. With this device, both the eye position and view point can be moved simultaneously. This results in easy and intuitive "fly-though" style movement.

To use a SpaceNavigator device:

1. Install the 3D Connexion driver software, and upgrade to the latest version.

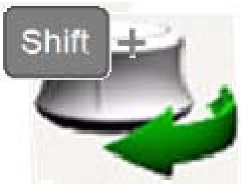
2. Connect the device via a USB port.

3. Start Discover 3D Viewer window. Select the **Tools>Options** menu option, and go to the **View** tab. Select the **Enable 3D Connexion devices** option.

4. Close and restart the 3D window.

The following SpaceNavigator™ controls are supported within Discover 3D Viewer:

Action	Device	Movement
Push/Pull		Move eye and view points forwards and backwards.

Action	Device	Movement
Slide		Move eye and view points to the left and right.
Tip		Rotate eye point downwards and upwards.
Spin		Rotate eye point around static view point.
CTRL **+ Push/Pull**		Zoom in and out (move the eye point closer to and farther from the view point.
SHIFT **+ Spin**		Rotate the eye position
Reset	LEFT	Press the left button to reset the 3D view.
Settings	RIGHT	Press the right button to display the device settings

3D Display Modes

The 3D **Display Window** in Discover 3D can be configured to operate using several view modes. The following view modes are available:

- *Normal View Mode*

- *Page Layout Mode*
- *Full Screen Mode*
- *3D Stereo Visualisation*

Normal View Mode

View>Normal

In normal mode, the 3D **Display Window** is maximised to fill the extents of the view area and provide optimal display of 3D objects.

Page Layout Mode

View>Page Layout

Page Layout mode enables you to see how objects are positioned on a printed page. This view can be used to modify the size and positioning of the 3D frame border, page margins and colour legend objects prior to printing. The size and orientation of the default page is defined by the printer settings specified under the **File>Page Setup** option.

Full Screen Mode

Discover 3D can be switched into full screen mode, where the 3D Map window will display full screen on the monitor that the Discover 3D window is currently positioned on.

To switch to full screen mode:

- Select **View>Full Screen** or select the toolbar button.

In this mode, the toolbars, status bar and workspace tree will be hidden. However by default the menu bar will remain. To re-enable controls such as the workspace tree, while in full screen mode, right-click on the menu toolbar and select the option.

To enable the 3D Navigation mode and button pad, Right click in the fullscreen map window and select the option.

To exit full screen mode:

- Locate the **Close Full Screen** floating toolbar button, or press ESC on the keyboard.

Additional options for fullscreen mode are available from **Tools>Options>Fullscreen**.

3D Stereo Visualisation

Discover 3D is capable of displaying full colour 3D stereo projection systems to create a semi immersive 3D visualization environment. The use of full colour 3D stereo visualisation is quickly becoming the preferred way for geoscientists to communicate and collaborate with each other. It allows the user to analyses and detect very subtle geometric relationships that are often overlooked in conventional 3D views. It is also a powerful presentation tool to display fly-through animations.

Discover3D supports three methods of stereo visualisation:

- *HDMI 3D*
- *Dual Projectors*
- *Anaglyph Mode*

HDMI 3D

This method will output the 3D map window to an external display connected by HDMI. The HDMI standards later than 1.4a support side-by-side viewing of 3D stereo. HDMI 3D ready devices include TVs, home theatre projectors and PC monitors. These generally are provided with Active shutter glass.

To enable 3D stereo on an external HDMI monitor:

1. Connect a HDMI 3D ready device to your computer, and extend the desktop to this.

Note Do not duplicate or mirror the desktop to the 3D display.

2. Select **View>3D Stereo** or right click on the 3D Map branch in the workspace tree, and select **Stereo View**.

3. In the Stereo Display dialog select the **HDMI 3D** mode.

4. Select the 3D ready HDMI device in the **3D Display** list.

5. Click **OK**.

6. The 3D Map will be duplicated as a side-by-side 3D stereo pair on the 3D display.

7. On your devices remote control, enable the 3D Mode and select the corresponding side-by-side mode, and wear the device's 3D glasses. Consult the device's documentation for help on doing this.

8. On your primary monitor you will be able to navigate and adjust the 3D display. It will also open a **Stereo Control** dialog. This allows adjustment of the separation (depth of view) and convergence (focus) angles. Adjust these as necessary to achieve a good display.

Dual Projectors

This method outputs the stereo pair to twoseparate external display. These will be two projectors in a specialized 3D projection cage. The cage aligns the two projectors to overlay the output frames exactly. They generally also have polarized filters on the projectors, and use passive polarized glasses. This is similar to what is used in commercial movie cinemas.

To enable 3D stereo on an dual 3D projector system:

1. Connect the two devices device to your computer, and extend the desktop to these.

Note Do not duplicate or mirror the desktop to the 3D display.

---

2. Select **View>3D Stereo** or right click on the 3D Map branch in the workspace tree, and select **Stereo View**.

3. In the Stereo Display dialog select the **Dual Projector** mode.

4. Select the two projectors in the two display lists.

5. Click **OK**.

6. The 3D Map will be duplicated as a side-by-side 3D stereo pair on the projectors.

7. Wear the projector polarized glasses to view in 3D.

8. On your primary monitor you will be able to navigate and adjust the 3D display. It will also open a **Stereo Control** dialog. This allows adjustment of the separation (depth of view) and convergence (focus) angles. Adjust these as necessary to achieve a good display.

Anaglyph Mode

This mode creates a pair of composite red and blue images in the 3D Display Window, which are slightly offset from each other. The composite anaglyph view is then visualised through a pair of red-blue glasses, which filter out the red image in the red eye and the blue image in the blue eye. The main disadvantage of anaglyph mode is that it does not allow the image to be viewed in true colour.

Cursor Plane

The Cursor plane is a user controllable and customizable plane in the 3D view, which has two main functions:

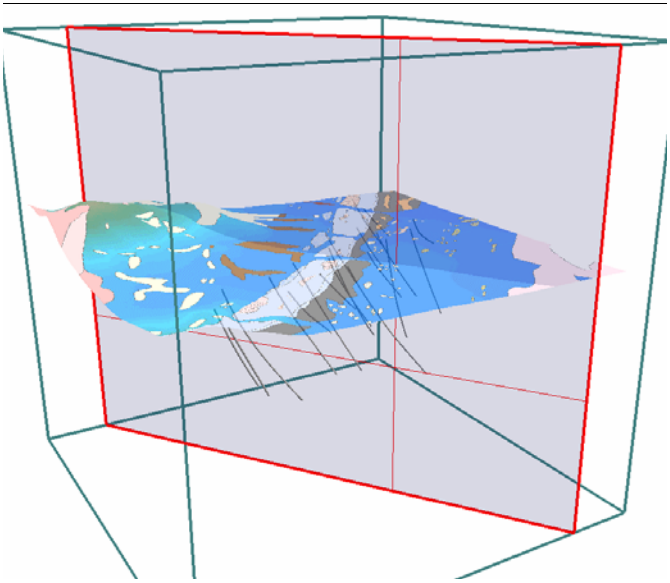
Note The Cursor Plane is not visible in Bond mode.

Digitizing

The Cursor plane operates as a drawing plane upon which Feature Objects are digitized in the 3D environment. Making a Feature Dataset editable automatically enables the Cursor Plane. For example, geological boundaries could be digitized from cross sections and subsequently formed in a solid object using the 3D Solid Generator.

Clipping

The Cursor plane can be used as a clipping plane to hide or obscure object data. This can be useful in complex datasets where a mass of drillholes obscures the data of interest. Alternatively, only a user-specified envelope or slice of data about the plane can be displayed; this is very useful when digitizing along sections, ensuring that only data within the specified envelope used in the interpretation.



Cursor plane (red outline) with bounding box (green box) and, 3D Cursor crosshairs (light red).

For more information on controlling and using the cursor plane, see:

- *Controlling the Cursor Plane Display*
- *Controlling the Cursor Plane Position and Orientation*
- *Navigating on the Cursor Plane*
- *Changing Cursor Plane Properties*

Controlling the Cursor Plane Display

The Cursor plane toolbar contains tools required for the operation of the Cursor plane.

Press the **Show Cursor Plane** button to display the default **Cursor Plane** location surrounded by a bounding Focus Box.

Press the **Cursor Plane Properties** button to modify the appearance of the Cursor Plane. For more information, see *Appearance* section of the Cursor Plane Properties dialog.

Press the **Plane clipping** button to hide all 3D data objects in front of the Cursor plane. This is useful when a slice view is required of the 3D datasets when digitizing.

Press the **Defined clipping** button to enable the supplementary clipping of a 3D dataset as defined by the current position of the Cursor plane. This option enables the current position of the Cursor plane to temporarily clip the 3D dataset whilst being able to move the Cursor plane to another position. This is useful when digitizing through a complex geometric object, or to reveal obscured features.

Press the **Bond** button to bond the Cursor plane to a selected georeferenced image or dataset (e.g. drillholes) in the Workspace Tree. This feature is useful for aligning the Cursor plane with images georeferenced in 3D for digitizing. It is also auto-enabled when selecting data in Spreadsheet mode—for interrogating drillholes dynamically, for example (see Browse tool in *Workspace Tree*).

Controlling the Cursor Plane Position and Orientation

Press the **Cursor Plane Properties** button to modify the position and orientation of the Cursor Plane. For more information, see *Plane* section of the Cursor Plane Properties dialog.

Press the **Lock Cursor plane** button to disable all Cursor plane movement controlled by the keyboard shortcuts, preventing the Cursor plane from being accidentally moved during digitization. This feature locks the Cursor plane origin X, Y and Z coordinate.

Press the **Cursor plane orientation** button to toggle the Cursor plane orientation around the X, Y and Z axes.

Press the **Perpendicular** button to orientate the view direction perpendicular to the Cursor plane. This is very useful when digitizing interpretations on adjacent sections, particular when used in tandem with the Orthographic View mode and Envelope or Slice clipping mode.

 Shrink

 Enlarge

 Fit

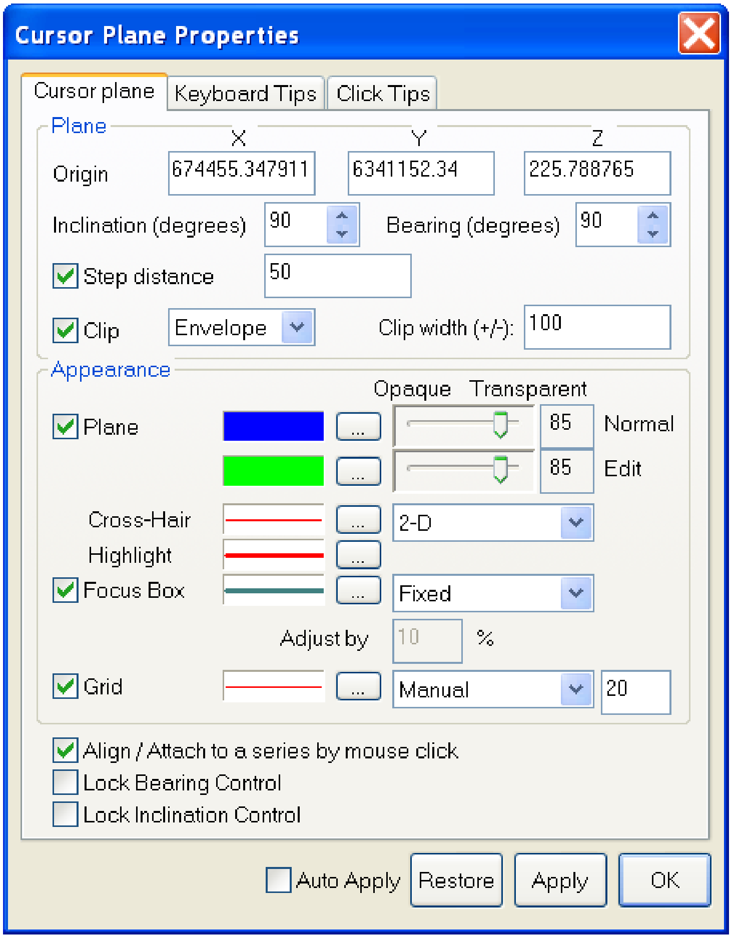
Press the **Shrink, Enlarge and Fit Cursor plane** button to Shrink, Enlarge and Fit Cursor plane controls resize.

Navigating on the Cursor Plane

3D navigation and selection on the Cursor plane can be achieved using the **Select/Navigate** control. When this control is enabled a set of crosshairs will appear on the Cursor plane, the crosshairs assist in the precise selection and digitisation on the Cursor plane. X, Y & Z coordinates for the crosshair position displayed are on the **Status Bar**.

Up Down	The up and down keyboard arrow keys control the inclination of the Cursor plane. This can change the X, Y and Z coordinates of the plane, as the rotation is applied about the centre of the bounding box, not the current centre point of the plane.
Left Right	The left and right keyboard arrow keys control the bearing of the Cursor plane. This can change the X, Y and Z coordinates of the plane, as the rotation is applied about the centre of the bounding box, not the current centre point of the plane.
Pg Up Pg Dn	The page up and page down keyboard keys move the Cursor plane laterally. The Cursor plane is kept parallel to but shifted left or right (up or down) from the current position, maintaining a fixed inclination and bearing.

Changing Cursor Plane Properties



Cursor Plane Properties dialog.

The Cursor plane properties dialog is divided into two sections:

- *Plane*
- *Appearance*

Plane

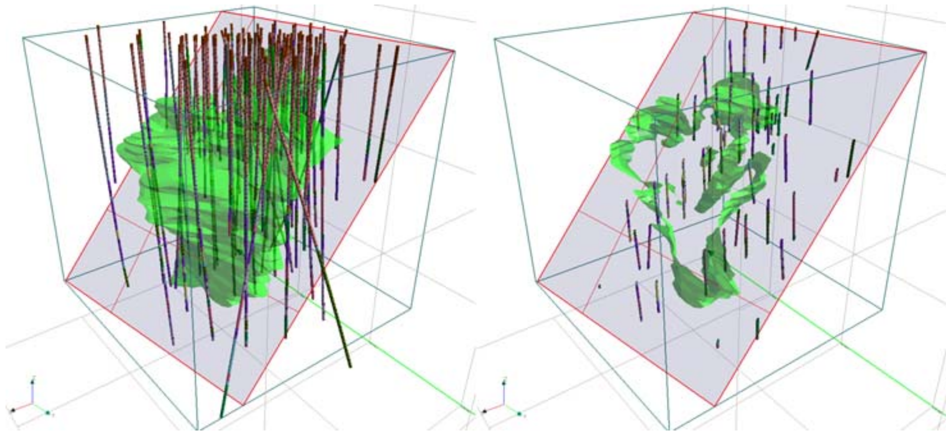
The 3D position of the Cursor plane can be set manually via the **Origin**, **Inclination** and **Bearing** options.

The cursor plane **Step Distance** option controls the distance it is shifted with each PAGE UP and PAGE DOWN key press. This is an excellent way to ensure feature object digitization occurs at a uniform spacing (e.g. 100m intervals).

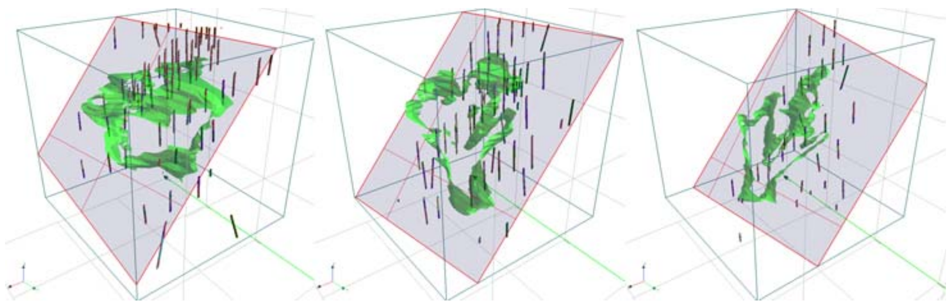
The **Clip** option refines the operation of the **Plane clipping** tool. Clipping is dynamic: if the cursor is moved (e.g. dip and azimuth, or lateral movement), the clipping region will follow.

Six clipping modes are available:

- **None** - No clipping is applied, replicates having the **Plane clipping** option disabled.
- **Nearest** - Hides all data in front of the Cursor plane, applies the clip dynamically when the **Display Window** is rotated.
- **Positive** - Hides all data in front of the Cursor plane from the current view point. Data clip is maintained even when rotating the **Display Window**.
- **Negative** - Hides all data behind the Cursor plane from the current view point. Data clip is maintained even when rotating the **Display Window**.
- **Envelope** - Displays data within a defined clip envelope from the Cursor plane. For example, setting a a +/-25 m envelope width will display a 50 m thick envelope (total) of data centred on the Cursor plane or 25 m either side of the Cursor plane..
- **Slice** - Displays data within a defined clip width behind the Cursor plane. For example, setting a 25m slice will display data up to 25m behind the Cursor plane.



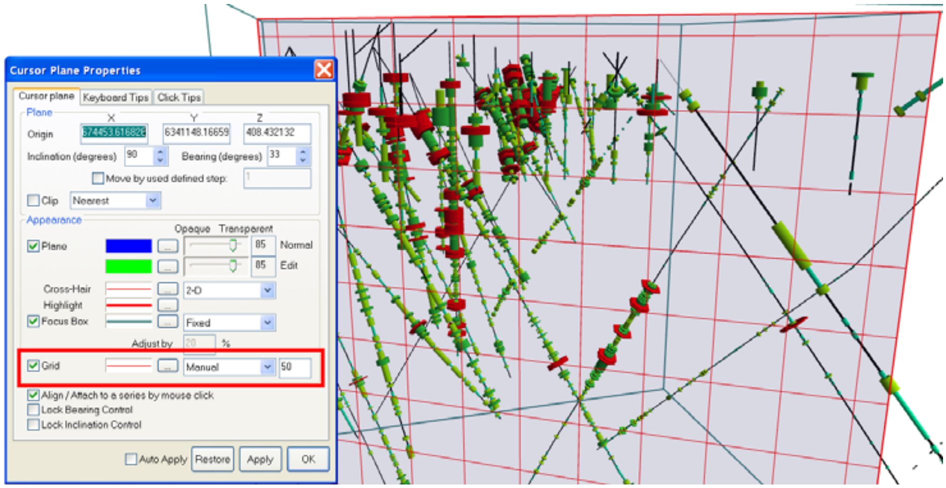
A skarn model (vector) and drillhole dataset before (left) and after application of a 40m wide clipping envelope centred on an inclined Cursor plane.



Dynamic clipping by using the Page Up/Down keys to shift the Cursor plane laterally whilst a clipping envelope is enabled.

Appearance

The **Appearance** section provides a range of cosmetic controls for the Cursor plane, cross hair and focus box colouring, size, style and transparency.



The **Grid** option (highlighted in the above image) allows the user to visualise a continuous square grid across the cursor plane surface, with a line spacing as set next next to the Manual option (dialog below has 50 map units set). This can be a powerful aid when interpreting/digitising to help visualise distances/sizes.

Disabling the **Align/Attach to a series by mouse click** option prevents the Cursor plane bonding to an image. This is useful when digitizing if a background image is behind an intended feature node.

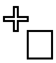
The **Lock Bearing** and **Inclination Control** options are provided to prevent the user inadvertently altering these parameters via the keyboard arrows during feature digitization. When a lock is enabled, the appropriate buttons will have no effect until the lock is disabled.

The **Keyboard Tips** and **Click Tips** tabs under the **Cursor Plane Properties** dialog provide a listing a keyboard shortcuts and mouse button combinations for Cursor plane control and feature editing. Refer to *3D Cursor Keyboard Shortcuts* for a complete list of shortcuts.

Floating Colour Bar

The Floating Colour Bar button enables a Colour Scale to be created for any open drillhole project, grid surface or voxel model. This makes it easy for the user to see the corresponding value for a particular grid colour (eg. DEM or magnetic susceptibility) or the corresponding lithology for a particular colour pattern in a drillhole series. Floating Colour Bars can be viewed in both and are particularly useful in 3D hard copy or digital Output.

Display>Floating Colour Bar – Add a Floating Colour Bar or data legend which can be link to surface, voxel or drillhole data types.



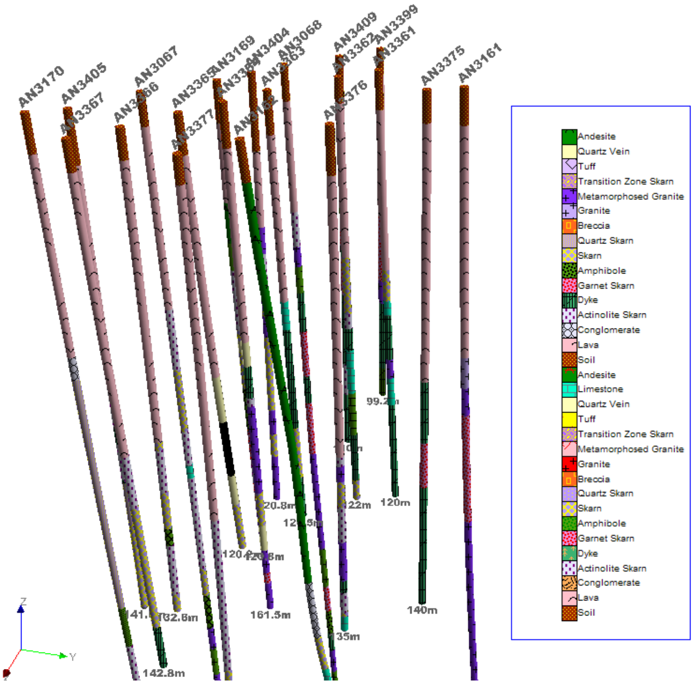
Activating the Floating Colour Bar button changes the cursor to the colour bar creation mode. This mode enables the position and size/shape of the Floating Colour Bar to be determined on the screen.

To modify the Floating Colour Bar single mouse click on the blue bounding box with the **Select/Navigate** tool. Click on a bounding box corner to resize the box or click and drag on a side wall of the bounding box to move the Floating Colour Bar to a new location.

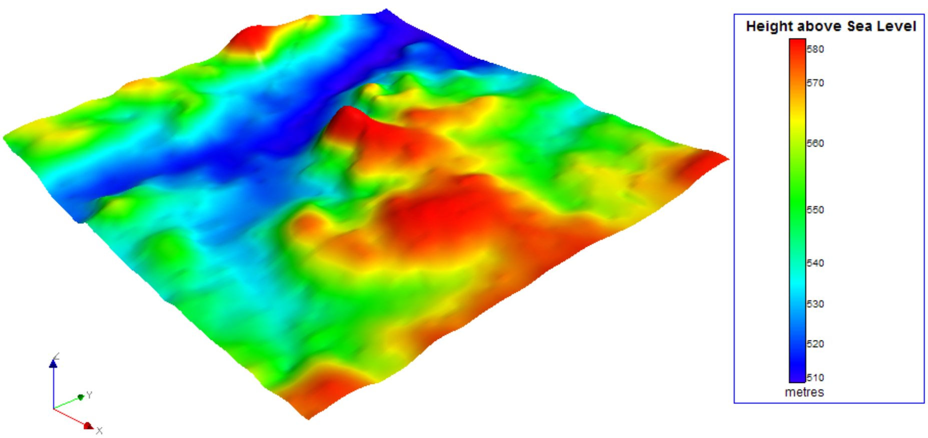
Once a Floating Colour Bar is created a corresponding branch is added to the **Workspace Tree**, with a Colour Legend sub-branch. A Floating Colour Bar can be deleted by highlighting this branch, and pressing the keyboard **Delete** key.

The **Connection** tab of the **Colour Legend Properties** dialog enables the selection of the source dataset, for example a drillhole project, voxel model or grid surface. Only one bolded dataset can be selected for each Floating Colour Bar. The remaining dialog tabs contain controls to apply and modify the **Stretch** (eg linear or non-linear), **Appearance**, **Title**, **Divisions** and **Ticks** of the colour axis.

The **Title** tab of the **Colour Legend Properties** dialog enables a Title to be added (via a user text string) and positioned for the entire Floating Colour Bar (note axis specific Title option in the **Colour Legend Properties** dialog).



Example of a Floating Colour Bar linked to a drillhole project colour modulated by geology. The displayed colour legend contains an entry for each rock type.

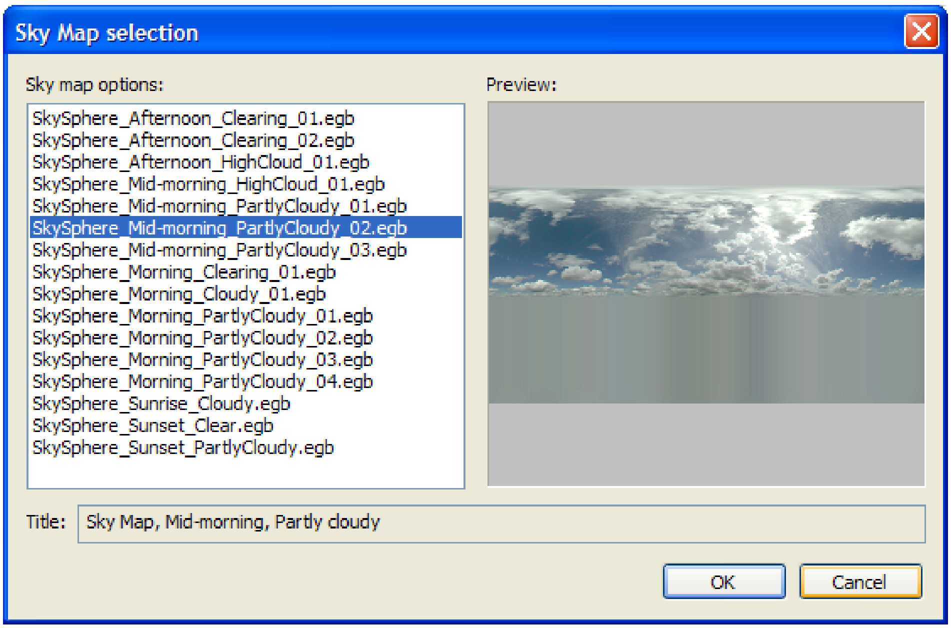


Example of a Floating Colour Bar showing elevation values linked to a DEM grid surface.

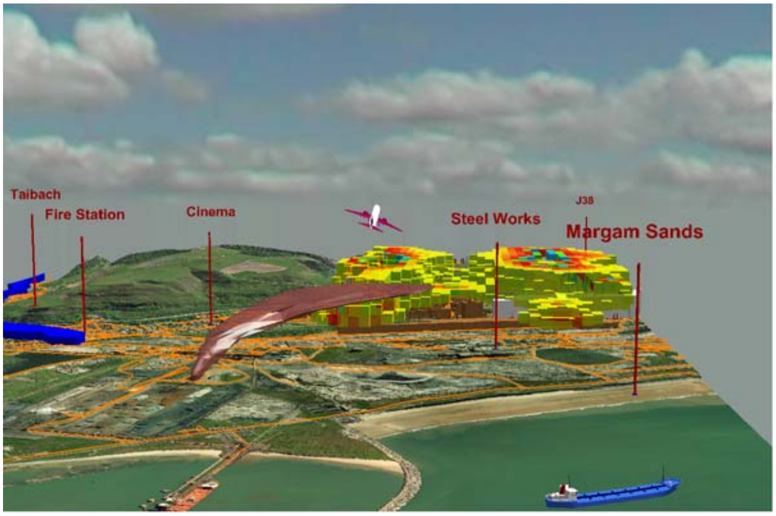
Sky Map

The Sky Map option (on the **Insert** menu) allows you to add a photo-realistic sky view backdrop to your 3D environment. This can be particularly valuable for producing impressive image/video output.

Selecting the **Sky Map** option from the Insert menu will prompt the user to select from the available sky map images (located in the `C:\Program Files\Encom\Common\SKYMAP` directory). Each image can be previewed within the selection dialog. Press OK to display the selected image. A Sky Map branch will be added to the Workspace Tree: its visibility can be toggled, and it can be either reordered (eg multiple Sky Maps listed) or deleted by accessing the right-click pop-up menu.



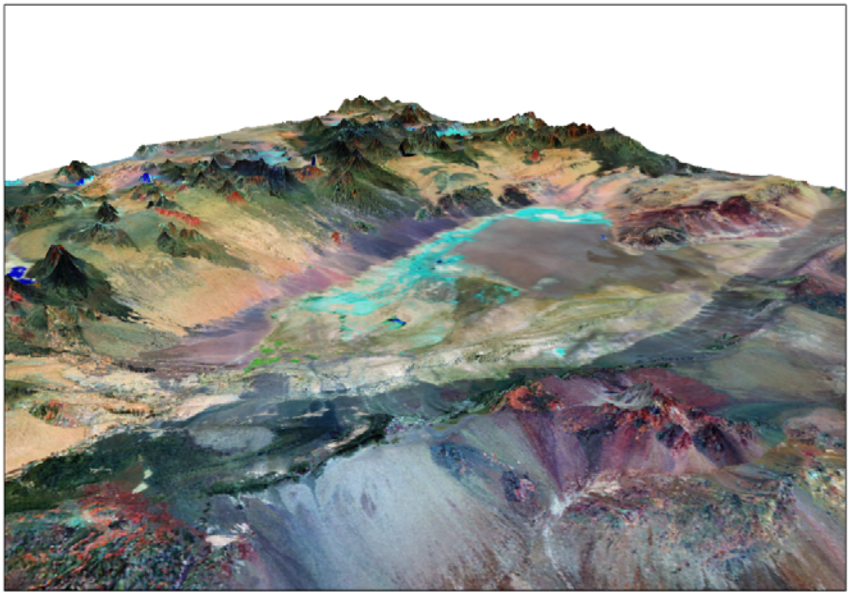
Example Sky Map to select for display.



Example of SkyMap background in complex 3D cityscape view.

7 Working with Raster Images in 3D

Bitmaps are simple images usually derived from a screen capture of a view or display. The view may be derived from Encom Discover or it may come from some other source, for example a scanned geological section or a scanned geophysical pseudosection such as IP or resistivity.



If an image is to be used in a three dimensional display in Discover 3D, the image is required to be accompanied by an *Encom Georeferenced Bitmap (EGB) Format* file . This small ASCII file defines the image source file by name, its type and its corner locations. The bitmaps formats available for display include BMP, PNG, JPG, TIFF, GIF and Windows Metafiles (.WMF).

A number of methods are available to create and display georeferenced bitmap images in Discover 3D:

- *Converting a Map Window into a 3D Image*.

  Organise multiple data tables in a mapper window in MapInfo, including vector, raster images, grid and/or labels, and automatically transfer an image of this mapper to Discover 3D using the **Discover 3D>View Map in 3D** menu option.

- *Draping Images Over a Grid*.

  If you have open in MapInfo a high resolution image (e.g. airphoto) and a gridded surface (e.g. topographic grid) of the same area, use the **Discover 3D>3D Utilities>Overlay Image on Grid** tool to create an .EGB header file linked directly to the source image file, thus preserving the image resolution.

- Add an existing georeferenced bitmap image to Discover 3D.

  Previously created EGB files can be opened directly from within Discover 3D, either by:

  - Dragging and dropping the .EGB file into the 3D window.
  - Using the **Display>Located Image** menu tool in Discover 3D.
  - Using the **Display Located Image** button in Discover 3D.

- Georeference an image from within Discover 3D.

  Vertical orientated images (such as scanned geophysical or historical cross sections) can be 3D georeferenced from directly within Discover 3D using the *Using the Georeferenced Image File Creation Wizard* to define the origin and location of one or more raster images.

- Georeference multiple vertical sections using MapInfo polylines.

  The **Multi Section Creator** (see *Displaying Other Types of Sections*) allows multiple vertical images (such as scanned long section from a 250K fact geology map) to be georeferenced. It requires a MapInfo table of polylines indicating these images spatial extents for georeferencing.

Encom Georeferenced Bitmap (EGB) Format

The .EGB file locates and references the bitmap. Below is an example of an .EGB format file.

```
GeoreferenceImage Begin
Comments = "Example Bitmap Section derived from Discover - MapInfo"
Version = "1.0"
Image = "Section2Map.bmp"
ImageFormat = "Bitmap"
CoordinateSpace Begin
Projection = "Non-Earth (meters)"
Datum = ""
Units = "m"
CoordinateSpace End
Registration Begin
TopLeft = 520814.36, 6302870.36, 340.418
TopRight = 522014.09, 6303861.59, 340.418
BottomLeft = 520814.36, 6302870.36, -597.606
BottomRight = 522014.09, 6303861.59, -597.606
Registration End
GeoreferenceImage End
```

The **CoordinateSpace** and **Datum** parameters are identical to those used within ER Mapper and can accommodate a wide range of projections.

Converting a Map Window into a 3D Image

Discover mapper windows (containing raster imagery, vector drawings, interpretative sections, drillhole sections or drillhole logs) can all be displayed in Discover 3D as georeferenced or located bitmap images. The best method of transferring a 2D display to 3D is to use the **Discover 3D** menu options.

The primary tool is the **View Map in 3D** tool (see *Displaying Map Window Views as 3D Images*). All data sets visible in the selected map window (e.g. grids, images, vector data, labels) will be incorporated into the 3D image. The image's RL can be assigned manually, or it can be 'draped' over a gridded surface (e.g. a DEM grid).

Additionally, the *Viewing Drillhole and Trench Sections* and *Viewing Drillhole Logs* options on the Discover 3D menu allow drillhole cross sections and logs displays to be converted into 3D images .

These options create a .PNG image of the source mapper window with an accompanying *Encom Georeferenced Bitmap (EGB) Format* file into either a temporary or permanent directory (enable the **Save Permanently** option in the appropriate dialog).

Once an image is displayed within Discover 3D, various properties such as transparency can be applied.

Maximising Image Resolution

The quality of the output 3D georeferenced image can be improved via:

- Utilising a larger MapInfo mapper window (but do not maximise it).
- Ensuring that your MapInfo mapper window is zoomed to the extents of the required data.
- Under **Discover 3D>Options**, increase the **Create Raster Images at …Screen Resolution** factor set in the **Display** tab of the dialog. This will increase the output image resolution. However it will also result in a larger output PNG image file size; displaying multiple high resolution images within Discover 3D may reduce the redraw speed and efficiency.
- If your input dataset in the map window is already an image (e.g. an air photo), and it will be draped over a grid (and no other data is involved), perhaps use the *Image on Grid Creator Tool*. This will create a .EGB header file referencing directly to the source image, thus preserving the source image's resolution.

Opening an Existing Georeferenced Bitmap Image

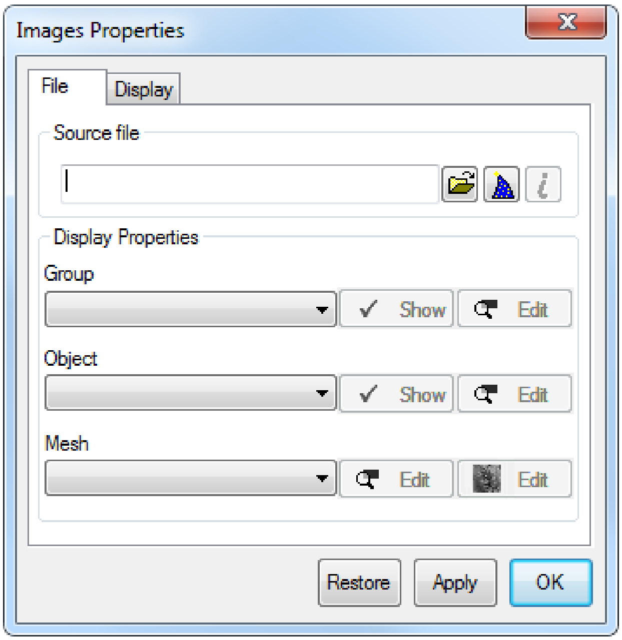
Previously created .EGB files (for instance, created in a previous session of Discover 3D or using utilities such as the *Multi Section Creator Tool* or *Object Manager Tool*) can be opened from directly within Discover 3D using any of the following options:

- Dragging the .EGB file from File Explorer or the **Explorer** tab of the Information window and dropping it into the 3D window.
- Using the **Display>Image** menu option to add an **Images branch** to the Workspace Tree. Within the File tab of the Images Properties dialog, use the Browse button to locate the target .EGB file.

- Using the **Display Located Image** button to add an **Images branch** to the Workspace Tree. Within the File tab of the Images Properties dialog, use the Browse button to locate the target .EGB file.

Using the Georeferenced Image File Creation Wizard

Vertically orientated images, such as scanned cross-sections from open file reports are other applications, can be displayed in Discover 3D using the **Georeferencing Wizard** button (shown left), available in the **Images Properties** dialog.



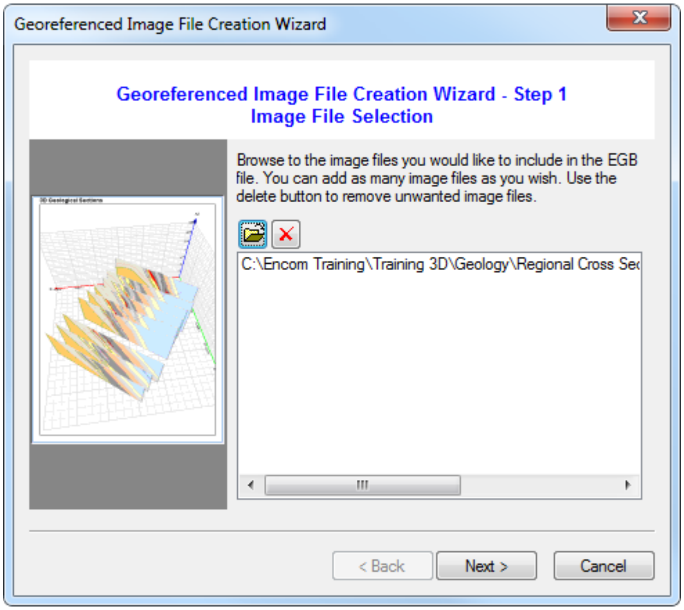
Images Properties dialog with the Georeferencing Wizard button highlighted

To georeference one or more vertical, planar images:

1. Create an **Image** branch in the Workspace Tree within Discover 3D by clicking the **Display Located Image** button on the **Data Objects** toolbar or the **Display>Image** item from the menu options.

2. Display the properties dialog from the created Image branch in the Workspace Tree and select the **Georeferencing Wizard** button.

3. Select the **Georeferencing Wizard** button and select the files required to be defined in an .EGB file. Note that a single .EGB file can specify the path, filename, location and type of any number of bitmap files. Choose the required files and add them to the screen as shown below:



Georeferencing Wizard screen 1

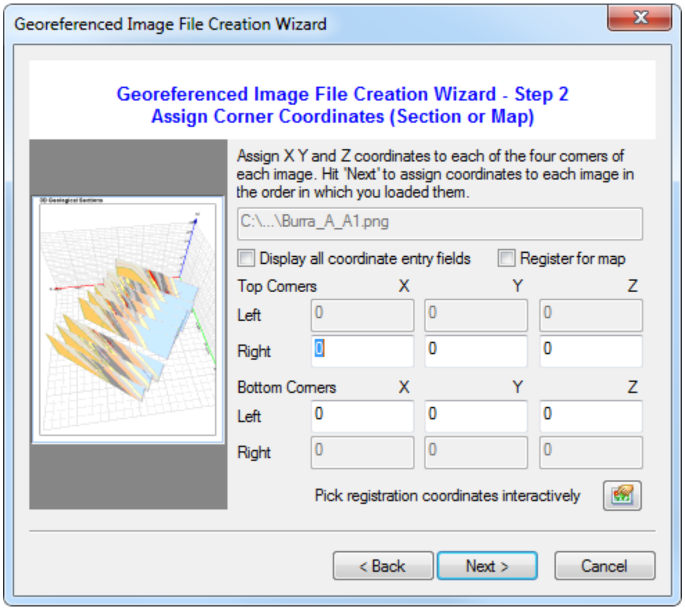
4. With the required image files selected in the list on Screen 1, click the **Next** button to access Screen 2. To pick your image registration points interactively (recommended), including using registration points inside the image boundary, select the interactive button at the bottom right of the dialog, which opens the Image registration dialog (see *Interactive Image Registration*).

   - To register your image as a vertical section/plane, enter the top right and bottom left corner XYZ coordinates.

   - To register your image as a horizontal non-rotated map or plan, enable the **Register for map** option, and enter the top right and bottom left corner XYZ coordinates (the Z values should be identical).

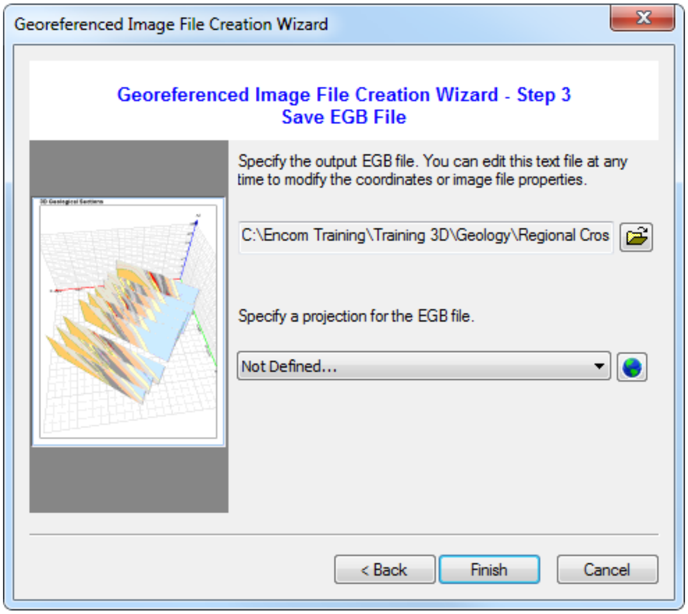
   - To register your image as as an inclined plane (e.g. inclined section), enable the **Display all coordinates entry fields** option, and enter all four corner XYZ coordinates.

If multiple images were selected in Step 1, click the **Next** button when the first image has been specified. The Step 2 screen will cycle through each of the selected bitmaps. To assist in defining their location, the corner coordinates are retained from the previous bitmap entries.



Screen 2 of the Georeferencing Wizard

5. Screen 3 of the **Georeferencing Wizard** requests an output .EGB file and path name as well as defining a projection for the output. You can use the **Open File** button to navigate to a suitable location and name the file. When completed, click the **Finish** button to create the .EGB file.



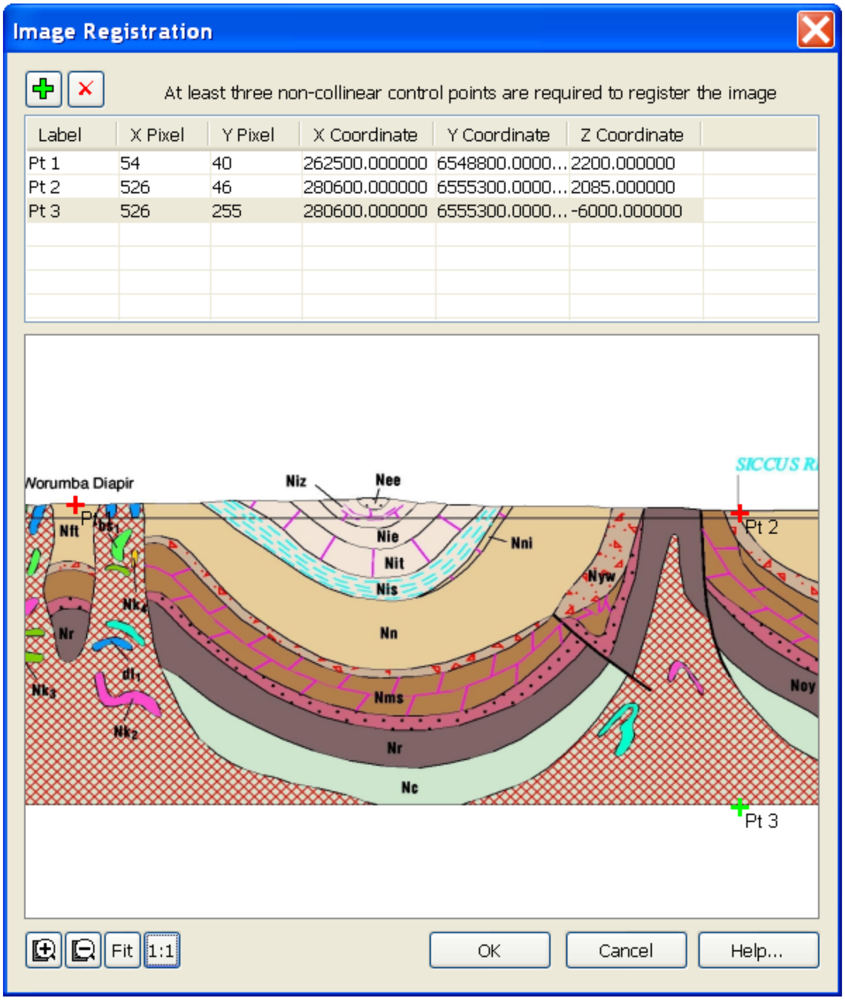
Output the .EGB file for use in a 3D display

6. After the .EGB file has been created, return to the File tab of the **Images Properties** dialog and use the Browse button to find and display the new file.

Interactive Image Registration

Interactive image registration in Discover 3D is very similar to the image registration process in MapInfo Professional:

- Allows control points anywhere on the image to be specified, unlike the other methods available, which can only use the image corner points
- Requires at least 3 control points to be specified, which must not lie on a straight line.
- For each control point, appropriate real world X, Y and Z coordinates must be specified

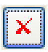


Interactive Image registration Dialog (Data courtesy of PIRSA).

To register an image interactively

1. Pan and zoom to the first control point on the image. Zoom, Fit to image and 1:1 view buttons are available at the bottom left of the dialog to help control the image preview window. The mouse wheel also functions as a zoom control. When not in Add control point mode, left-clicking and dragging the image will pan the image view

2. Press the **Add control point** button at the top of the dialog
3. Click on the image: a red cross with the control point name will be added, as well as a new row in the data sheet at the top of the dialog
4. Type in the real world X,Y & Z coordinates in the appropriate columns in the data sheet (double-click in the target cell to edit).
5. Repeat steps 1-4 for each additional control point



6. Incorrectly placed or unnecessary control points can be deleted by highlighting the appropriate data sheet row, and pressing the **Delete** button
7. When all control points have been added, press the OK button to return to the main dialog

Directly Importing Data as EGB Images

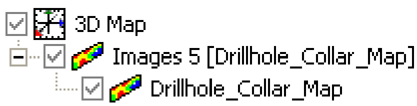
Discover can directly load the following data formats using the browse button in the Image Properties dialog:

3D DXF vector (.DXF)	Geosoft grid (.GRD)
3DS vector (.3DS)	GeoTIFF (.TIF .TIFF)
Arc ASCII grid (.ASC)	JPEG (.JPG)
Arc binary grid (.ADF)	Landmark grid (.GRD)
ASEG GXF grid (.GXF)	MapInfo grid (.MIG)
BIL (.BIL)	Minex grid (.XYZ)
Compuserve GIF (.GIF)	ModelVision TKM (.TKM)
Datamine vector (.DM)	Portable network graphics (.PNG)
Encom georeferenced image (.EGB)	Surfer ASCII grid (.GRD)
Encom grid (.GRD)	Surfer binary grid (.GRD)
ERMapper ECW (.ECW)	TAB raster (.TAB)
ERMapper algorithm (.ALG)	TIFF (.TIF)

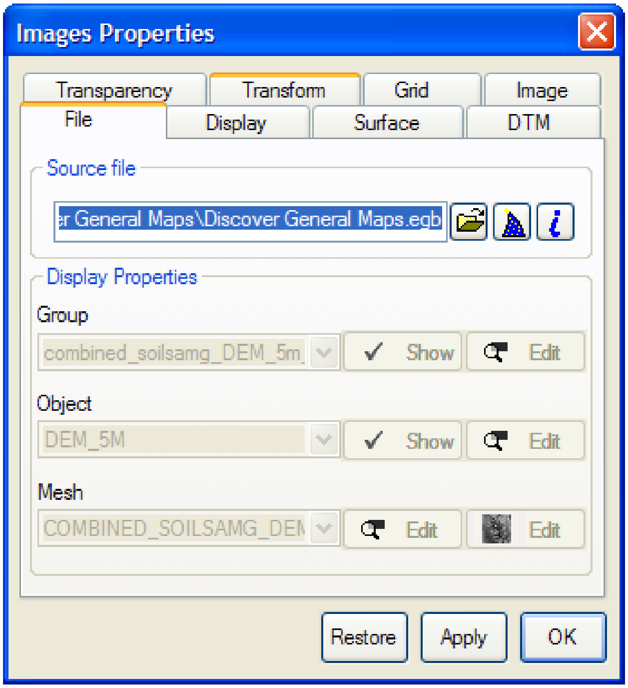
ERMapper grid (.ERS)
ESRI TIN (TDENV.ADF)
Gemcom vector (.BT2)
GeoTIFF (.TIF .TIFF)
GoCAD vector (.TS .PL .VS)
Geopak grid (.GRD)
USGS DEM grid (.DEM)
USGS grid (.USG)
USGS SDTS grid (.TAR)
Vertical Mapper grid (.GRD .GRC)
Vulcan vector (.00T)
Windows bitmap (.BMP)

Changing Image Display Properties

When a **Georeferenced Image** is displayed in Discover 3D an **Images** branch appears in the Workspace Tree. To display the properties dialog of the image, select the **Images** branch and either double click with the left mouse button or highlight and click right. From the displayed shortcut menu, select the **Properties** item.



The **Image Properties** dialog is a dynamic dialog; the property tabs displayed will vary depending on the data types incorporated into the EGB file. The following screenshot illustrates the full range of property tabs available when an EGB created by draping the contents of a map window over a grid is selected:



The Image Properties dialog with the various controlling tab sections.

The bitmap source path and file is listed under the **Source File** section. If you need to display an alternative Georeferenced Image, use the **Browse** button (shown left). The **Georeferencing Wizard** is also available (see *Using the Georeferenced Image File Creation Wizard*).

The **Images Properties** dialog provides various control tabs for image display. These are divided into:

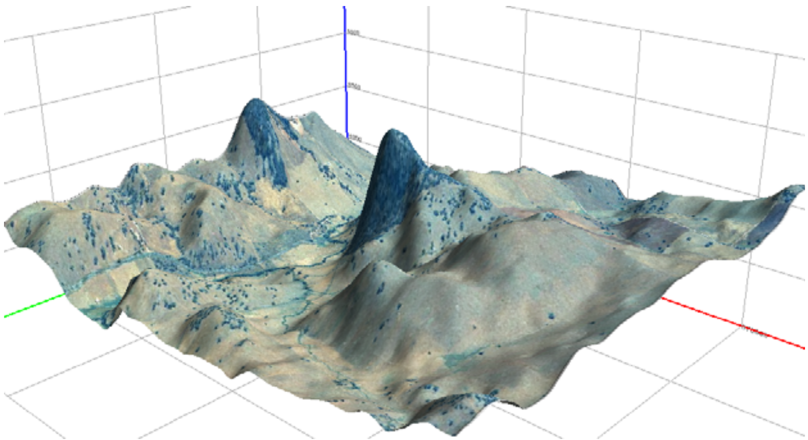
- **File** - initially displayed, allows file selection, group controls and image registration.
- *Display Tab* – provides clipping options.
- *Surface Tab* – provides options for image colour, modulation and lighting.
- *DTM Tab* – allows vertical section images to be offset using a digital terrain model (DTM).
- *Transparency Tab* – controls the image transparency.
- *Transform Tab* – provides scaling and translation controls.

- *Image Tab* – image stretch and interpolation controls, as well as transparent colour assignment.
- *Grid Tab* – incorporates grid compression controls.

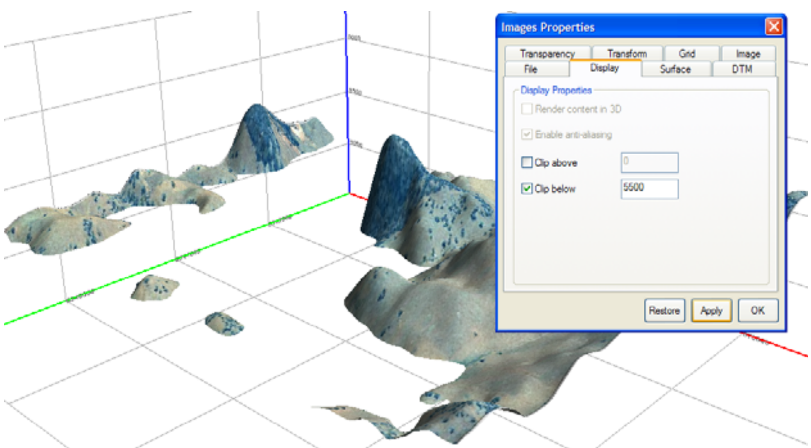
Display Tab

The **Display** tab allows the image to be clipped above and/or below two user-specified horizontal planes (i.e. Z values).

For instance, if a Clip below value of 235 was set, an airphoto draped over a DEM grid would have all portions of the image with Z value less than 235 hidden.



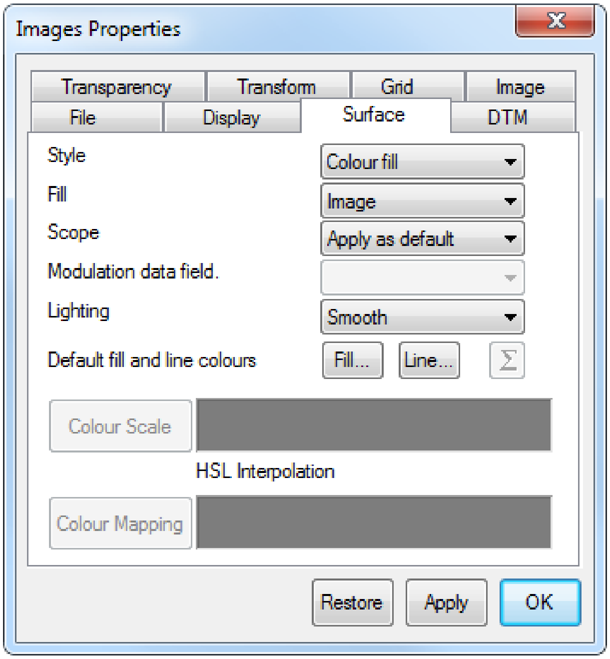
An airphoto draped over a DEM grid



The same .EGB with all portions below 5500 clipped.

Surface Tab

The **Surface** tab specifies aspects of colour, modulation and lighting.



Surface tab defining colour usage and appearance of Located Images

Parameters that can be controlled include:

- **Style** – Controls the basic surface rendering style and allows the triangle meshes to be displayed using fill (colour or textured), fill with triangle mesh wireframes or simply as coloured wireframes.

- **Fill** – Controls the source of the fill colour, if used. There are three possible colour sources – an image, a user-defined solid colour or solid colour modulated by a field in any of the ancillary data tables (Object, Mesh, Surface, Face and Vertex). In the latter case you must select a field from the table and define a colour table and colour mapping. If you are only displaying wireframes then the settings will apply to the line colour.

- **Scope** – Allows you to override default settings (defined in the source files) or only apply the settings to those meshes for which no defaults were specified in the source file.

- **Modulation Data Field** – Lists the fields in the ancillary data table, if selected. Note that the system will create many fields automatically. One such field is the '_Colour' field which will contain any colour information recovered from the source file.

- **Lighting** – Provides None, Smooth or Flat and affects the display of the facets of the objects.

- **Default fill and line colours** – Specifies the user-defined colours for solid fill and wireframes.

- **Colour scale** – Control of the colour look-up table is provided.

- **Colour mapping** – The colour mapping controls the mapping of the colour modulation field data to the colour table. In general this will be a linear transform but other options such as histogram equalisation can be chosen to stretch the colour table. Note that a special transform called 'None' is available. This will bypass the colour table and is automatically used when the '_Colour' field is selected.

Colour mapping applies a transform to the distribution of colours assigned by the specified look-up table. The dialog is displayed from the **Edit** button of **Colour Mapping** and offers band-pass cut-off of colour plus a range of transforms to map the colours. Band-pass operations determine the percentage of attribute data that is colour scaled. You can specify an upper and lower cut-off as required.

Available **Colour Mapping transforms** include**:**

- **None** – The source data is used as an RGB colour value directly with no modification or reference to the colour table.

- **Linear (Increasing)** – Within the prescribed colour range (as determined by the Band-Pass Cut Off) the data (from the attributes of the voxel model) are linearly stretched from lowest data value to lowest mapped colour, to highest value mapped to highest mapped colour.

- **Linear (Decreasing)** – The colour values are inversely mapped to the Linear Increasing option.

- **Log (Increasing)** – The log value of the data are linearly mapped from the lowest ranked colour, to the highest.

- **Log (Decreasing)** – The converse of the Log – Increasing option.

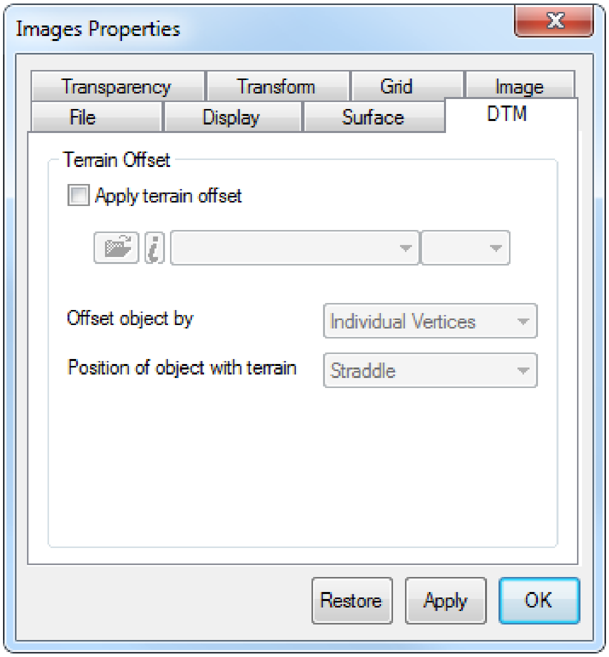
- **Histogram** – An equal area histogram operation to map the display.

- **User Defined** – Specify the range from the settings in the colour scale. To enable this feature use the ..specified in data values option and enter appropriate data values in the Lower and Upper Value entries.

DTM Tab

The **DTM** tab is used solely for offsetting planar vertical images (such as scanned geophysical sections imported using the Georeferencing Wizard) using a DTM (Digital Terrain Model) grid.

The offset can be computed for each vertex of the image or an offset for the centre of the image can be computed and applied equally to all vertices. The vertical image can sit on top of the DTM, hang from it or be centred on the DTM.

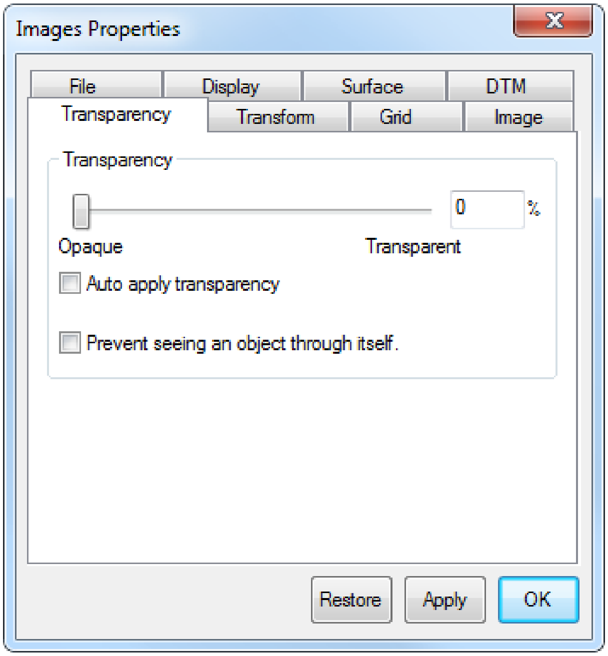


The DTM tab and defining a surface to alter the top surface

Transparency Tab

Image Transparency is controlled from the slider bar or by entering a specific percentage entry. An entry of 0% has no transparency while 100% displays the object with total transparency (invisible).

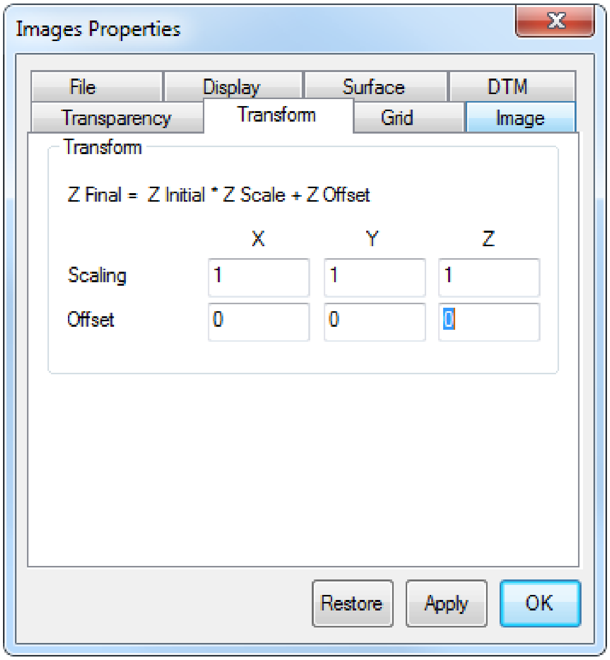
For certain objects (such as a sphere, that has a surface behind any other point), you can remove the effect of seeing a second surface behind the first.



The Transparency property tab

Transform Tab

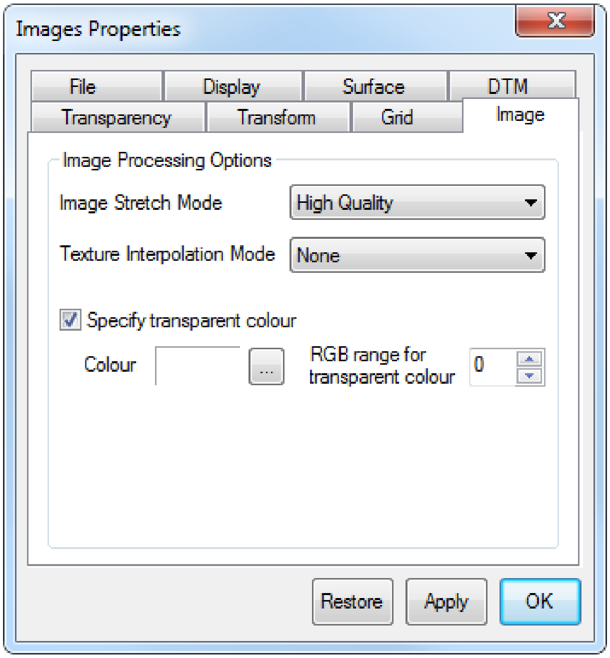
The **Transform** tab provides image scaling and translation controls. The scaling is applied as the last transformation prior to the DEM offset transform. The translation operation is applied as the first transformation after the DEM offset transform.



The Transform property tab

Image Tab

The **Image** tab provides Image Stretch and Texture Interpolation controls, as well as allowing the specification of a colour or colour range to set as transparent.



The Image Processing tab

Image Stretch allows for the preservation of either Colour, Black or White data, or for these to be displayed in the highest possible quality. **Texture Interpolation Modes** of an image can be:

- **High Quality** – displays interpolated images at suitable levels of resolution at all times to prevent aliasing artefacts.
- **Interpolate** – applies an interpolation algorithm to ‘smooth’ the display.
- **None** – displays the bitmap as is with no interpolation applied.

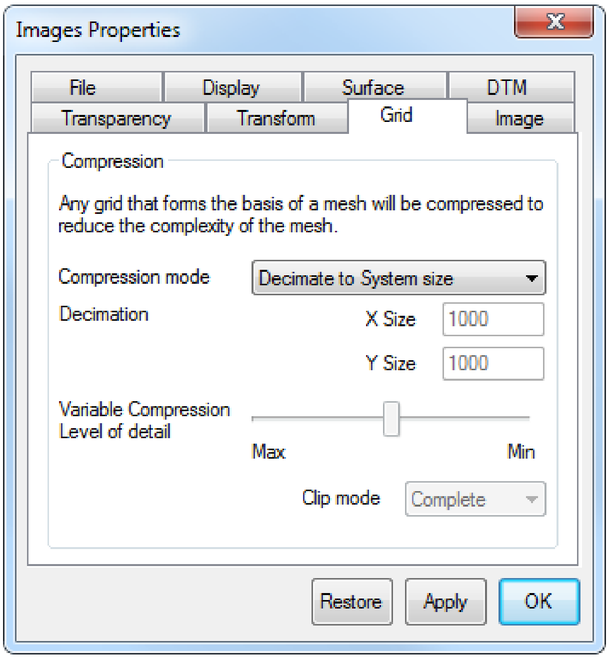
The **Specify transparent colour** option allows the selected colour to be rendered transparent. This is particularly useful when the image has been captured with a homogenously coloured mask (typically a white mask, e.g. scanned government 250000 interpreted geology maps).

Also available is a transparency range, allowing a ‘range’ of colours to be made clear. If you set this value, the RGB combinations above the colour specified to be transparent are also set transparent. For example, if you request the colour white to be transparent (RGB colour 255:255:255), and set a range of 5, then all colour combinations with RGB from 250 are set transparent (that is, 250:255:255 and 251:255:255 etc). This option is especially useful when some pixels of a bitmap are nearly white but not exactly as is often the case in a scanned product.

Grid Tab

The **Grid** tab is only displayed when a grid surface is incorporated into the EGB file, e.g. when a map window had been draped over a grid surface.

The range of grid compression options available allow Discover 3D to display and manipulate large, complex surfaces quickly with little loss of surface detail. Grid compression reduces the complexity of a grid, thereby improving display redraw speeds and navigation efficiency.



The Grid Compression tab

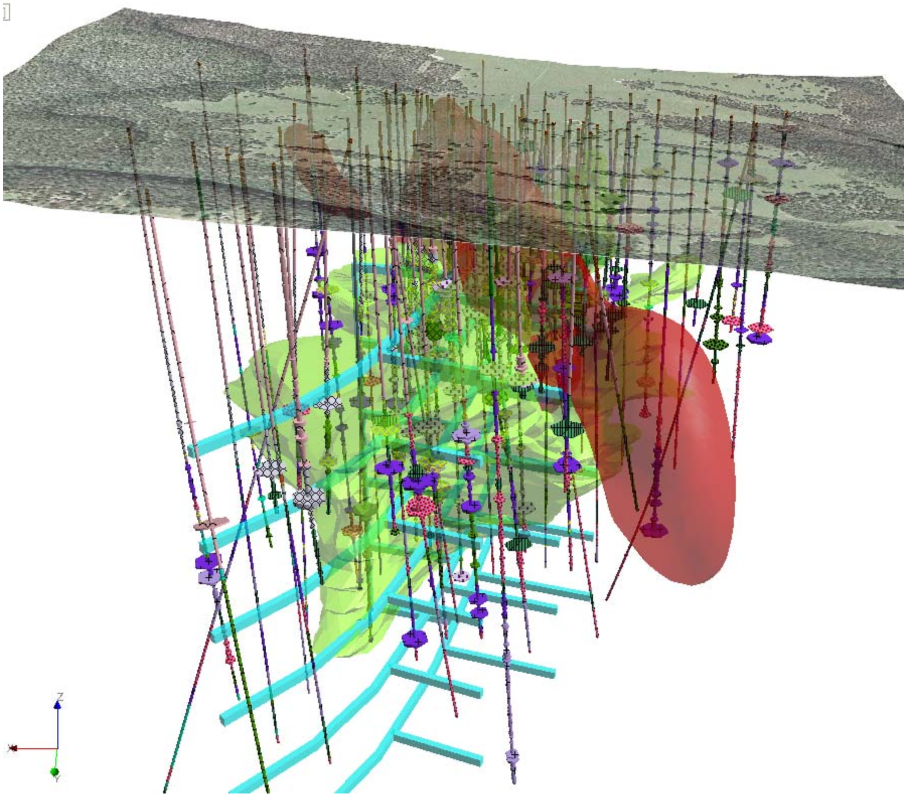
The **Compression Mode** pull-down list provides five compression options:

- **None** - No compression is performed and the grid is displayed at full resolution.

- **Decimate to System size** - The grid is decimated to the System size set via the **Discover 3D> Options** menu option, under the System tab.
- **Decimate to User size** - The grid is decimated to the X and Y Size values assigned by the user within this dialog
- **Decimate to User Factor** - The grid is decimated according to the X Skip and Y Skip values assigned by the user.
- **Variable Compression** - Attempts to retain the boundary of the grid, including internal holes. The compression level is adjusted via the slider bar. Three clipping levels are available:
    - **Complete** - the grid boundary and holes are preserved,
    - **Partial** - the grid boundary and holes may not be perfectly preserved, or
    - **None** - the grid is rendered as a convex hull with no holes preserved.

8 Working with Vector Data in 3D

Discover 3D can display objects as both 3D and 2D vector files. An example of 3D vector models imported into a 3D space is shown below and illustrates the use of 3D vector files with other data types.



An example of underground workings (light blue) and an ore body outline (light green) displayed as 3D vector files with drillholes, located bitmaps, and a voxel model isosurface (red).

- *Displaying 3D Vectors*
- *Changing 3D Vector Display Properties*
- *Displaying 2D Vectors in 3D*
- *Manipulating, Transforming and Reprojecting Vectors*
- *Creating Orientated 3D Vector Symbols*
- *Extruding 2D Vectors into 3D*

- *Generating 3D Models from Vectors*

Displaying 3D Vectors

There are four options for displaying 3D vector files in Discover 3D:

1. From within Discover, 3D DXF tables or selections may be created using the **Discover 3D>View Objects in 3D** tool (see *Displaying Map Objects as 3D Vectors*), which then transfers the output to Discover 3D automatically. The DXF files created via this option can also be opened directly into Discover 3D at a later time.

2. By selecting the **Display Vector** button from the **Main** toolbar.

3. Using the **Display>3D Vector** menu option.

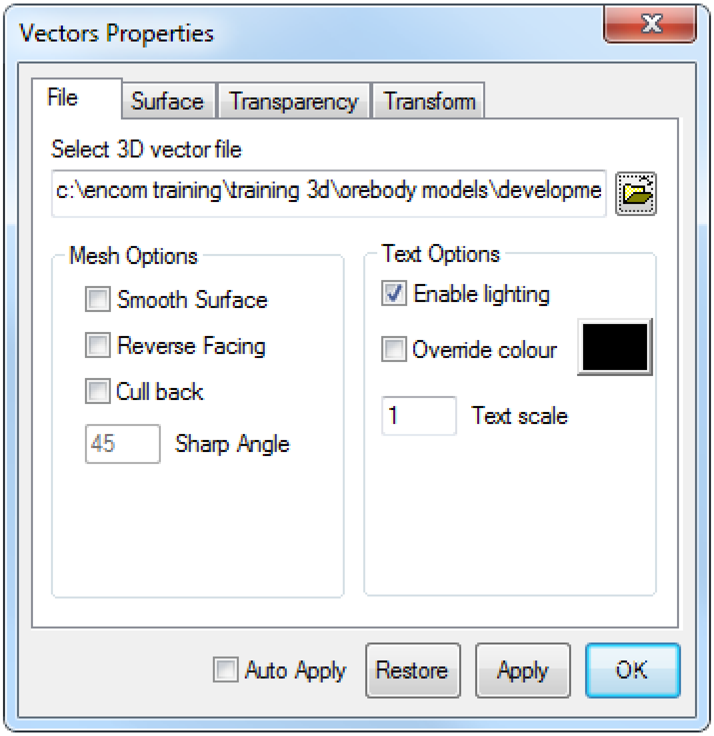
4. Selecting the **File>Open** menu option and setting the File of Type to **All supported 3D Vector formats**.

Discover 3D supports the following 3D vector file formats:

AutoCAD (.DXF)
GoCAD vector files (.TS, .PL, .VS)
Gemcom (.BT2)
ESRI TIN (.ADF)
ESRI 3D shapefiles (.SHP)
Datamine wireframe (.DM)
Surpac DTM files (.DTM)
Surpac string files (.STR)
Vulcan triangulation files (.00T)
3D Studio files (.3DS)

Note Additional vector file formats can be imported and converted using the *Transform Vector File* tool detailed in the 3D Utilities chapter.

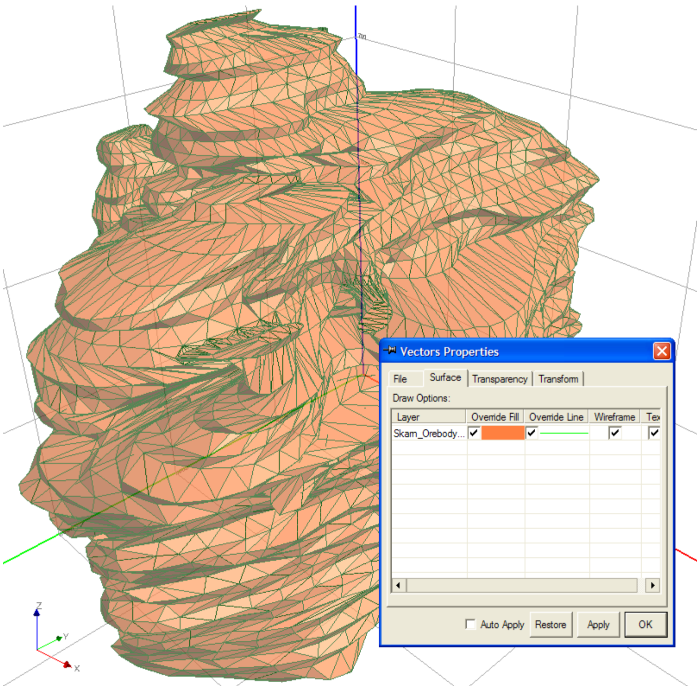
When opening 3D vector files from within Discover 3D (using the import options 2 & 3 above) a new, empty **Vectors** branch is added to the Workspace Tree. Display the **Vectors Properties** dialog of this layer and use the **Browse** button under the **3D Vector File** tab to specify a 3D vector file after navigating to the appropriate location. Click the **Apply** button after selection and the 3D object file is displayed in Discover 3D.



Vectors Properties dialog used for specifying a 3D vector format file

Changing 3D Vector Display Properties

The **Vectors Properties** dialog provides a number of controls (in four tabs) to improve or alter the appearance of displayed 3D vector files.



Display of an ore body 3D vector file with altered appearance

The **File** tab has the following options:

- Apply a **Smooth Surface** to the object if the surface is faceted. The smoothing can be applied to facet angles sharper than a defined angle (Sharp Angle in degrees).
- **Reverse Facing** - The defining facets can be reversed to allow the internal surface of the 3D file to be viewed.
- **Cull Back** - To improve the speed of rendering, the back (away from the display), can be culled.

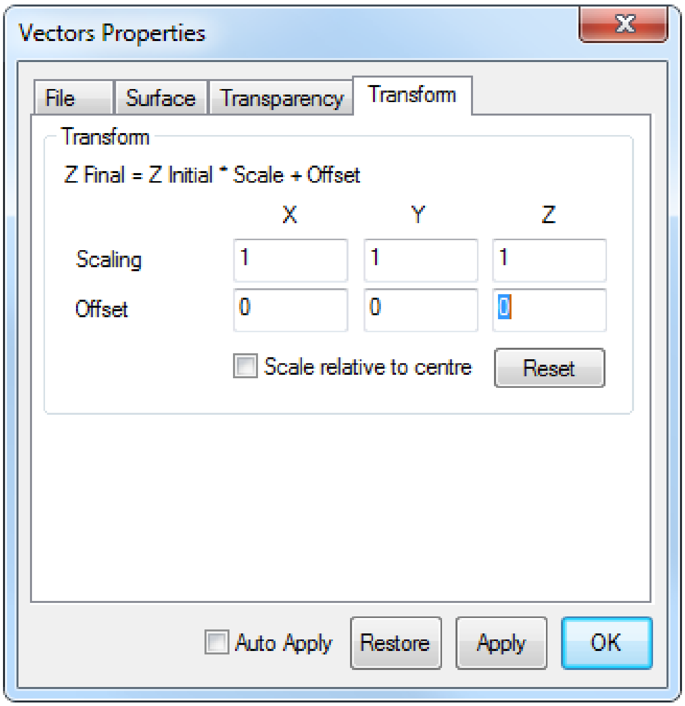
A 3D vector file can be composed of multiple layers, the display attributes of which can be controlled via the **Surface** tab. For each layer, the following options are available:

- Specification of an **Override Fill** .

- Specification of an **Override Line** colour, weight and pattern.
- Display a **Wireframe** of the surface faceted shape in addition to the surface.

The **Transparency tab** enables the transparency of a vector object to be altered via a slider bar or manual entry of a transparency percentage. An option to prevent the vector object being visible through itself is also available.

The **Transform** tab allows **Scaling** and **Translation** parameters to be assigned individually to the X, Y and Z axes. A **Reset** button allows these values to be restored to their original (un-transformed) values.



3D Vectors Transform tab, incorporating scaling and translation controls

Displaying 2D Vectors in 3D

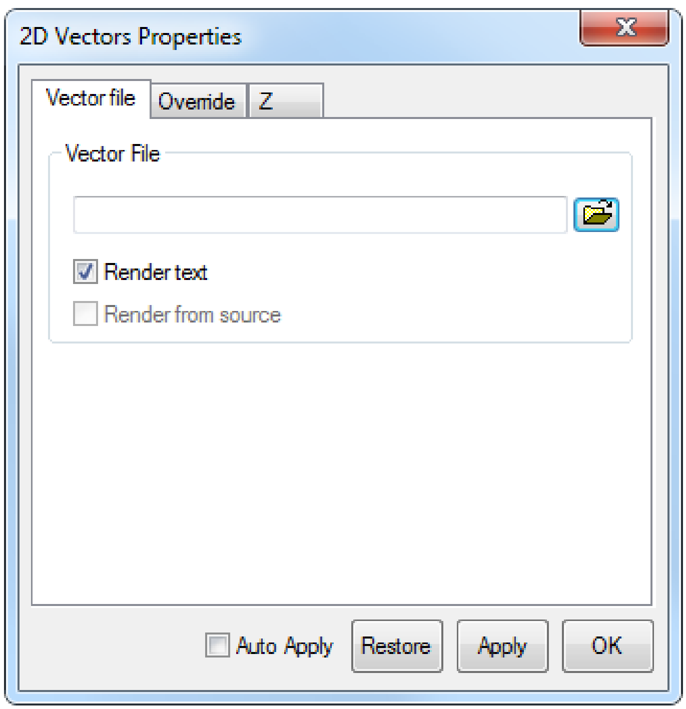
Discover 3D enables the display of vector data as 2D vectors without creating a DXF file. Vector data text, line and fill colours can be modified within the 3D view. Discover 3D supports the following formats for display as 2D vector files:

AutoCAD 2D DXF
Encom GSF files
ERMapper ERV file
ESRI SHP files
MapInfo Professional MIF and TAB files

There are three options for importing data into Discover 3D as 2D vectors:

- Use the **Display>2D Vector** menu option,
- Right-click on the 3D Map branch in the Workspace Tree and select **Display>2D Vectors** option, or
- Select the **File>Open** menu option and set the Files of Types to **All supported 2D Vector Files**. Browse for the relevant file.

These import options will place a new **2D Vectors** branch in the Workspace Tree. Double click on this branch to open the **2D Vectors Properties** dialog. The first two import options require data file selection using the **Browse** button in the **Vector File** tab of this dialog.



2D Vectors Properties dialog

The **Override** tab of the **2D Vectors Properties** dialog provides override controls for the text, line, fill and background fill colours. Polygons with a solid fill colour can be instantly made hollow using the **Make Hollow Fill** option.

The **Z** tab incorporates **Offset** options (DTM offset via a specified grid, or a Constant Offset), as well as allowing **Z scaling** specification.

Manipulating, Transforming and Reprojecting Vectors

Discover 3D provides the following additional tools for vector file manipulation:

Transform Vector File

This tool allows the following functionality:

- Import and conversion of a wide range of vector file formats.
- Reprojection of vector files to different coordinates systems.
- Positioning in 3D of vector objects such as vehicle models, including separate rotation and scaling options along each axis.

See the *Transform Vector File* section of the 3D Utilities chapter for further information.

Combine DXF

Combines separate DXF data files into a single DXF file, preserving the location and properties of objects. See the *Combine DXFs* section of the 3D Utilities chapter for further information.

Creating Orientated 3D Vector Symbols

This tool converts 2D point data into 3D symbols in an output DXF vector file. It includes controls for Bearing and Inclination of the symbols, as well as shape, colour, and size controls for the symbols. See the *3D Point Symbols* section of the 3D Utilities chapter for further information.

Extruding 2D Vectors into 3D

Converts 2D input objects into a 3D objects in a vector DXF file, by extruding the objects in the Z axis. This is a powerful way of visualising faults from surface trace mapping, or buildings from polygon outlines. This tool includes controls for the primary and secondary heights, bearing and inclination options, etc. See *Extruding Models from Points, Lines and Polygons* for more information.

Generating 3D Models from Vectors

Produces interpolated solid 3D objects (e.g. fault planes, ore zones) from input polygons or polylines. See the *Modelling Triangulated Surfaces and Solids* chapter for further information

9 Working with 3D Points

Point data may include data that has been collected systematically at sample locations along linear traverses or measurements taken randomly across an area. Point data tables require each record to have an **X** (Easting), **Y** (Northing) and **Z** (RL) field, with an optional line identifier field. A soil sampling survey or traverses of height measurements are examples of point data tables that can be viewed in Discover 3D.

To display point data in Discover 3D, use the **Discover 3D>Create 3D Points** tool (see *Displaying Points in 3D*) on the MapInfo/Discover 2D interface.

Alternatively within Discover 3D window, use the **Display Points** tool from the **Data Objects** toolbar or the **Display>Points** menu item to display data already open in the 3D window.

The source of data to be used can be from a table in MapInfo Professional or a data *Selection*. When you request Points be added to the Workspace Tree, a **Points** branch is created.

This is a powerful way of visualising surface geochemical datasets (e.g. soils or rock-chips) over your drillholes, gridded surfaces and imagery.It can also be utilised to indicate points of interest (e.g. prospect locations). Displaying data as 3D Points provides the following capability:

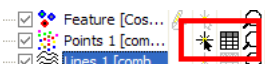
- Change the point *Symbol Style* type, colour and size.
- Display points as 3D symbols based on True Type fonts, with controls over symbol orientation, extrusion and positioning.
- Display *Labels* for the dataset.
- Modulate point data colour (*Colour Modulation*) or symbol size (*Size Modulation*) using values from other fields in the table and colour tables and legends.
- *Rotating Point Symbols* to modulate point by independent Bearing, Inclination and Tilt fields (ZXY components).

To control these and many other parameters, see the *Changing Point Display Properties* section below.

Interrogating Point Information

3D Point datasets displayed in Discover 3D can be dynamically interrogated, allowing a particular point's attributes in the source .TAB file to be instantly viewed.

To enable point interrogation:



1. In the *Workspace Tree*, make the Point branch both **Selectable** and **Browsable**.

2. Display the Data Window using either the toolbar button or the **View>Data Window** menu option.

3. Enable the **Select/Navigate** button on the *Zoom Controls Toolbar.* Place the mouse cursor over a point location – the Data tab will scroll to the appropriate point record. The Cursor Plane and its Bond (see Bond option on *Cursor Plane Toolbar*) function will be auto-enabled for this process.

   The fields displayed and their order in the Data window spreadsheet can be customised by right clicking in the Data window and selecting the **Customise** option. These Data window customisation options (including field widths) are preserved as long as the current Discover 3D session is open.

Changing Point Display Properties

The display of points in 3D is controlled from the **Points Properties** dialog as shown below. To display the **Points Properties** dialog highlight the **Points** branch in the Workspace Tree and select **Properties** from the right-mouse click shortcut menu (or double-click on the branch).

Clicking the **Apply** button will update the 3D display to reflect chosen options. Enabling the **Auto Apply** option will automatically update the display after every change. For large datasets it is recommended this feature be disabled, allowing multiple changes to be applied at the one time rather than waiting for the display to redraw between each change.

The following parameters can be controlled with this dialog:

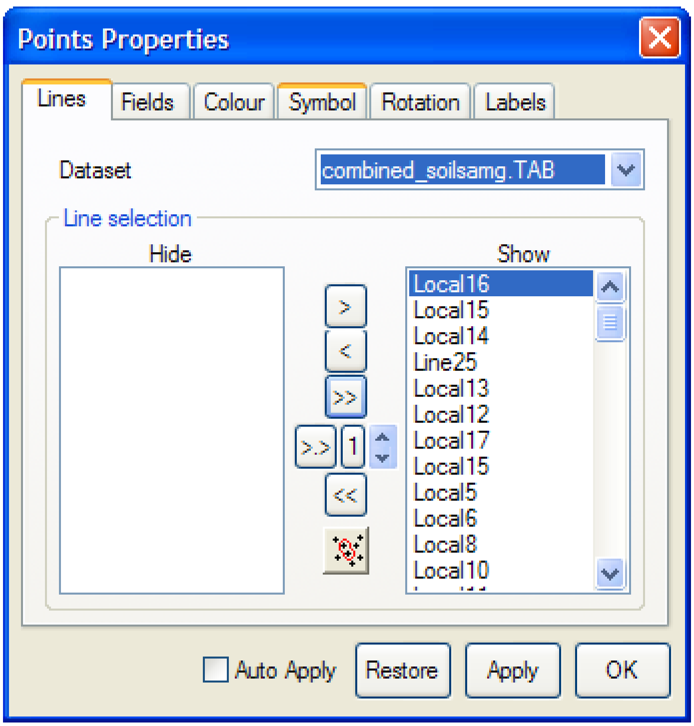
- *Data Selection*
- *Offsetting and Scaling Data*
- *Symbol Style*

- *Labels*
- *Colour Modulation*
- *Size Modulation*
- *Rotating Point Symbols*
- *Decimation*

Saving or Loading Point Display Settings

Any display settings customized in the following tab controls can be saved for later reuse. To save or load any current settings, move the cursor over the Points branch in the Workspace tree and right click. Select either **Save Properties** or **Load Properties** from the menu.

Data Selection

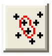


Lines selection dialog

The **Lines** tab provides data subsetting control. Select the Lines dataset (if more than one is open) from the **Dataset** pull-down list.

Select the lines to display in the 3D window from the **Line Selection** list. Use the arrows to select or deselect lines by moving them between the **Hide** and **Display** windows.

You can quickly select every 2nd line from the dataset by using the subset arrows button. At 1 this selects every line, 2 every second, 3 every third line etc.



Alternatively, use the button to make a map window selection of point data to display.

Offsetting and Scaling Data

The **Fields** tab allows the user to set both:

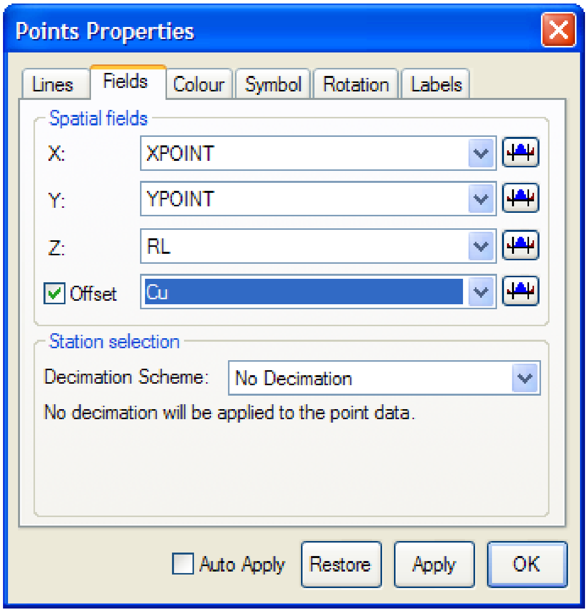
- **Z field** – typically an RL or Elevation field in the dataset. If this does not exist, perhaps use the **Discover>Surfaces>Assign Values from grid** tool to populate a field with RL values from a coincident topographic grid.
- **Offset** field – a separate field to offset the points (in the vertical) from the Z value. For example, using a geochemical assay field to in essence create a stacked profile view.

The **Field Data Conditioning** button adjacent to each of the field selection lists opens the **Field Data Conditioning** dialog. This dialog allows the selected field to be transformed via a number of methods:

- It is of most use for applying a scaling factor to the **Offset** field, as it is unlikely that the magnitude of the offset field (e.g. an assay or geophysical field) will be comparable to the Z field.
- It may also be useful in applying a **Translation** factor to your Z field when data is coincident with a draped aerial photo (and therefore partially obscured: e.g. translating the data by 2m above the Z field value.

For more information on data conditioning, see *Field Data Conditioning Tool*.

To scale the Offset field:



Setting an Offset field in the Fields tab

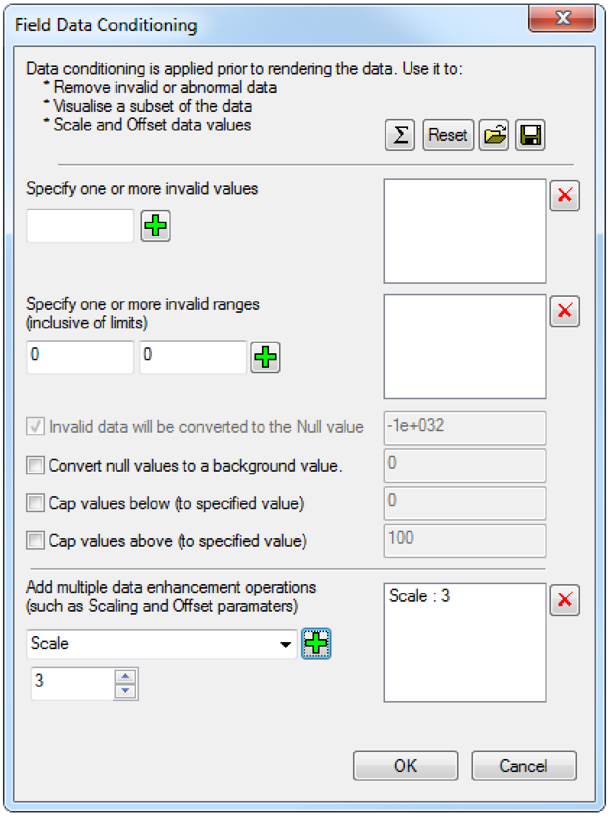
1. Enable the **Offset** tick box.

2. Select the field you wish to Offset the point data with (e.g. a geochemical field) from the adjacent pull-down list.

3. Select the adjacent **Field Data Conditioning** button. See *Field Data Conditioning Tool* for more information.

4. In the following dialog, select the **Scale** option from the pull-down list at the bottom left of the dialog.

5. Set the scaling factor in the window below this list, and press the adjacent Add button to add this setting to the right hand list. Press OK to close the Field Data Conditioning dialog.



Setting a Scale factor in the Field Data Conditioning dialog

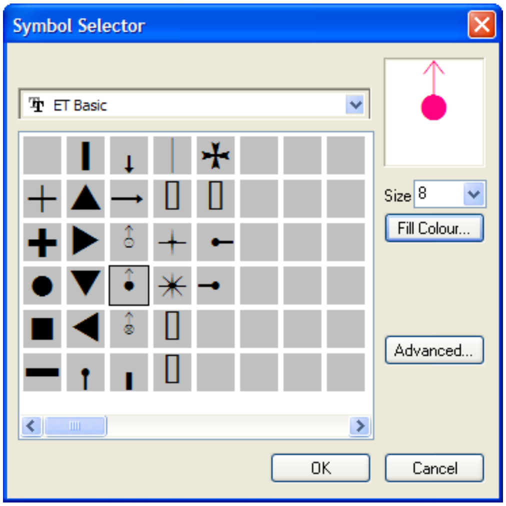
6. Press **Apply** to visualise this in 3D.

Symbol Style

The point symbol style is controlled under the **Symbol** tab of the point properties dialog.

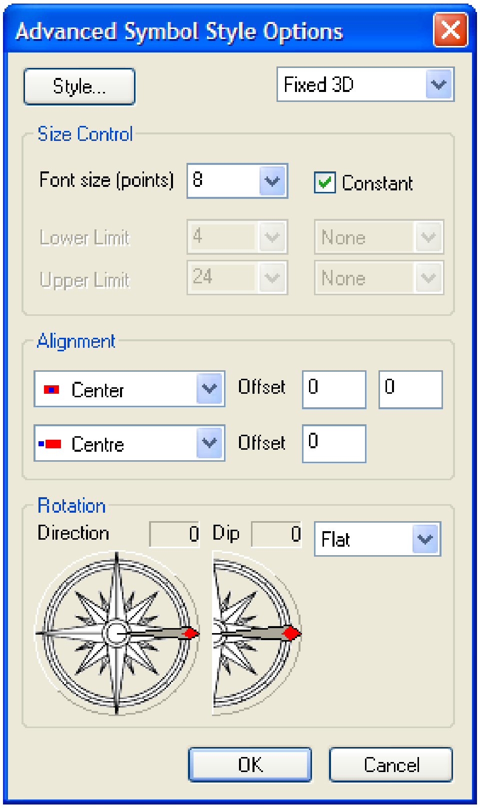
To access the symbol controls ensure the **Show Symbols** option is ticked. To quickly display point data using the default square symbol check the **Fast Symbol** option. This is a memory-efficient display method, and recommended when using very large datasets.

To customise the symbol display, deselect the **Fast Symbol** option and click in the **Symbol** box. This will open the **Symbol Selector** dialog, allowing you to choose from an extensive library of symbols based on installed True Type fonts. Choose a symbol from the list; it will be previewed to the right. The symbol **Size** and **Fill Colour** can also be altered here.



Symbol Selector dialog

Advanced Symbol Style Options are available via the **Advanced** button.



Advanced Symbol Style Options dialog

A range of symbol orientation options is presented in the pull-down list at the top right of the dialog:

- **Facing Viewer** (default) – Symbols are aligned parallel to the viewing/ screen plane, so that they always face the viewer.
- **Fixed 3D** – Symbols are fixed in the 3D environment. The pull-down list in the Rotation panel at the bottom of the dialog controls the initial orientation: Flat (the XY plane) or Upright (XZ plane).
- **Fast 3D** – identical to the Fixed 3D except that no Style controls are available; this is a very fast and memory-efficient option.

The **Style** button is only available for the **Facing Viewer** and **Fixed 3D** orientation options. It enables symbols to be extruded either as filled Polygons or Line Segments (wireframe) using the **Format** pull-down list. The depth of the symbol is set using the **Extrusion** control, expressed as a percentage of the symbol size. To display a flat symbol, set the **Extrusion** to 0%. The symbol is extruded perpendicular to its display plane.

A range of **Size Controls** is available. For the **Fast 3D** orientation option, only the **Font Size** control is available (in points). Enabling the **Constant** checkbox (for the other orientation options) will keep the symbols at the specified size (relative to the screen) regardless of zoom level. If the Constant option is disabled, **Lower** and **Upper Limits** can instead be set:

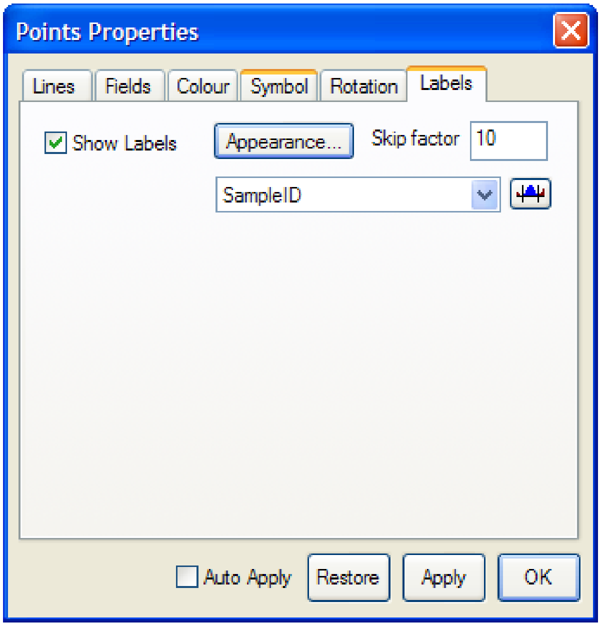
- Block: symbols will disappear when the applied zoom level takes the symbol past the specified limit,
- Clamp: symbols will be locked to the specified limit when the applied zoom level takes the symbols past the specified limit.

The symbol position relative to the data location can be altered using the **Alignment** controls (either preset or manual positioning). The first row of controls concern symbol positioning in the symbol plane (i.e. the relative XY components), whilst the second row controls the vertical height of the symbol with respect to its initial plane (i.e. the relative Z component).

The angle of the symbols can also be set by moving the red-tipped arrow on the compass in the **Rotation** panel at the bottom of the dialog. The **Fixed** and **Fast 3D** labelling options also provide a **Dip** control (half-compass) in this panel.

Labels

The labelling of data points in a 3D window display is useful for applications where visualising the actual values is important.



The Label control dialog, with 1 in every 10 points to be displayed with an arsenic value

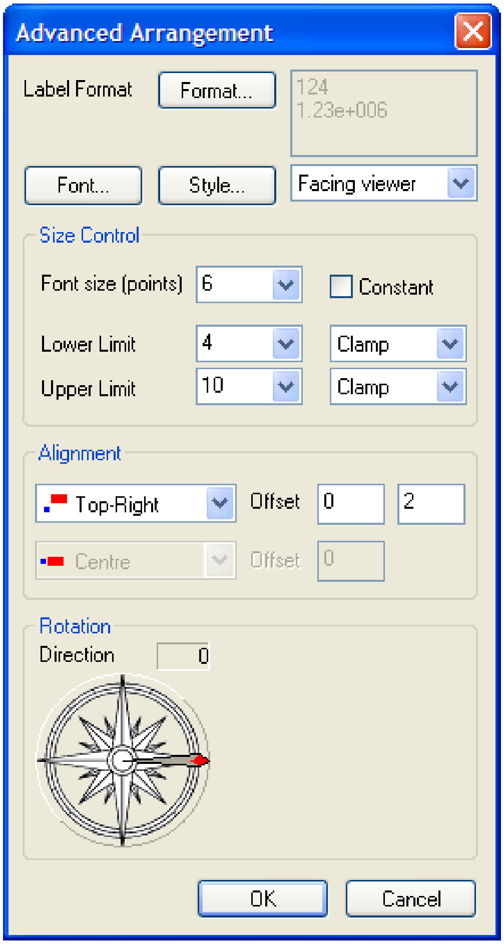
Enable the **Show Labels** tick box to activate the dialog. From the adjacent pull-down menu, select the data field to display as a label.

Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

The **Label Skip Factor** controls how often labels are displayed. For large datasets, it may be necessary to display the labels for only 1 in every 5 or 10 points.

Note Rendering labels as extrusions is memory intensive and may affect 3D performance. It is not recommended for large numbers of labels: the Label Skip Factor control can help minimise this issue.

The **Appearance** button opens the **Advanced Arrangement** dialog.



The Advanced Arrangement dialog for line labels

The **Format** button provides numeric formatting options (scientific, various DMS formats, general, etc), decimal place allocation as well as suffix and prefix specification.

Standard **Font** controls are provided, as well as the following range of orientation options in the adjacent pull-down list:

1. **Facing Viewer** (default) – Labels are aligned parallel to the viewing/ screen plane, so that they always face the viewer.

2. **Fixed 3D** – Labels are fixed in the 3D environment. The pull-down list in the Rotation panel at the bottom of the dialog controls the initial orientation: Flat (the XY plane) or Upright (XZ plane).

- **Fast 3D** – identical to the Fixed 3D except that no Style controls are available; this is a very fast and memory-efficient labelling option.

The **Style** button is only available for the **Facing Viewer** and **Fixed 3D** orientation options. It enables labels to be extruded either as filled Polygons or Line Segments (wireframe) using the **Format** pull-down list. The depth of the label is set using the **Extrusion** control, expressed as a percentage of the label size. To display a flat label, set the **Extrusion** to 0%. The label is extruded perpendicular to its display plane.

A range of **Size Controls** is available. For the **Fast 3D** orientation option, only the **Font Size** control is available (in points). Enabling the **Constant** checkbox (for the other orientation options) will keep the labels at the specified size (relative to the screen) regardless of zoom level. If the Constant option is disabled, **Lower** and **Upper Limits** can instead be set:

- **Block**: labels will disappear when the applied zoom level takes the label past the specified limit,

- **Clamp**: labels will be locked to the specified limit when the applied zoom level takes the label past the specified limit.

The label position relative to the data (collar) location can be altered using the **Alignment** controls (either preset or manual positioning). The first row of controls concern label positioning in the label plane (i.e. the relative XY components), whilst the second row controls the vertical height of the label with respect to its initial plane (i.e. the relative Z component).

The angle of the labels can also be set by moving the red-tipped arrow on the compass in the **Rotation** panel at the bottom of the dialog. The **Fixed** and **Fast 3D** labelling options also provide a **Dip** control (half-compass) in this panel.

Colour Modulation

The **Colour** tab allows point data to be coloured modulated using a specified field.

To colour modulate point data:

1. Enable the **Colour** option, and select the source data field from the adjacent pull-down list.

2. Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

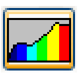
3. Specify a colour lookup table from the **Edit Colour Scale** button: this will open the **Colour Scale** dialog.

4. Select one of the four methods of colour scale definition:

    - **RGB Interpolation -** interpolates between two colours in Red:Green:Blue colour space.
    - **HSL Interpolation -** interpolates between two colours in Hue:Saturation:Luminosity colour space.

For the RGB and HSL Interpolations, set the first and last colours of the colour scale by selecting the **Colour Browse** buttons at the bottom or top of the colour bar. When clicked, a standard Windows colour selection dialog is displayed allowing colour specification. These can be reset by clicking the **Set Default Colours** button.

- **Look Up Tables** - the standard look-up table formats are supported and are installed as part of your Discover 3D installation. These can be created or edited using the Colour Look-Up Table Editor (see *Using the Colour Look-Up Table Editor*).
- A custom **Legend** created using the Legend Editor (see *Using the Legend Editor*).



5. Specify a non-linear mapping of the data, such as histogram or log, by selecting the **Edit Colour Mapping** button. This can assist in achieving an even stretch of the colour scale across the data range. See *Advanced Colour Mapping* for more details.

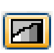
6. Press OK twice to apply.

Size Modulation

The point symbols can be size modulated using the controls in the **Symbol tab.**

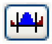
1. Enable the **Size** option.

2. Enter the desired minimum and maximum point sizes in the From and To areas.



3. Advanced size mapping options are available from the adjacent button which opens the **Data Mapping** dialog. See *Advanced Colour Mapping* for more details

4. Select the field to size modulate by from the associated pull-down list.

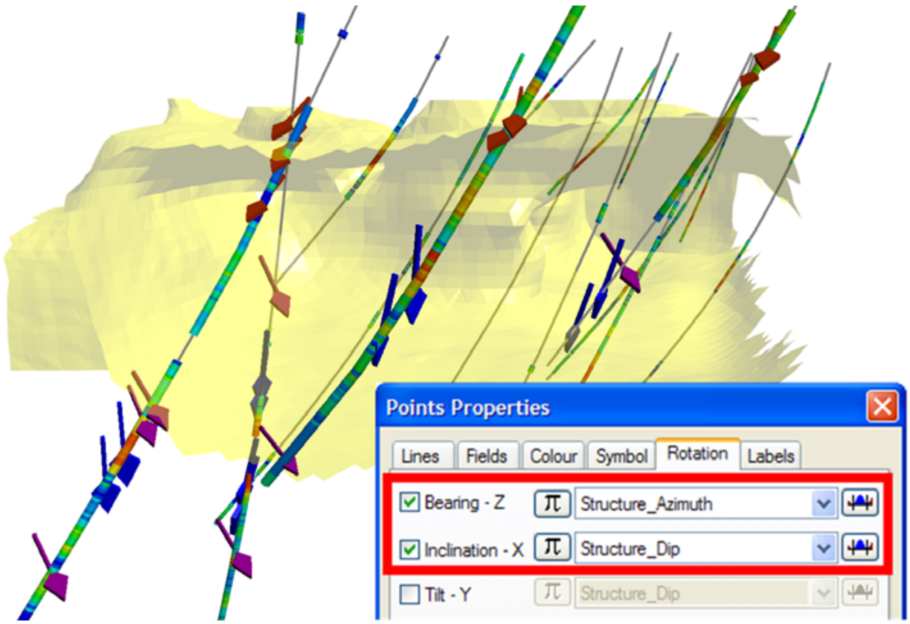


Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

Rotating Point Symbols

The **Rotation tab** allows point symbols to be rotated by up to 3 fields using:

- Bearing (i.e azimuth)
- Inclination (i.e. dip)
- Tilt



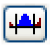
Drilling downhole structural measurements displayed as orientated points using azimuth and dip fields, with colour modulation by type (e.g. fault, bedding,vein). The point data was spatially referenced using the Drillholes>Calculate 3D Coordinates tool (see the Discover Reference Manual)

To apply rotation:

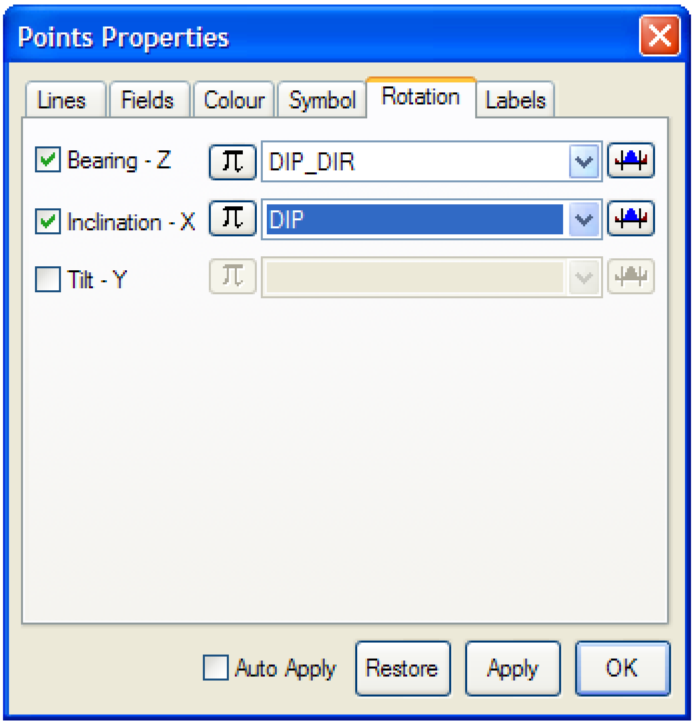
1. Enable the rotation types required.
2. Set the appropriate fields to each rotation type.

3. Modulate Rotation select the data field to use for the modulation.

The adjacent Pi button opens the **Rotation Parameters** dialog. This allows the direction of rotation to be altered (for instance to orientate positive dip values downwards, set this to -1 to reverse the angle direction). The angle units can also be set to radians or a fixed offset applied.



Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.



The Rotation tab configured for a dataset of structural measurements

Use the *3D Point Symbols* utility with point or line data to create orientated vector symbols for display in Discover 3D. This can be a less memory intensive way to display large amounts of orientated point data in 3D.

Decimation

Your dataset can be decimated using the options at the bottom of the **Fields tab.** The **Decimation** or station selection options allow the 3D rendering performance to be improved when dealing with very large datasets. A number of decimation options are available, including **Fixed rate** (where you specify a sample ‘skip’ factor) and **Compression**.

10 Working with 3D Lines

In Discover 3D, line data generally refers to data that has been collected systematically along linear traverses with samples or measurements taken at discrete intervals. The data is stored in exactly the same way as a MapInfo Professional point data table and requires a data field that can be used as a Line descriptor. When point data is displayed in Discover 3D it is presented as a single continuous line for each sample traverse.

Line data tables require each record to have an X (Easting), Y (Northing) and Z (RL) field along with a line identifier field. Elevation or geochemical survey line profiles are examples of line data that can be viewed in Discover 3D.

To display line data in Discover 3D use the **Discover 3D>Create 3D Lines** tool (see *Displaying Lines in 3D*) in the MapInfo/Discover 2D interface.

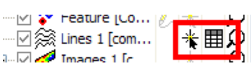
Alternatively use the **Display Lines** tool from the *Data Objects Toolbar*, or the **Display>Lines** menu item to display data already open in the 3D window.

The source of data to be used can be from a table in MapInfo Professional or a data *Selection*. When you request Lines be added to the Workspace Tree, a **Lines** branch is created.

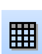
To modify line parameters such as applying colour modulation, tube styles, applying fill colours above or below a nominated threshold or removing lines from the 3D map view, refer to *Changing Line Display Properties* below.

Interrogating Line Information

3D Line datasets displayed in Discover 3D can be dynamically interrogated, allowing a each particular point’s attributes in the source .TAB file to be instantly viewed. To enable this:



1. In the *Workspace Tree*, make the Line branch both Selectable and Browsable.



2. Display the **Data Window** using either the toolbar button or the **View>Data Window** menu option.

3. Enable the **Select/Navigate** button on the *Zoom Controls Toolbar.* Place the mouse cursor over a point location along a line – the Data tab will scroll to the appropriate point record. The Cursor Plane and its Bond (see Bond option on *Cursor Plane Toolbar*) function will be auto-enabled for this process.

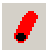
The fields displayed and their order in the Data window spreadsheet can be customised by right clicking in the Data window and selecting the **Customise** option. These Data window customisation options (including field widths) are preserved as long as the current Discover 3D session is open.

Displaying Lines as 3D Tubes

Closely related to the display of lines in three dimensions is the conversion of lines to a different object type called **Tubes**. A tube display may be useful in situations where a zone or 3D volume display is required that follows a line trace but has thickness represented by a cylindrical column surrounding the line.

The representation of data collected along a line by 3D tubes can often be used to advantage. The **Lines Properties** dialog enables the 3D tubes to be displayed and provides the following capability:

- Modify and import the tube shape
- Change the size and colour of the 3D tube
- Modulate line data colour, tube thickness or from other fields in the table and colour tables and legends.



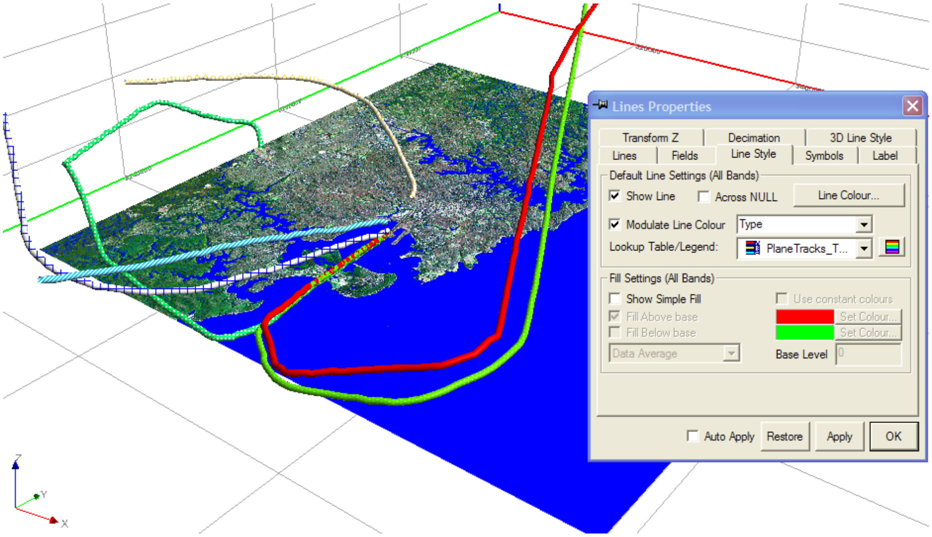
The *TubeShape Manager* is used to import and manage the tube shapes that are used to display both 3D Lines and drillhole traces.

The conversion of **Line** objects to **Tubes** is controlled from the **Appearance** tab of the Lines Properties dialog (see *Changing Line Display Properties*).

In the **Appearance** tab the tube thickness and shape can be modified. All other line options are available to alter the tube display, including colour and modulation options under the Lines Properties dialog. See the *Line and 3D Tube Style* section for displaying 3D Tubes.

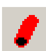
Note The **Tube Style** button is only available when the **Discrete samples** option is disabled.

An example of a patterned tube presentation where the tubes represent flight paths of specific aircraft types above an airport are shown below.



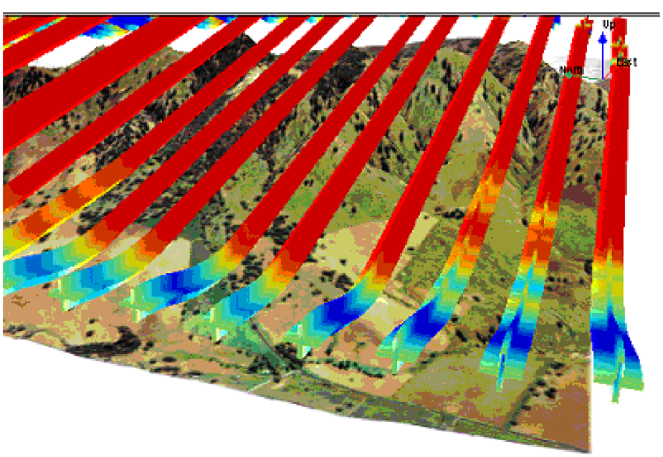
Aircraft flight paths presented as 3D tubes with pattern

TubeShape Manager



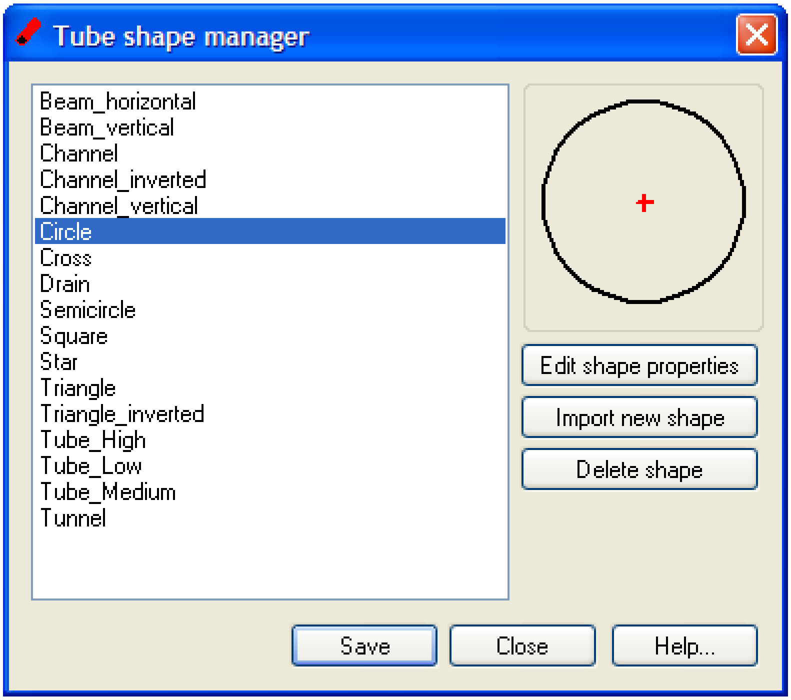
3D data lines can be displayed as tubes using a variety of tube shapes. The tube shapes can be either open or closed and any number of tube shapes can be created. The **TubeShape Manager** is used to import and manage the tube shapes that are used to display the 3D traces of any linear data, e.g. grid sample lines.

In the example below a cross-shaped tube has been used to display a series of flight lines in the Discover 3D map window.



Example flight lines using a colour modulated “cross” tube shape

The **TubeShape Manager** dialog displays a list of available tube shapes and enables new tube shapes to be added or deleted. The tube shapes are saved to the .ETS file located in the (Windows XP) `C:\Documents and Settings\All Users\Application Data\Encom\Common\TubeShapes` or (Windows 7 and 8) `C:\Users\All Users\Encom\Common\TubeShapes` directory.



Tube Shape Manager showing selection of tube shapes

To alter the name and description of individual shapes use the **Edit shape properties** button. Click on the **Import New Shape** button to import a tube shape created in a MapInfo Professional TAB or MIF file. Remove any shapes using the **Delete shape** button.

Linear data (including drillhole traces) is displayed in the Discover 3D map window from a **Lines** or **Drillholes** data type entry in the Workspace Tree. To select a tube shape to display with line data select the **Appearance** tab from the **Properties** dialog. See *Displaying Lines as 3D Tubes* for more information.

Changing Line Display Properties

The display of lines is controlled from the **Lines Properties** dialog as shown below. To display the **Lines Properties** dialog highlight the **Lines** branch in the Workspace Tree and select **Properties** from the right-mouse click shortcut menu. The **Lines Properties** dialog contains eight tabs to modify or control line data in the Discover 3D window.

Clicking the **Apply** button will change the display to reflect chosen options. Enabling the **Auto Apply** option will automatically update the display after every change. For large datasets it is recommended this feature be disabled, allowing multiple changes to be applied at the one time rather than waiting for the display to redraw between each change.

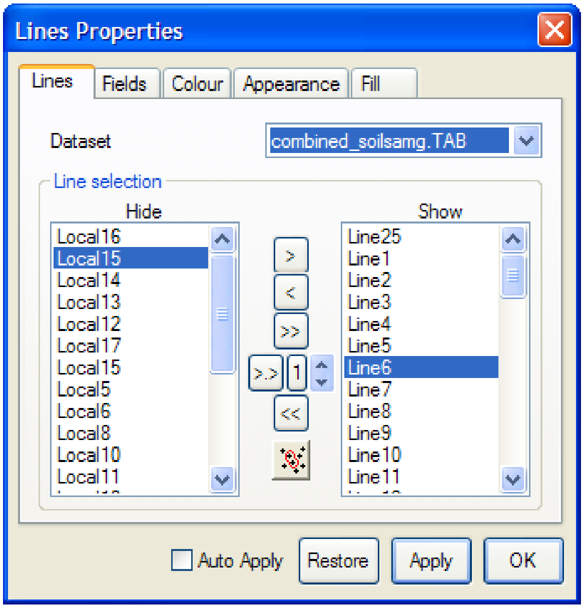
The following parameters can be controlled with this dialog:

- *Line Selection*
- *Offsetting and Scaling Data*
- *Line and 3D Tube Style*
- *Colour Modulation*
- *Thickness Modulation*
- *Colour Filling Line Profiles*
- *Decimation*

Saving or Loading Line Display Settings

Any display settings customized in the following tab controls can be saved for later reuse. To save or load any current settings, move the cursor over the Lines branch in the Workspace tree and right click. Select either **Save Properties** or **Load Properties** from the menu.

Line Selection

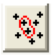


Lines selection dialog

The **Lines** tab provides data subsetting control. Select the Lines dataset (if more than one is open) from the **Dataset** pull-down list.

Select the lines to display in the 3D window from the **Line Selection** list. Use the arrows to select or deselect lines by moving them between the **Hide** and **Display** windows.

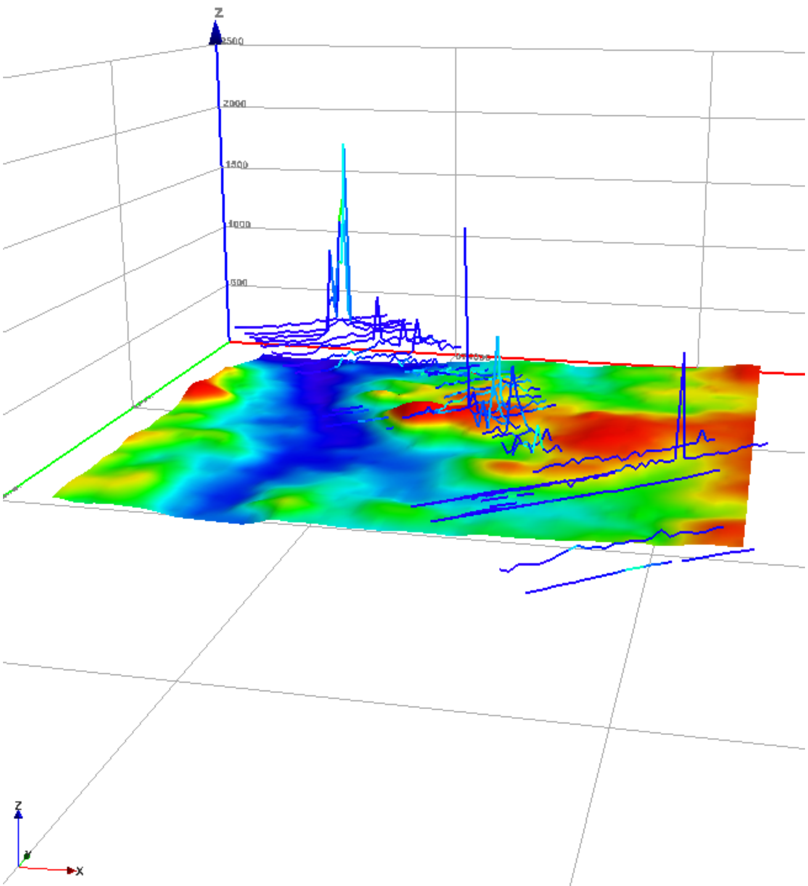
You can quickly select every 2nd line from the dataset by using the subset arrows button. At 1 this selects every line, 2 every second, 3 every third line etc.



Alternatively, use the button to make a map window selection of point data to display.

Offsetting and Scaling Data

An **Offset** can be applied to line data; for example by using a geochemical assay field as an offset.



Using the Fields tab to offset line data by the Copper field

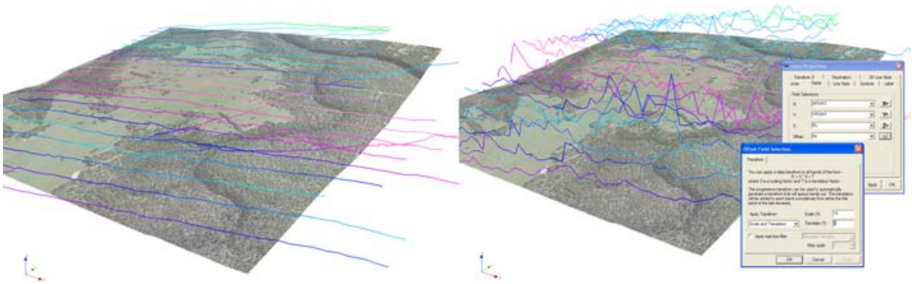
The **Fields** tab allows the user to set both:

- **Z field** – typically an RL or Elevation field in the dataset. If this does not exist, perhaps use the **Discover>Surfaces>Assign Values from grid** tool to populate a field with RL values from a coincident topographic grid.
- **Offset** field – a separate field to offset the points (in the vertical) from the Z value. For example, using a geochemical assay field to in essence create a stacked profile view.

The **Field Data Conditioning** button adjacent to each of the field selection lists opens the **Field Data Conditioning** dialog. This dialog allows the selected field to be transformed via a number of methods:

- It is of most use for applying a scaling factor to the **Offset** field, as it is unlikely that the magnitude of the offset field (e.g. an assay or geophysical field) will be comparable to the Z field.
- It may also be useful in applying a **Translation** factor to your Z field when data is coincident with a draped aerial photo (and therefore partially obscured: e.g. translating the data by 2m above the Z field value.

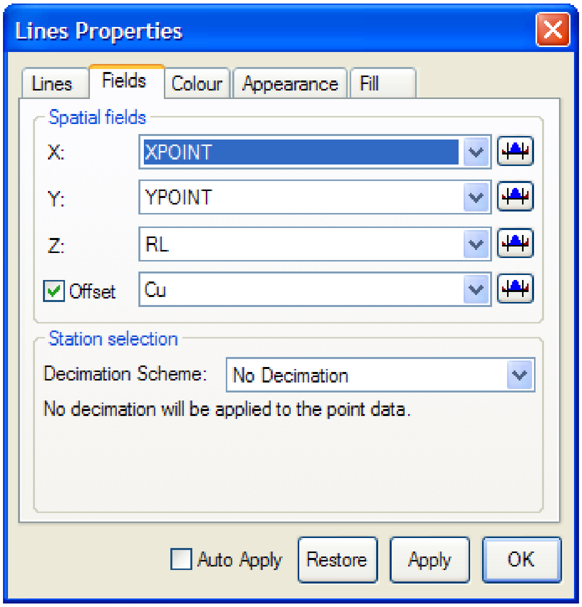
For more information on data conditioning, see *Field Data Conditioning Tool*.



An example of Offset field scaling. The left image shows soil samples offset from an airphoto by the Fe field. Due to the difference in magnitude between the Fe assays and the RL values, the soil lines appear very flat and uninformative. Using the Offset Field Selection dialog (right), this offset has been scaled by a factor of 10 (1000%) to give the viewer a better appreciation of the Fe distribution over the project area.

To scale the Offset field:

1. Enable the **Offset** tick box.
2. Select the field you wish to Offset the point data with (e.g. a geochemical field) from the adjacent pull-down list.

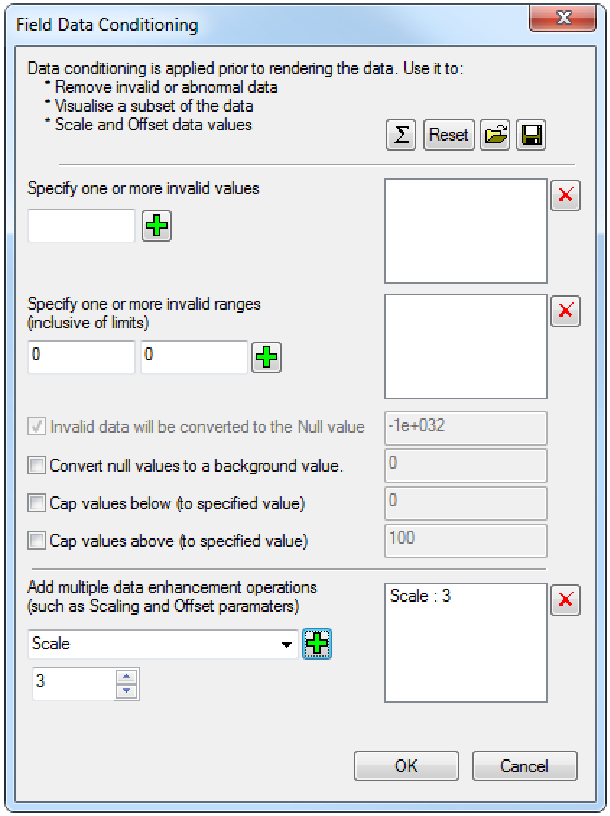


Setting an Offset field in the Fields tab

3. Select the adjacent **Field Data Conditioning** button. For more information, see *Field Data Conditioning Tool*.

4. In the following dialog, select the **Scale** option from the pull-down list at the bottom left of the dialog.

5. Set the scaling factor in the window below this list, and press the adjacent Add button to add this setting to the right hand list. Press OK to close the Field Data Conditioning dialog.



Setting a Scale factor in the Field Data Conditioning dialog

6. Press **Apply** to visualise this in 3D.

Line and 3D Tube Style

Under the **Appearance** tab the line style options can be set. Choose from either Line or 3D tube to display. The style of these can be set under the **Line Style** or **Tube Style** buttons.

Note The **Tube Style** button is only available when the **Discrete samples** option is disabled.

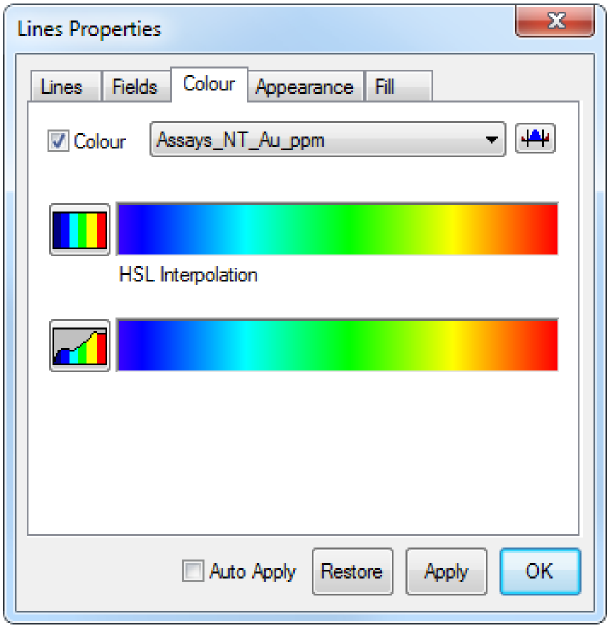
Note that Line thickness is set in the Line Style dialog, but the Tube thickness is set in the **Appearance** tab.

To edit the available list of tube shapes, use the *TubeShape Manager*.

Data gaps can be including or ignored in the line by the **Show Gaps** and **Discrete Samples** options.

Colour Modulation

The **Colour** tab allows line data to be coloured modulated using a specified field.



Line style controls with Au HSL lookup table and a histogram colour map applied

To colour modulate line data:

1. Enable the **Colour** option in the Colour tab, and select the source data field from the adjacent pull-down list.

2. Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

3. Specify a colour lookup table from the first **Edit Colour Scale** button: this will open the **Colour Scale** dialog.

4. Select one of the four methods of colour scale definition:

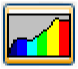
   - **RGB Interpolation -** interpolates between two colours in Red:Green:Blue colour space.

- **HSL Interpolation -** interpolates between two colours in Hue:Saturation:Luminosity colour space.

For the RGB and HSL Interpolations, set the first and last colours of the colour scale by selecting the **Colour Browse** buttons at the bottom or top of the colour bar. When clicked, a standard Windows colour selection dialog is displayed allowing colour specification. These can be reset by clicking the **Set Default Colours** button.

- **Look Up Tables** - the standard look-up table formats are supported and are installed as part of your Discover 3D installation. These can be created or edited using the Colour Look-Up Table Editor (see *Using the Colour Look-Up Table Editor*).

- A custom **Legend** created using the Legend Editor (see *Using the Legend Editor*).



5. Specify a non-linear mapping of the data, such as histogram or log, by selecting the **Edit Colour Mapping** button. This can assist in achieving an even stretch of the colour scale across the data range. See *Advanced Colour Mapping* for more details.

6. Press **OK** twice to apply.

Thickness Modulation

3D Lines displayed as 3D Tubes (not Lines) can have their thickness modulated by a field, using the controls under the **Appearance** tab.

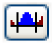
1. Enable the **3D Tube** style from the pull-down list.

2. Enable the **Modulate** thickness option.

3. Enter the desired minimum and maximum thickness in the From and To areas.

4. Advanced size mapping options are available from the adjacent button which opens the **Data Mapping** dialog. See *Advanced Colour Mapping* for more details.

5. Select the field to thickness modulate by from the associated pull-down list.

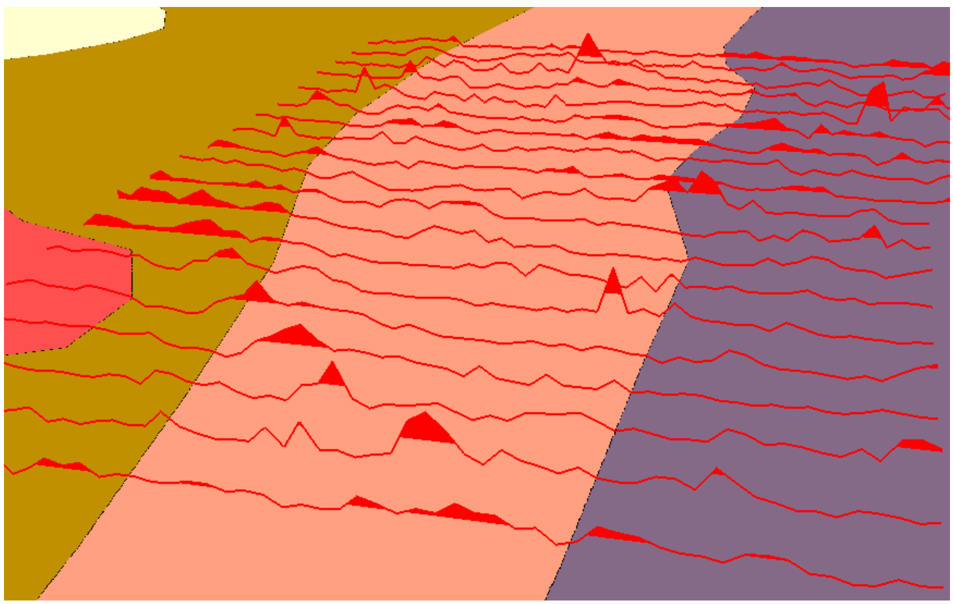


6. Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

Colour Filling Line Profiles

Lines can also be colour filled under the **Fill** tab. This is similar to applying a colour fill to 'Stacked Profiles' in 2D (see the *Table Utilities>Create Stacked Profiles* section of the *Discover Reference Manual*), but with the 2D profiles displayed in the 3D space.

An example might be displaying the copper results of a prospect-scale soil sampling program as 3D Lines, and colour filling all sections of the profiles above a certain anomalous copper value.



Copper soil profiles colour filled above an anomalous value.

To colour fill your 3D line profiles:

1. Ensure the target data field is set as the Z field in the Fields tab, with no offset field assigned.

2. Enable the **Show Fill** option in the Fill tab.

3. Select to colour fill either above or below a base level.

4. Define the **Base level** position. This can be a user set value (eg 50ppb) or a number of Data options based on the Z value of the line. This includes the minimum, maximum and average data values.

Note If a Scale or Translate factor has been applied to the Z field, and/or an Offset field set, the User Defined Base value will need accommodate these modifications.

5. Enable the **Constant colours** option, and set the colour using the appropriate button.

6. Press Apply or OK to visualise the colour fill.

Decimation

Your dataset can be decimated using the options at the bottom of the **Fields tab.** The **Decimation** or station selection options allows the rendering performance of Discover 3D to be improved when dealing with very large datasets. A number of decimation options are available, including **Fixed rate** (where you specify a sample ‘skip’ factor) and **Compression**.

11 Working with Directional Vectors

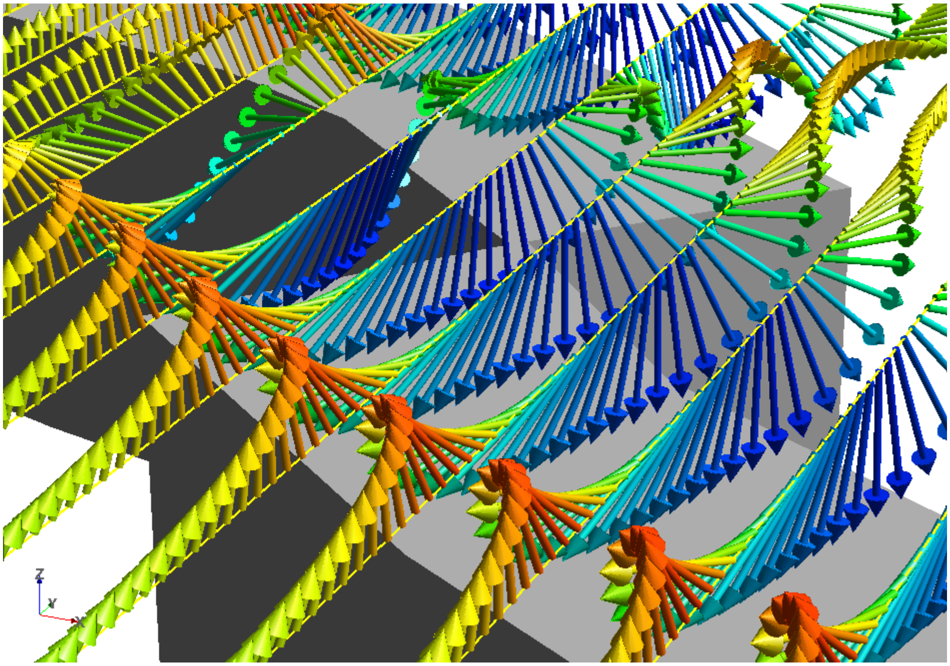
A Directional Vector data object type is available in Discover 3D, designed to display vector and tensor data:

A 3D vector is defined by its origin and the x, y, z components or an origin with amplitude, dip and azimuth. The first is used to visualise vector magnetometer data and the second for dip and strike style measurements.

A tensor on the other hand can be thought of as a collection of three vectors all located at the same origin.

Discover 3D uses a database to provide the X, Y, Z coordinates and component fields. The direction of the vector can be defined by rotation angles (origin and rotations), requiring two angles – a Y-axis rotation and a Z-axis rotation.

Use the **Display Directional Vectors** tool from the Data Objects toolbar, or the **Display>Directional Vectors** menu item to display data already open in the 3D window.



An example of 3D Vector series with modulated colour and size applied.

Changing Directional Vector Display Properties

The display of directional vectors is controlled from the **Directional Vectors Properties** dialog. To display the **Directional Vectors Properties** dialog highlight the **Directional Vectors** branch in the Workspace Tree and select **Properties** from the right-mouse click shortcut menu. The **Directional Vectors Properties** dialog contains eight tabs to modify or control vector data in the Discover 3D window:

- *Vector Tab*
- *Lines Tab*
- *Origin Tab*
- *Components Tab*
- *Tail Tab*
- *Head Tab*
- *Colour Tab*

Clicking the **Apply** button will apply the chosen options to the display. Selecting the **Auto Apply** option will automatically update the display after every change. For large datasets, turn off Auto Apply so that multiple changes are applied at the one time rather than waiting for the display to redraw between each change.

Vector Tab

The following parameters can be controlled with this dialog:

Mode

- **Vector rendering**: When rendering vectors, define the origin coordinate and the components. The components can be sourced from different fields.
- **Tensor rendering**: When rendering tensors, define the origin coordinate and up to three sets of components (one for each vector in the tensor). The components for each vector in the tensor must be sourced from a single field. It follows that field needs to be multi-banded.

Data source

Click the box and select how the vectors are defined. A vector can be defined by two points in space (origin and head positions). In this case the ‘components’ specify an additional point in space. It can be defined by vector components (origin and components) where each component specifies the displacement of the head from the origin in a certain dimension. Finally, the direction of the vector can be defined by rotation angles (origin and rotations). In 3D this requires two angles – a Y axis rotation and a Z axis rotation. In 2D this requires a single Z axis rotation.

In the first two cases, the length of the vector is defined by the specification of the origin and components. In the last case the length remains undefined and will initially be assigned a unit length.

Centre the vectors on the origin position

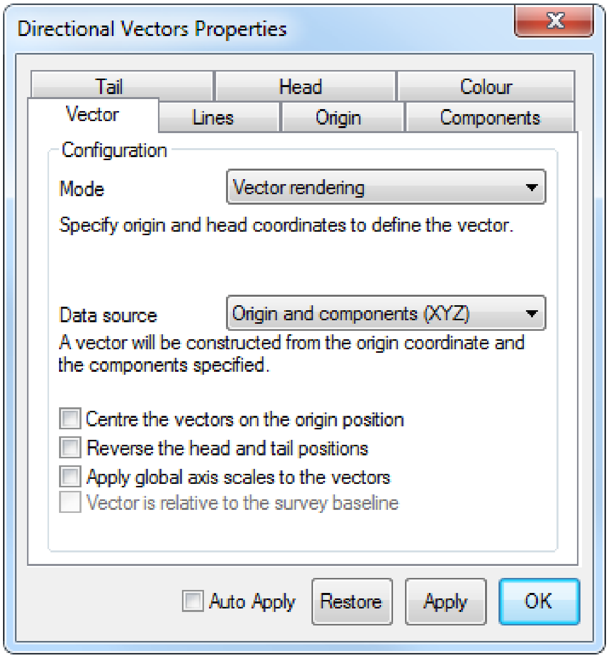
Moves the origin and head positions after the vector has been defined so that the vector is centred on the origin position. This is useful when displaying tensors and will result in the familiar tensor ‘hedgehog’ display.

Reverse the head and tail positions

As a final processing step, reverses the head and tail positions so that the head is rendered at the origin.

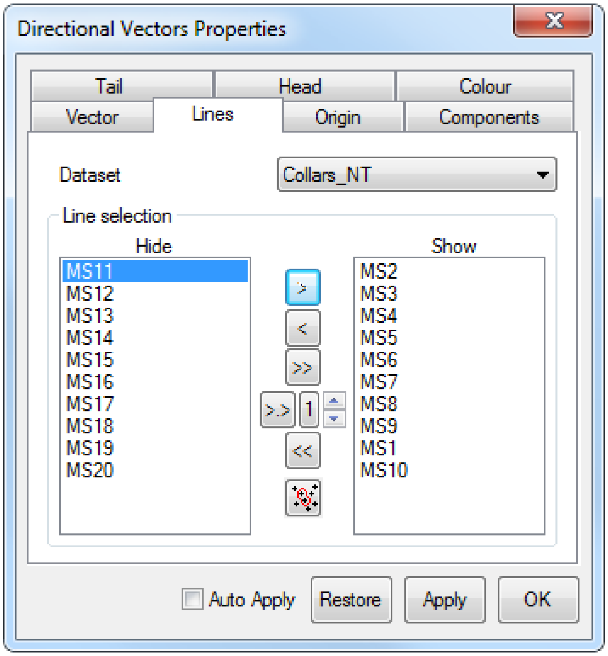
Apply global axis scales to the vectors

Generally we want to see vectors displayed in isotropic space. If this option is ‘off‘ then regardless of the scaling of the X, Y or Z axes, the vectors will always be displayed in isotopic space. If turned on the vectors will be scaled along with the space.



The Vector tab dialog specifying the mode and data source.

Lines Tab

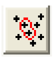


Lines selection dialog

The **Lines** tab provides data subsetting control. Select the Lines dataset (if more than one is open) from the **Dataset** pull-down list.

Select the lines to display in the 3D window from the **Line Selection** list. Use the arrows to select or deselect lines by moving them between the **Hide** and **Display** windows.

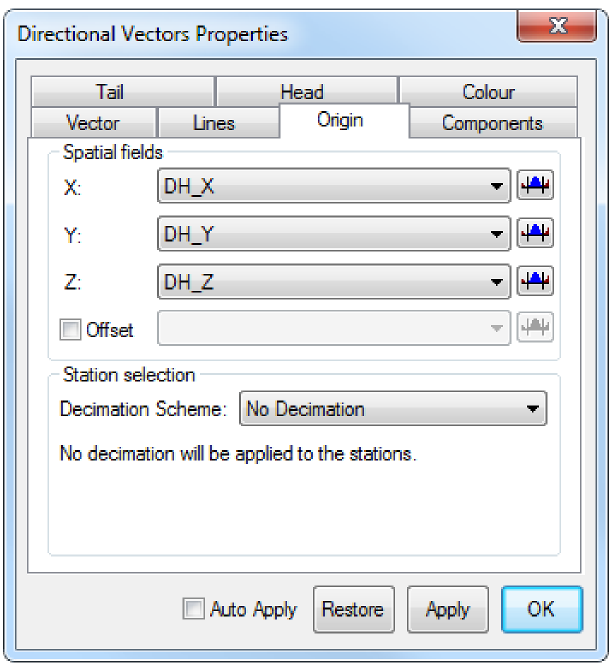
You can quickly select every 2nd line from the dataset by using the subset arrows button. At 1 this selects every line, 2 every second, 3 every third line etc.



Alternatively, use the button to make a map window selection of point data to display.

Origin Tab

The source data for the origin of the vector or tensor are defined on the **Origin** tab dialog. Specify the X, Y and Z fields if these have not been automatically selected for you. The optional offset field defines an additional offset which is applied to the Z coordinate by simple addition. If the vector components are defined as an additional coordinate then this offset will be added to the Z coordinate of the component data as well.



The Origin tab for the allocation of the spatial fields

The **Field Data Conditioning** button adjacent to each of the field selection lists opens the **Field Data Conditioning** dialog. This dialog allows the selected field to be transformed via a number of methods:

- It is of most use for applying a scaling factor to the **Offset** field, as it is unlikely that the magnitude of the offset field (e.g. an assay or geophysical field) will be comparable to the Z field.
- It may also be useful in applying a **Translation** factor to your Z field when data is coincident with a draped aerial photo (and therefore partially obscured: e.g. translating the data by 2m above the Z field value.

For more information on data conditioning, see the *Field Data Conditioning Tool* section in this chapter.

Decimation

On the **Origin** tab, the **Decimation** or station selection options allows the rendering performance of Discover 3D to be improved when dealing with very large datasets. A number of decimation options are available, including **Fixed rate** (where you specify a sample ‘skip’ factor) and **Compression**.

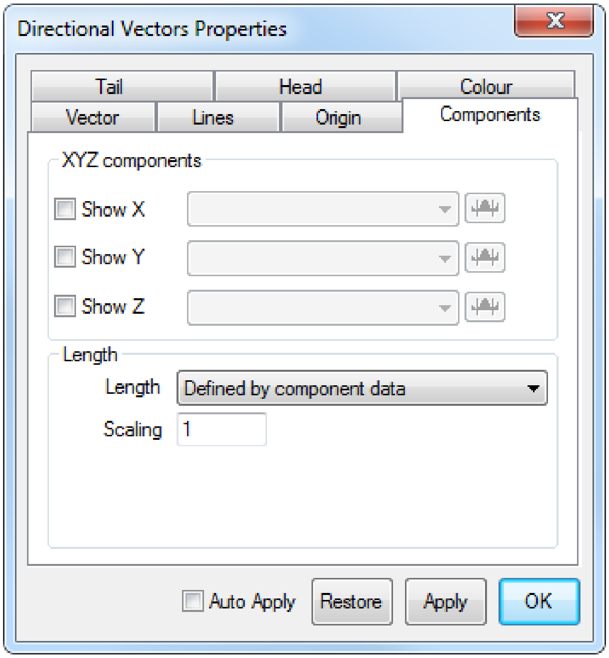
Components Tab

The **Components** property tab allows you to define the data source for up to three vector components or the data source for up to three vectors of a tensor.

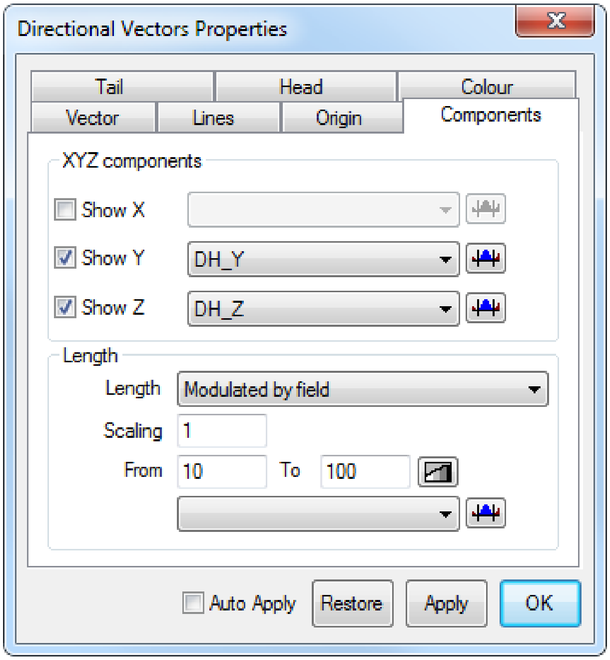
Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

When rendering a vector select the field combination for each of the components. If specifying rotations the X field buttons will be grayed out.

When rendering tensors you can specify the components or rotations for up to three vectors which comprise the tensor.



The Components tab dialog for specifying the X, Y and Z components of the vectors.



The Components tab dialog for specifying the Y and Z rotations of the vectors.

In rotation mode, the adjacent Pi button opens the **Rotation Parameters** dialog. This allows the direction of rotation to be altered (for instance to orientate positive dip values downwards, set this to -1 to reverse the angle direction). The angle units can also be set to radians or a fixed offset applied.

Length Modulation

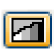
The total length of the vector or tensor will be determined by the choice of components or set to a unit length if the components are defined by angles. For display purposes this is usually inappropriate and so the length of the vector can be set, scaled and modulated in a variety of ways.

On the **Components** property tab the total length of the vector/tensor can be set to:

- **Fixed Value** – with user defined scale and length.
- **Defined by Component data** – calculated from input components with user defined scaling.
- **Defined by field** – length of each vector is defined in another dataset field using the absolute value with user defined scaling applied.

Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

- **Modulated by field** - length of each vector is defined in another dataset field mapped to a user entered maximum and minimum length and scaling.



Advanced size mapping options are available from the adjacent button which opens the **Data Mapping** dialog. For more information, see *Advanced Colour Mapping*.

Tail Tab

The **Tail** tab page allows you to define if the vector tail is shown as a **Line** or **3D Tube**.

The style of the Line or the 3D tube shape is set under the Line Style or Tube Style button.

If you are rendering a tensor then there will be three buttons allowing you to set the colours of each vector in the tensor individually. This is useful when rendering tensor ‘hedgehogs’ as the X, Y and Z vectors can be rendered using a different colour or style.

The thickness of the tail can either be a fixed value or it can be proportional to the length of the tail. In this case you must specify the minimum and maximum thickness values. Thickness modulation is only available when rendering the tails as tubes.

Head Tab

The **Head** tab page allows you to specify both the size and the rendering style of the head.

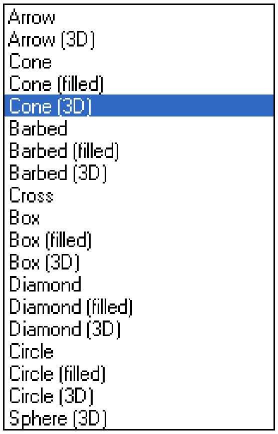
There are two rendering styles:

- **Display at tip** - the most common style where the head symbol will be rendered at the tip of the vector.

- **Display tail to tip** - the tail of the vector is not displayed and the head is rendered at the origin position and stretched over the length of the vector. You can use this option to create a ‘solid’ vector display.

When activated the **Anchor at head tip** check box option will ensure the tip of the head coincides with the head of the vector, hence the head symbol will not “lengthen” the vector. If this option is turned off the symbol is rendered beyond the end of the vector.

The head can be rendered using a variety of symbols listed below. Some of these symbols are simple line-work, others are flat filled polygons and others are 3D shapes. Only the 3D shapes use lighting.



The Vector head symbol style options list.

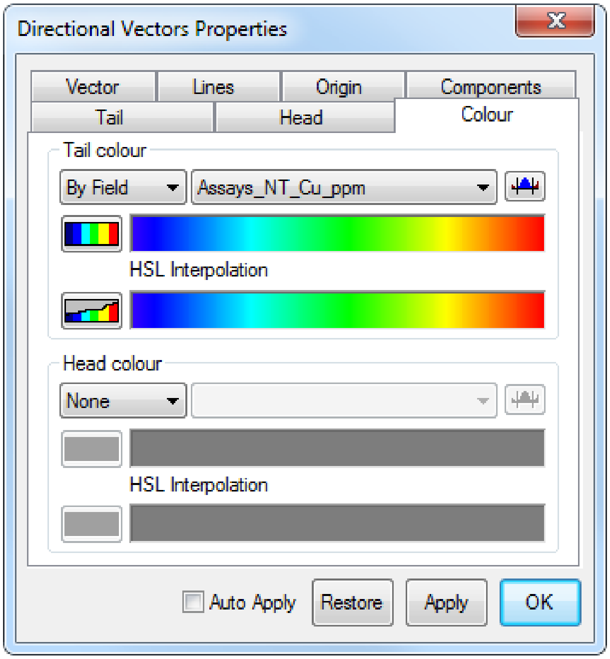
If you are displaying tensors and you choose the *Tail to tip* style option then only the *Diamond 3D* symbol option is available from the list. In this case a diamond is constructed from the three vectors of the tensor. The diamond is centred on the origin in this case.

The head size can either be a fixed size, or it can be proportional to the length of the tail. In this case you must specify the minimum and maximum size values. The size can also be modulated by a dataset field.

Advanced size mapping options are available from the adjacent button which opens the **Data Mapping** dialog.

Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

Colour Tab



The Colour tab dialog of Directional Vector Properties.

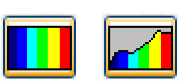
The **Colour** tab page allows you to define the colour modulation setting for both the tail and the head individually.

Colouring options for the Vector/Tensor Tail and Head include:

- **None** - the element (either tail or head) will be rendered using the line style properties defined in the Tail tab dialog.

- **By Length** - the colour of the head or tail is modulated by the length of the vector. Data mapping can be applied but note that the mapping dialog will only be able to display statistics of the data in this case if the vectors have already been displayed. Prior to this no length information is available.

- **By Field** - choose any field in the loaded database to colour modulate the tails and heads by. Data conditioning can be applied to the field and a data transform and mapping must be specified.

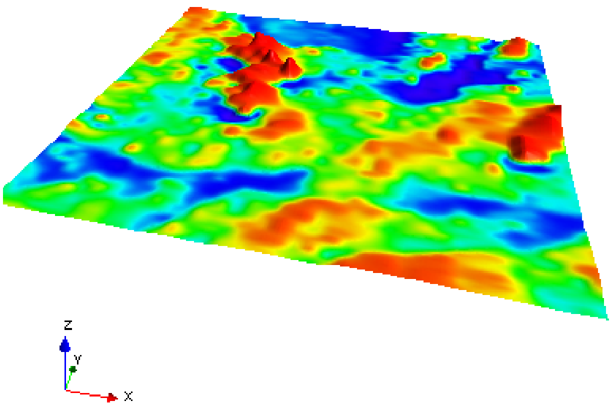


The colour scale used and colour mapping for modulating by length or a field, can be set under the respective buttons for both the head and the tail.

Specific ranges or attributes of the dataset can be selected with the adjacent Data Conditioning button, as well as null values removed. See *Field Data Conditioning Tool* for detailed information on this tool.

12 Displaying Gridded Surfaces in 3D

Discover 3D allows gridded surfaces created with Discover (**Surfaces>Create Grid**) or Discover 3D (**Grids>Surface Gridding**) to be displayed and modified in the 3D environment in association with other data sets (e.g. drillholes, imagery, etc). Surfaces may represent topography, geochemistry, geophysics, depth-to-basement modelling, etc., with a wide range of supported grid formats (see *Supported Grid Formats*).



Initial view of a 3D magnetics surface

In this section:

- *Supported Grid Formats*
- *Loading a Grid into 3D*
- *Grid Flipper*
- *Creating a Grid Legend*
- *Grid Groups and Surfaces Branches*
- *Viewing Grid Information*
- *Changing Surface Display Properties*

For information on creating surface grids with Discover 3D, see *Creating Gridded Surfaces*. For information on creating grids with the Discover Surfaces module, refer to the *Discover User Guide*.

Supported Grid Formats

A wide range of industry-standard grid formats are supported and can be imported for use in Discover 3D. Compatible grids include:

ASEG GXF (.GXF)
Band Interleaved by Line (.BIL including .HDR)
Encom grid (.GRD)
ERMapper (.ERS)
ESRI/Arc Binary (.ADF)
ESRI/Arc ASCII (.ASC)
Geopak (.GRD)
Geosoft (.GRD)
Landmark (.GRD)
MapInfo grid (.MIG)
Minex (.XYZ)
Surfer ASCII (.GRD)
Surfer binary (.GRD)
USGS (.USG)
USGS DEM (.DEM)
USGS SDTS (.TAR)
Vertical Mapper (.GRD)

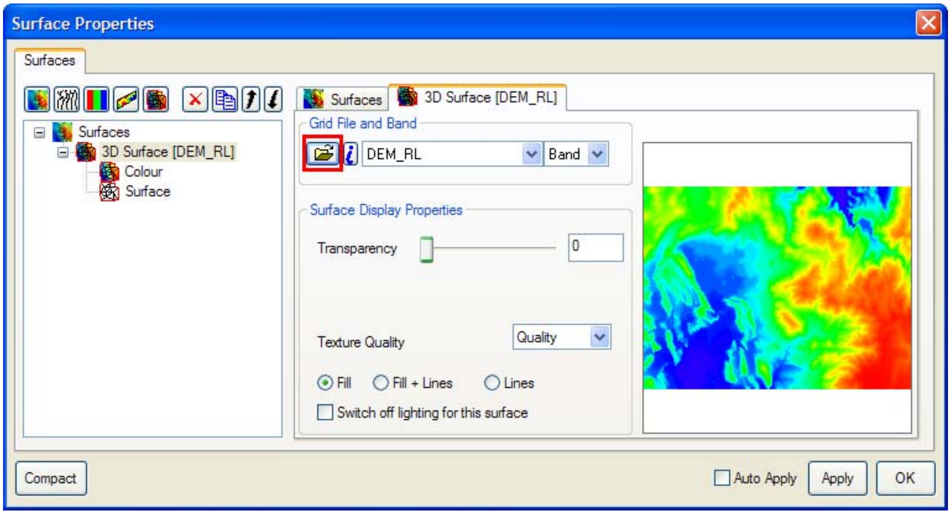
Loading a Grid into 3D

The easiest way to display a gridded surface in 3D is to have it open in MapInfo/ Discover, and use the **Discover 3D>View Surface in 3D** tool (see *Displaying Surfaces in 3D*). Alternatively, you can drag-and-drop surface files into the 3D window from File Explorer.

Gridded surfaces can also be opened from directly within Discover 3D by adding a **Surface** branch to the Workspace Tree and browsing for the target grid. There are three methods of creating a Surface branch:

- Using the **Display Surface** button.
- Using the **Display>Surface** menu option.
- Right-clicking on the **3D Map** branch in the Workspace Tree and selecting **Display Data>Surface**.

Double click on this Surface branch to open the **Surface Properties** dialog.



Surface properties dialog with the File Open button highlighted

Use the **File Open** button within this dialog (middle of the dialog) to browse for the target grid file. Access to a range of different grid formats is possible (see *Supported Grid Formats*). Once a grid file is specified and the **Open** button is clicked, a preview of the image is displayed in the right of the dialog.

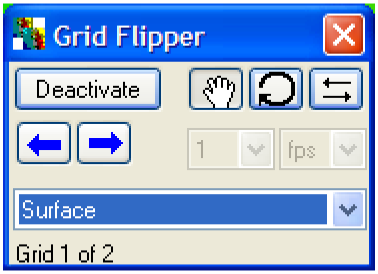
Note

Control changes made in the **Surface Properties** dialog can be invoked immediately they are changed if the **Auto-Apply** option (at the base of the dialog) is enabled. This option allows real time manipulation of imagery enhancement such as shading, colour changes etc.

If experimenting with the display options of a large grid, disable the **Auto Apply** option to prevent having to wait for the 3D Display to update/redraw after each change. Instead use the **Apply** button when the previewed image is satisfactory.

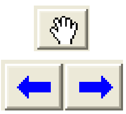
Grid Flipper

Within the Discover 3D window, the **Grid Flipper** allows the view to be rapidly switched between any open grid surfaces within a single Surface group (see *Grid Groups and Surfaces Branches*), i.e. multiple surfaces need to have been added to a single Grid Group as for example 3D Depth or Pseudocolour Surfaces. This tool can be accessed via right-clicking on the Surface branch in the Workspace Tree and choosing **Open Grid Flipper**, or by selecting **Tools>Grid Flipper**. It requires multiple grid surfaces to be available in the selected Grid Group before being enabled.



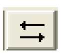
Grid Flipper dialog

Three modes are available once the **Activate** button is pressed:



- **Manual** mode allows grid selection either via **Previous** and **Advance** buttons, or through the pull-down list.

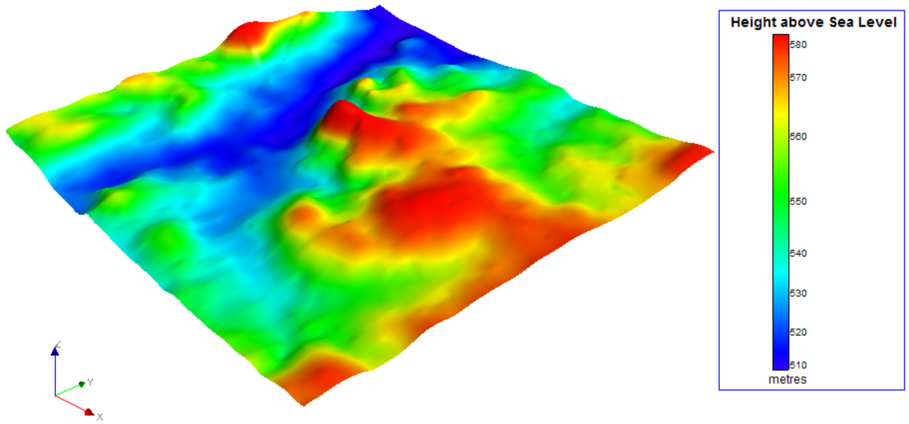
- **Continuous** mode will flip through each grid in turn with a user specified delay. The delay control has 2 time types: fps (flips per second) and spf (seconds delay per flip).



- **Strobe** mode flips to the next grid and then back to the starting grid again. The delay control has two time types: 'fps' (flips per second) and 'spf' (seconds delay per flip).

Creating a Grid Legend

A floating Colour Bar can be added to the 3D display and referenced to a grid surface, allowing easy visualisation of the relationship between colours and numeric data ranges. This is detailed in the *Floating Colour Bar* section of the **Discover 3D Interface** chapter for further information.



Example of a Floating Colour Bar linked to a DEM grid surface. A colour scale indicating elevation values is displayed

Grid Groups and Surfaces Branches

A **Grid Group** consists of a variable number of surfaces, image layers and vectors (such as contours) that make up a surface display. The structure of the display is shown in the tree view of the dialog. By selecting any item in the tree, the displayed controls change to show the properties of the selected item.

A view is rendered by drawing all surfaces from the bottom branch of the tree upwards. Surfaces at the top of the tree obscure surfaces lower down the tree (although the transparency control can modify this).

There are four types of image surfaces that can be draped over a **3D Depth Surface** and each can have one or more layers. These types include:

- **Pseudocolour** with Colour and Intensity layers.
- **RGB** surface has Red, Green, Blue and Intensity layers.
- **Located Bitmap surface** contains a layer that is assigned an ER Mapper algorithm/ECW.
- **Contour** surface with a Vector layer.

Surfaces branches can be moved up or down the **Grid Group** to control the draw order. Visibility of individual surfaces and layers can also be turned on or off from the tree branch shortcut menu.

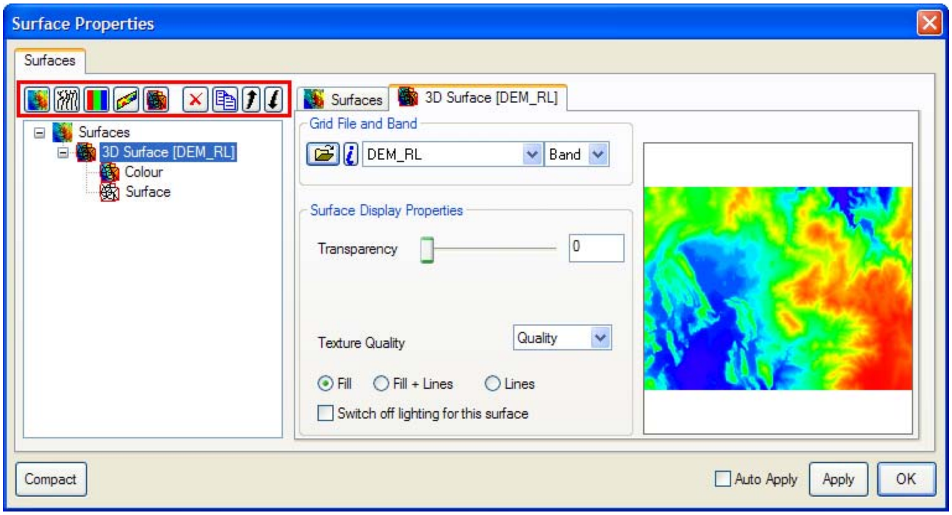
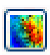


Image surface control.

The dialog allows you to directly add one of the supported display types to the surface tree by selecting one of the following buttons:



Add a new **Pseudocolour** surface to the surface tree.

Add a new **Contour** surface to the surface tree.

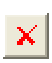
Add a new **Red:Green:Blue** surface to the surface tree.

Add a **Located Bitmap surface** (ER Mapper) to the surface tree.

Add a **3D Depth Surface** to the surface tree.

Other buttons in the control area provide the following functions:

Add a new surface to a view.



Delete the selected surface.

Duplicate the selected surface.

Move the selected surface up or down.

Display **Information** (location extent and row/column details) about the selected grid.

The surface tree provides access to properties of the various branches of 3D Surfaces. If you select the Pseudocolour layer and right click a shortcut menu is presented as shown.

Options provided by the shortcut menu are:

Turn On **Turn Off**	A selected plot, surface or layer can be switched on or off. If switched off, it will not be rendered.
Move Up **Move Down**	Move Up and Move Down duplicate the functionality of the arrow buttons shown above. Move surfaces up to ensure they overdraw surfaces lower down. Contours should generally be at the top of the tree.
Duplicate Surface	Duplicates a surface. Duplication of a surface copies all the attributes of the original surface and layers.
Delete Surface	Deletes the selected surface. Identical function to Delete button shown above.

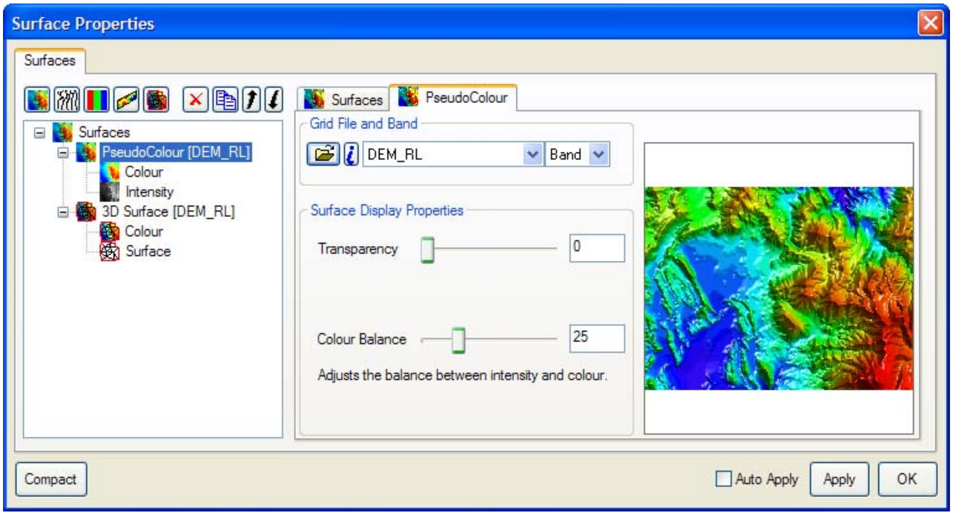
Located Bitmap Surface	Change the surface type to ER Mapper Algorithm or ECW file. An existing ER Mapper algorithm (.ALG) or .ECW file can be used to display an image layer. Refer to *Draping Located Images* for additional information
Pseudocolour	Change the surface type to Pseudocolour. The Pseudocolour image representation uses both colour and intensity control branches to render an image. The colour of the image is determined by a look-up table specified in the colour branch. This form of image display is the default style. Refer to *Pseudocolour Surfaces*.
Red:Green:Blue	Change the surface type to RGB. This form of display uses multi-band grid files (or multiple grid files) to present images. Examples of this form of display include satellite or digital photography. Refer to *RGB Image Surfaces*.
Contours	Change the surface type to Contour. Contours can be added to image maps to enhance displays. Refer to *Surface Contours* for additional details.
3D Surface	Change the surface type to 3D Surface.
Export to EGB (only available for 3D Surface)	Opens the **Located Image Settings** dialog allowing the selected surface to be saved as a Georeferenced Image at a user-defined resolution and image type.

Transparency and Colour-Intensity Balance

There are **Transparency** and **Colour–Intensity** balance slider controls on the **Surfaces Properties** dialog. Transparency is not available for contour surfaces. Colour-Intensity balance is not available for ER Mapper algorithm surfaces and contour surfaces.

Each surface is combined with the underlying surfaces using a transparency transform that allows underlying imagery to be seen through overlying surfaces. To ensure a surface completely obscures underlying surfaces, set the transparency value to minimum (opaque). To view underlying surfaces and hide the selected surface, set the transparency value to maximum (invisible).

Pseudocolour and RGB surfaces have colour-intensity balance slider controls on the **Surfaces Properties** dialog. This allows you to easily adjust the balance the contributions the colour and intensity layers make to the final image. When set to minimum, colour and intensity are in balance. As the balance level is increased the intensity is progressively washed out and the colour saturation increases.



Pseudocolour Properties dialog for a 3D surface display

Select the button to specify a grid to be assigned to a Pseudocolour draped surface. Note that **Colour** and **Intensity** levels appear beneath the **Pseudocolour** surface.

Texture Mapping

Within Discover 3D a computer display method called ‘texture mapping’ is used to quickly render images that drape over a surface. A compromise is made between the quality (resolution) of the drawn image and the speed of drawing. An upper limit on the resolution that can be physically supported is imposed.

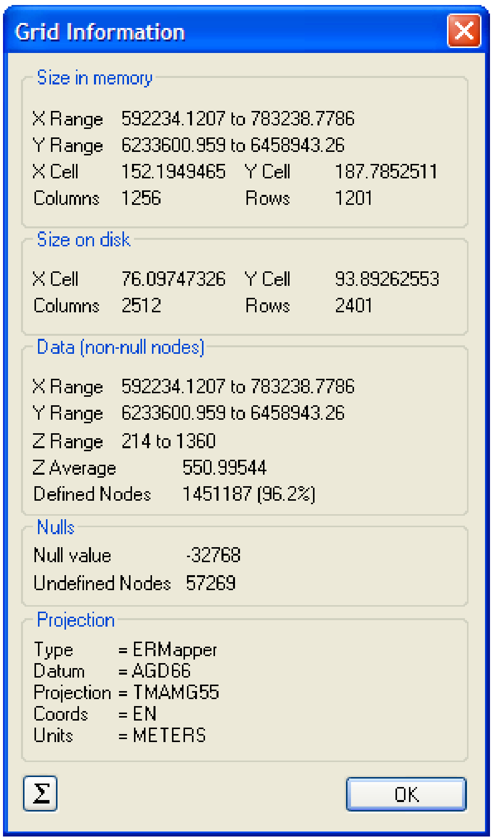
This limit is imposed by the installed graphics card drivers not by the software or its OpenGL display system. An upper display limit (not data limit) of 1024 x 1024 was found to be the maximum reliable texture size several years ago. Limits imposed by modern graphics cards are now higher.

Two options within Discover 3D are provided to control texture map size:

- One method uses a simple **Texture Quality** setting in the properties dialog of the 3D Surface. This allows Low/Medium/High Quality settings.
- The second method provides greater control and is available via the **texture size** option on the **System** tab of the **Options** dialog in the Discover 3D menu in the 2D interface.

Viewing Grid Information

It is often useful to obtain information about a grid prior its use in a display. The **Grid Information** button can be used to obtain statistics that detail the location, projection and grid attributes.



Grid information resulting from selecting the button on the Grid Display dialog

Changing Surface Display Properties

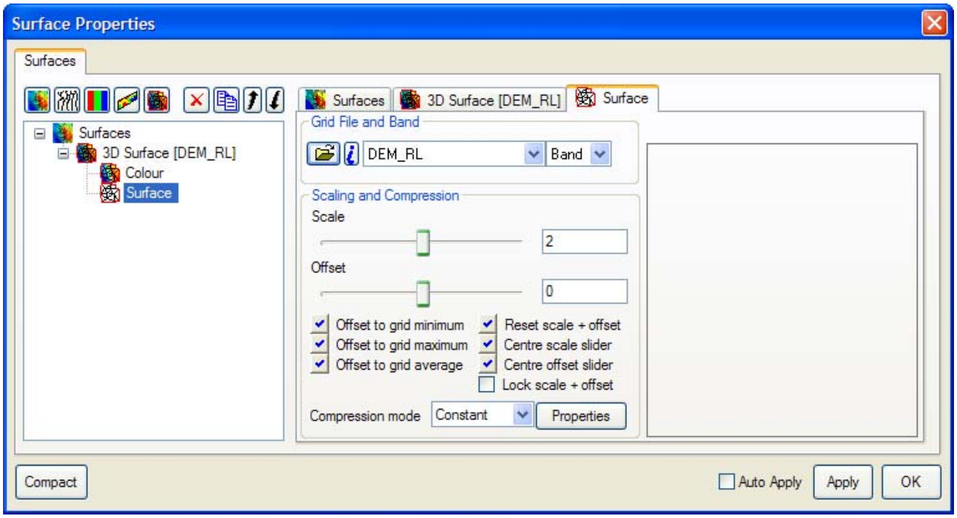
- *Scaling Grids*
- *Grid Compression*
- *Offsetting a Grid*
- *Rendering Surfaces*
- *Pseudocolour Surfaces*
- *RGB Image Surfaces*
- *Draping Located Images*
- *Surface Contours*

Scaling Grids

You may wish to exaggerate the vertical scale of a surface by changing the Z axis scaling. This can be done on the **Surface Properties** dialog by either manually entering a Scale value or by using Scale slider bar. For application of larger scaling factors, entering a **Multiplier** will multiply the **Scale** factor by this value. Note that when adjusting a Z Scale, this applies only to the 3D Surface of that surface layer.

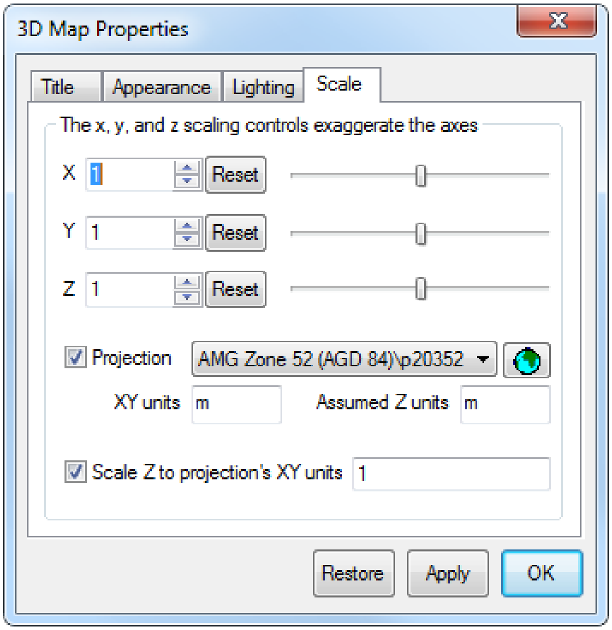
Note Surface scaling applies an exaggeration to the surface, by subtracting the minimum values of the grid before mulitplying:
`Zfinal = ((Zinitial - Zmin) * Zscale) + Zmin + offset`

Once a scale is determined it can be locked by enabling the **Lock scale + offset box**. This will ensure that introducing other objects (like lines, other surfaces or graphical objects) does not readjust the scale.



Surface dialog controls for scaling applied

If you want to adjust the Z scaling of all surfaces in a view, this can be done from the **Scale** tab of the **3D Map Properties** using the Z slider bar.



Adjusting the 3D Scaling item in the 3D Map Properties dialog

Grid Compression

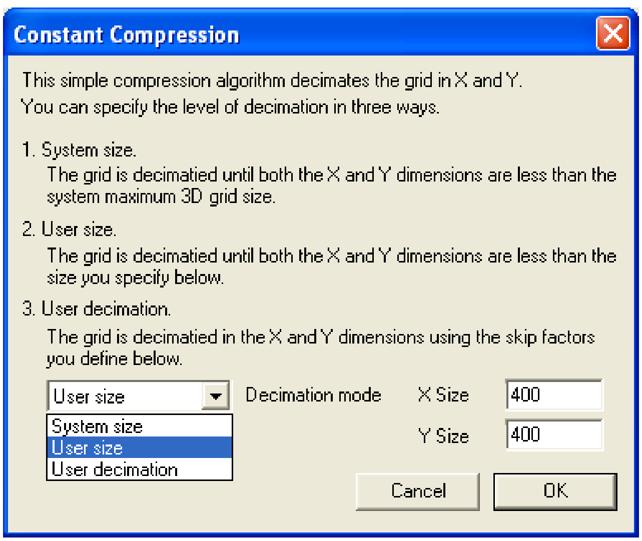
Another control on the **Surface** tab allows compression of grid surfaces. This feature permits Discover 3D to display and manipulate large, complex surfaces quickly with little loss of surface detail. When Discover 3D displays surfaces, it applies a display algorithm that by default allows the presentation to be exceptionally fast but also accurate in its representation of a surface.

The **Compression Mode** pull-down list provides three compression options:

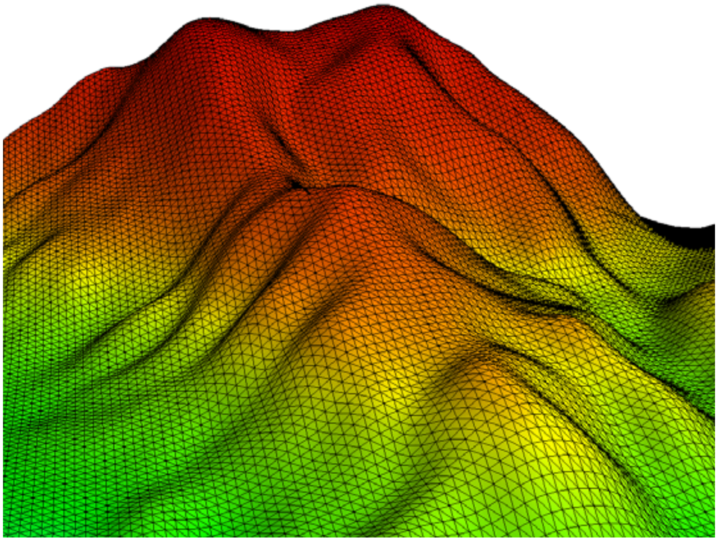
- **None** - no compression is performed and the grid is displayed at full resolution.
- **Constant** - a simple compression algorithm decimates the grid in X and Y. See *Constant Compression*.
- **Variable** - attempts to retain the boundary of the grid, including internal holes. See *Variable Compression*.

Once a mode is selected, click the adjacent **Properties** button to open the relevant compression controls.

Constant Compression

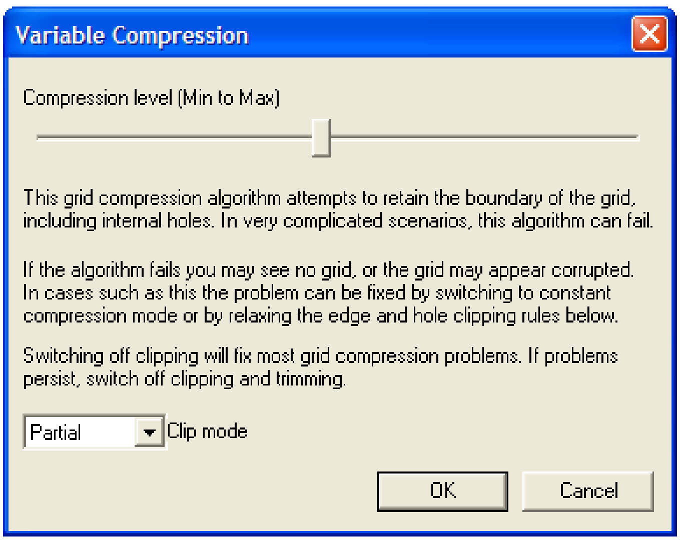


A simple compression algorithm decimates the grid in X and Y. The decimation parameters can be based on the **System size**, **User size** or **User decimation**. The final two options require user definition of parameters.



Line display of a grid illustrating Constant compression

Variable Compression



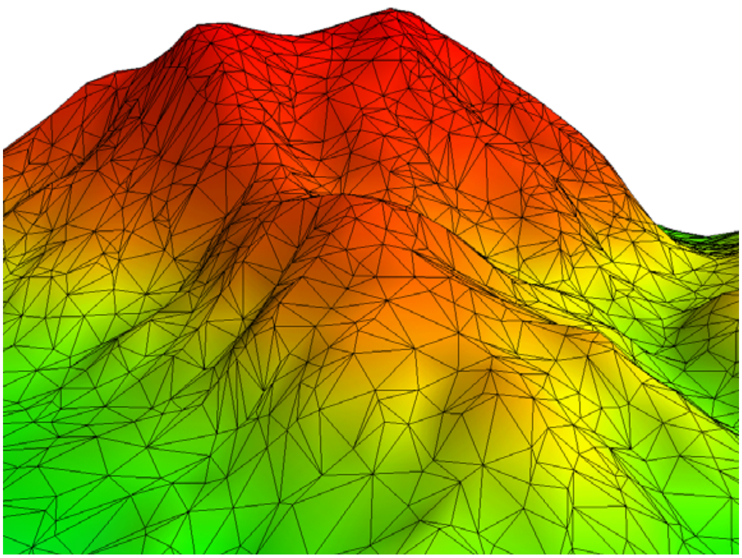
Variable compression uses the Tomek algorithm, which variably adjusts to the data; this attempts to retain the boundary of the grid, including internal holes. The compression level is adjusted via a slider bar and three clipping levels are available:

- **Complete** - the grid boundary and holes are preserved,

- **Partial** - the grid boundary and holes may not be perfectly preserved, and
- **None** - the grid is rendered as a convex hull with no holes preserved.

The degree of compression can also be assessed by displaying the Lines (triangulation) used in the surface display.

It is recommended you maintain surface compression above 80% for best data results. If you want less than 80% compression, switch all compression off.

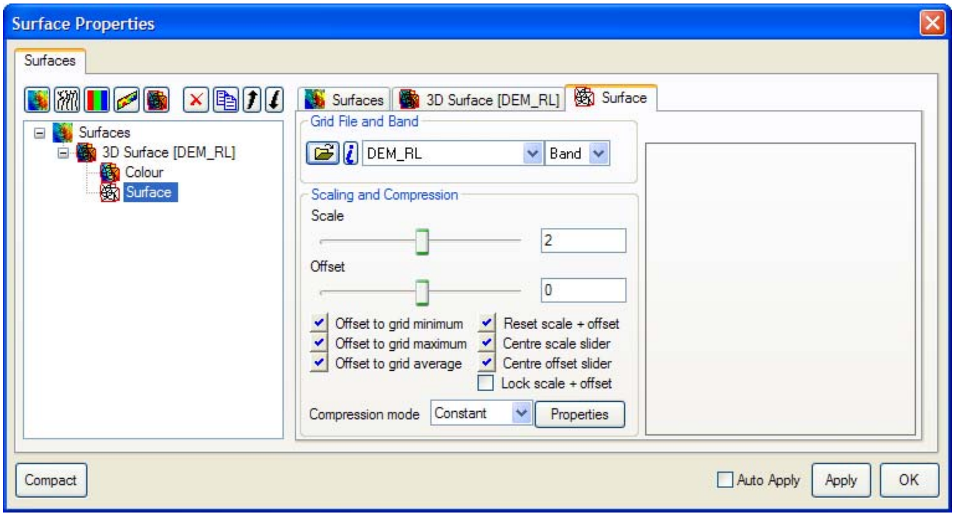


Line display of a grid illustrating Variable compression

Offsetting a Grid

The position of a grid surface in space depends on the data values assigned to the Z field of that surface. For example, the effective elevation of a topography surface may be, say 300 metres. However, it may be desirable to place a soil geochemical surface above the topographic surface and the amplitude or Z values of the geochemical surface may be for example, 2,000.

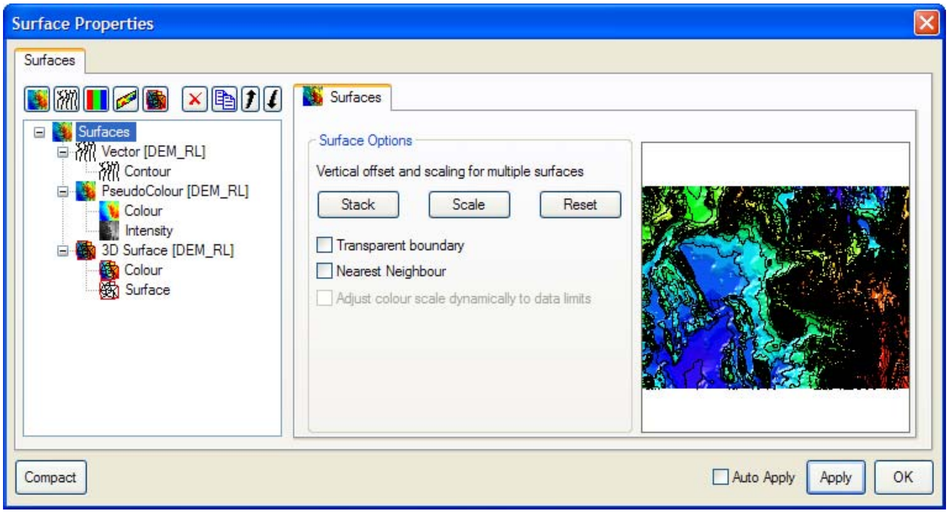
You can control the Z offset of surfaces by entering a specific offset amount in the **Surface** tab, or by interactively raising or lowering the surface using the slider bar. Alternatively, alter the Z offset automatically to the grid surface minimum, maximum or average value by selecting the appropriate check box.



Scale and Offset controls in the Surface Properties dialog

Once an offset is determined it can be locked by enabling the **Lock scale + offset box**. This will ensure that introducing other objects (like lines, other surfaces or graphical objects) does not readjust the offset.

Note To automatically allow these surfaces to be brought together and stacked for viewing, select the **Grid Group** branch of the image tree and click the **Stack** button followed by the **Apply** button. Similarly, to present the surfaces with suitable scaling for the display window chosen, you can click the **Scale** button. The **Reset** button returns the surfaces to their initial default scales and offsets.



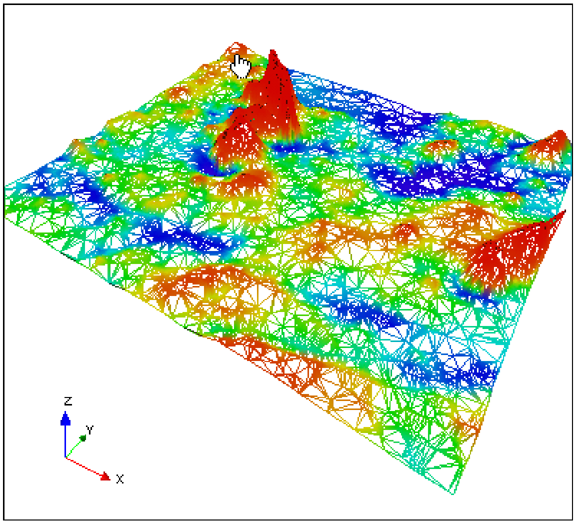
Using the Stack, Scale and Reset buttons of the Plot image dialog

The **Stack** button effectively adjusts the surface offsets to force all surfaces to lie within the display area. If you need to slightly move them up or down from their recomputed positions use the **Z Offset** controls (see above).

The **Scale** button operation adjusts the vertical exaggeration of the surfaces to optimally suit the 3D display. If adjustments to the exaggeration are still required, use the **Z Scale** slider controls (see above).

Rendering Surfaces

The **3D Surface** tab of the **Surface Properties** dialog has three options for **Fill**, **Fill + Lines** and **Lines**. These options relate to displaying the surface as a filled surface, or with the surface being presented as the triangulated surface when compression is used. By displaying the **Line** option only, you can make a judgement about the level of grid compression that is being used by the **Surface Compress** option (see above). The grid display compression tends to have few triangular facets in a presentation in low gradient areas and a high number in steep gradient areas. An example of **Line** display is shown below.



Lines displayed instead of the filled surface to assess the grid compression

On the top level branch in Surfaces, there are controls to:

- **Display Null Values as transparent** – any null values in the grid, which normally appear as white background, will be automatically set to transparent
- **Apply Nearest Neighbour Smoothing** – applies a real time smoothing filter

Note 3D Surfaces are always smoothed, instead display the layer as a psuedocolour surface along with the 3D surface to display unsmoothed.

Pseudocolour Surfaces

A pseudocolour surface consists of one or both of:

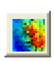
- A Colour layer in which a predefined RGB or HSL colour table is used to render the data from a single banded grid or a single band of a multi-banded grid. Colours can be assigned from user-specified RGB or HSL colour space stretches or from colour look-up tables which can display any colour combinations including greyscales. For more information see *Controlling the Colour Layer*.

- An Intensity layer that introduces a greyscale shading scheme based on an artificial sun azimuth and inclination. This feature is a powerful method of introducing colour gradients to an image to highlight trends and patterns in the data. Usually the grid used as a data source for the intensity layer will match the colour layer, but a different grid or grid band may be used if required. Note that with the **Auto-Apply** option enabled, real time sun shading is available in the displayed image plot as well as the preview. For more information see *Controlling the Intensity Layer*.

To create a Pseudocolour image:

1. Create a **Surface** branch in the Workspace Tree as described above.

2. Highlight the created **Surface** branch and access the **Properties** dialog.

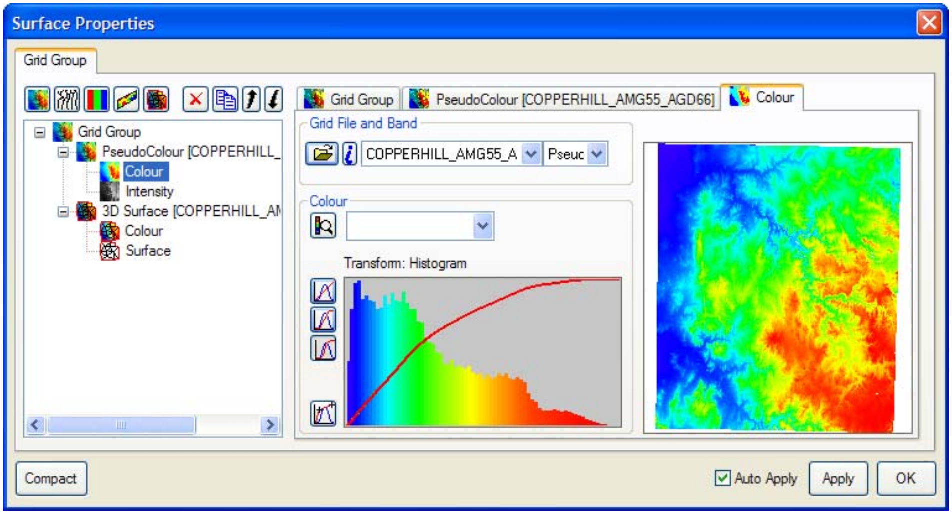



3. Click the **Pseudocolour** button to create a new Pseudocolour branch in the image tree. Select the **Grid Selection** button to specify a grid for both the Colour and Intensity layers (see *Supported Grid Formats*).

4. Once a grid is specified, a **Pseudocolour** preview image appears in the right side of the **Surface Properties** dialog.

Controlling the Colour Layer

Select the **Colour** branch in the surface tree to show the layer colour properties.



Colour layer controls for the Pseudocolour image

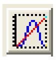
The properties available include:

Data Source – You can select and load a grid using the Load button. In most cases it is sufficient to load a grid from the Pseudocolour surface property page which loads the same grid into the colour and intensity layers. If the grid has multiple bands, the required band can be selected.

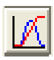
Colour – There are three methods of colour table specification, all of which can be accessed via the **Advanced Colour Properties** button:

- Specify the upper and lower colours and interpolate a specified number of colours in RGB colour space.
- Specify the upper and lower colours and interpolate a specified number of colours in HSL colour space.
- Load a colour look-up table that defines the RGB colours. The number of colours in the table is defined by the number of colour entries in the LUT file. This option can also be accessed from the pull-down list to the right of the Advanced Colour Properties button. If there are no colour tables in the list, Discover 3D has failed to identify any LUT files on your system.

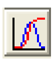
A histogram of the data distribution is displayed. The histogram shows the frequency of data values in the grid as well as the colour of the data values. The cumulative value of the histogram is also shown as a black line on the histogram graph. The transform from data to colour space is defined by the transform line that is shown on the histogram. By modifying the transform you can change the colour distribution across the image. Three simple options are available to do this:



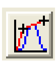
- **Linear Colour Mapping** – Click the button to enable linear colour mapping.



- **Band Pass Mapping –** Click the button to apply a bandpass filter to the transform (either Linear or Histogram). The percentage or value limits of the bandpass range can be specified in the **Colour Mapping** dialog accessed via **Advanced Colour Mapping** button (below). All data values outside the specified range will be assigned the minimum or maximum colour value and the colour stretch will then be restricted to the data within the bandpass. Bandpass mapping replaces a linear transform, but can be combined with a histogram colour mapping (as described below).



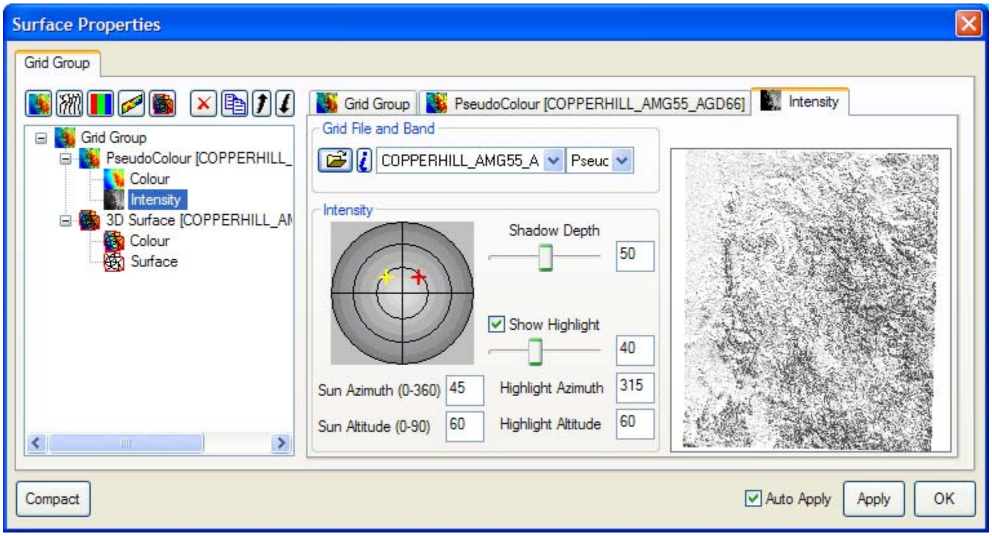
- **Histogram Colour Mapping** – Click the button to apply a histogram equalised colour transform.



More advanced data transform controls are accessed via the **Advanced Colour Mapping** button. This opens the **Colour Mapping** dialog, which can also be opened by double-clicking in the histogram preview area. For more information, see *Advanced Colour Mapping*.

Controlling the Intensity Layer

The **Intensity** layer of a Pseudocolour image is used for applying shading to the display. The shading applies a greyscale colour modulation to the pseudocolour surface.



Pseudocolour Intensity control dialog

Properties available include:

Data Source – You can select and load a grid using the **Load New Grid** button. In most cases it is sufficient to load a grid from the Pseudocolour tab in the **Surface Properties** dialog which loads the same grid into the colour and intensity layers. If the grid has multiple bands, the required band can be selected.

Intensity – The shading control of an intensity layer is interactive. A red cross displayed in a mapped ‘shade ball’ is used to control the graphic location of azimuth and altitude of a virtual sun. These two angles control the sun shading process. Specific angles can be entered in the appropriate fields beneath the ‘shade ball’. To graphically operate, select the red cross with the cursor and with the left mouse button pressed move the cross to an azimuth/altitude combination that best enhances the features required. The image displayed in the Preview window updates in real-time. If the **Auto-Apply** option is enabled, the main display window is updated in real-time similarly to the **Preview** area.

Note If the **Auto-Apply** option is enabled, adjustments to the shading intensity can be viewed in real-time.

The intensity of the shadow can be enhanced using the **Shadow Depth** slider bar and the **Enhance** setting. The shadow depth option can increase or decrease the intensity of the shadows and the result is displayed after you move the slider control. For a large grid, this update may take a few seconds. The **Enhance** setting can improve the shadow intensity of grids that have a very high dynamic range – for example, derivatives of magnetic data. Generally, this setting is not required.

The **Show Highlight** option allows the application of a second virtual sun/light source. This allows the lightening of desired shadow regions and further lightening of illuminated regions. The graphical location of the azimuth and altitude of this **Highlight** source is represented by the yellow cross in the ‘shade ball’. The **Highlight** positioning can only be altered by entering parameters into the **Highlight Azimuth** and **Altitude** fields.

RGB Image Surfaces

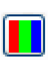
Red:Blue:Green (RGB) imagery surfaces are used when more than one band of data is to be used in the image display. RGB colour mode uses the fact that Red, Green, Blue and Intensity colour receptors are used to display up to four separate bands or layers of data at the same time.

Areas in an RGB image that appear predominantly red would have high values in the red layer of data and low values in the other layers and similarly for the other colours. Interpretation becomes more difficult when there are high values in two or more layers of data at the same time because composite colours are created that are not intuitive to understand. If the same image is loaded into each of the Red, Green and Blue layers the resulting image appears as shades of grey.

The RGB image format is especially useful for presentation of multi-band datasets such as Landsat or Spot satellite imagery or for geophysical interpretation, the multi-band data of radiometrics (Potassium, Uranium and Thorium) can be mapped to the various colours.

To create a Red:Green:Blue image:

1. Create a **Surface** branch in the Workspace Tree (see *Loading a Grid into 3D)*.
2. Highlight the created 3D Surface branch and display the **Surface Properties** dialog.

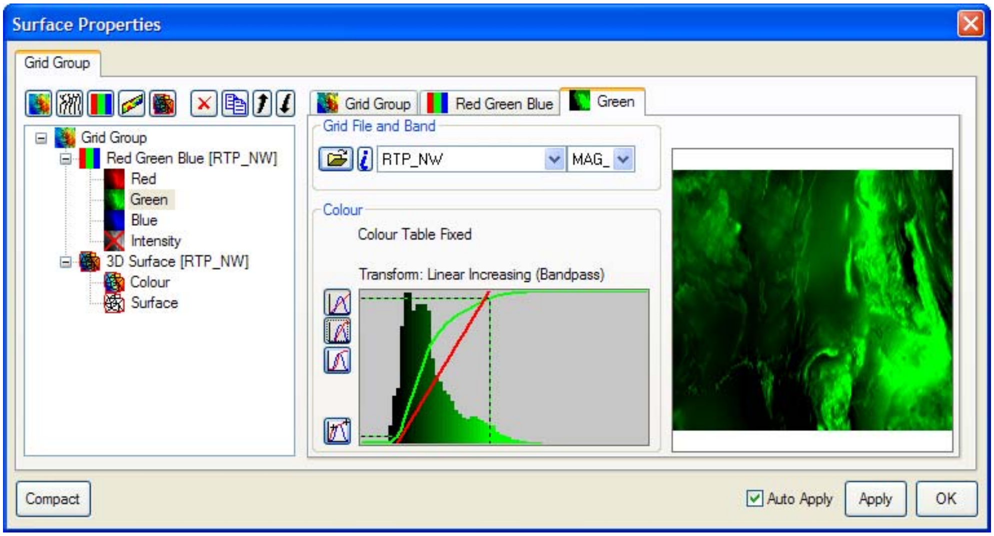


3. Select the 3D Surface in the Grid Group (see *Grid Groups and Surfaces Branches*).

4. Click the grid selection button to choose the required input multi-band grid (see *Supported Grid Formats*). Discover 3D assigns grid bands to the Red, Green and Blue layers by default.

5. Once a grid is specified, select the **Red:Green:Blue** layers to display the properties of the relevant layer. A preview image appears in the right side of the dialog.

The **Red:Green:Blue** layer has Red, Green, Blue and Intensity layers associated with it. By default, the Intensity layer is switched OFF (see *Controlling the Intensity Layer*).



Example of the Green Layer controls of a Red:Green:Blue image

Data Source – Select and load a grid using the **Load** button. In most cases it is sufficient to load a grid from the RGB **Surface Properties** tab that loads a multi-banded grid and assigns grid bands to the red, green, blue and intensity branches. If the grid has multiple bands, the required band can be selected.

Branches – Each of the Red, Green and Blue branches need to have their individual colour distributions adjusted according to a Linear, Band Pass or Histogram Equalisation process. Refer to *Controlling the Colour Layer* for details on this. In each case the look-up table appropriate to the colour layer is fixed.

Note You can only preview the combined RGB surface from the RGB **Surface Properties** tab (or the main plot window if **Auto-Apply** is enabled). Select the surface and ensure the **Show Preview** setting is enabled.

Draping Located Images

The **Add Located Bitmap Surface** option allows GeoTIFF, ERMapper Algorithm (.alg) , JPEG 200 (.jp2) and ECW georeferenced images (as well as images with a MapInfo Professional TAB file) to be draped over a grid.

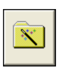
Discover 3D makes use of an ER Mapper software plug-in that displays ER Mapper algorithms and imaging including ER Mapper Compressed Wavelet (ECW) imagery. Also provided with the ER Mapper software support are a wide range of look-up tables that can be used in colouring images in Discover 3D.

The main benefit of ER Mapper support is to provide direct interpretation of ER Mapper algorithms. This means the complex image processing techniques made available from ER Mapper can be read and displayed within Discover 3D.

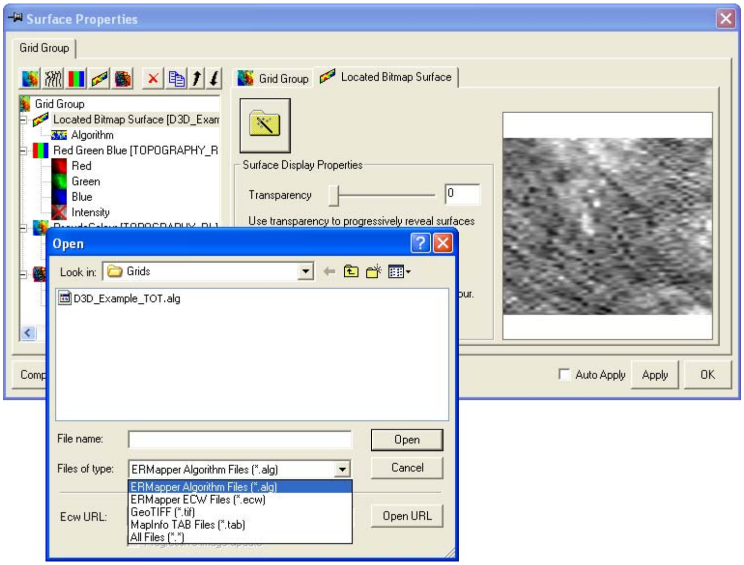
To display a georeferenced image over a grid surface:

1. Create a **Surface** branch in the Workspace Tree.
2. Highlight the created **Surface** branch and access the **Surface Properties** dialog.

3. Select the **3D Surface** branch and click the **Add Located Bitmap Surface** button.



4. Select the **Grid Selection** button and choose the appropriate file type (.TIF, .ALG, .ECW or .TAB). Browse for the appropriate file and click **Open** when selected. Alternatively, a URL can be entered from which to download an ECW image.



Bitmap selection and preview of the image

Once an image has been loaded a preview of the surface is displayed. Click the **Apply** button to redraw the Map.

Note that an ER Mapper algorithm cannot be modified. You cannot modify the histogram, colour look-up tables or data content. However, you can zoom, roam and pan in the same manner as other displayed images and maps.

Surface Contours

Grid files can be displayed in a Grid Group as an imaged contour vector layer.

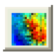
Note

Contours can be also be displayed as **vector data**. Create contours for the grid surface using the **Contour a Grid** option in the Discover Surfaces module. Then use the **Discover 3D>View Map Objects in 3D** tool to convert these contours into a 3D vector file. This option is recommended as it prevents the pixilation/ rasterization of contour lines as apparent with a **Vector Contour** branch.

Vector layers (such as contours) and image layers are treated separately because an image occupies all pixels of its display area and therefore it can hide any vector layers drawn below it. Consequently, the drawing order of images and contours is important. To ensure vectors are visible, they should lie ABOVE images in the plot tree. The **Move Up** and **Move Down** buttons (or shortcut menu options) should be used to ensure the correct order is created.

Note also that **Transparency** and **Colour-Intensity Balance** slider controls are not available for contour surfaces.

To create a Vector Contour branch:



1. Create a **Surface** branch in the Workspace Tree using the **Surface** button or the **Display>Surface** menu item.

2. Highlight the new **Surface** branch and display the **Properties**.

3. Create a new surface using the **New Surface** button.

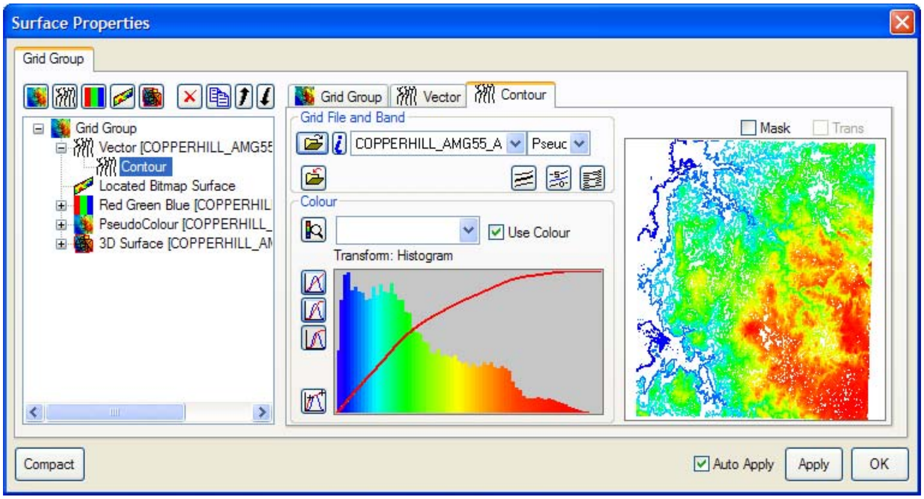
4. Add a **Vector** branch in the surface tree by clicking the **Add Contour Surface** button (shown left) or clicking the right mouse button with the **Grid Group** highlighted and selecting the **Contours** option.

   Click the **Grid Selection** button to choose the required input multi-band grid (see *Supported Grid Formats*).

5. Once a grid is specified, select the **Contour** layer to display the properties of the contours. A preview contour image appears in the right side of the **Properties** dialog. As contours can take some time to generate, disabling the **Auto Apply** option will prevent the 3D window display updating with any changes until the **Apply** button is selected.

Controlling the Contour Layer

A large range of controls for contours is available. They can be displayed as monochrome or coloured (according to a colour mapping scheme) and the contour interval and dropout can be specified. The contour dialog is shown below:

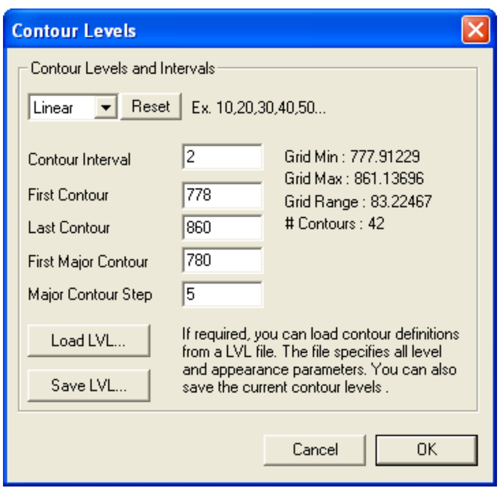


Contour control dialog and specifications for look-up tables

Available controls include:

Data Source - Select and load a grid using the button. It is convenient to load a grid from the **Contour** tab of the **Surface Properties** dialog. If the grid has multiple bands, the required band can be selected.

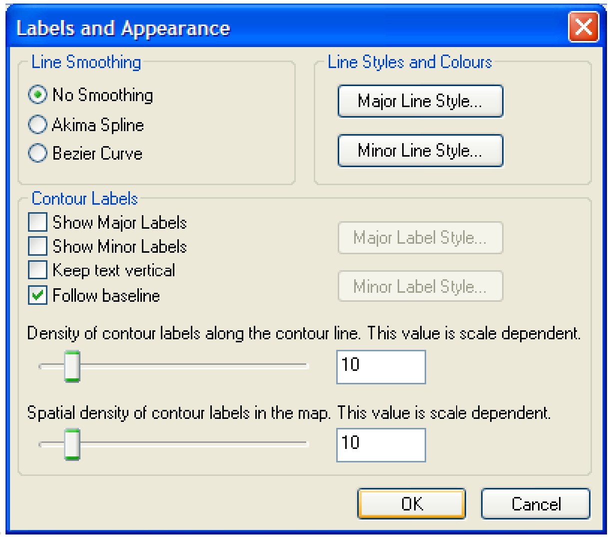
Contour Levels - You can specify contour intervals, levels and their distribution from the dialog displayed when this button is selected.



Select the distribution, contour levels or save/restore from a file

Contours can be distributed linearly, logarithmically or from an external file defining the contour levels (a .LVL file). Contour levels can also be limited to a range of data specified in the **First** and **Last** contour values.

Contour Appearance – Control of the contour appearance and labelling is provided in a dialog when the appearance button is clicked.



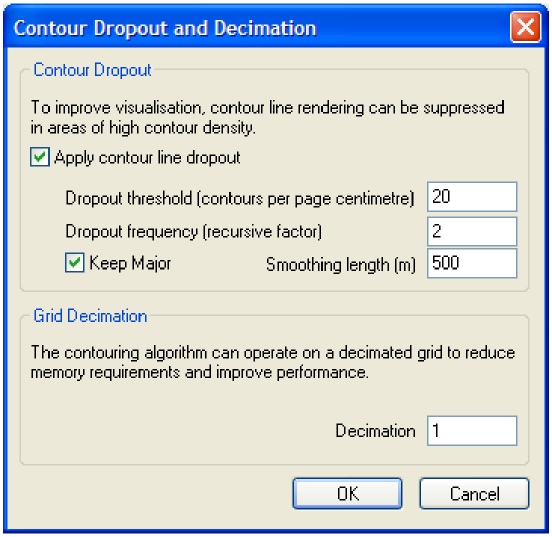
Appearance of labels of contours

The **Line styles** of major and minor contours can be defined and their smoothness determined by one of three types of contour **Line Smoothing** (None, Akima or Bezier).

The **Major** and/or **Minor Labels** drawn along contours can have a specific style and be either straight text or follow the curve of the contour (**Follow baseline** option). By default, contour labels are created relative to the gradient of the slope they represent, however, if you would rather they all be drawn such that the text is read from the base of a map, the **Keep text vertical** option can be used.

The density of contour labels within a map area and/or along contour lines is controlled by slide bars.

Contour Dropout – Options are accessed by clicking the button. When contour dropout is applied, contours are not drawn in areas of high data gradient to improve the look of the final contour view. The dialog used to control dropout is shown here.



Control dialog for contour dropout and grid decimation

Parameters that can be specified from this dialog include:

Dropout Threshold – Specifies the maximum number of contours that is rendered in a centimetre on the page. A setting of 20 is the default that allows each contour line half a millimetre of page space. Decrease the setting to dropout contours more frequently and increase it to dropout contours less frequently.

Dropout Frequency – Specifies how contours are dropped. The default is a setting of two (2). If Discover 3D decides dropout should occur it will dropout every second contour, then three in four, then seven in eight, 15 in 16 etc until sufficient dropout has been achieved. A setting of 10 would dropout every 9 in 10, then every 99 in 100 etc.

Grid Decimation can be used to enhance redraw and refresh performance on large grids. It operates by using only every 2nd, 3rd etc cell/row of an image when drawing. The **Decimation** entry is always a compromise between speed of redrawing and image honouring.

A **Colour** scale can be applied to a **Contour** surface by enabling the **Use Colour** option and selecting a colour scale from the pull-down list or via the **Advanced Colour Properties** dialog (shown left). A colour histogram graph is displayed and full control of the colour tables and colour transform is available. The colour controls are exactly the same as for a **Pseudocolour** image (see *Controlling the Colour Layer*).

Mask – Above the Preview window is the **Mask** check box. It is disabled by default. When enabled, a contour surface will erase the grid image area before it renders the contours. Consequently, when multiple contour surfaces are overlain, the contour lines from overlapping grids do not overdraw each other. It is purely a cosmetic feature. However, when enabled, any contour surface obliterates underlying surfaces of all types and so this option generally cannot be used in conjunction with colour grid images.

Exporting Contours

When contours are overlaid on a grid surface, gradients of the 3D display may distort the contour appearance since a contour vector line may be required to draw across grid cells many pixels wide but which represent a constant contour value.

A method of displaying ‘clean’ contours, or for use in other software, is to export the derived contours. The supported export formats include:

- Attributed .DXF (contour values are saved within the DXF file)
- ESRI .SHP files
- MapInfo Professional .TAB format
- MapInfo Professional .MIF format

To export the contours, use the **Export Contours** button and select the desired output format. If a MapInfo Interchange (.MIF) or TAB format is selected, a **Choose Projection** dialog appears after clicking the **Save** button. This allows the exported file to be assigned a projection for use within MapInfo Professional or other software.

13 Creating Gridded Surfaces

Discover 3D's extensive modelling capabilities include the ability to smoothly interpolate between 3D datasets to create gridded surfaces, using interpolation algorithms such as Minimum Curvature and Inverse Distance Weighting. This is a powerful way of constructing fault planes or water table boundaries from digitized drillhole intercepts.

This capability is an excellent alternative to wireframe modelling (see *Modelling Triangulated Surfaces and Solids*). Unlike wireframing, surface interpolation:

- produces a smoother and more realistic looking model between data points
- is not limited to the bounds/extents of the data points, so a continuous model can be created to cover the zone of interest

Discover 3D's **Surface Gridding** module under the **Grids** menu can support the following data sources:

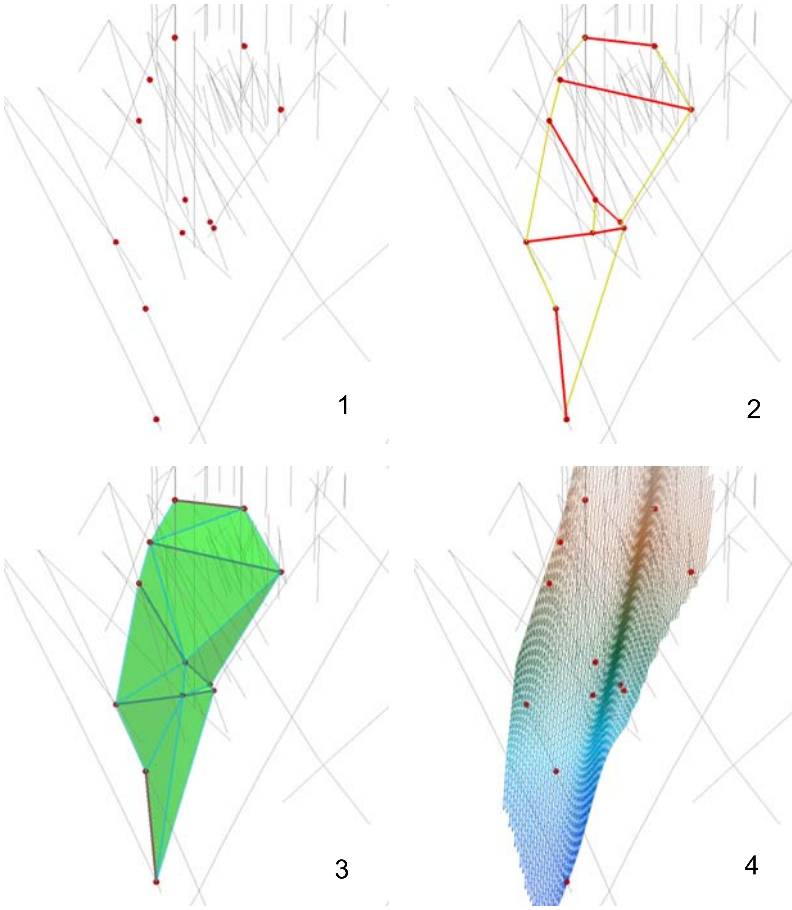
- Feature Points, Polylines, Polygons and other objects (e.g. snapped to drillholes in 3D).
- 3D Points (e.g. specific downhole intervals visualized with *View Intervals as 3D Points*).
- 3D Lines

The basic gridding work-flow is described in *Interpolating a Gridded Surface from Digitized Drillhole Intercepts*, and additional detail about the options avaialbale from the gridding tool can be found in *Using the Surface Gridding Tool*.

Examples of input datasets include:

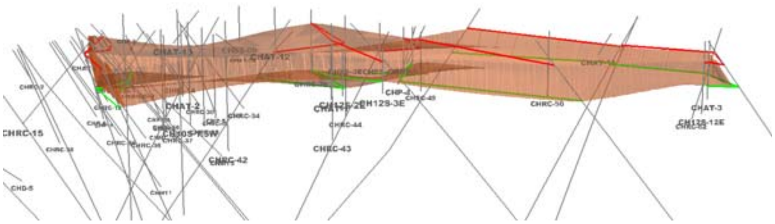
- Snapping feature points, polylines or polygons to drillholes in 3D to digitise:
    - particular fault or water table intercepts
    - the bottom of supergene mineralisation (aided by drillhole colour and thickness modulation)
    - a primary quartz vein/gold mobilisation system (interpreted between lithology colour modulation and thickness modulated assay results)

- Querying out the intervals at the top of a heavy mineral sands layer in a 2D drillhole project, and the viewing these as 3D Points (using the **View Intervals as 3D Points** menu in 2D).



A comparison of the modelling techniques: 1/. Initial digitised feature points snapped to drillhole fault intercepts in 3D 2/. Wireframing requires join lines and tielines to be created by the user to help define the structure 3/. Wireframing with the 3D Solid Generator creates a triangulated surface: the surface displays large triangular facets and is limited to the data bounds 4/. Using the Surface Gridding tool, a smooth grid is interpolated from the source feature point intercepts (there is no need for user-created tie-lines).

If there are multiple gridded surfaces generated for a body of interest (e.g. top and bottom surfaces), these can be converted to triangulated (TIN) surfaces and used to build a closed polyhedral volume.



A 3D volume constructed from 2 surface grids generated from drillhole intercepts

The Surface Gridding tool can create grids in the ER Mapper (.ERS), Geosoft (.GRD) and Surfer Binary (.GRD) uncompressed grid formats.

Interpolating a Gridded Surface from Digitized Drillhole Intercepts

Digitising intercept locations

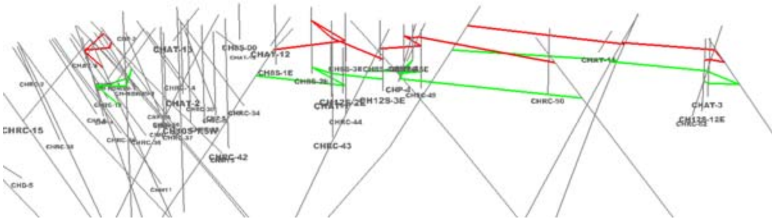
1. Load your 2D drillhole project into Discover3D using **Discover3D>View Drilllholes**.

2. Adjust the drillhole display properties to visualise the desired downhole data (e.g. colour and/or thickness modulation), and/or enable browsing for the drillholes in the datasheet window. Note that you may instead simply load a previously saved display setting (see *Saving and Applying Display Settings*).

3. In the workspace tree, make the feature database into which you will digitize **Editable** (i.e. either the cosmetic layer or a new feature database). Also make the drillhole branch **Selectable**.

4. Enable either the points or polylines tool in the feature toolbar. Also enable snapping by pressing the S key or the **Snapping** button.

5. Digitize the appropriate intercept locations. Using a 3DConnexion SpaceNavigator (see *Using the 3DConnexion SpaceNavigator™*) can help to precisely identify and snap to the intercept location in more complex 3D environments, and is essential when digitising polylines.



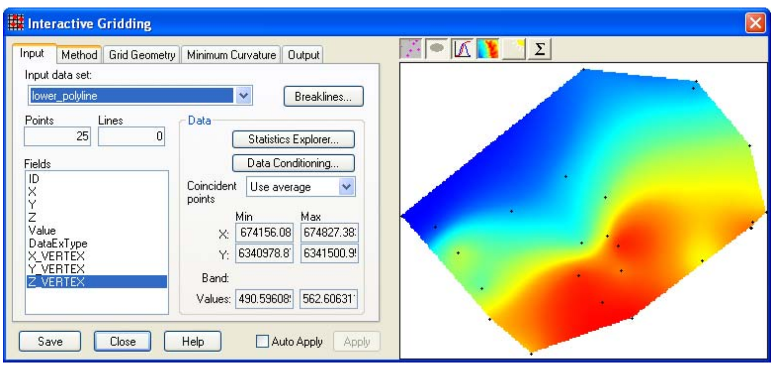
Digitizing the top and bottom of a heavy mineral layer in a mineral sands project as polylines. The red (top) and green (bottom) polylines have been digitized into two separate feature databases.

Interpolating a gridded surface

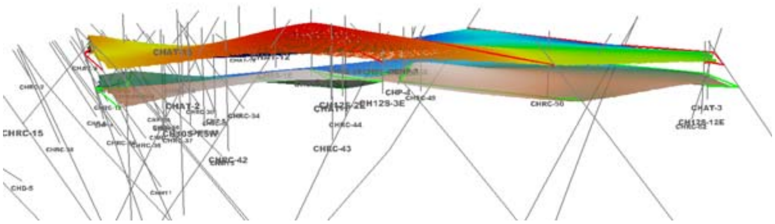
6. Select the **Grids>Surface Gridding** menu option.

7. In the *Gridding Tool Input Tab*, select the feature database containing the digitized intercepts under **Input data set**. Under the **Fields** area, select the Z_VERTEX option; as this will produce a grid using the elevation of the digitized features.

8. In the *Gridding Tool Method Tab*, select an appropriate method to use; Minimum Curvature is recommended as the initial method to experiment with, in order to quickly create a viable surface. Adjust the method's various parameters as required in the following tabs.



9. In the *Gridding Tool Output Tab*, specify a location and filename for the output grid using the **Browse** button at the bottom right of the dialog. If required, enable smoothing and clipping if required. Press **Save** to create the grid in the 3D window.

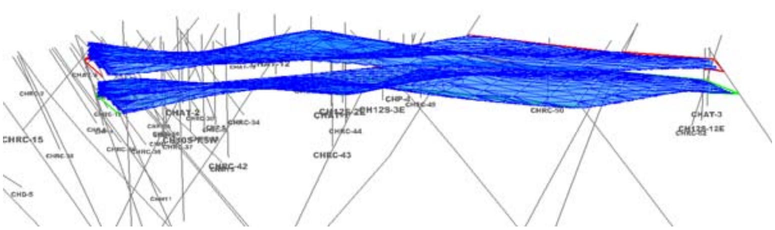


Two gridded surfaces created from the poylines in the previous example (representing the top and bottom of a heavy mineral layer). Minimum curvature interpolation was used.

Converting grids into triangulated surfaces, and building polyhedral solids

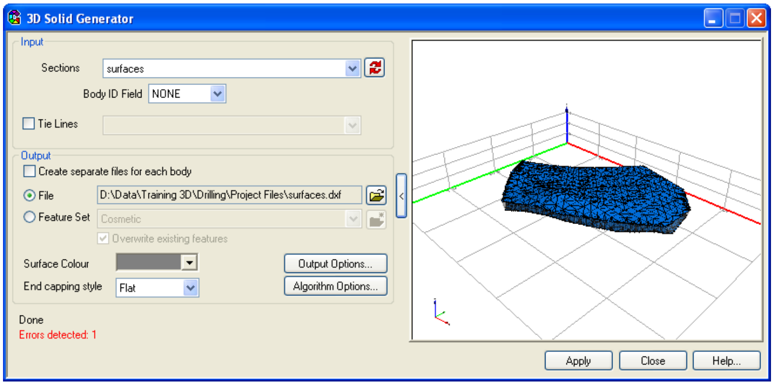
10. Converting a gridded surface into a triangulated feature surface means it can be processed with Discover 3D's *Advanced Editing Functions*. For example, to use a fault surafce to clip a mineralisation solid model.

    - Open the **Features>Import** menu option
    - In Step 1, set the type of file to import as **Grid files**.
    - In Step 2, browse for the gridded surface created previously. Also specifiy an output feature database for the new triangulated surface: it is recommended to create a new FDB rather than add the output to the cosmetic layer
    - In Step 3, leave the default option enabled (“Let the Import System...") and press **Finish** to create the TIN surface.

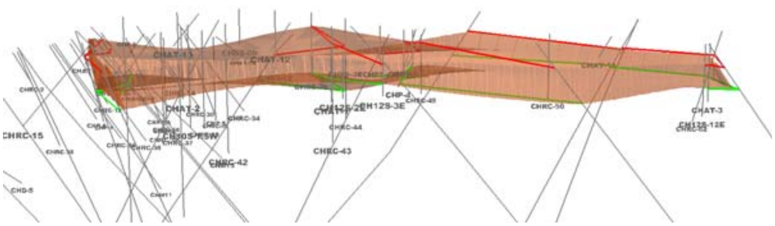


The two gridded surfaces created for the heavy mineral layer imported as two triangulated feature surfaces in the same new feature database

11. If multiple surfaces have been generated for an anomaly/rock unit/target, these can be converted into a closed TIN/polyhedral surface (i.e. a volume) by using the 3D Solid Generator (see *Modelling Triangulated Surfaces and Solids*).



Loading the two feature TIN surfaces into the Solid Generator (both are in the same feature database)

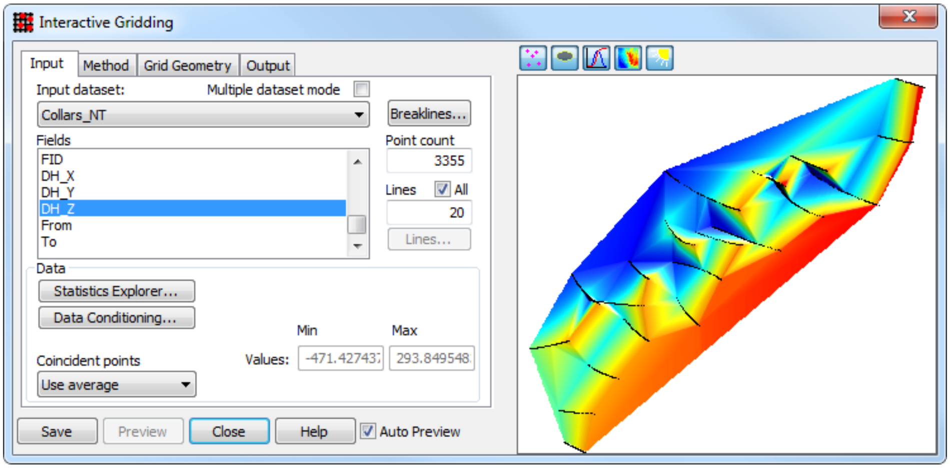


The resulting closed polyhedral surface (volume) with the original digitized polylines displayed for reference.

Using the Surface Gridding Tool

The Surface Gridding tool is accessed via the **Grids>Surface Gridding** menu option. The gridding tool is controlled via a series of tab pages along the top of the dialog. The number of visible tabs in the dialog varies depending on the gridding method selected, however four main tabs (Input, Method, Grid Geometry and Output) are always visible irrespective of the gridding method selected. Operation of each tab control is described in:

- *Gridding Tool Input Tab*
- *Gridding Tool Method Tab*
- *Gridding Tool Geometry Tab*
- *Gridding Tool Output Tab*



The Surface Gridding tool in Discover3D

On the bottom-left side of the dialog are Save, Cancel and Help buttons:

- The **Save** button creates an output surface as specified on the Output tab. The preview window does not have to complete drawing before saving the output grid file. If the initial appearance of the grid is satisfactory click the Save button and the grid is saved in the same folder as the input data points. The saved grid is then loaded automatically into Discover3D.

- The **Cancel** button dismisses the grid tool without creating an output surface

- The **Help** button displays the on-line help.

Two additional controls at the base of the dialog determine the operational mode of the Gridding. The two operational modes are described below:

- **Automatic mode** - This is specified with the **Auto Apply** checkbox enabled and is the default behaviour. As changes to any control parameter are made, the gridding process automatically re-computes and updates the preview display. The Gridding Tool can detect when a parameter change has been made and will apply this change when the cursor is moved to another tab page or field in the dialog.

- **On Demand mode** – Disabling the **Auto Apply** checkbox places the gridding tool in manual mode. No grid processing is commenced until the Apply button is selected or the Auto Apply mode is enabled. This mode is best used for situations where a number of parameters are to be modified prior to gridding, or the dataset is very large and the gridding operation may take some time. If the button is disabled then no changes are pending and no grid computation is performed.

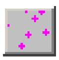
Note The Gridding tool will automatically switch to **On Demand** mode if a large dataset is loaded. In this mode, select a field to grid and press the **Apply** button to display the grid before proceeding beyond the **Input** tab page.

A vertical progress bar is displayed next to the preview window in both modes and will show the percentage progress of the gridding operation. The progress can also be monitored by observing the grid re-draw in the preview area. See *Gridding Tool Preview Display* for a description of the controls available.

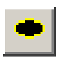
Gridding Tool Preview Display

The appearance of the preview display can be controlled by four toggle buttons and a pull-down list located above the preview window.

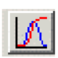
These control buttons are:



View input points in the preview window. Click button to remove input points from view.



View search ellipse in the preview window. Only available if Inverse Distance Gridding method is selected.

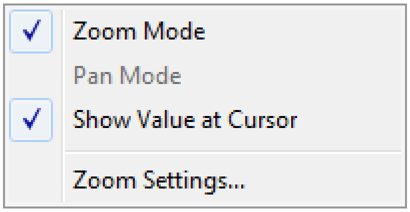


Apply a Histogram Equalisation stretch to the image in the preview window. This button can be used to distribute colours more evenly across the image and is particularly useful for data with poor dynamic range.

Display images in the preview window as either coloured or monochrome (e.g. greyscale from black to white). This button toggles the image between monochrome and colour.

Apply a sun illumination to the image in the preview window. This can be used to enhance detail within a gridded image. The sun angle is fixed from the north-east direction.

A pop-up menu can also be accessed by right-clicking with the mouse in the preview window:



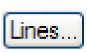
Preview window right-click mouse options

The **Zoom** and **Pan** controls enable the grid to be examined prior to saving. To zoom in (x4 magnification) on an area position the cursor over the area of interest and click the left mouse button. Click the left mouse button again to zoom out. The zoom factor can be changed by selecting the **Settings** menu option.

Pan a zoomed image by selecting **Pan Mode** and holding down the left mouse button. The cursor will change to a hand when in Pan Mode. When the button is released the image redisplays at the panned location. Other options include **Show Value at Cursor** when the cursor is placed over a grid cell and **View Input Points** to display the original data point locations. If the interpolation method chosen is Inverse Distance Weighting the **View Search Ellipse** option is enabled to view the search radius used to calculate the grid cell values.

Gridding Tool Input Tab

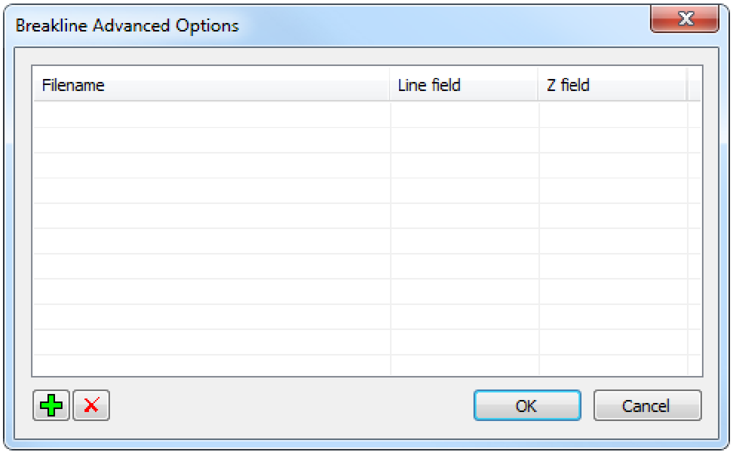
Data fields from the Input dataset are listed in the **Fields (bands) to grid** list. To change the column to be gridded select the column name so it is highlighted. Multiple fields (or bands) may be gridded at the same time. The data range for the selected column(s) is automatically displayed in the text boxes to the right and if the **Auto Apply** option is enabled the grid is regenerated for the new column. If the **Auto Apply** option is turned off then click the **Apply** button to display the grid in the preview window. To view multiple gridded fields select the field from the pull-down list displayed above the preview window.



A sub-selection of certain Lines or Drillholes from the dataset can be selected by clicking the **Lines** button.

Breaklines

Breakline data can be read from a MapInfo TAB file. Break lines are defined as multi-segment lines in which the slope is monotonically increasing or decreasing along each segment. Examples of breaklines include stream or river traces, topographic ridge lines or roadways. By incorporating break lines into the gridding process the output grid can be forced to conform to certain slope requirements in critical areas.



The Breakline options dialog

The **Breakline Advanced Options** dialog enables you to browse and select an input TAB file. To identify the breaklines it is necessary to select a **Line field** from the TAB file. All vertices from all features with the same LINE identifier will be considered to be a single break line. This means that a collection of points can be grouped into polylines but it also means that polylines will need to have a unique LINE identifier if they are to be considered unique. A **Z field** must also be specified. It is assumed that the Z data will be compatible with the data field selected to grid. In order to define breaklines with a variable Z attribute for each node (or vertex) in MapInfo the data must be represented as a collection of points. These points must be assigned a LINE identifier to distinguish them as being part of a particular break line. By defining the break lines in MapInfo as a collection of points identified using a LINE identifier it is possible to represent complex 3D geometries such as streams, slope ridgelines or cliffs.

Coincident points

The **Coincident points** pull-down list controls the handling of multiple data points at the same location. Coincident points may be averaged, the first or last point value that does not contain a null value used, the minimum or maximum point value taken or all points kept.

Statistics Explorer

The Statistics Explorer is a utility for examining and understanding the spatial and statistical aspects of a data set. It has been designed in a way that allows for intelligent interrogation of a data set to help identify spatial and frequency relationships which may be important when choosing and configuring an appropriate gridding method. The operation and functionality of the Statistics Explorer is described in *Statistics Explorer Tool*.

Data Conditioning

The Data Conditioning options enable data to be modified prior to gridding by applying one or more of the following processes:

- **Clip Input Data** - Specify maximum and minimum X and Y extents to conduct a data clip. The source dataset is clipped to the X and Y extents by checking the Apply data clip option. These values can be reset to the initial dataset extents by pressing the Reset extents to input button. All source data points outside the defined region are ignored and do not contribute to the gridding.

- **Define NULL Values -** Null values are used to flag specific values in a dataset that are not to be included in gridding process. These may include values indicating Sample Not Received (SNR) or Below Detection limit (BDL), etc. These samples may be attributed with a negative numerical value such as -9999 or -0.5 (detection limit). Failure to remove such artefacts can result in meaningless output grids. The Null values are set for one or all bands in data to be gridded in the F**ield Data Conditioning** tool (see *Field Data Conditioning Tool*)..

- **Cap Input Data Bands** - Data capping options can be set to prevent outlier values in the dataset from being included in the gridding process. For example a maximum cap value can be set for gold assays which occur in mineralisation systems prone to nugget effects. Capping data to remove very high or very low values is used to remove samples which may unduly influence the cell values in the output grid. Setting a Cap minimum or maximum value will cap source data outside the set limit to the limiting value. For example, if the Cap maximum value is set to 500, a gold assay with a value of 725 ppm will be handled during gridding as having a value of 500 ppm value.

- **Specifying background values -** It is also possible to check the Convert NULL to background box and enter a user specified value in order to constrain the gridding. For example, if gridding drillhole geochemical assays, much of the hole may not have been sampled and in these areas the assay result may be assumed to be equal to the background value. This helps prevent anomalies ‘ballooning’ into areas with no source data coverage.

The options presented in the **Grid Conditioning** dialog are global settings. If multiple data fields have been selected in the *Gridding Tool Input Tab* (i.e. in order to create a multi-banded grid), these settings will affect all fields equally.

Field Data Conditioning

Further data conditioning options for each individual field selected fro gridding are available under the **Field Data Conditioning** button. See *Field Data Conditioning Tool* for more information.

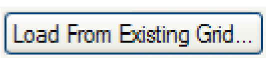
Gridding Tool Method Tab

Eight gridding methods are available in Discover3D. The methods can be selected via the **Method** tab on the gridding tool dialog. The gridding method chosen will determine the number of additional dialog tabs and control properties that are displayed along the top of the gridding tool.

Located on the **Method** tab is an option to save the grid parameters or to apply pre-defined parameters.

The **Load Settings** button will present the user with a list of user-defined grid parameters.

The **Save Settings** button is used to save settings from the existing Create Grid session, such as gridding method and search parameters (it does not save the grid cell size). These settings can therefore be applied to different bands or different datasets.



The **Load From Existing Grid** option is designed to load gridding parameters from an existing grid file. By default a Discover-created grid will have an associated XML configuration file located in the same folder as the parent grid; to load these settings navigate to the associated XML file. Note that this option should only be used on the same dataset, as it will load all settings for the grid such as grid extents, cell size and band used.

The available methods are described below:

- *Minimum Curvature*
- *Kriging*
- *Inverse Distance Weighting*
- *Spatial Neighbour*
- *Triangulation*
- *Density Grid*

- *Distance Grid*
- *Bi-Directional Spline*

Minimum Curvature

The Minimum Curvature gridding method is widely used in many branches of science and research. This method creates an interpolated surface similar to a thin, linearly elastic plate passing through each of the data values defined in the input dataset. An important criterion in creating a surface is that it has a minimum amount of bending forced upon it to conform to the data points. The degree of bending is constrained by a ‘tension’ parameter and this can be specified both within the data area and along the edges. Minimum curvature gridding generates the smoothest surface possible while attempting to honour the data as closely as possible. Like all gridding methods, minimum curvature gridding is not an exact interpolation technique and therefore some error may occur between the input data point values and the interpolated surface values.

The Minimum curvature algorithm attempts to fit a surface through all of the data points without putting any abrupt kinks in the surface. Between the fixed observation points, the surface bends according to the Interior tension. This parameter is used to control the amount of distortion on the interior with the higher the tension, the less the distortion. The Boundary tension controls the amount of distortion at the edges. By default, the boundary tension is set to 0.

The minimum curvature method produces a grid by repeatedly applying an equation over the data in an attempt to produce a smoothly varying grid. Iteration is used to describe the number of times the equation is to be applied to the grid. The Maximum iterations number can be specified in the Minimum Curvature tab. The grid node values are recalculated until successive changes in the error between successive iterations are minimised, or the maximum number of iterations is reached.

Kriging

Kriging is a geostatistical gridding method which has proven popular across a variety of industries due to its flexibility and data driven approach to surface interpolation. Kriging is an advanced technique which is based on the assumption that the spatial variability in the measured property of a data set is neither due to totally random nor deterministic constraints. The main advantage of Kriging over simpler interpolation techniques such as IDW (Inverse Distance Weighting), is that it uses a weighting model which is adaptive to the inherent trends in a data set rather than imposing a set of fixed conditions upon them. The process of using Kriging in interpolation can be complex and requires an intimate knowledge of the structure and variability in the data set so that an appropriate sample model and set of gridding properties can be chosen.

Over the past several decades kriging has become a fundamental tool in the field of geostatistics. The method of interpolating a surface using kriging is generally performed as a two stage process:

1. Analyse the input data to establish the spatial predictability of the measured values in the study area. This analysis generally focuses on the spatially correlated component of the data by means of determining the degree of spatial dependence among the sample points. The average degree of spatial dependence among variables is summarised in a plot known as the semi-variogram. The semi-variogram is a concise means of representing the average intersample variation according to sample separation distance and direction. In order to use the sample variance as part of the interpolator in the Kriging process it is necessary to model the semi-variogram in order to define a mathematical function which optimally describes the underlying structure in the data. This process is known as variogram modelling which in itself and can be a very involved and complex task. Once an appropriate model has been chosen it can then be used to estimate the semivariance or weighting at any given sample distance.

2. Interpolation or estimation of values at locations which have not been adequately sampled. This process is known as interpolation 'kriging'. The simplest technique known as "ordinary kriging" uses a weighted average of the neighbouring samples to estimate the unknown value at a given grid node. The weights are optimized for each node using the variogram model, the distance to the surrounding samples and the inter-sample variance.

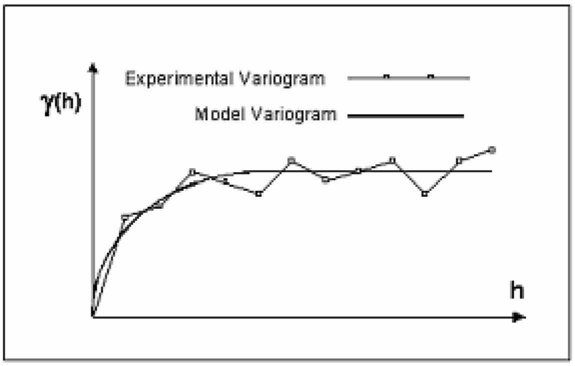
The first step in kriging is to construct a variogram (or semivariogram) from the input data which describes the spatial correlation between the sample points. A variogram generally consists of two parts:

- an experimental (or sample) variogram and
- a model variogram (a descriptive function which mathematically models the experimental variogram).

The degree of spatial dependence among sample points is measured by the average semi-variance:

$$\gamma(h) = \frac{1}{2n}\sum_{i=1}^{n}[z(x_i) - z(x_i + h)]^2$$

Where h is the distance or lag between sample points, n is the number of samples separated by h and z is the attribute value of interest. The computation of $\gamma(h)$ is performed in two steps. First pairs of sample points are grouped together by distance. For example, if the distance interval (or lag) is 1000m then pairs of points separated by less than 1000m are grouped together into a 0-1000m lag, samples separated by a distance of between 1000 and 2000m would be grouped into a lag of 1000-200m and so on. Next the average distance h and the average semivariance $\gamma(h)$ is calculated for each group. If spatial dependence exists among the samples, then pairs of points closer together will have more similar values than pairs that are further apart. The semivariogram is a plot which has the average semivariance $\gamma(h)$ along the y-axis and the separation distance h along the x-axis.



Experimental and Model Variogram used in kriging

The semi-variogram can be broken down into thee main components; the Nugget, Sill and Range.

- Nugget - is the semi-variance at a distance of zero and represents the degree of sample repeatability or spatially uncorrelated noise.
- Range – is the spatially correlated portion of the semi-variogram that exhibits an increase in the semi-variance with distance. Towards the limit of the range the semi-variance levels off such that with additional increases in distance it is indistinguishable from one point to the next. This point of flattening is called the sill.
- Sill – is the point at which the semivariance (range) levels off to a relatively constant value.

Once an experimental variogram has been computed, the next step is to define a model variogram. A model variogram is a mathematical function that models the trend in the experimental variogram. Once the model variogram is constructed, it is used to compute the weights which are used in the kriging interpolator. The basic equation used in ordinary kriging can be described as follows:

$$F(x, y) = \sum_{i=1}^{n} w_i f_i$$

Where n is the number of points in the data set, f_i are the attribute values of these points, and w_i are weights assigned to each point. This equation is essentially the same as the equation used for inverse distance weighted interpolation except that rather than using weights based on an arbitrary function of distance, the weights used in kriging are based on the model variogram.

The creation and analysis of the sample and model variograms is done using the *Statistics Explorer Tool*.

Selection of Kriging on the Method tab opens both the Search and Kriging tabs on the gridding tool interface. Kriging uses the same searching mechanism as the Inverse Distance Weighting method.

The gridding tool supports two types of kriging estimation; Point and Block. Point kriging estimates the values of the points at each of the grid nodes. Block kriging on the other hand estimates the average value of points which fall within a rectangular block centred on each of the grid nodes. Because Block kriging estimates the average point value for each block it tends to generate smoother grids. In addition because block kriging uses an average of the input sample points it is considered an imperfect interpolator even when an input sample falls exactly on a grid node. The Block Kriging method provides controls to decimate (or break up) the blocks into a smaller mesh of sub cells, defined by the X and Y increment values. When a decimation factor is applied in Block Kriging the value assigned to each block is determined as the average of the sub cells inside the block rather than the average of the entire block.

In addition to the point and block estimation types the Interactive gridding tool supports two methods of Kriging; Ordinary and Simple. Ordinary Kriging focuses on the spatial correlation component between the measured values but ignores drift (drift is a regional trend in the data which exists is addition to spatial correlation between samples). Simple Kriging is similar to Ordinary Kriging with the exception that the weights used do not sum to unity and the average of the entire data set is used in the interpolation of each grid node rather than the local average of points that fall within the Search Distance of the node. Consequently Simple Kriging can be less precise than Ordinary Kriging but generally produces smoother grids.

Inverse Distance Weighting

Inverse Distance Weighting (IDW) is a universal technique that can be applied to a wide range of spatial data. IDW uses weighted average interpolation to estimate grid cell values and can be used as either an exact or a smoothing interpolator. Each grid cell value in an output surface is calculated using a weighted average of all data point values surrounding the grid cell that lie within a specified search radius.

The IDW method is optimal when the data has a fairly uniform distribution of input points across the area to be gridded, and some degree of smoothing is beneficial; for example soil samples taken over a regular grid. With this type of data, a repeat measurement at a point does not necessarily give the same results as the first measurement. If the input data points are not evenly distributed then using an oriented search ellipse may produce a more representative grid.

Selection of the Inverse Distance Weighting method opens both the Search and Inverse Distance tabs.

The weighting value assigned to each point within the search ellipse is determined by the distance from the data point to the grid node being interpolated. The further away a data point lies from the grid node, the less the point value will contribute to the final value assigned to that node. The distance weighting parameters can be adjusted under the Weighting Model controls in the Inverse Distance tab.

A search ellipse of fixed size and orientation can be defined in a similar manner to the Kriging method using the Search tab (below); a grid cell value is then calculated from the weighted average of all data points that lie within the ellipse centred on that grid cell.

Weighting Model

The weighting of an input data point is (by default) inversely proportional to its distance from the grid node (a Power weight model). This can be varied by choosing a different Weight Model on the Inverse Distance tab, and altering the model's parameters where applicable. The following models are available for selection:

- Linear

Each input point's weight is proportional to its Euclidian distance from the grid node being interpolated. The linear weight model enables the Nugget and Range parameters to be adjusted in order to vary the weight assignments. At distances less than the Nugget distance the weight model will be 1 – i.e. all data will contribute equally. The Range parameter is used to set the outer distance threshold for which the weight model is applied. Any samples which exceed the Range and are less than the Search Distance (Search tab) will be assigned an equal weight.

- Exponential

  Each input point's weight is proportional to its distance from the grid node being interpolated raised to the specified power. Increasing the power value will cause smaller weights to be assigned to closer points and more distant points to be assigned equal but large weights. Increasing power values will therefore cause each interpolated grid node to more closely approximate the sample values closest to it. As with the Linear model the Nugget and Range properties can be modified to constrain that distance over which the exponential weight model is most effective.

- Power

  The default option, each input point's weight is proportional to the inverse of its distance to the specified Power from the grid node. Increasing the weighting power reduces the influence distant points have on the calculated value of each grid node. Large power values cause grid cell values to approximate the value of the nearest data point, while smaller power values will result in data values being more evenly distributed among neighbouring grid nodes. The weighting value defaults to 2 (i.e. the weight of any data point is inversely proportional to the square of its distance from the grid cell) which is appropriate for most situations. If required, the weighting value can be altered to any positive value.

- Gaussian

  The weight assigned to each input value is determined according to a 2D Gaussian function centred on the grid node. The shape and standard deviation of the Gaussian function is proportional to the Range with larger values producing a flatter function and a smoother grid.

Weighting Options

The **Elliptical weighting** option is only available when the **Elliptical Search** option is enabled (in the **Search** tab). It adjusts the distance weighting function for data points within the search ellipse depending on their relative position with respect to the elliptical shell. Points located on the same elliptical shell will be assigned equivalent weighting even though their distance from the ellipse centroid may be different.

The **Density corrections** control dynamically adjusts the search algorithm to optimise grid cell interpolation in areas of data clustering. Activating density corrections can help to enhance detail in datasets where sample points are unevenly spaced (e.g. regional geochemistry sampling) and may in some cases produce a smoother or more representative grid. The density correction modifies the weights for each contributing point based on the sample density at that point.

Note If you have enabled the Use nearest neighbours option (in the Search tab) in conjunction with four search sectors then you will have effectively removed clustering from the input data point distribution. The Density corrections option is not available when the Use nearest neighbours option is enabled.

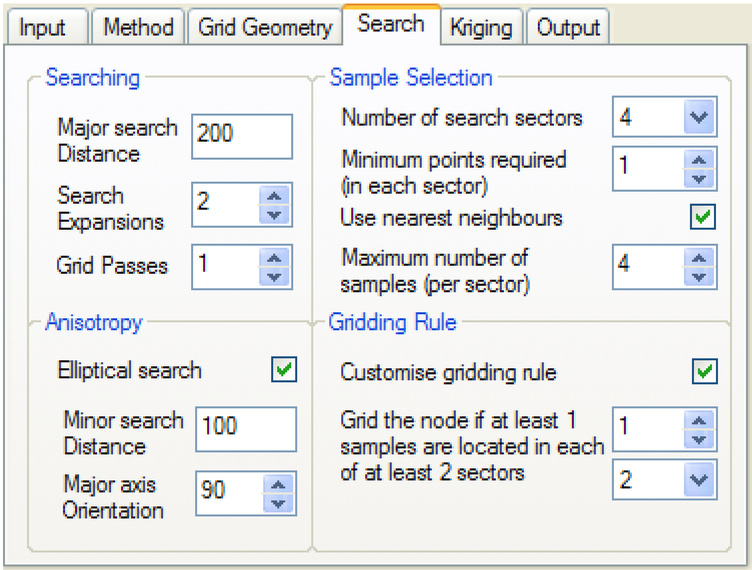
The **Exact hit distance** is a tolerance distance for assigning actual input data values to coincident grid nodes. As the inverse distance gridding technique is attempting to interpolate a continuous surface through the data, a certain number of grid nodes coincide with the input data points. Where grid nodes and data points coincide, the distance between them is zero, so by default the data value is assigned a weighting of 1.0 and all other data points in the search radius are given a weight value of zero. This means that grid nodes that are coincident with input data points are assigned the value of the coincident data point rather than an interpolated (averaged) value derived from the data points surrounding it.

This effect can produce significant ‘spotting‘ in the output grid, particularly if the data value of the coincident point/grid cell deviates significantly from the points surrounding it. By adjusting the exact hit distance it is possible to increase the tolerance distance in which input data values are assigned to grid nodes. Assigning this value to a high number can produce unacceptable spotting or concentric banding in the output grid, while reducing the value below 1 has little or no effect.

The Taper controls allow you to apply a taper function to the interpolated value of each grid node based on its distance to the nearest valid sample point. The taper function is applied using a linear weighting model thereby adjusting the expected grid node values towards the background value. Between a distance of zero and the FROM distance the taper function is assigned a constant value of 1 (i.e. no modification is made to the grid node). Between the FROM and TO distance the taper function is applied as a linear weighting between the grid node value and the background value. Beyond the TO distance grid nodes are assigned the background value.

Search Tab

The Kriging and IDW methods, when selected, both display the **Search** tab, along with their individual control tabs. These methods, if not optimised, can quickly becomes unworkable as the number of input data points increases beyond a few thousand. To improve the performance of these algorithms and to ensure these methods are suitable for large datasets, a search radius can be used to restrict the number of input points that contribute to each interpolation. This introduces a number of problems. For example the algorithm may not find a sufficient number of points within the search radius to make a reasonable estimation or, the spatial distribution of the points within the search radius may not be uniform so that the estimation becomes directionally biased.



The Search dialog

The Search tab provides controls to resolve these issues by determining the shape, size and orientation of the search ellipse used to locate data points during interpolation. Specifying an appropriate size and orientation for the search ellipse is important. Setting it smaller than the average data spacing may result in a large number of the interpolated grid cells being assigned a null value and therefore displayed as white in the output grid. Conversely, if the search ellipse is set to be too large then significant edge effects or grid artefacts may result around the edge of the grid. The **Search** tab is subdivided into a number of sections:

- *Searching*
- *Anisotropy*
- *Sample Selection*
- *Gridding Rule*

Searching

By default Discover3D uses a circular search with a radius specified via the Search Distance option. If the node cannot be estimated from the points located within the search radius then the search radius can be incrementally increased and the searching repeated using Search Expansions. The increased radius is likely to encompass more input points and consequently the node may be able to be interpolated. At each stage the actual search radius used will be equal to the stage number multiplied by the initial search radius. The number of allowable increments is limited because after a while this process becomes self defeating and it is wiser to specify a larger initial search radius.

To optimise performance, choose an initial search radius that is likely to encompass the minimum number of required input points most of the time. It can sometimes be very difficult to make this decision but the tool will always make a suggestion to get started with.

If the spatial distribution of the data points is not uniform (or not uniformly random) then the use of search expansions may not be enough to populate the grid successfully. For example, a dataset may have regional data located on two kilometre centres and local data in parts of the study area on 100 metre centres. To produce a suitable grid of the whole region that characterises the detail in the high resolution areas would require small search radius in these areas and a large search radius elsewhere.

The solution is to use additional refinement Grid Passes which grid the data multiple times - once for each pass - at increasingly higher resolution. The gridded results from each pass are then used as additional input data for the next pass. The grid cell size and search parameters are scaled up by a factor of two for each additional grid pass – for example if you use three additional passes then the first pass scales up these parameters by a factor of eight, the second by a factor of 4, the third by a factor of two. The final pass grids the data at the requested resolution with the specified searching parameters.

Anisotropy

By default the search radius is isotropic creating a circular search area. However directional bias can be applied by enabling an Elliptical Search.

The dimensions of the search ellipse can be controlled by specifying the length of the major and minor axes. The major axis is defined by the Search Distance value in the Searching section (above), whilst the minor axis is governed by the Minor search distance option. The Major axis Orientation control determines the rotation angle of the major axis.

If elliptical searching is used with the Inverse Distance Weighting method, it also allows the use of the Elliptical weighting option under the **Inverse Distance** tab. This option modifies the data point weighting so that they are isotropic with respect to angle within the search ellipse – in other words it removes the directional bias from the weighting

Sample Selection

Input points ‘close’ to the grid node may not be uniformly distributed, e.g. they may all be on one side of it. This will introduce a directional bias into the estimation. This can be resolved by using search sectors.

Discover3D provides options for specifying 1, 2 or 4 search sectors. By adjusting the Number of search sectors and Minimum points required (in each sector) the appearance and smoothness of the output grid can be varied. If any of the sectors contain fewer than the minimum number of specified points, the interpolated grid cell value for that node is assigned a null value.

If four sectors are used then each covers 90 degrees of arc (centred about NE, NW, SE, SW). If two sectors are used then each covers 180 degrees of arc (centred about North, South). Using only one search sector effectively turns the option off.

Using 2 or 4 search sectors can significantly improve the appearance of a grid if the input data has been collected on widely spaced lines. Using a one sector search ellipse may result in grid node values being estimated from data points from a single direction. This might generate unrealistic or sharp slopes between the lines producing a rough or stepped grid. Using a two or four sector search with an appropriate search distance should generally eliminate or reduce this effect. Experiment with the use of search sectors and examine the difference these can have on the output grid.

Specifying the **Use nearest neighbours** option enables you to use only the closest Maximum number of samples found within each search sector in each estimation. Contributions from other points within the search radius are ignored.

The **Use nearest neighbours** option controls the Maximum number of samples that are used in each sector when interpolating each grid node value. When this option is selected Discover3D uses the closest points (up to the maximum specified in each sector) to interpolate each grid cell. Any excess data points within the search ellipse are ignored in the calculation.

Gridding Rule

Enabling the **Customise gridding rule** option allows a node to be gridded only if a specified minimum number of sample points are located in at least a specified minimum number of sectors. If a node fails to meet this rule, it will be assigned a null value.

Spatial Neighbour

This method is similar to the Inverse Distance Weighting method described above but does not use the search radius to interpolate data values from surrounding cells. For each grid cell, neighbouring input points are located based on a ‘spatial neighbours’ selection criterion. The value of the computed grid cell is the average of the neighbours, weighted such that the closer the neighbouring point, the greater the influence than points further away.

Search criteria are only radial in this method but the distance and applied weighting can be specified in the **Spatial Neighbour** tab displayed when this gridding method is selected.

Triangulation

The Triangulation method produces a regular gridded surface through a set of data points by using an optimised Delaunay triangulation algorithm. The triangular mesh is created by drawing lines between adjacent input data points and forming an irregular network such that no triangle edges are intersected by other triangles. A regular grid is then computed from the triangular irregular network and grid cell values computed using a natural neighbour interpolation process. As the original data are used to define the triangles, this method is very useful for situations where the data must be honoured very closely (e.g. elevation data in a digital terrain model).

The triangulation method is best applied to data that is evenly distributed over the gridded area. If there are large areas of sparse or missing data distinct triangular facets may appear in the output grid. As triangulation uses all the input data to construct the triangular mesh, the only parameter that needs to be adjusted is the grid cell size. Grid cell size can be adjusted to an appropriate value for the dataset.

Density Grid

The Density gridding method produces a grid which records a measure of the point density at each grid node. The density at each grid node is determined independently using an estimator function. Two estimators are available:

- Radial Density Estimator
- Kernel Density Estimator (KDE)

The **Radial Density Estimator** method returns a true measure of the point density at each grid node (measured as the number of samples per square area unit – usually metres). It is a simple method that counts the number of input samples within a specified radius of the grid node position and then normalizes that count by the area of the search.

The **Kernel Density Estimator** method is a non-parametric density estimator. It uses a similar approach but it weights the input samples by a kernel function that is normally a function of the normalized distance of the sample to the grid node. To achieve a good result with the KDE function it is more important to choose an appropriate search radius – sometimes referred to as the bandwidth – than to choose an appropriate kernel function. If the bandwidth is too small the density will be under-smoothed whereas if the bandwidth is too large the density will be over-smoothed and lacking in resolution.

Given a kernel function K and a search radius (or bandwidth) h, the estimated density at any point x is given by –

$$\hat{f}(x) = \frac{1}{n}\sum_{i=1}^{n} K\left(\frac{x - x(i)}{h}\right)$$

where n is the number of samples. The following kernel functions are supported:

Kernel	K(*u*)
Uniform	$\frac{1}{2} I(\lvert u \rvert \le 1)$
Triangle	$(1 - \lvert u \rvert) I(\lvert u \rvert \le 1)$
Epanechnikov	$\frac{3}{4}(1 - u^2) I(\lvert u \rvert \le 1)$
Quartic	$\frac{15}{16}(1 - u^2)^2 I(\lvert u \rvert \le 1)$
Triweight	$\frac{35}{32}(1 - u^2)^3 I(\lvert u \rvert \le 1)$
Gaussian	$\frac{1}{\sqrt{2\pi}} \exp(-\frac{1}{2} u^2)$
Cosinus	$\frac{\pi}{4} \cos(\frac{\pi}{2}) I(\lvert u \rvert \le 1)$

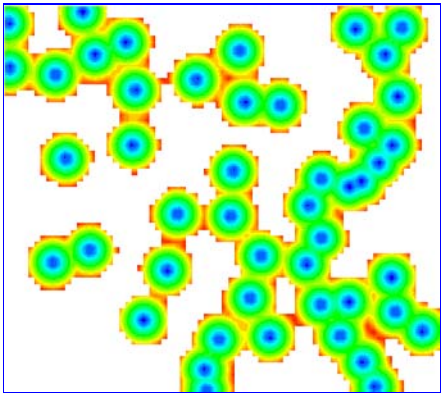
If you have taken multiple samples at each input data location and this information is recorded in the input data then you can use this information to bias the density estimation. To enable this option, check the Interpret selected data channel as a count frequency box and on the *Gridding Tool Input Tab* ensure the frequency or count field is selected as the input data field.

Distance Grid

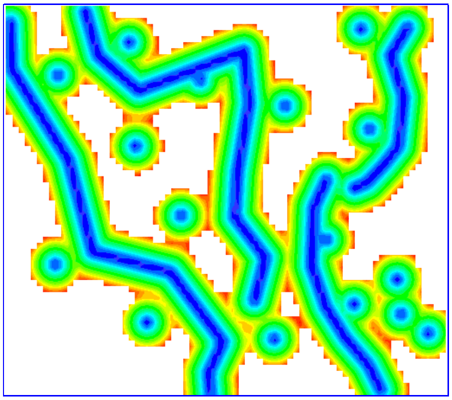
The Distance gridding method produces a grid which records the minimum distance to the input data features at every grid node. A distance envelope can be specified to clip the grid at a maximum distance from the nearest feature. Any grid node that is outside this envelope will be assigned a null value.

When the input data is loaded it is classified as either point data or polyline data depending on the source of the data and the type of object that was loaded. By default the method will consider all input data as point locations. Optionally, you can add to this the input polyline data. In this case the method will also check the perpendicular distance to the nearest polyline in addition to looking at all point data.

In the examples below the first grid has been computed considering all the input data as point locations. The second grid has added to this the polyline information. In both cases a distance envelope has been applied.



Distance Grid points only



Distance Grid points and polylines

Bi-Directional Spline

This method of gridding is fast and uses an accurate approach with few side effects. It has a small memory foot print and is suitable for gridding very large datasets collected on lines with a common azimuth.

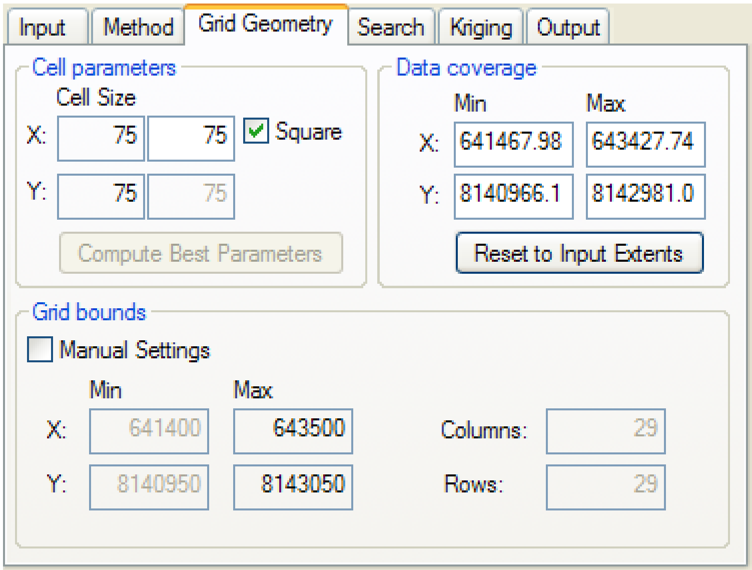
The bi-directional spline employs a form of damping called the Akima spline. The Akima spline interpolates a smoothed curve through a series of given points. The interpolation approximates a manually drawn curve better than many other ordinary splines. In the Gridding Tool, an Akima spline is applied first along input data lines and then across them in alignment with the grid cells in either the North, North East or East directions.

For best results, choose multiple lines of data that have similar direction azimuths. Best results will be obtained for lines that have azimuths close to North, North East or East.

No additional tabs are displayed for this option because it does not require any parameters.

Gridding Tool Geometry Tab

This tab controls the main parameters for determining the geometry of the output grid. The Cell Parameters control determines the size of each grid cell in the output image and is measured in the same data units as the input dataset. For most geographic data these units are in metres; however Discover3D supports all of the MapInfo units of measurement. During the loading process, Discover3D automatically computes an optimised grid cell size based on the distribution and density of the input data. This is displayed in the first pair of boxes. The cell size can be adjusted in the second pair of boxes.



The Grid Geometry dialog

Note It is recommend to define a square cell size, as Discover and MapInfo Professional only supports square grid cells.

When a large dataset is loaded the **Auto Apply** box is disabled and the **Compute Best Parameters** button is active. Use this button to calculate an optimal grid cell size.

The extents of the **Data coverage** to be gridded can be modified by entering new co-ordinates into the Min and Max X and Y columns. The full extents of the original data coverage are displayed by default. If the coordinates have been modified select the **Reset to Input Extents** button to return to the original data coverage.

The **Grid bounds** parameters control the boundary extents of the output grid and can be used to reduce the size of the output grid if required. The number of **Rows** and **Columns** in the output grid is also displayed. These values are calculated from the grid extents and cell size and cannot be edited.

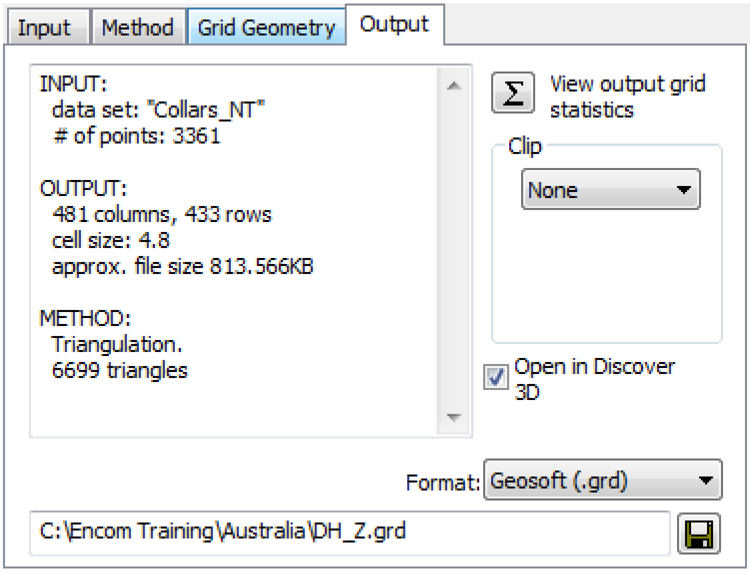
Note When creating a surface, the grid cell size selected is important. As a general rule of thumb the grid cell size should not be made smaller than approximately one fifth of the average data spacing. Reducing the grid cell size beyond this limit may cause the grids appearance to become smoother but can also introduce unwanted irregularities in the output image. Assigning an excessively small grid cell size will also increase computation time and file size. Discover3D suggests an optimal grid cell size for the data but experiment with alternative values.

Gridding Tool Output Tab

The **Output** tab provides a number of controls for saving the gridded image. The left side of the dialog provides a summary report for the grid process and lists the primary gridding parameters chosen. This information may be copied to the windows clipboard by selecting the information with the mouse and using the Windows Copy and Paste commands. This may be useful for reporting or archive purposes.

Below the summary box is a pull-down list containing the Discover3D supported grid formats. Discover3D currently creates grids in ER Mapper (.ERS), Geosoft (.GRD) and Surfer Binary (.GRD) uncompressed grid formats. To save a grid, select the appropriate format from the list. Click the **Save** button. The output grid file is assigned a default name. The file name and directory path by clicking on the small button at the right end of the path name box. The Output tab is accessible at any stage during the gridding process so it is not necessary to wait for the preview window to complete drawing before saving the final grid.

The **Smooth grid** control allows the appearance of the output image to be smoothed by applying a Gaussian smoothing filter to the grid. In most situations, enabling the smooth grid function removes high frequency noise in the grid and enhances the appearance of your image.



The grid output dialog, showing the clip options availble

The **Clip** control provides a number of options for clipping the extents of the interpolated grid, so that it more closely approximates the distribution of the input data. Enabling this option can improve the appearance of the output grid for irregular input data where the gridding method (usually Triangulation or Minimum Curvature) has interpolated the grid over large gaps within the data. Options available include:

- Creating a **Buffer** around the input data at a user-specified distance. The gridded data outside the buffer is then removed.
- The Near value of the **Near/Far** option is the distance the grid is to be clipped back to from the convex hull of the data points. The Far distance is the distance between the points to interpolate between. Areas in the grid which lie between data points greater than the Far distance value grid will be displayed as null or “white”.
- The **Convex Hull** option clips the output grid to the smallest convex region/polygon enclosing the dataset. This convex region is found conceptually by stretching a rubber band around the points so that all of the points lie within the band. The convex region can be expanded by a specified Buffer value.
- The **Concave Hull** option starts with a Convex Hull region, and then removes/erodes triangles on its edge that have an outward facing angle greater than a defined tolerance (specified under the Options button). This new concave region can then be expanded by a specified Buffer value.

If multiple fields have been selected for gridding then three output options are available:

- **Separate grids** - each selected field is created as a separate grid which is named using the original source table with the field name extension. E.g. Geochem_Cu, Geochem_Pb, etc.
- **Multi-banded grid** - multiple fields are gridded and saved to a single ERMapper Multi-banded grid.
- **Create RGB Image** - if three fields are selected for gridding they can be saved as an RGB image. Each field is saved as a red, green or blue channel in the resulting image.

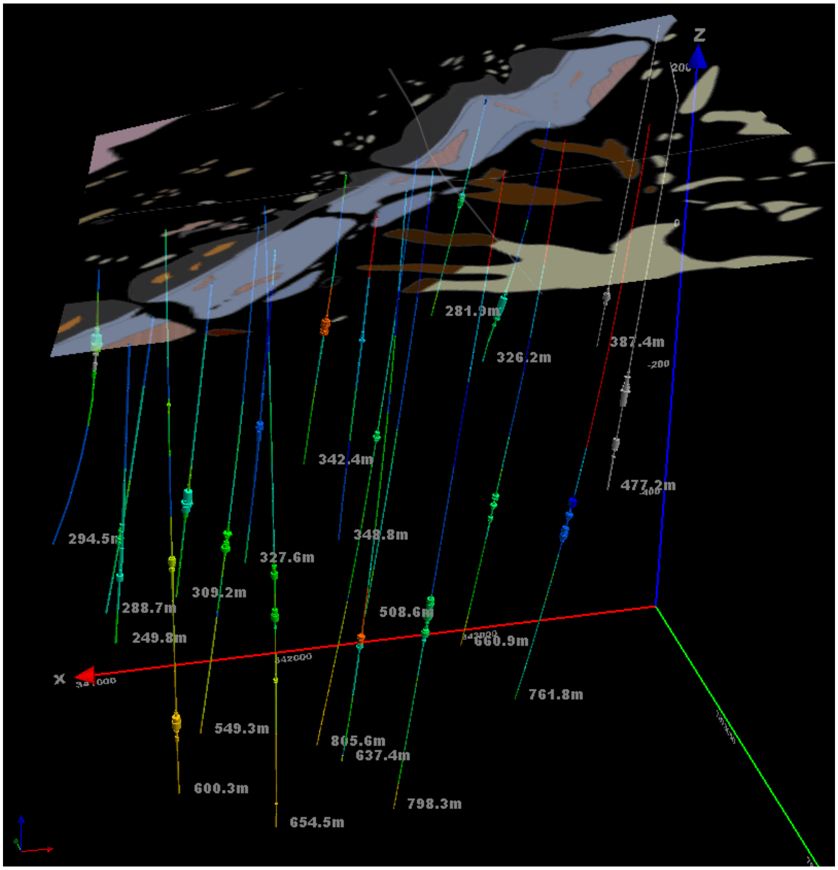
When the Gridding Tool parameters have been selected and the grid surface is completed, press the **Save** button. This button closes the Gridding Tool dialog, saves the grid surface into the specified file and updates the original map window with the surface grid. The surface grid is displayed using the same look-up table as selected in the Gridding Tool dialog prior to saving.

14 Working with Drillholes and Trenches in 3D

Discover 3D allows drillholes and trenches and their associated downhole datasets to be quickly and easily displayed, allowing you to view your drilling in ‘real-space’.

Combining this with gridded surfaces, raster images, voxel models and point/line data allows the geologist to view their exploration projects in a complete and coherent 3D environment. This allows the geologist to start assessing and interpreting their drillhole project’s geology and mineralization in the 3D world only previously accessible in high-end mining packages.

In addition to Discover Drillholes projects, any Trenches or Costeans associated with these project can also be visualized in 3D, including face/chip sampling, or well sampling. It is also an effective method to display Boreholes/Drillholes with XYZ gyroscopic coordinates provided.



Drillhole and trench data can be displayed in 3D via:

- Opening a Drillhole Project in MapInfo/Discover and using the **Discover 3D>View Drillholes** tool (see *Displaying Drillhole and Trench Data in 3D*), or
- Directly opening a *Discover Drillhole Project* or a *Geosoft WHOLEPLOT Database* in Discover 3D.
- Opening a Drillhole Project with associated Trenches/Costeans in MapInfo/Discover, then use the **Discover 3D>View Trenches** (see *Displaying Drillhole and Trench Data in 3D*)

Once drillhole or trench data is displayed in Discover 3D, there is an extensive range of display controls available for both the drillholes and the data that may be associated with them. For example:

Drillholes

(see *Changing Drillhole Display Properties*)

- *Trace Style -* trace paths can be displayed to show the accurate location of the drillholes.
- *Labelling* - drillholes can be labeled with information such as hole names, EOH depths, downhole depths, etc.
- *Colour Modulation* - drillhole traces can be colour modulated (including patterns) using downhole attribute (e.g. geology) or numeric (e.g. mag sus) information.
- *Thickness Modulation* - drillhole traces can be thickness modulated using one or more downhole numeric fields, such as assay or geophysical measurements.
- *Display Drillhole Logs Images* - Logs can be quickly displayed as georeferenced images in 3D This function is not available with trenches and costeans.

Trenches

(see *Changing Trench Display Properties*)

- *Trace Style -* trace paths can be displayed to show the accurate location of the drillholes.

- *Labelling* - trenches can be labeled with information such as names, lengths, etc.
- *Colour Modulation* - trench traces can be colour modulated (including patterns) using attributes or numeric information.
- *Thickness Modulation* - trench traces can be thickness modulated using one or more numeric fields, such as assays.

Displaying Drillholes and Trenches in 3D

A drillhole or trench dataset can be displayed in Discover 3D via two methods:

- From the within the 2D MapInfo/Discover interface.

  If a Discover Drillhole Project has been created and is open in MapInfo (**Drillholes>Project Setup**), use the **Discover 3D>View Drillholes** tool (see *Displaying Drillhole and Trench Data in 3D*). This is generally the easiest and the recommended method or drillhole loading.

  Upon loading into Discover 3D, a new Drillhole branch will be added to the Workspace Tree, and all drillholes within the project will be displayed in the 3D view. See the *Changing Drillhole Display Properties* section below for visualising and controlling the drillholes.

  The same method can be used to display any Trench associated with the Drillhole project, by using **Discover 3D>View Trenches** and a Trench branch will be added to the workspace tree.

Note Trenches in 3D does not support Bearing and Distance Trench project with Segment distances. It also does not support Bearing and Distance Survey (either cumulative or segments) when a Topographic DEM surface grid is associated with the project.

- From within the Discover 3D window.

Discover 3D supports the direct loading of two drillhole data base types:

- **Discover Drillhole Projects.** Select the **Open Discover Drillholes**…button of the main toolbar to browse the target drillhole tables and click **Open**. See the *Discover Drillhole Project* section below for further information.

- **Geosoft WHOLEPLOT databases** Select the **Open Geosoft Drillholes**…button of the main toolbar to browse the target drillhole tables and click **Open**. See the *Geosoft WHOLEPLOT Database* section below for further information.

Once a drillhole database is loaded , add a new **Drillholes branch** to the Workspace tree by selecting the **Drillholes** button (on the Data Objects toolbar)or choose the **Display>Drillhole** menu item.

A **Drillholes** branch is created in the Workspace tree and the first of the available holes is displayed as a 3D hole.

Note To update a 3D Drillhole display after altering drillhole data in MapInfo/Discover, use the **Discover 3D>Refresh All 3D Data** option in the 2D interface

Drillhole Dataset Format

Typically drillholes and associated data require the following information:

- Collar positions in X, Y and vertically (Z) referenced elevation. Names of the drillholes are also associated with drillhole collars. Usually the total distance along the hole axis from the collar to the base of the hole is included in this file.

- Downhole Survey data must be associated with each Hole_ID to define the location of points down the hole. This data defines the shape of the hole and is usually derived from a survey undertaken in the field using a downhole camera and survey software. The data requires a distance down the hole axis, a dip and a compass azimuth reading at the measured survey point.

  A minimum number of two points can be used to survey a hole. For a straight hole, only a single entry is required as the azimuth and dip at the collar is assumed to the base of the hole. Note that the survey describes the shape of the drillhole, not the points at which assays or data readings or samples are made.

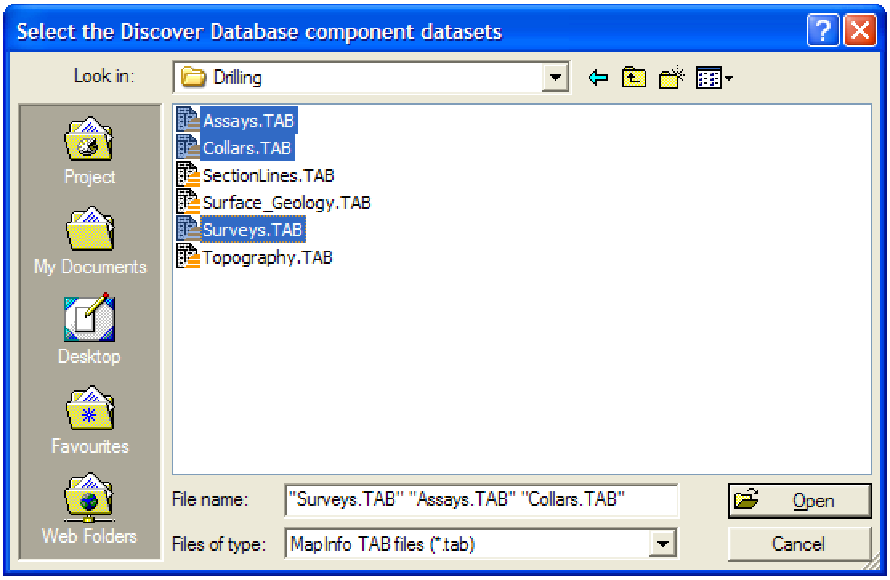
- Downhole Data such as assay samples, readings or analyses can be done at measured depths of the hole. These readings may be geophysical (eg magnetic susceptibility, density, gamma etc) or they may be geological (eg lithological, assay or geochemical). Readings can be taken at a specific distance point down the axis of the hole, or over an interval that is using from-to terminology (for example, lithology between 50.0 and 50.5 metres distance the hole intersected mudstone).

Discover Drillhole Project

Refer to the *Discover documentation* for information regarding how to create a *Drillhole Project*. Usually, the Drillhole Project consists of three or more MapInfo Professional .TAB files similar to:

- COLLARS.TAB – defines the Hole_ID, collar location (XYZ) and depth of holes.
- SURVEY.TAB – specifies the survey points down the various holes. This file defines the shape of the drillholes.
- DATA.TAB – typically these files contain assay or lithological data.

Select **File>Open>Discover Downhole** or click on the button (shown left)) on the Main Toolbar. Browse to the drillhole folder, choose the MapInfo Drillhole tables and click **Open**.

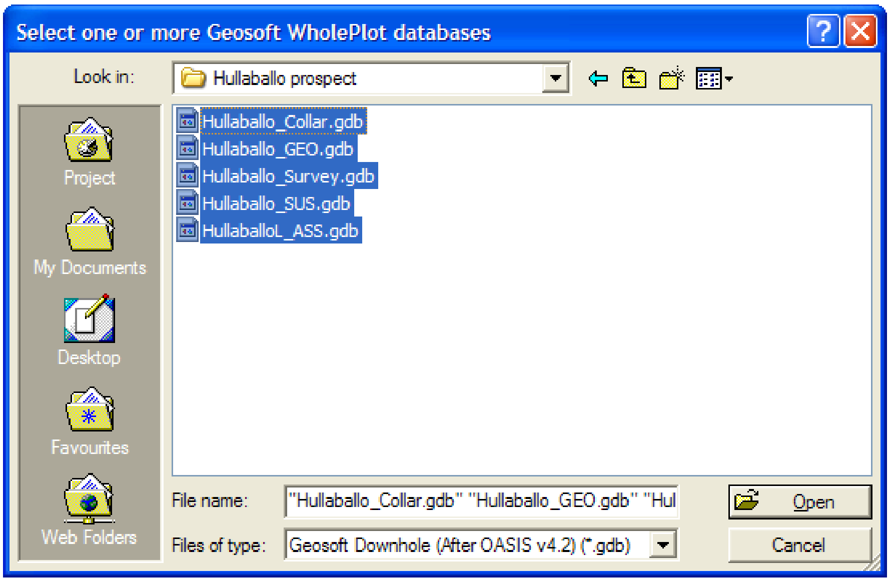


Selecting the Discover Downhole database files

Geosoft WHOLEPLOT Database

Geosoft drillhole databases are created by the WHOLEPLOT option of Oasis montaj™ and usually have a .GDB name extension. Depending on the version of montaj™ used to create the database, the collar, survey and data content may be contained in a single .GDB file or multiple .GDB files.

Select **File>Open>Geosoft Downhole**…or click on the button (shown left) on the Main Toolbar. Browse to the drillhole folder, choose the Geosoft Database and click **Open**.



Selecting the Geosoft WHOLEPLOT database file

Geosoft databases generally define downward dipping drillholes as negative values so Discover 3D assumes the Geosoft database adheres to this format. If this needs to be reversed, the **File>Modify Dataset Field and Survey Properties** tool (from the 3D interface) provides an option to do so for any Geosoft drillhole database loaded.

Note For the above changes to take effect any existing drillhole branch containing the Geosoft drillhole project must be deleted first and a new branch opened.

For additional information on this data format, please refer to *the Geosoft WHOLEPLOT documentation.*

Desurvey Methods

Upon loading a drillhole project into Discover 3D, the drillholes are desurveyed using the **Back Calculation** method (detailed below) by default, as this is the method used by Discover.

The user can however choose a different method by opening the **File>Modify Dataset Field and Survey Properties** dialog, and selecting the desired method from the pull-down list of **Desurvey Methods** at the bottom of the dialog. This change will apply permanently to the drillhole project each time it is opened into Discover 3D, until a further change is made in this dialog.

The methods available are:

- *Back Calculation* (Discover)
- *Segments*
- *Minimum Curvature*
- *Akima Spline*
- *Bezier Spline*
- *XYZ (trenches only)*

Back Calculation

This is the only method offered in Discover and is the default method in Discover 3D. Although it is a simple method it does produce reasonable results. This method is also known as mid-point tangential averaging.

Each survey point is considered to lie in the middle of a drillhole segment which has the dip and azimuth of the survey location. The dip and azimuth of the hole changes automatically halfway between each survey point. This has the affect of intersecting a midpoint in between the actual survey points, and may not actually intersect a survey point. It also means that the first and last interval of the trace will be half their expected length.

Put simply, if your collar is at 0 m and finishes at 40 m and you have down hole surveys at depth intervals of 10 m, 20 m and 30 m, then Discover will compute the the end X,Y,Z coordinate of the first line segment using the midpoint distance between each depth interval (e.g. 0-5 m). It then uses corresponding dip and azimuth values from the surveys (10 m measurement), to project the second line segment (5-15 m). At this point it then uses the next Survey dip and azimuth record (20 m) to generate the next segment.

This has the effect of making 'curved' holes more curved and a hole will diverge further from the 'straight line' (called segments in Discover 3D) location at increasing depth.

Note that if you have a high frequency of survey data (e.g. 1 m intervals), then the effect of the interpolation will be negligible.

Note If performing data interpretation on a drillhole project in both the 2D (MapInfo/Discover) and 3D environments, it is recommended to utilise the default Back Calculation method.

Segments

This is the simplest method available. It is similar to back calculation but produces less acceptable results.

Each survey point is considered to be the start of a drillhole segment which has the dip and azimuth of the survey location. The dip and azimuth of the hole changes automatically at each survey point.

The Segments method is also used to display Trenches measured by distance and bearing in 3D.

Minimum Curvature

This is the most robust method and is highly recommended. It is also known as radius of curvature and is considered the standard method in the oil and gas industry.

Each survey point is considered to lie at the start point on a spherical arc (or straight line) whose radius is defined by the change in the direction of the hole from the survey point to the next survey point. The dip and azimuth of the hole varies smoothly between survey points.

The Discover 3D implementation is based on the formulae presented by Sawaryn & Thorogood; A Compendium of Directional Calculations Based on the Minimum Curvature Method presented in the *Society of Petroleum Engineers Journal*.

Akima Spline

This method is also very robust and produces very similar results to the minimum curvature method.

Smooth Akima spline functions are fitted to the dip and azimuth survey data for the length of the hole. The hole is then desurveyed using the segment method and a small, constant depth interval where the dip and azimuth for each segment is acquired from the spline functions.

Bezier Spline

This method is exactly the same as the Akima spline method except that a Bezier spline function is used instead of an Akima spline.

Use of this method is not recommended.

XYZ (trenches only)

This method is only available for trenches, measured by XYZ coordinates for each survey point.

It simply plots each survey point's XYZ coordinates, and then join these with a straight line.

Interrogating Downhole Information

Two modes of dynamic drillhole information are available in 3D:

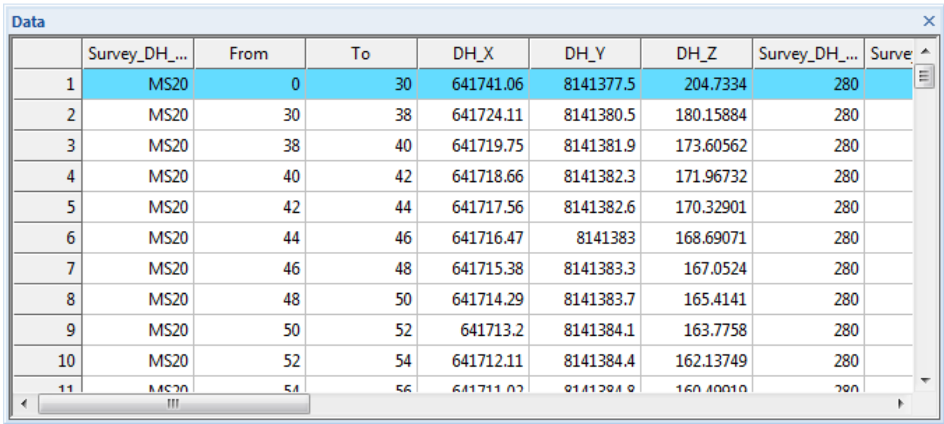
- **Basic** - when the **Select/Navigate** button is enabled on the *Zoom Controls Toolbar,* place the mouse cursor over a drillhole trace to display the drillhole name and the downhole depth of the drillhole. The results are continuously displayed in the status bar at the bottom left corner of the 3D window.
- **Advanced** – allows detailed downhole data to be dynamically displayed in the Information sheet, including 3D coordinates, from/to intervals, geology, assays, etc.

To enable advanced interrogation:

1. In *Workspace Tree*, make the Drillhole branch both Selectable and Browsable.

2. Display the Data Window using either the toolbar button or the **View>Data Window** menu option.

3. Enable the **Select/Navigate** button on the *Zoom Controls Toolbar.* and click on a drillhole trace (this will auto-enable both the Cursor Plane and its Bond function on the *Cursor Plane Toolbar*. The Data window will scroll to the appropriate downhole record for the coincident hole interval.


Data

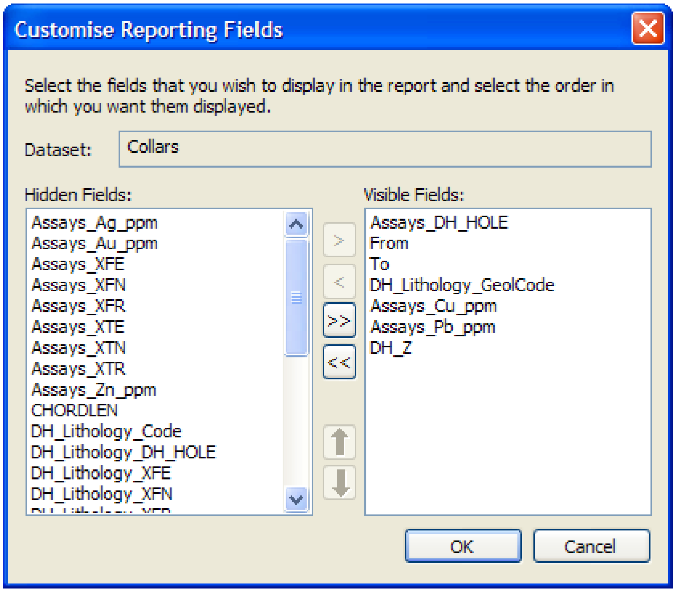
	Survey_DH_...	From	To	DH_X	DH_Y	DH_Z	Survey_DH_...	Surve
1	MS20	0	30	641741.06	8141377.5	204.7334	280	
2	MS20	30	38	641724.11	8141380.5	180.15884	280	
3	MS20	38	40	641719.75	8141381.9	173.60562	280	
4	MS20	40	42	641718.66	8141382.3	171.96732	280	
5	MS20	42	44	641717.56	8141382.6	170.32901	280	
6	MS20	44	46	641716.47	8141383	168.69071	280	
7	MS20	46	48	641715.38	8141383.3	167.0524	280	
8	MS20	48	50	641714.29	8141383.7	165.4141	280	
9	MS20	50	52	641713.2	8141384.1	163.7758	280	
10	MS20	52	54	641712.11	8141384.4	162.13749	280	

Using the Data tab of the Information sheet to evaluate downhole data.

The Data window provides a single combined spreadsheet view for all downhole data tables for the drillhole project. Discover 3D segments the various tables to the shortest interval over each segment and provides the following information at the start of the spreadsheet, in addition to the fields inherent in each table:

- Hole ID
- From/To fields for the segment intervals
- 3D coordinates (DH_X, DH_Y and DH_Z fields)

This single combined spreadsheet can result in a large number of fields being displayed, with the actual desired fields scrolling out of view. The spreadsheet can be customised in terms of actual downhole fields displayed and their order, by right clicking in the Data window and selecting the **Customise** option. These Data window customisation options (including field widths) are preserved as long the as the current Discover 3D session is open.



Customising the Data tab view

If a group of cells is highlighted in the data window, right-clicking will present a **Copy selected cells to clipboard** menu option. This can be useful for extracting data into Excel for data analysis or other uses.

This is also a useful way of merging multiple downhole datasets into a combined table processed to the smallest interval in every table.

Changing Drillhole Display Properties

Most aspects of drillhole displays can be controlled from the **Properties** dialog opened from the **Drillholes** branch of the Workspace Tree. To display the **Drillholes Properties** dialog, either:

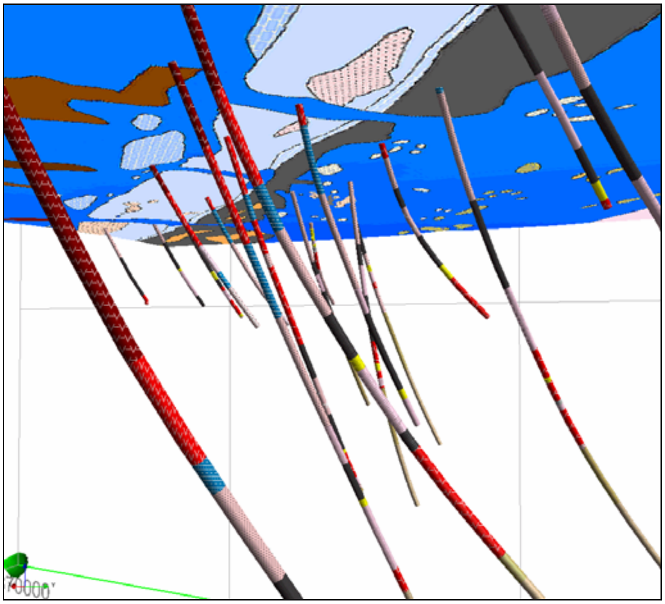
- Move the cursor over the Drillholes branch in the Workspace tree and double click with the left mouse button, or
- Highlight the Drillholes branch in the Workspace tree and click the right mouse button. From the resulting shortcut menu, select the Properties item.

The display settings made on the Drillholes Properties dialog box can be saved or loaded from a settings file (see *Saving and Applying Display Settings*). The display controls available depend on the type of data and may include:

- *Selecting Drillholes*

- *Trace Style*
- *Labelling*
- *Symbols*
- *Colour Modulation*
- *Thickness Modulation*
- *Display Drillhole Logs Images*
- *Data Compression*

An example of a downhole display with lithology is shown below.



Example drillholes with downhole lithology displayed

Saving and Applying Display Settings

Any display settings customized in the Drillhole Properties tab controls (e.g. colour or thickness modulation) can be saved for later reuse. This is an excellent way to maintain the same display settings between drillhole project data updates, or between projects sharing the same data structure.

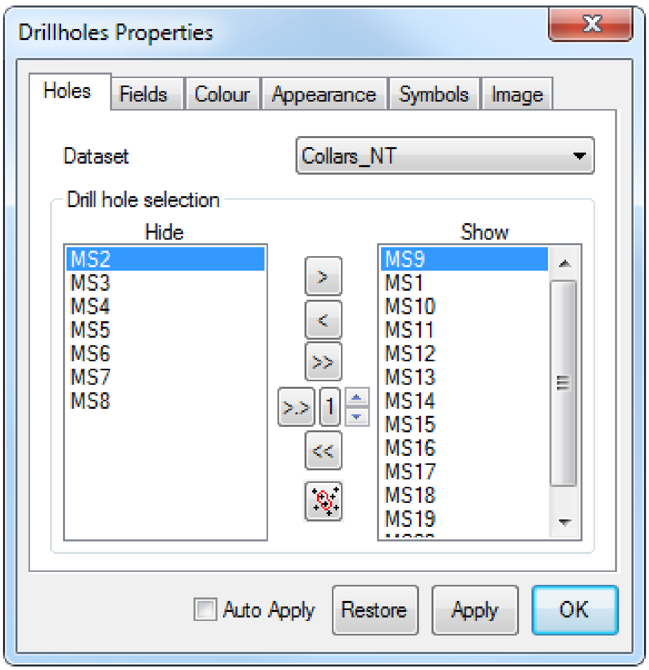
To save any current display settings, move the cursor over the Drillholes branch in the Workspace tree and right click. Select the **Save Properties** option, and specify a name and location for the output display settings .XML file (e.g. a under the drillhole project root directory, or in a user-created settings folder elsewhere).

To reapply display settings at a later date (e.g. for an updated version of the same drillhole project, or another project), first load the drillhole project into 3D. Move the cursor over the Drillholes branch in the Workspace tree and right click. Select the Load Properties option, and browse for the desired settings .XML.

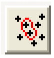
Note Display Settings can ONLY be applied to drillhole projects that have exactly the same table names and field names and types. Any differences in table names or structure will likely cause settings to not display.

Selecting Drillholes

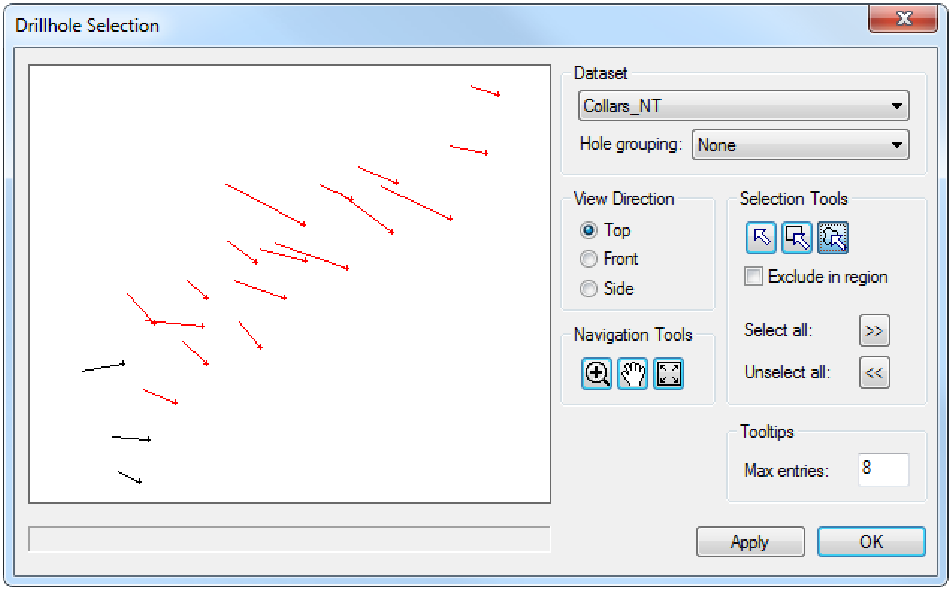
The **Holes** tab of the **Drillholes Properties** dialog allows the user to control which holes are displayed.



Drillhole selection using the Holes tab



Specify the holes to be displayed by moving the drillhole name(s) from the **Hide** to the **Show** list. Alternatively, you can graphically select the holes to be displayed from a collar map using the **Graphical Selection** button.

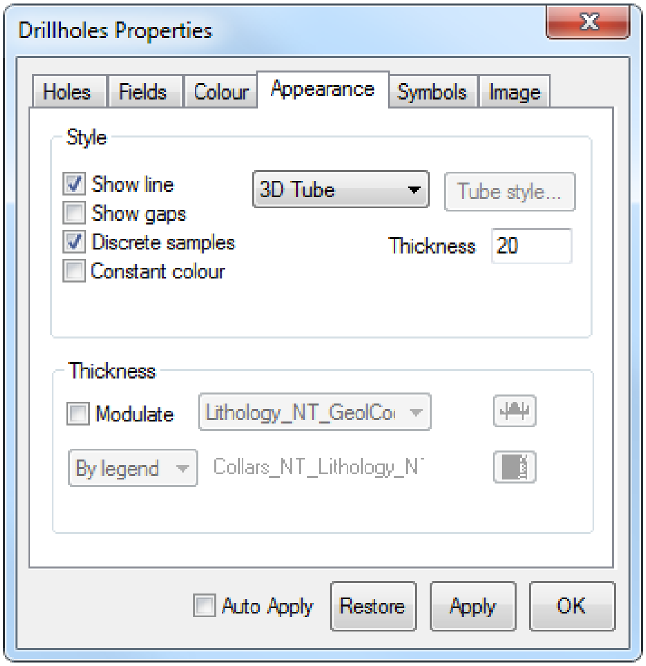


Drillhole graphical collar selection

Note Drillhole collar names can be displayed in the graphical selection dialog by placing the cursor over a drill hole and viewing the name in the left corner of the status bar at the bottom of the dialog.

Trace Style

The **Appearance** tab allows the trace style of drillholes to be altered.



Appearance tab of Drillhole Properties

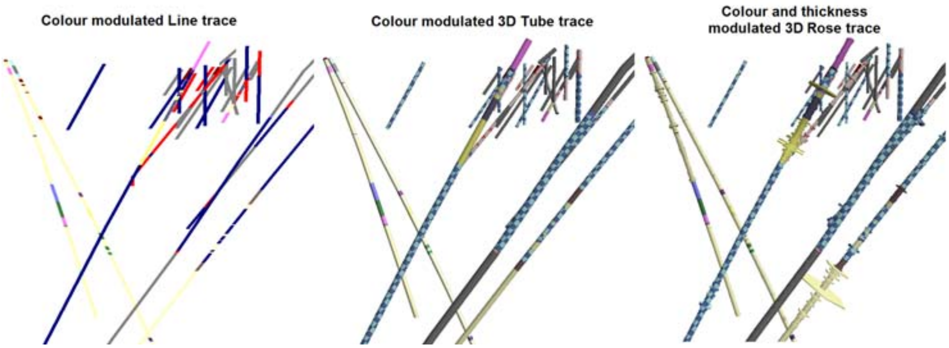
To alter the **Trace Style**, ensure the **Show Line** option is enabled, and choose one of the following styles from the adjacent pull-down list:

- **Line** - This is the most efficient display style, and is recommended as the initial style when displaying large datasets. Hole traces are displayed as a simple line, with colour, line weight and basic pattern options under the adjacent **Line Style** button. *Colour Modulation* is possible but thickness modulation is not.

- **3D Tube** - Drillholes can be displayed as solid tubes, with a cross-section profile (and colour) set via the adjacent **Tube Style** button (only available when the **Discrete samples** option is disabled: default Tube_Medium style). Tube thickness (in projection units) and colour can also be specified. *Colour Modulation* is possible, as well as *Thickness Modulation* by a single field.

- **3D Rose -** This is the most memory intensive of the trace styles, allowing the width of the trace to be modulated based on multiple downhole numeric data fields (such as an assay or geophysical fields). These are set using the adjacent **Petals button** as discussed in the *Thickness Modulation* section below. It requires specification of the Thickness range (minimum and maximum widths) at the bottom of the tab. An Initial Angle control is also available, controlling the initial starting angle for rose segments when more than one numeric field is specified. A *Colour Modulation* can also be applied, or a constant colour set via the **Colour** button.

Different drillhole appearances are shown here:

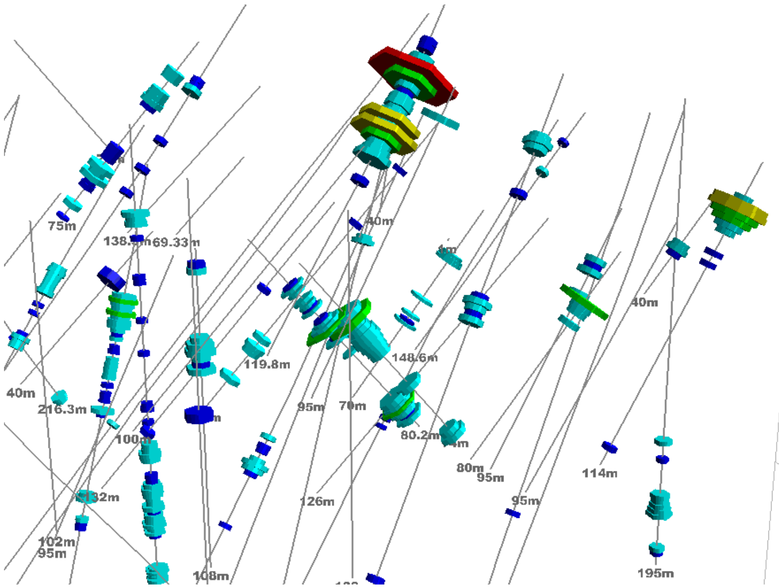


Examples of colour modulation of drillholes as applied to Line, 3D Tube and 3D Rose traces using lithological data. The thickness modulation of the 3D Rose traces has been applied using an assay field.

The **Show Gaps** option is only available (not greyed out) when a *Colour Modulation* and/or one or more *Thickness Modulation* fields has been assigned. Deselecting this option (un-ticking it) will result in any sampling gaps not being displayed; i.e. drillhole traces will be discontinuous if sampling gaps exist. This is an excellent way of visualising only populated sample intervals, particularly after using Data Conditioning (see *Field Data Conditioning Tool*) e.g. nulls removed or only anomalous data intervals displayed.

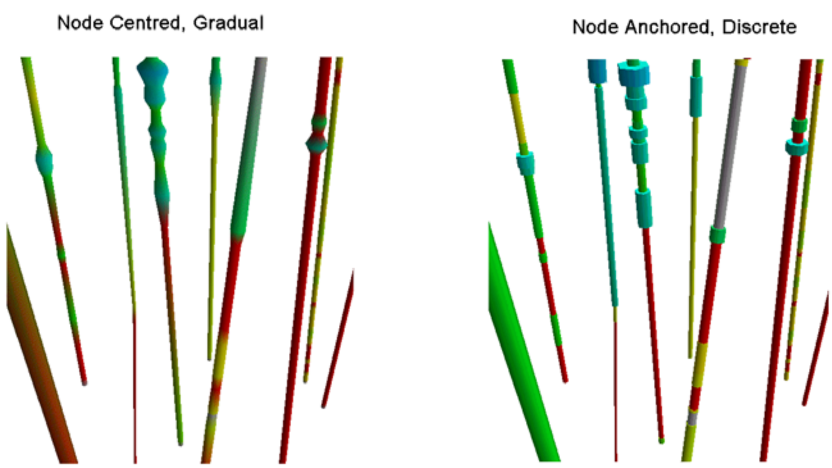
Tip

If after applying **Data Conditioning** in tandem with disabling **Show Gaps**, you see only isolated downhole intervals and require the spatial reference of the entire drillhole trace, try adding another Drillhole branch to the workspace tree, and populating it with the same drillhole dataset. Set the display **Style** of this second branch as **Lines**- it will present your entire dataset as drillhole trace ‘skeletons’ over which your isolated intervals will be located, as pictured below.



The same drillhole dataset displayed in two separate branches in the workspace tree. One branch utilises a thin Line Style display to show the drillhole skeletons (thin grey lines); the other is both thickness and colour modulated, with the Show Gaps option disabled and Discrete samples enabled.

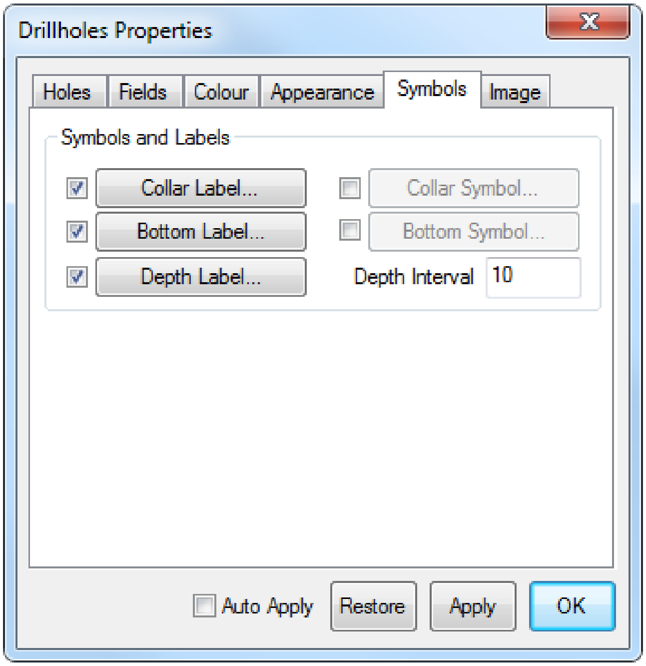
Discrete samples are only applicable to the **3D Tube** or **3D Rose** trace styles**.** When enabled, each sample interval is displayed as a uniform thickness across the interval (i.e. discs as pictured above). Assay intervals are displayed as steps/ disks of varying thicknesses. Disabling this option enables the hole thickness to be altered gradually/ continuously across sample intervals. Examples of these modes are presented below.



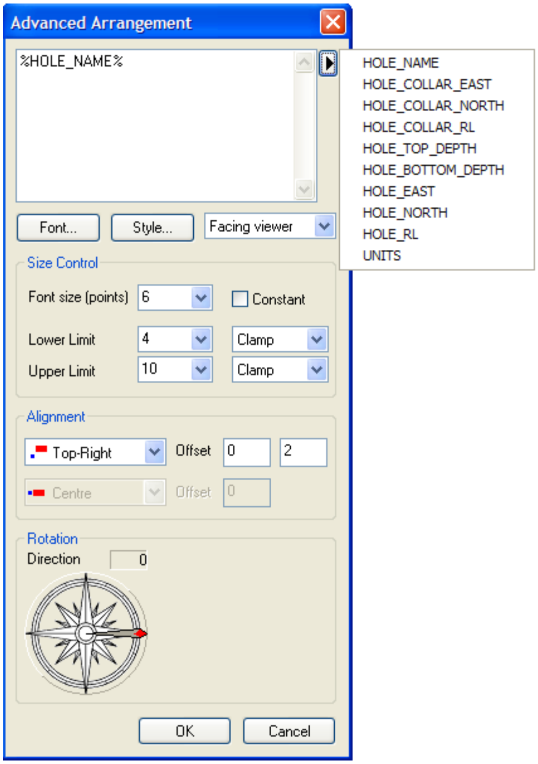
Examples of Gradual and Discrete thickness modulated (3D Rose) downhole assay *displays*

The **Override background colour** option will only have an effect when a **Legend** is applied as *Colour Modulation* (3D Tube or 3D Rose styles). It will replace the background colour of each legend entry with the colour assigned in either the **Tube Style** or **Colour** buttons. Foreground colours will be preserved.

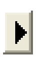
Labelling



Drillhole collars and bottoms can be annotated with **labels** by enabling the appropriate tick box in the **Symbols** tab. To change or customise the label display, click the adjacent **Collar Label**, **Bottom Label** or **Depth Label** button to open the **Advanced Arrangement** dialog.



The Advanced Arrangement dialog for applying drillhole collar and bottom labels



The **Collar** and **Bottom Label** buttons display the current label in a text box at the top of the dialog; to change this, highlight the entire text entry and click the **Field Selection** button (shown left). Select the desired field from the displayed list (populated from the drillhole collar file). Multiple fields can be displayed together by placing the cursor at the desired insertion point within the text window (without highlighting any existing text) and then selecting extra fields from the field selection list (without highlighting any existing text) and then selecting extra fields from the filed selection list.

The **Depth Label** button relaces the text box with a **Format** button; this can be used to set numeric formatting, prefixes and suffixes. Activating the **Depth Label** option also requires setting a **Depth Interval** in the **Appearance** tab (i.e how often downhole depths are displayed).

For all label types, standard **Font** controls are provided, as well as the following range of orientation options in the adjacent pull-down list:

- **Facing Viewer** (default) – Labels are aligned parallel to the viewing/ screen plane so that they always face the viewer.

- **Fixed 3D** – Labels are fixed in the 3D environment. The pull-down list in the Rotation panel at the bottom of the dialog controls the initial orientation: **Flat** (the XY plane) or **Upright** (XZ plane).

- **Fast 3D** – identical to the Fixed 3D except that no **Style** controls are available. This is a very fast and memory-efficient labelling option.

The **Style** button is only available for the Facing Viewer and Fixed 3D orientation options. It allows labels to be extruded either as filled Polygons or Line Segments (wireframe) using the **Format** pull-down list. The depth of the label is set using the **Extrusion** control, expressed as a percentage of the label size. To display a flat label, set the **Extrusion** to 0%. The label is extruded perpendicular to its display plane.

Note Rendering labels as extrusions is memory intensive and will affect 3D performance. It is not recommended for large numbers of labels.

A range of **Size Controls** are available. For the **Fast 3D** orientation option, only the **Font Size** control is available (in points). Enabling the **Constant** checkbox (for the other orientation options) will keep the labels at the specified size (relative to the screen) regardless of zoom level. If the Constant option is disabled, **Lower** and **Upper Limits** can instead be set:

- **Block** - labels will disappear when the applied zoom level takes the label past the specified limit,

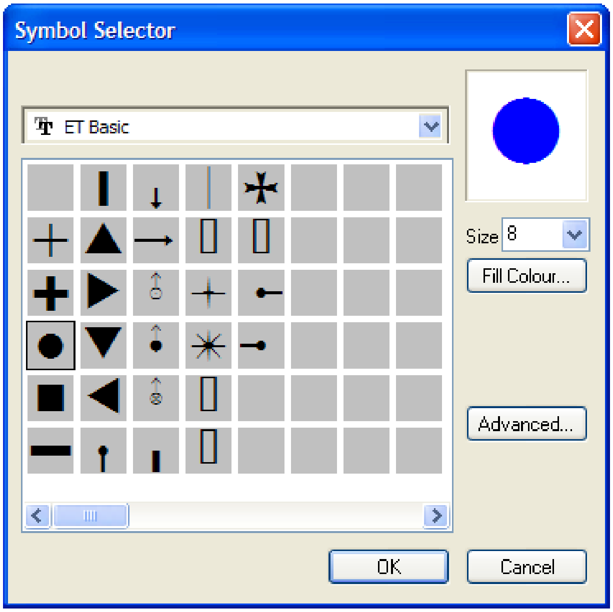
- **Clamp** - labels will be locked to the specified limit when the applied zoom level takes the label past the specified limit.

The label position relative to the data (collar) location can be altered using the **Alignment** controls (either preset or manual positioning). The first row of controls concern label positioning in the label plane (i.e. the relative XY components), whilst the second row controls the vertical height of the label with respect to its initial plane (i.e. the relative Z component).

The angle of the labels can also be set by moving the red-tipped arrow on the compass in the **Rotation** panel at the bottom of the dialog. The **Fixed** and **Fast 3D** labelling options also provide a **Dip** control (half-compass) in this panel.

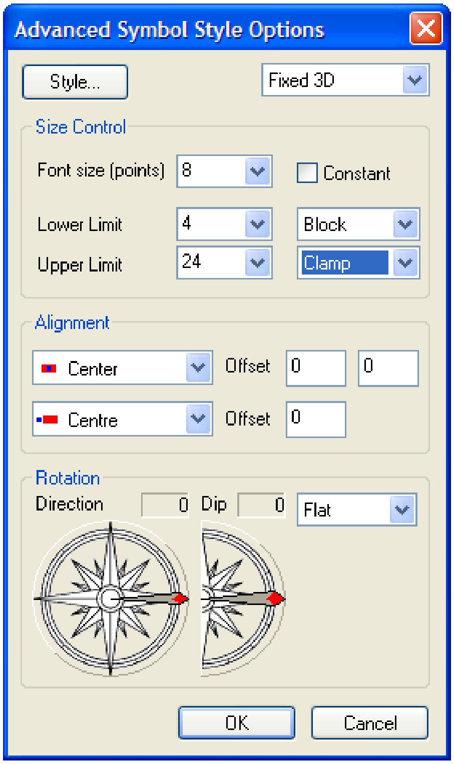
Symbols

The collars and/or bottoms of drillholes can be annotated with a symbol by enabling the **Collar Symbol** or **Bottom Symbol** options in the **Symbol** tab. The adjacent buttons open the **Symbol Selector** dialog allowing you to choose from an extensive library of symbols. Choose a symbol from the list to view a preview on the right side of the dialog. The symbol **Size** and **Fill Colour** can also be modified.



Symbol Selector dialog

Advanced Symbol Style Options are available via the **Advanced** button.



Advanced Symbol Style Options dialog

A range of symbol orientation options are presented in the pull-down list at the top right of the dialog:

- **Facing Viewer** (default) – Symbols are aligned parallel to the viewing/ screen plane so that they always face the viewer.
- **Fixed 3D** – Symbols are fixed in the 3D environment. The pull-down list in the Rotation panel at the bottom of the dialog controls the initial orientation: Flat (the XY plane) or Upright (XZ plane).
- **Fast 3D** – identical to the Fixed 3D, except that no **Style** controls are available. This is a very fast and memory-efficient option.

The **Style** button is only available for the **Facing Viewer** and **Fixed 3D** orientation options. It allows symbols to be extruded either as filled Polygons or Line Segments (wireframe) using the **Format** pull-down list. The depth of the symbol is set using the **Extrusion** control, expressed as a percentage of the symbol size. To display a flat symbol, set the **Extrusion** to 0%. The symbol is extruded perpendicular to its display plane.

Note Rendering symbols as extrusions is memory intensive, and will effect 3D performance. It is not recommended for large numbers of symbols.

A range of **Size Controls** are available. For the **Fast 3D** orientation option, only the **Font Size** control is available (in points). Enabling the **Constant** checkbox (for the other orientation options) will keep the symbols at the specified size (relative to the screen) regardless of zoom level. If the Constant option is disabled, **Lower** and **Upper Limits** can instead be set:

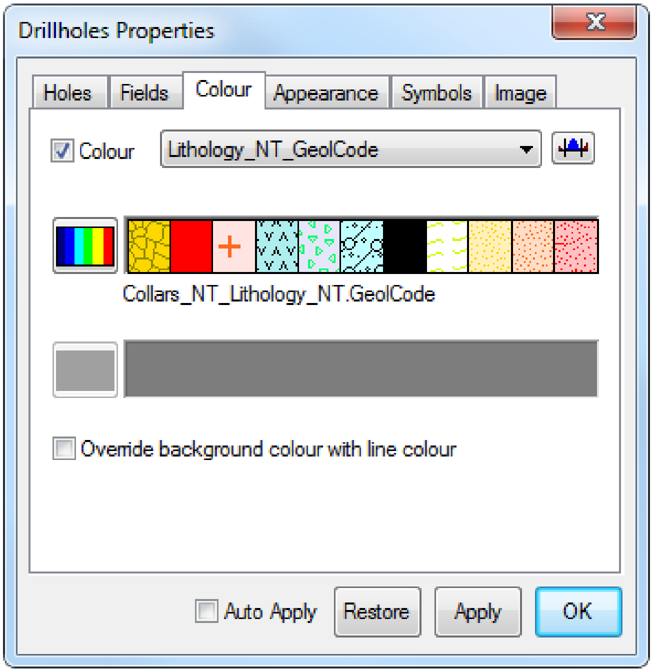
- **Block**: symbols will disappear when the applied zoom level takes the symbol past the specified limit,
- **Clamp**: symbols will be locked to the specified limit when the applied zoom level takes the symbols past the specified limit.

The symbol position relative to the data location can be altered using the **Alignment** controls (either preset or manual positioning). The first row of controls concern symbol positioning in the symbol plane (i.e. the relative XY components), whilst the second row controls the vertical height of the symbol with respect to its initial plane (i.e. the relative Z component).

The angle of the symbols can also be set by moving the red-tipped arrow on the compass in the **Rotation** panel at the bottom of the dialog. The **Fixed** and **Fast 3D** labelling options also provide a **Dip** control (half-compass) in this panel.

Colour Modulation

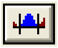
Drillhole traces can be colour modulated using controls provided under the **Colour** tab of the **Drillholes Properties** dialog. For example, colour drillholes based on a downhole lithology, assay, or alteration field.



The Colour tab configured to colour modulate drillhole traces using a lithology field and a custom colour legend.

Enable the **Colour** tick box and set the desired data field from the adjacent pull-down list (available data fields are named with the following convention: *Tablename_Fieldname*). Click the first **Edit** button to open the **Colour Scale** dialog.

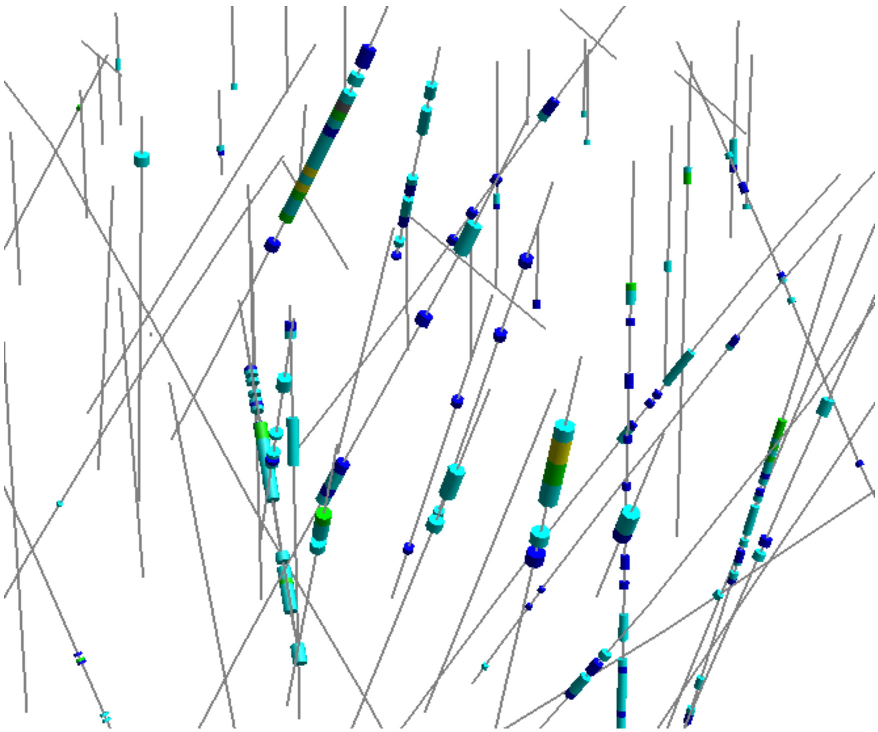
In the Colour Scale dialog, select a custom **Legend** if one has already been created using the Legend Editor (see *Using the Legend Editor*), either in 3D or more typically in 2D as part of Discover Drillhole Project. A range of standard colour interpolations (RGB & HSL) and **Look-Up Tables** (see *Using the Colour Look-Up Table Editor*) are also available.



The **Field Data Conditioning** button at the far right of the Colour tab is used to apply conditions to the selected colour field. Conditions available include data capping (i.e. setting upper data limits to minimise nugget effect for assay data) or excluding unwanted categories from the display (e.g. overburden such as soil and alluvials). See *Field Data Conditioning Tool* for use of this tool.

Note The **Field Data Conditioning** dialog is also an excellent way to visualise only the desired subset of a dataset: for instance only Quartz_Vein and Fault lithological intercepts, or only Au assays between 5 & 15g/t. Once a subset is specified, try turning the **Show Gaps** option Off (under the **Appearance tab**), then add another Drillholes branch to the Workspace Tree populated with the entire dataset, but displayed as simple **Lines**.

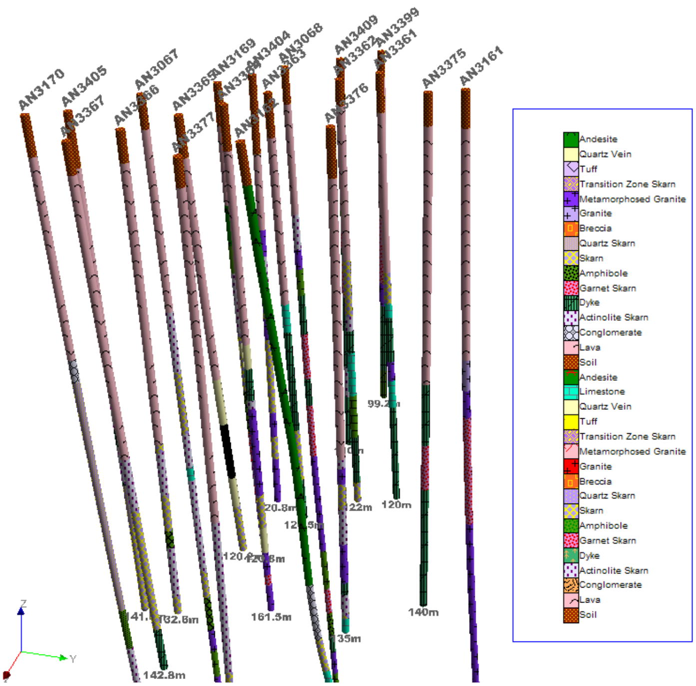
Note Activating the **Override background colour** option in the **Colour** tab will replace any background colours applied via a Legend in the **Appearance** tab with the colour specified in the **Appearance** tab.



Displaying only particular colour modulated geological intercepts by i/. Selecting only target rock types using Field Data Conditioning; ii/. Disabling the Show Gaps option in the Appearance tab & iii/. Displaying the same drillhole project as another branch in the workspace tree, using a Lines display style.

Drillhole Colour Legends

An onscreen colour legend **Floating Colour Bar** can be added to the 3D display which is referenced to a colour modulated drillhole project. This enables the user to easily visualise the relationship between different drillhole colours and the corresponding lithology or assay range values. See the *Floating Colour Bar* section of the **3D Interface** chapter for further information.



Example of a Floating Colour Bar linked to a drillhole project colour modulated by geology; a colour legend with an entry for each rock type is displayed

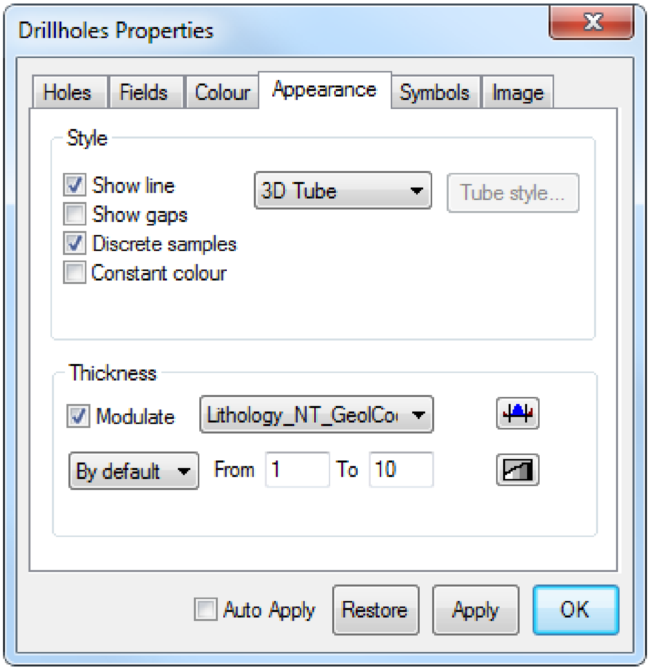
Thickness Modulation

Drillholes can be thickness modulated in the **Appearance tab** either by:

- Single field modulation using the **3D Tube** style.
- Multiple field modulation using the **3D Rose** style.

To vary thickness of drillholes by a single numeric field:

(e.g. assay or geophysical downhole data)



1. In the **Appearance** tab set the hole style to **3D Tube**.

2. Enable the **Modulate Thickness** option at the bottom left of the dialog

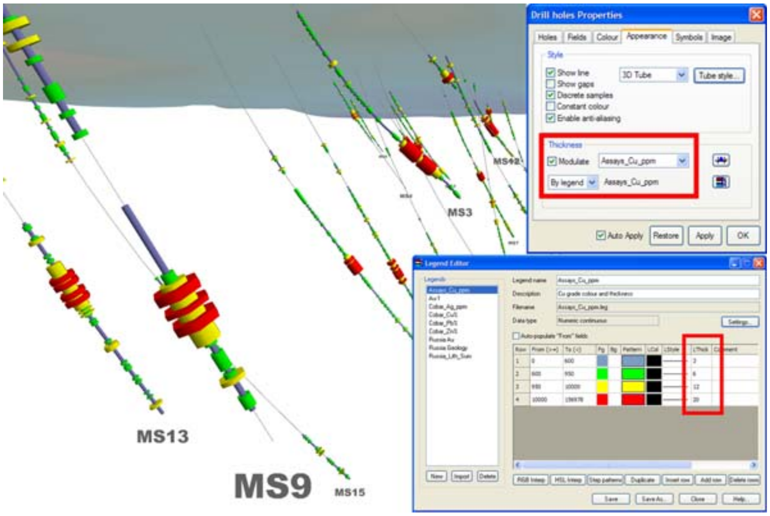
3. Select the required downhole value field from the adjacent pull-down list

4. Use the adjacent **Field Data Conditioning** button (see *Field Data Conditioning Tool*) to remove any null values from the dataset, or to limit the data range to that of interest (e.g. all Au values above 50ppb)

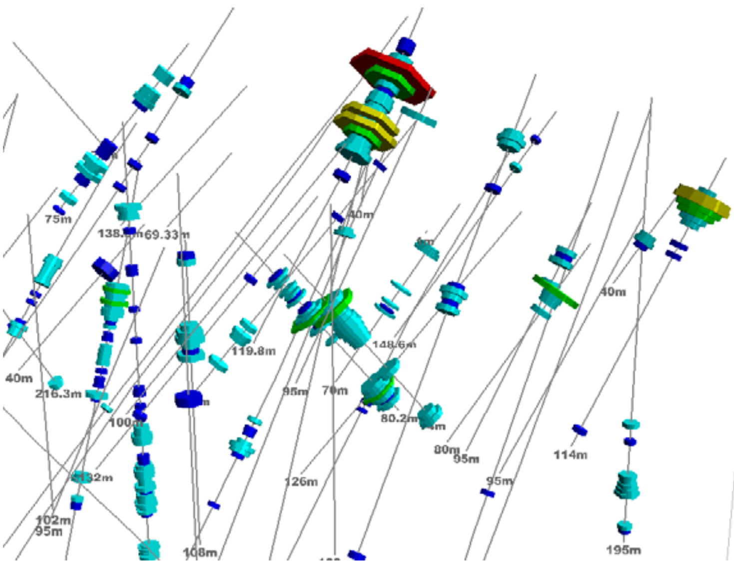
5. The thickness range can be applied either by:

   - A **default** linear transform between the set From and To values. The transform method can be altered using the adjacent Data Mapping button (options include log and histogram transforms). For more information, see *Advanced Colour Mapping*.

   - A custom **legend** of data ranges and associated line thicknesses selected using the adjacent **Legend selector** button, created with the Legend Editor (see *Using the Legend Editor*).



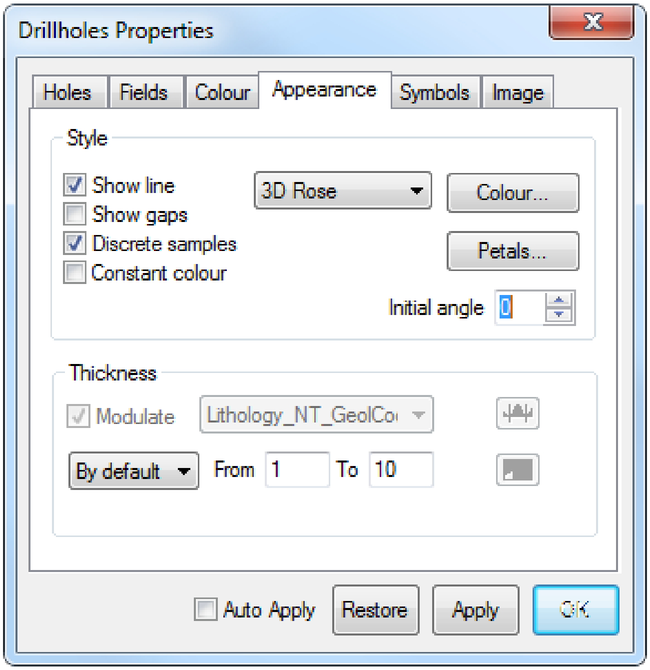
6. Press Apply or OK to complete this process.



An example of drillhole thickness modulation using a single assay field. This dataset has also been colour modulated to reinforce the visual display. Additionally data conditioning has been applied to remove all non-anomalous results (and the Show Gaps option disabled), thereby focussing attention on the significant intercepts. The same dataset was also displayed as a second Drillhole branch in the Workspace Tree, using a Line display style, to provide a continuous spatial reference/skeleton for each hole.

To vary thickness of drillholes by multiple numeric fields simultaneously:

(e.g. assay or geophysical downhole data)

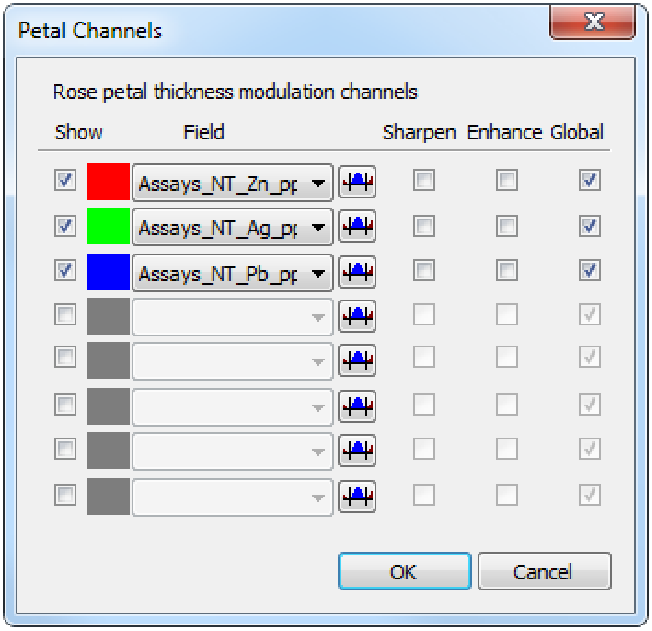


1. In the **Appearance** tab set the hole style to **3D Rose**.

2. In the Thickness section of the tab, leave the modulation to **By default**, and set the From and To values as required (these are the minimum and maximum thickness values for the drillholes).

Note It is not recommended to use the **By legend** option for multiple field modulation.

3. Press the **Petals** button (top right of the dialog) to open the **Petal Channels** dialog.



Specify the thickness of selected drillhole database fields for 3D Hole display

Up to 8 numeric downhole data fields can be specified by checking a **Show** box and selecting the required field to display from the adjacent pull-down lists. Available data fields are named with the following convention: *Tablename_Fieldname*. For each field the following options are available:

- **Colour** - specify a standard or customised colour to display the data in the selected field.
- **Sharpen** - this option squares the data to give greater weighting to larger values. This is a useful way of exaggerating the visual display of high grade intercepts.
- **Enhance** - display the value for each depth using a logarithmic expansion.
- **Global** - this option scales the data in each hole with respect to the entire dataset (as imported into Discover 3D), rather than scaling each hole individually.

In addition each field has a **Field Data Conditioning** button which can be used to condition the source data. Type of conditioning available include data capping (removing nugget effects), null value handling and invalid data assignment (e.g. removal of large negative values representing BDL or SNR that affect thickness distribution). See *Field Data Conditioning Tool* for use of this tool.

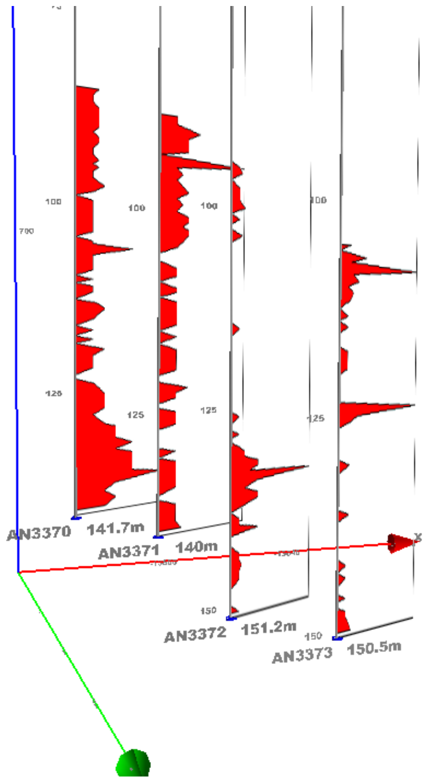
Note The Field Data Conditioning dialog is also an excellent way to visualise only the desired subset of a dataset: for instance only Au assays between 5 and 15g/t. Once a subset is specified, try turning the **Show Gaps** option Off (under the **Appearance tab**), then add another Drillholes branch to the Workspace Tree populated with the entire dataset, but displayed as simple **Lines**.

The **Appearance** tab also contains the **Thickness** range, **Initial angle** and **Discrete samples** controls relating to this thickness modulation functionality.

Note The colours set in the **Thickness** tab will be overwritten if a colour **Legend** has been selected in the **Fields** tab. Setting multiple downhole fields to display using thickness modulation in this instance is not recommended as it will be impossible to determine the individual thickness modulation fields in the 3D display.

Display Drillhole Logs Images

A useful 3D display technique for drillholes is to show a downhole log of measured drillhole data. This can be done by adding a bitmap of a drillhole log that has already been created using Encom Discover into the Discover 3D display, using the **Discover 3D>View Logs** menu option. The Drillhole Log is created as a georeferenced bitmap image (.EGB) and is rendered as an image curtain next to the 3D Drillhole.



Multiple drillhole logs displayed in Discover 3D

Step 1 - Open the drillhole project

Open the Drillhole Project using the Encom **Discover>Drillholes>Project Setup** menu option.

Step 2 - Select drillholes

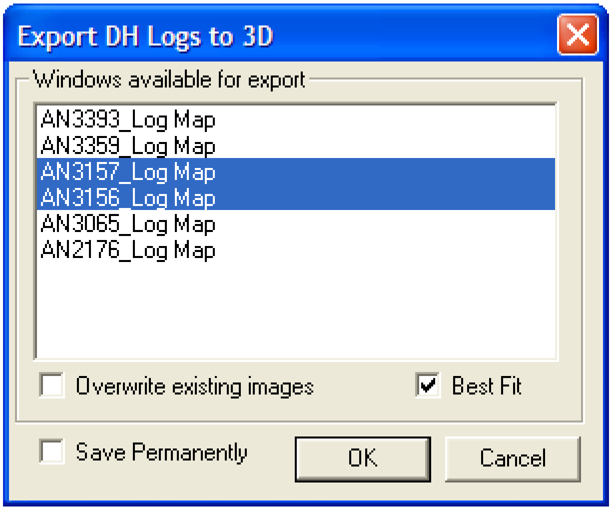
Select the drillhole(s) in the collar map window to create a log display. The hole trace selected must also be visible in Discover 3D before a drill log can be displayed or rendered. Use the **Discover 3D>View Drillhole** menu item in Encom Discover to display the selected drillhole trace(s).

Step 3 - Create drillhole logs

Select the **Discover>Drillholes>Log Display** menu option. Using the dialog, create a log using the settings available. Refer to the *Drillhole Log Display* section of the *Discover Reference Manual* for instructions on defining a log display.

Step 4 - View logs in 3D

When the drillhole log displays open in a map windows in Discover, select the **Discover 3D>View Logs** menu option. This will open the **Export DH Logs to 3D** dialog. Highlight the logs to be displayed in the 3D environment.



View Logs dialog

If drillhole log images are already displayed within Discover 3D, enabling the **Overwrite existing images** option will replace them with these newly selected logs. Otherwise, the newly selected logs will simply be added to the 3D display in addition to those already displayed. Enabling the **Save Permanently** option allows you to specify the output directory for these files (this defaults to the drillhole project directory) and retain these images at the end of the 3D session; otherwise the files will be saved to the assigned temporary directory. Click **OK** to display the selected logs in Discover 3D.

Alternatively, place the cursor in the log display window and right click to select the **View in 3D** menu item. This will add just the selected log display to the 3D window in a cumulative fashion.

Note The zoom level used for the display of the log in Discover determines the resolution of the log in Discover 3D. If the highest resolution is required in Discover 3D, zoom into the view in Discover.

Note If you do not have the selected drillhole trace displayed in Discover 3D before trying to display a drill log, the log is not displayed in Discover 3D.

Note To place log displays directly adjacent to their associated drillholes (without a transparent buffer between them), enable the **Best Fit** option.

Step 5 - Set the display properties

Within Discover 3D you can use the **Image** tab of the **Drillhole Properties** dialog to control the orientation, scale and quality of the view.

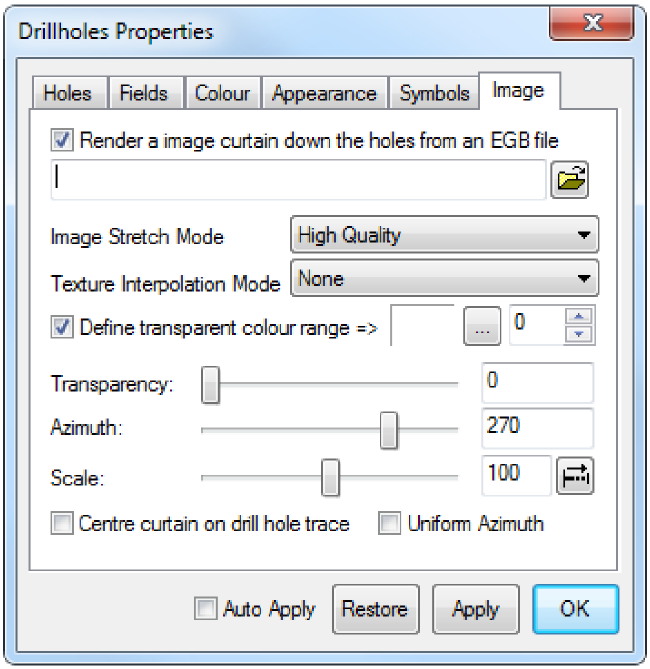
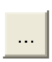
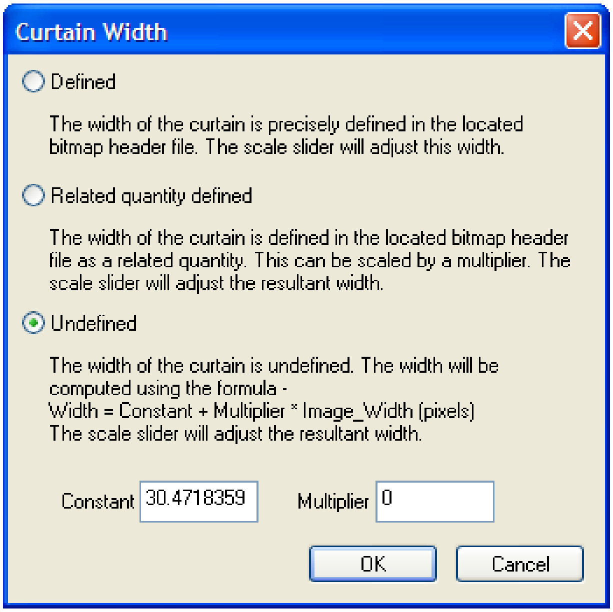


Image tab of the drillhole property dialog



The bitmap created by Discover is automatically listed and the colour of ‘white’ is selectable to be transparent. Other colours can be chosen using the **Browse** button (shown left). The degree of transparency can be controlled with the **Transparency** slider to allow view of items behind the log.

The log images are automatically orientated in Discover 3D according to the average azimuth of the corresponding drillhole. The **Azimuth** slider enables this initial azimuth to be modified by a constant value. Enabling the **Uniform Azimuth** option will force all images to orient along the same specified azimuth as set with the Azimuth slider (0^o and 360^o is north). The horizontal **Scale** can also be adjusted either via the **Scale** slider, or the **Precision Scale** button (shown left).



Curtain width dialog enabling precise control of the image curtain scale

The **Curtain Width** dialog enables the width of the image curtain to be precisely controlled via one of three options:

- As Defined in the located bitmap header file.
- Related Quantity Defined by applying a user-defined Multiplier to the header file definition.
- Undefined - the width is calculated using the formula: Width = Constant + Multiplier * Image_Width, with the highlighted parameters user-defined.

Additional **Image** tab controls include:

- Option to **Centre** the curtain on the drillhole trace, or having its margin lie along the trace.
- Refine the quality of the bitmap .
- Adjust the **Texture Interpolation Mode** of the log. Lower quality texturing provides faster bitmap display and manipulation.

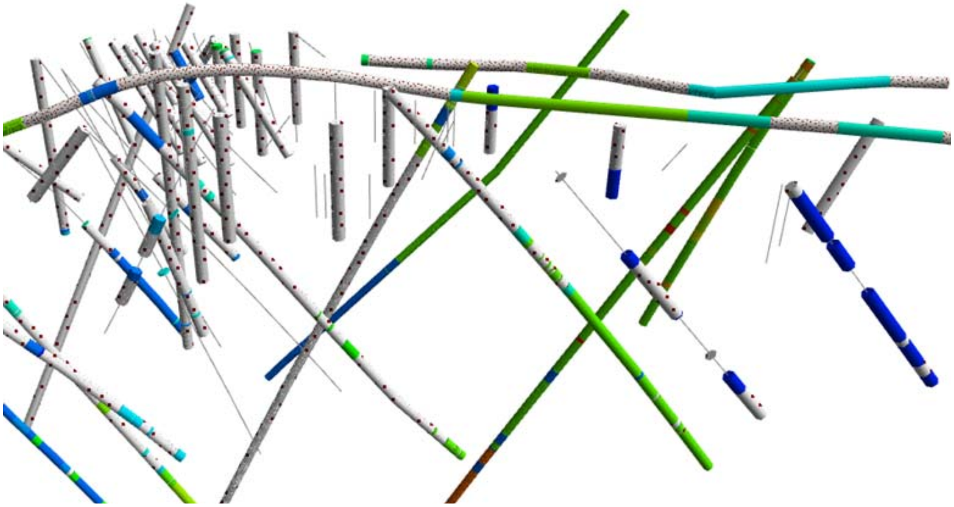
Data Compression

The **Decimation Scheme** option in the **Fields** tab enables the display rendering performance to be improved by reducing the number of samples displayed down the hole. This option is generally only appropriate for larger datasets where geophysical data has been collected at a small scale (e.g. centimetre or decimetre) rather then geological data collected at a larger scale (eg metre).

Three options are available:

- **No decimation** – all samples are utilised i.e. no compression applied (the default option).
- **Fixed rate decimation** – utilises 1 in every *n* samples, where *n* is a user-defined value.
- **Compressed** - applies variable compression using the Tomek algorithm, which variably adjusts to the data via a user-specified compression factor.

Changing Trench Display Properties



Most aspects of Trench displays can be controlled from the Properties dialog opened from the Trenches branch of the Workspace Tree. To display the Trenches Properties dialog, either:

- Move the cursor over the Trench branch in the Workspace tree and double click with the left mouse button, or

- Highlight the Trench branch in the Workspace tree and right-click to display the shortcut menu, and then select **Properties**.

Most settings available for Trench Properties are also found for the Drillholes Properties.

The display settings made on the Trench Properties dialog box can be saved or loaded from a settings file (see Saving and Applying Display Settings). The display controls available depend on the type of data and may include:

- *Selecting Trenches*
- *Trace Style*
- *Colour Modulation*
- *Thickness Modulation*
- *Labelling*
- *Symbols*

Saving and Applying Display Settings

Any display settings customized in the **Trench Properties** tab controls (e.g. colour or thickness modulation) can be saved for later reuse. This is an excellent way to maintain the same display settings between Trench project data updates, or between projects sharing the same data structure.

To save any current display settings, move the cursor over the Trench branch in the Workspace tree and right-click. Select the **Save Properties** option, and specify a name and location for the output display settings .XML file (e.g. a under the Trench project root directory, or in a user-created settings folder elsewhere).

To reapply display settings at a later date (e.g. for an updated version of the same drillhole project, or another project), first load the Trench project into 3D. Move the cursor over the Trench branch in the Workspace tree and right click. Select the **Load Properties** option, and browse for the desired settings .XML.

Note Display settings can only be applied to Trench projects that have exactly the same table names and field names and types. Any differences in table names or structure will likely cause settings to not display.

Selecting Trenches

The **Trench** tab of the **Trench Properties** dialog allows the user to control which trenches are displayed. Specify the trenches to be displayed by moving the trench name(s) from the **Hide** to the **Show** list. Alternatively, you can graphically select the holes to be displayed from a collar map using the **Graphical Selection** button.

Trace Style

The trench trace styles are the same as for Drillholes; see *Trace Style* for more information.

Colour Modulation

The trench trace styles are the same as for Drillholes; see *Colour Modulation* for more information.

Thickness Modulation

The trench trace styles are the same as for Drillholes; see *Thickness Modulation* for more information.

Labelling

Trench start and ends can be annotated with labels by enabling the appropriate tick box in the **Symbols** tab. To change or customise the label display, click the adjacent **Start Label**, **End Label** or **Distance Label** button to open the **Advanced Arrangement** dialog.

The **Start** and **End Label** buttons display the current label in a text box at the top of the dialog; to change this, highlight the entire text entry and click the **Field Selection** button (shown left). Select the desired field from the displayed list (populated from the trench collar file). Multiple fields can be displayed together by placing the cursor at the desired insertion point within the text window (without highlighting any existing text) and then selecting extra fields from the field selection list (without highlighting any existing text) and then selecting extra fields from the filed selection list.

The **Distance Label** button replaces the text box with a **Format** button; this can be used to set numeric formatting, prefixes and suffixes. Activating the **Distance Label** option also requires setting a **Distance Interval** in the **Appearance** tab (i.e how often trench distances are displayed).

For all label types, standard **Font** controls are provided, as well as a range of orientation options in the adjacent pull-down list.

Symbols

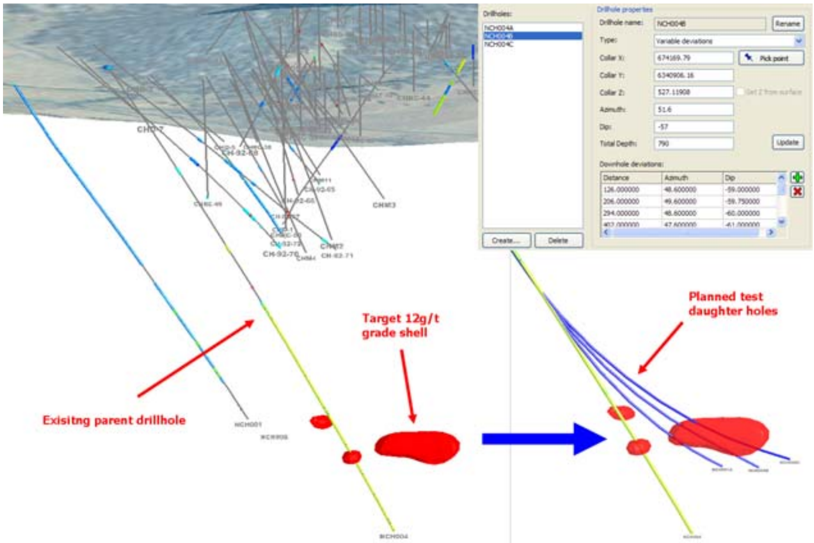
The collars and/or bottoms of trenches can be annotated with a symbol by enabling the **Start Symbol** or **End Symbol** options in the **Symbol** tab. The adjacent buttons open the **Symbol Selector** dialog allowing you to choose from an extensive library of symbols. Choose a symbol from the list to view a preview on the right side of the dialog. The symbol **Size** and **Fill Colour** can also be modified.

Advanced Symbol Style Options are available via the **Advanced** button.

Drillhole Planner

Utilities>Drillhole Planner

Discover 3D provides the powerful capability to dynamically position and plan new drillholes directly within the 3D environment. This allows the user to target 3D objects such as voxel isosurface grade shells, extruded quartz veins, .DXF solids (such as alteration zone volumes generated from digitized section boundaries) or feature object interpretations. Additionally, the drillhole planner can automatically calculate a collar location using a specified target's coordinates, intercept dip and azimuth, and will take into account real-world hole deviations.



The **Drillhole Planner** is accessed from the **Utilities** menu in the 3D interface.

Planning Drillholes in 3D

1. Open all required targeting data into the 3D window (e.g. existing drillhole project, grade shell isosurfaces, mineralization solids, etc).

2. If a topographic/DEM grid is to be used for for the collar location, see the *Collar RL/Z Value Assignment* section:

3. In 3D, open the **Utilities>Drillhole Planner** menu option.

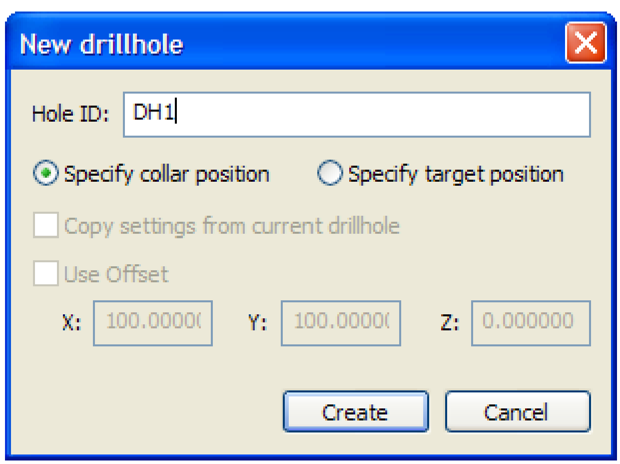
4. On the **Manage Project** tab, use the **New** button to create a new **Drillhole Planner Project** to add the new drillholes to. Specify a project name and location and click **Save**. This will create two files: `projectname_collars.TAB` and `projectname_surveys.TAB`.

5. Set the required **Dip Sense** (i.e. whether down dip is negative or positive) and **Drillhole Units** (default metres).

   The Drillhole Planner can construct drillholes either from a specified collar or to a specified target.

Planning a drillhole from a specified collar location.

1. Use the 3D Navigation controls (see *Navigating in 3D*) to orientate the view to the first proposed collar location and press the **Create** button on the **Plan Holes** tab of the **Drillhole Planner** dialog.



2. The **New Drillhole** dialog will open, with a default Hole ID ‘DH1’ populated. Change this if required and select the **Specify collar position** option. Press **Create**. The new hole will be listed in the Drillhole Planner dialog, with various default attributes.

3. The drillhole collar can be positioned either:

   - **Manually** by entering the Collar X & Y coordinates (and Z value if no grid is being utilised - see *Collar RL/Z Value Assignment*). Press the **Update** button to visualise the planned hole location (a point symbol will be added to a new temporary Feature Database called *DHPlanner_Collars*)

   - **Dynamically** by pressing the **Pick Point** button, and clicking on the desired location in the 3D window. If no gird is being utilized (see *Collar RL/Z Value Assignment*), the collar Z value will be populated using the z value of the cursor plane at the location selected. A point symbol will be added to a new temporary Feature Database called `DHPlanner_Collars`. The hole location can be then manually edited or dynamically repositioned again using the Pick Point button. Press **Update** to preview any manual changes to the location (e.g. subtle RL changes).

4. Set the hole **Type** as either Straight, Constant Deviation or Variable Deviations. *Drillhole Types* are discussed fully below; this example will use the straight type.

5. Type the desired **Collar Azimuth**, **Collar Dip** and **Total Depth (Length)** values. The current view direction bearing and inclination is displayed at the bottom right of the 3D window in the status bar; this can be used as a guide to the required azimuth and dip parameters.

6. Press the **Apply** button at the bottom of the dialog to create the drillhole. A new Drillhole item will be added to the workspace tree, called `projectname_collars,` and the planned drillhole will be displayed.

7. If any aspect of the drillhole is incorrect, modify as necessary on the **Plan Holes** tab (e.g. Total Depth/Azimuth/Dip) and press **Apply** again to recreate the hole.

8. Repeat steps 1 through 8 for each additional drillhole required.

Planning a drillhole using a specified target location.

1. Press the **Create** button on the **Plan Holes** tab of the Drillhole Planner dialog; the New Drillhole dialog will open, with a default Hole ID ‘DH1’ populated. Change this if required and select the **Specify target position** option. Press **Create**. The new hole will be listed in the Drillhole Planner dialog, with various default attributes.

2. The drillhole target can be positioned either:

    - **Manually** by entering the Target X, Y & Z coordinates . (To visualize your target location in 3D, select the Pick Point button, click roughly in the area of interest, then manually edit the X, Y & Z coordinates and press the **Update** button. A point symbol will be added to a new temporary Feature Database called DHPlanner_Collars)

    - **Dynamically** by pressing the **Pick Point** button, and clicking on the desired location in the 3D window. The target Z value will be populated from the cursor plane Z value at that location. A point symbol will be added to a new temporary Feature Database called `DHPlanner_Collars`. The target location can be then manually edited (press Update to alter the corresponding 3D location) or dynamically repositioned again using the Pick Point button.

3. Select the hole **Type** as either **Straight** or **Constant Deviations**. Drillhole Types are discussed fully below; this example will use the straight type.

4. Enter the desired **Target Intercept Azimuth** and **Target Intercept Dip** angles. These represent the orientation of the hole when it intercepts the target location; this is particularly useful when trying to intercept planar features at a right angle (e.g. quartz vein).

Note The current view direction bearing and inclination is displayed at the bottom right of the 3D window in the status bar; this can be used as a guide to the required azimuth and dip parameters.

---

5. To ensure the hole adequately penetrates the target body, enter an appropriate **Extension Depth (Length)**; this is the distance the hole will continue past the target location. A value of 0 will terminate the hole at the target coordinates.

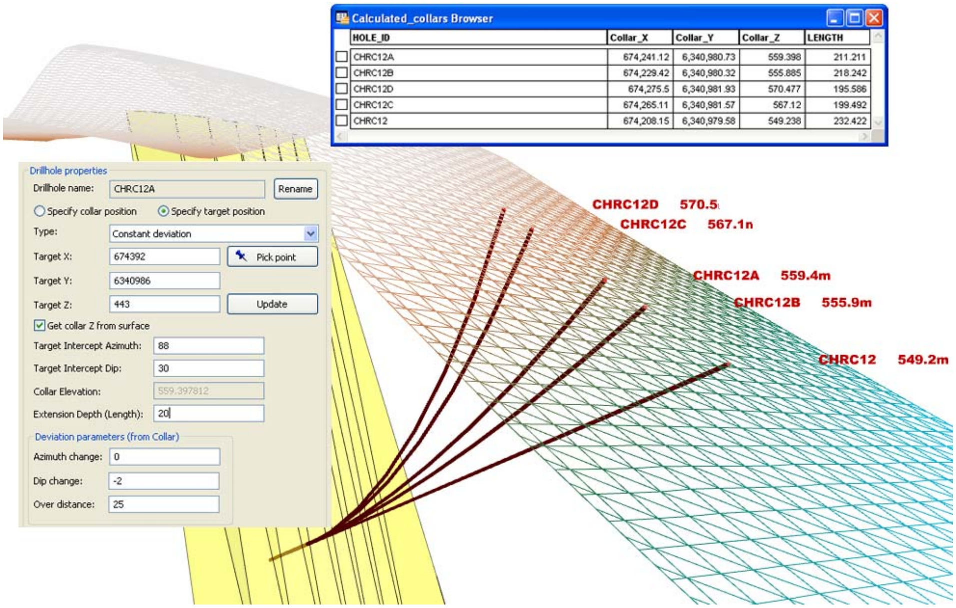
6. The drillhole planner will construct a drillhole upwards from the target location; it therefore requires a collar RL to finish the hole construction at. The recommended option is to specify a topographic grid surface (e.g. such as a digital elevation mode (DEM) created or imported with Discover's Surfaces module) by enabling the **Use surface for collars** option at the top of the dialog, and browsing for the grid file. If no surface grid is specified for the collars, you will need to manually enter a **Collar Elevation**. More information on these options can be found under *Collar RL/Z Value Assignment*.

7. Press the **Apply** button at the bottom of the dialog; the drillhole planner will calculate (using all hole parameters specified) a hole upwards from the target location to either the specified topographic surface or collar elevation. A new Drillhole item will be added to the workspace tree, called `projectname_collars` (i.e. the project name specified in Step 4 above), and the calculated drillhole will be displayed.

8. If any aspect of the drillhole is incorrect, modify as necessary in the Drillhole Planner (e.g. Azimuth/Dip parameters) and press **Apply** again to recreate the hole.

9. Repeat steps 1 through 8 for each additional drillhole required.

Note If you are evaluating possible collar locations for different azimuth/dip parameters on the same target, simply create a series of potential holes (e.g. 12A, 12B, 12C 12D) using the same target location information. Select the first hole containing the target location information in the Drillhole Planner, press **Create** and enable the **Copy settings from current drillhole** option in the New Drillhole dialog. This will create a new hole duplicating the selected holes target parameters; modify the azimuth/dip parameters.



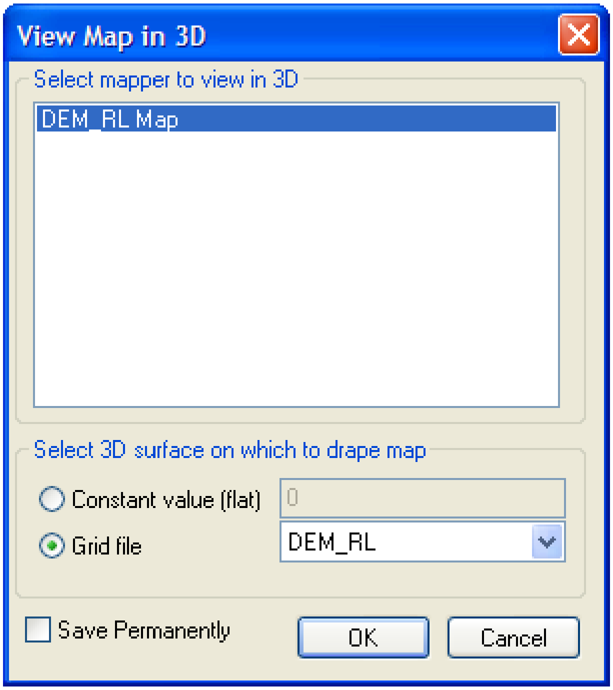
Experimenting with different hole orientations and their resulting collar locations in order to intercept a fault plane target location.

Collar RL/Z Value Assignment

Collar Z/RL values can be assigned to your planned collar locations in four ways:

1. By utilising a topographic/DEM grid imported as an image. When the **Pick Point** button on the **Plan Holes** tab is used, the drillhole location is snapped to the image and therefore uses the images Z value (i.e. the Z value of the grid the image is draped over). To utilise a surface grid in this way -

    - In 2D, open the topographic grid into a MapInfo map window, and use the **View Map in 3D** menu option to capture a view of the grid draped over itself.



- In 3D, alter the transparency of this image to between 40% and 60%.
- Make the image **Selectable** in the Workspace Tree.
- Click the **Bond to Image** button on the Cursor Plane toolbar. The cursor plane is now bonded to the DEM image.

2. Manually by entering the required value in the Collar Z or Collar Elevation field on the **Plan Holes** tab. This also allows an existing Z value (e.g. one captured using a grid in first method above) to be modified

3. By specifying an alternative grid from within the **Manage Project** tab of the Drillhole Planner dialog via enabling the **Use surface for collars** option. This allows grids that have not been imported into MapInfo Professional/Discover to be utilised for the Z value. When using the Pick Point button, Discover 3D will first determine the XY coordinates of the selected point on the horizontal cursor plane, and then determine the Z value for the selected grid for these coordinates. Therefore the most accurate way of dynamically capturing a point location for this method is to use the Down view button on the **View Direction** toolbar.

4. Using **Pick Point** button on the **Plan Holes** tab to select a location on the cursor plane and capture this points RL value (as displayed in the bottom middle of the 3D window). See *Cursor Plane* for more information on controlling the cursor plane position.

Drillhole Types

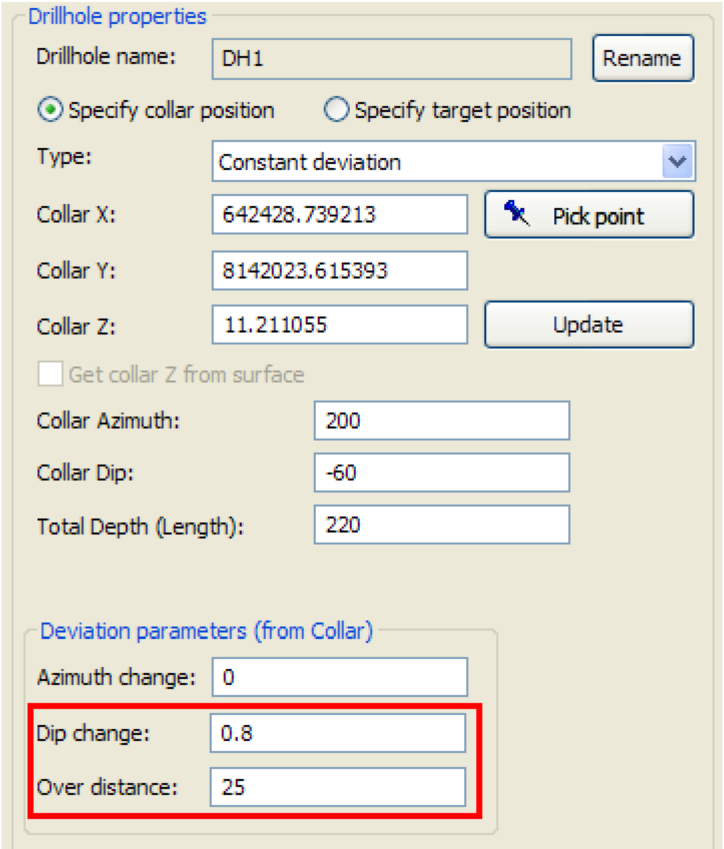
The Drillhole Planner allows 3 types of drillholes to be created. All types require a collar or target Azimuth and Dip value to be specified, as well as the Total Depth of the drillhole.

Straight

A simple straight-line drillhole from the collar location, requiring only collar azimuth and dip values to be specified, as well as the total depth/length of the hole. Examples include boreholes, vacuum drilling and possibly RAB (Rotary Air Blast) reconnaissance drilling.

Constant Deviation

This allows a planned hole to accommodate a constant **dip** and/or **azimuth change** per specified **over distance** (over the entire length of the drillhole). For example, an expected shallowing/rising hole might have a dip change of 0.8 degrees every 25m set (based on previous drilling in the area).



A constant deviation hole set to rise 0.8 degrees every 25m.

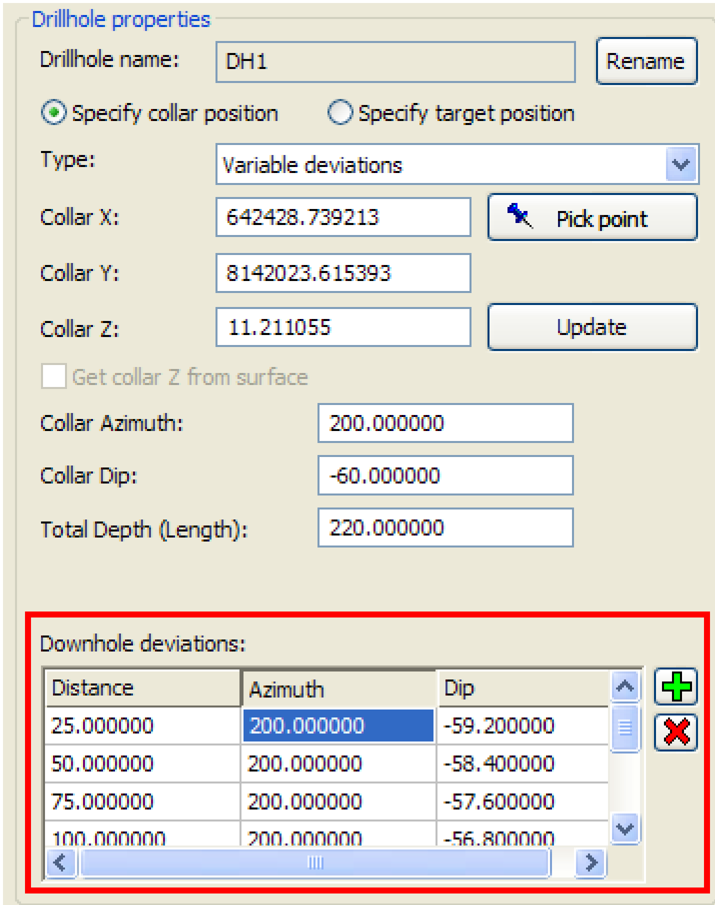
Note Changes are added to the existing azimuth/dip value. For instance if the collar dip is -60, and a dip change of +1.1 degrees is set every 25m, at 50m downhole depth the dip will be -47.8.

Variable Deviations

Individual downhole azimuths and dips can be set at specific depths for the planned drillhole. This allows a drillhole to mirror an existing drillholes downhole variations.

Note This option is currently not available when when specifying a target XYZ location

Use the **Add row** button to add each required downhole survey entry, and enter the appropriate Distance, Azimuth and Dip values.



A variable deviation hole with a series of downhole variations entered.

Targeting Example

The following images illustrate a simple but powerful use of the Drillhole Planner. They present a common situation where the geologist is using an existing drill pad to target mineralization at depth. In this case the target is a 5g/t gold grade shell generated by exporting an isosurface from a drillhole gold distribution voxel model (created by 3D gridding of existing drilling).

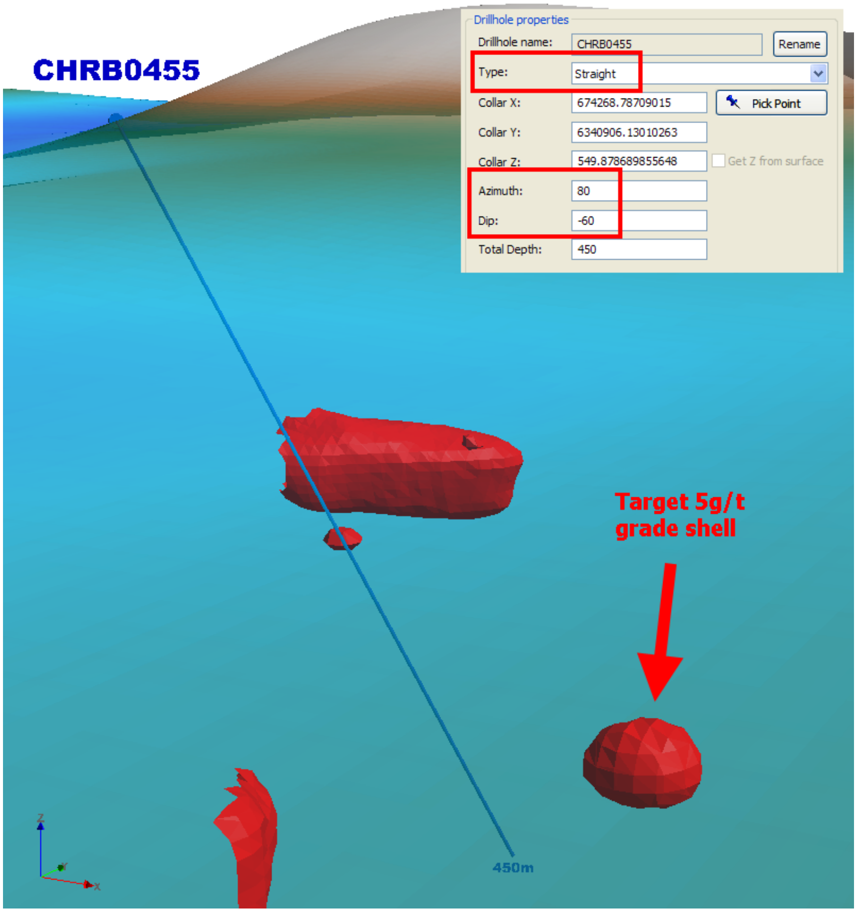


Image 1: Initial hole generation as a straight hole, using the standard collar dip of -60 degrees orientated towards the target.

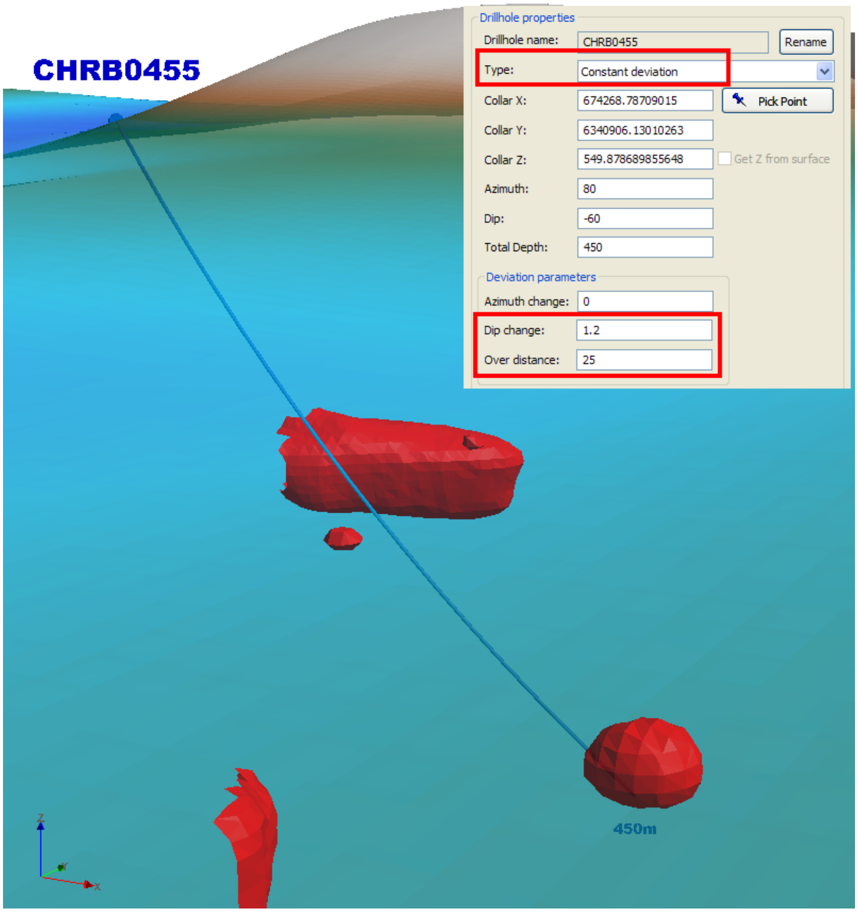


Image 2: Using existing drillhole variations as a guide, the proposed hole is changed to a Constant Variation type, and set to rise 1.2 degrees every 25m. The hole now intersects the target, but barely.

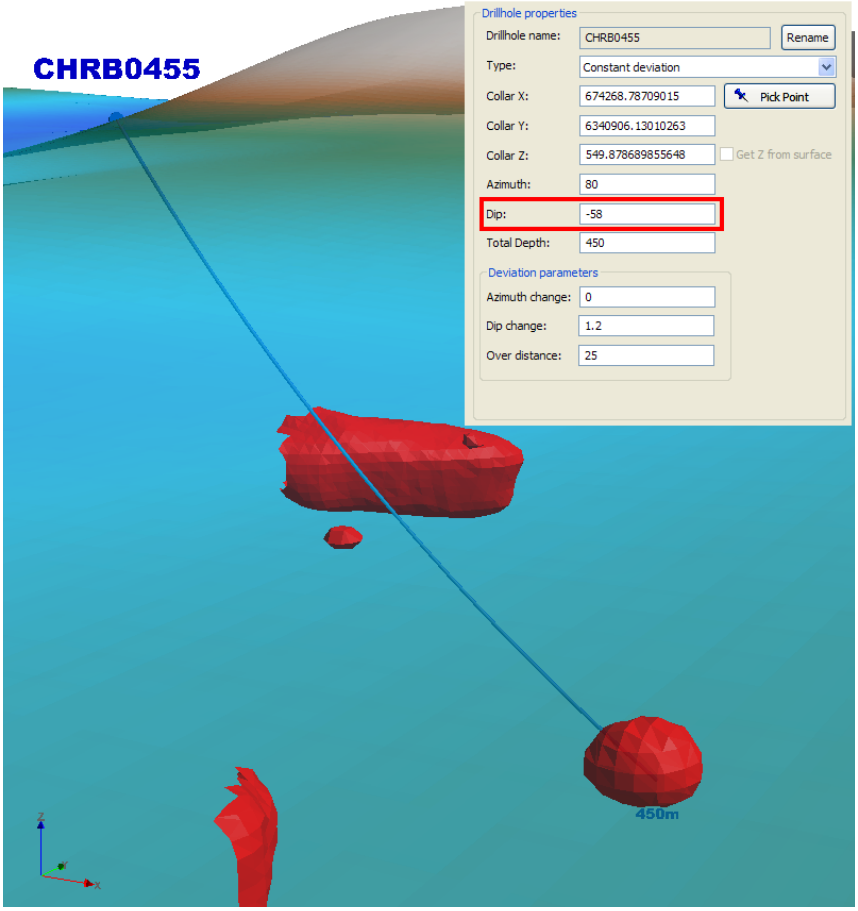


Image 3: The collar dip angle is now refined to maximise target intersection, eventually settling on a -58 degree dip.

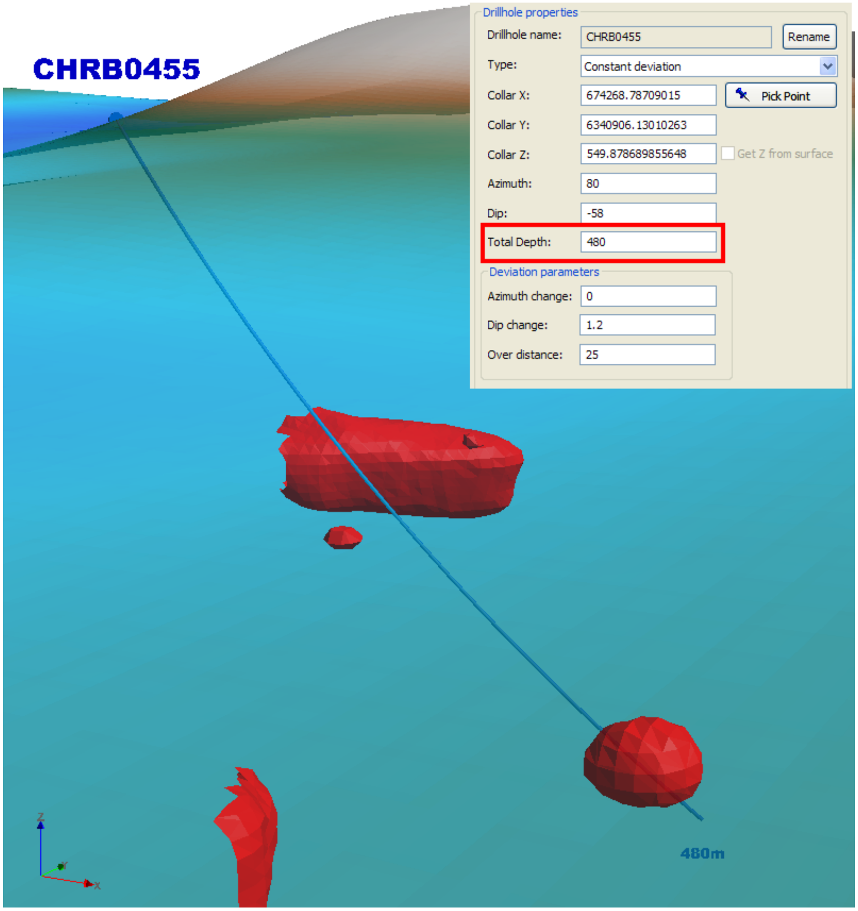
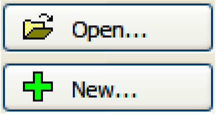


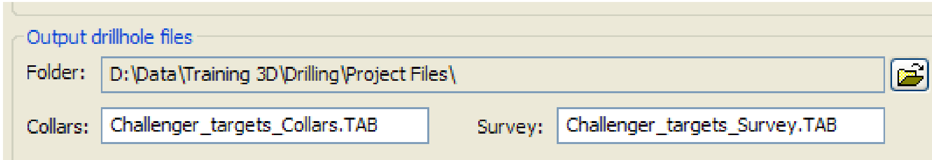
Image 4: The proposed hole depth is extended to 480m to ensure the target is fully tested.

Drillhole Planner Projects



The Drillhole Planner tool creates Drillhole Planner Project (.DPP) files to dynamically handle drillholes. These can be created and reopened using the **New** and **Open** buttons on the **Manage Project** tab.

After a hole's attributes have been specified on the **Plan Holes** tab, pressing the Apply button uses the .DPP file to generate the necessary MapInfo collar and survey tables; these can then be used to visualise these holes both Discover and Discover 3D. The output .TAB names and location are displayed on the **Manage Project** tab:



A drillhole planner project can also be created from an existing Discover Drillhole Project. This allows a number of existing real-world holes to be utilised as templates for proposed holes i.e. to mirror an existing drillholes downhole azimuth and dip variations for a hole planned in a proximal location. It can also be used to design daughter or wedge holes off an existing drillhole.

To use existing drillholes as templates:

1. On the **Manage Project** tab, click the **Import** button and browse for the source collar and survey files (these must be part of an existing Discover drillhole project).

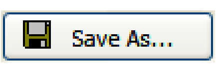
2. A drillhole **Selection** dialog will then be displayed- select only the required drillholes to be used as templates.

3. Specify a new .DPP file to store these templates in.

4. The selected holes will be displayed on the **Plan Holes** tab. It is recommended to use the **Rename** button to rename these holes (to save confusion with the original holes (e.g. rename CHRC112 to CHRC112A).

5. If designing multiple daughter or wedge holes off a parent hole, select the parent hole in the Drillholes list, and press **Create**. Assign a new hole name (e.g. CHRC035B) and enable the **Copy settings from current drillhole** option – this will copy the parent holes deviation data to the new hole. Disable the **Use Offset** option. Press **Create**.

6. Alter the holes as necessary and visualise.



An open drillhole planner project can be renamed and saved as a new project on the **Manage Project** tab.

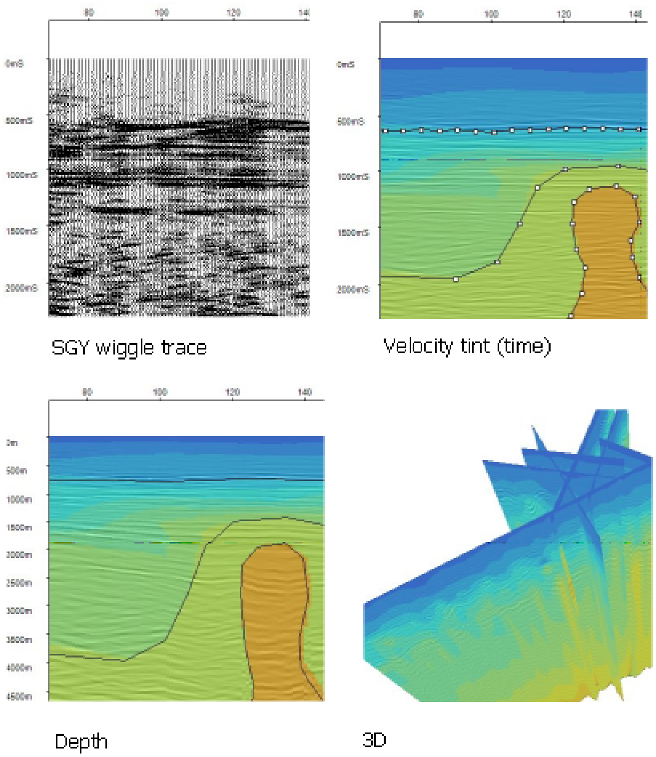
Note You should only copy settings from Target specified holes to Target specified holes and Collar specified holes to Collar specified holes.

---

15 Viewing Seismic Sections

Discover3D>View Seismic Sections

The Seismic Sections tool allow you to create projects by directly opening SGY files (also known as SEG-Y), along with support for various navigation or velocity models files. Sections can be viewed as either two way travel time or a depth axis. You can then manually or auto-pick horizons and define velocities for each facies or region within the horizons. Sections can be exported as 3D geo-located vectors and images or a 2D Plan map of the survey lines.



Seismic Workflow

When seismic data is received, in the form of SEG-Y (.SGY, .SEG or .SEGY files), there is a general workflow to generate Discover 2D plan maps or Discover3D depth image of this data.

The key steps are:

A. Create and configure a new seismic project and add SEG-Y files:

B. Assign navigation and coordinate information

C. Visualise section data

D. Digitize horizons on a section

E. Assign velocity values to a section.

F. Generate 2D plan of the track lines

G. Generate 3D output for the depth sections

A. Create and configure a new seismic project and add SEG-Y files:

1. Select **Discover3D>View Seismic Sections**
2. Start a new project with **File>New Project**, or open an existing project with the **File>Open Project**. When Seismic Sections starts up it will automatically load the last project worked on.
3. Set a project name (SPR file) in the project directory, under which all the SGY files are located.
4. Tick the **Find and add all SGY files** for auto import of SGY files already in the project directory.
5. Set the **Line Name Pattern** to assist in file name matching. For example if your lines are ordered -

   WB-401, WB-402, WB-403 etc

6. Then set the line pattern to be

   AA-NNN where A=alpha and N=numeric and *=skip(wildcard)

7. Any SGY files with suffixes after the Line name will be included in the line's metadata, for example -

   `WB-401.sgy, WB-401_part2.sgy, WB-401_part3.sgy`

8. Will all be reduced to the line number WB-401 and used to match other files like `WB-401.west` and extract coordinates from a Nav text file that has columns containing line number (WB-401) and trace (shot) number.

Note The SGY file must begin with the line number. If you have files like “lineWB-401.sgy” then remove the prefix, i.e. ****AA-NNNN

9. If WEST files are present, tick the **Find West file per SGY** so that .west files matching SGY files will be imported. The scanning of files by default will look for both `<linenumber>.west` and `<SGYFile>.west` files (i.e. the asterisk implies both <linenumber> and <SGYFile> names will be attempted).

10. You can use one Navigation (text) file for every SGY file in the project, or you can let the dialog scan for a file per SGY file for the available navigation types. See *B. Assign navigation and coordinate information* for more details.

11. The scanning of per-SGY navigation files by default will look for both `<linenumber>.west` and `<SGYFile>.west` files (i.e. the asterisk implies both <linenumber> and <SGYFile> names will be attempted).

12. Select **Projection Override** and define projection to use for all sections in the project.

13. Select **Test Line Name to Nav matching**, to ensure all SGY files and lines have valid navigation information.

Note If new SGY or metadata files are added to the project, the **File>Project Settings** can be re-run at any time to create a new project, which will allow batch importing and editing of Section Properties.

---

14. If new data is received, you can use **File>Open Section** or **Section>Add** to add SEG-Y files to your existing project (select multiple by holding the Ctrl key).

B. Assign navigation and coordinate information

1. After a project is created you can review and edit the Coordinate information for individual Sections by selecting **Section>Properties**.

Note If you wish to batch edit section coordinates settings, new the **File>Project Settings** can be re-run at any time to create a new project, which will allow batch importing and editing of Section Properties.

---

2. The Coordinates formats supported per Section file are:

   - SGY: The SGY file contains typically long integers at trace header bytes 73,77
   - West: A supplied .west file

- Nav: A column delimited text file
- UKOOA: A UKO format file
- Ends: Manually enter the coordinates of 2 traces in the section.

3. For all coordinates types select the **Config**… button to configure the format of the source regarding the decoding of values (decimal, DMS, Easting/Northing). The following formats of coordinate data can be decoded

   - Easting / Northing
   - Longitude / Latitude from decimal
   - Longitude / Latitude from [D]DDMMSS
   - Longitude / Latitude from [D]DDMMSSF (fraction tenths)
   - Longitude / Latitude from [D]DDMMSS.FFA A=N/S or E/W

4. For Text Files (Nav) select **Config** to specify the header size as **Skip Lines**. Also specify any comments by the **Comment** character. Select the applicable **Delimited** characters or **Fixed width** sizes between the columns (separate the sizes with commas).

5. In the **Header** text box, you will need to provide names for each column/ field in the file, separated by a comma. The **Header** column names must include following column names:

   - LINE
     The line number column (also known as Track)
   - TRACE
     The trace number column (also known as Shot, for example in un-stacked data)
   - X
     The source of Longitude or Easting
   - Y
     The source of Latitude or Northing
   - SKIP
     All other columns should be labelled as skip

Note It is strongly recommond to define a Project Navigation under **File>Project Settings>Nav**

6. If the SGY data file does not start at the first shot (trace) recorded in the navigation, define the shot that it does start at in the **First Trace** box.

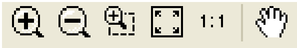
C. Visualise section data

1. Select the section file to view from the list on the toolbar.

2. The current view style can be changed by selecting from **View>Wiggle Trace** (time axis), **View>Time** or **View>Depth**.

3. The velocity value can be visualised by selecting **View>Velocity tint.**

4. Zoom and pan the view by selecting the various controls from the toolbar.



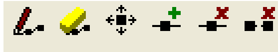
5. The view can also be scrolled using the horizontal scrollbar located below the section.

6. A dynamic profile of individual traces can be viewed by select **View>Profile**.

D. Digitize horizons on a section

1. With the auto-pick tool, simply click on the event (pick) in any trace in the section, and the horizon will be automatically generated for this event across the section.

2. Alternatively you can create the polylines (horizons) manually by creating a series picks (points). Make sure you draw past the edges of the data or cut though another horizon so the horizons define enclosed velocity regions. You can cut lines, and you can join lines by moving the end point of one line onto the end point of another line.



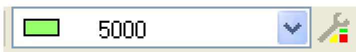
3. You can undo any changes by selecting **Edit>Undo** or **Edit>Redo**.

4. Label your horizons and change their line style by double-clicking on a horizon line.

E. Assign velocity values to a section

1. By default a section will display with a single default velocity as set under the project settings. Alternatively if a .WEST file is available for the section, this information will be displayed.

2. Define the available velocities for the project by selecting the edit velocities button on the toolbar. Velocities can be given a name, as well as the speed and colour to display.



3. To assign a velocity to an area in the section, enclosed by horizons and/or the edge of the section, select the velocity from the list and use the colour fill tool to click in the area.

F. Generate 2D plan of the track lines

1. Select **View>Plan View** to open a plan view of the sections navigation/ coordinate data.

2. Select the **Open** tick box to auto-open the output TAB file in MapInfo Professional.

3. Click **Make TAB file** to generate a MapInfo vector file for the seismic lines.

4. After the TAB file is created, the Section SEG-Y files can be reopened form within MapInfo Professional and Discover, by selecting the SEG-Y line, then select **Discover>Table Utilities>Open Linked Documents**. The associated section will be automatically opened for viewing.

G. Generate 3D output for the depth sections

1. Select **Section>Generate 3D**

2. To generate an image for the currently viewed **SGY** select **Current**. To generate image layers for all sections in the project select **All**.

3. Select the output style as **Depth (Greyscale)**.

4. (Optional) For the output style, select **Velocity Tint** .The tint transparency will be the same as defined under **File>Project Settings>Appearance**.

5. Select an output directory for the files. By default, it will in the project directory in a new "out" folder. The files will be automatically named by the seciton or the project.

6. Select an output type: Horizons lines, Velocity regions or Images of the section. The digitized horizons and velocity regions can be output as either DXF vectors or an editable FDB file.

7. Select **Generate 3D** to create the image files.

Seismic Project Files

A project (.spr file) can contain many seismic lines (.sgy files), as well as associated files for coordinate information and velocity models.

In addition the following files may be present:

- A source of velocity for time to depth conversion (.west file or manual velocity creation in .vel).
- A source of navigation coordinates (in SEGY, .nav, UKOOA or manual input)
- A file to store any digitized horizons, .bdy.
- A file to save the current view, .spv.

If your section file has a matching .west velocity file it will be loaded and the velocity tint displayed by default. You can change a section’s velocity and coordinates source using **Section>Properties**. If there are no velocity files you can create horizon boundaries and colour fill between the boundaries with a velocity value.

File format details of .west

West navigation files do not strictly follow a standard format. In standard files the trace index is read from columns 5-15 (in bold):

SPNT	**72**	100 0 0
VELF	**72**	100 5000 600 5900 1400 …
VELF	**72**	3000 9800 400010800 580011800

In non-standard files the trace index is read from columns shown in bold:

CDFN	**40**
VELF	0 1891 212 2054 455 2289 …
VELF	1864 4211 2992 5029

For these cases with VELF data, it is consistently located as a maximum of 5 pairs (5 chars per value) from column 21

VELF	216	**100 5000 500 6100 800 6225 1100 6525** …
VELF		**0 1921 208 2116 476 2535 627 3209** …

If neither of these are present, then often SPNT X and Y data is present instead of VELF. SPNT is inconsistent and so is scanned in from column 29 (in bold):

SPNT		119.00	**631317**	**8057146**
SPNT	72	100	**0**	**0**

File Format details of layers

The layers are stored in the .SPR XML project file.

File Format details of .vel

The .vel file is a run length encoded grid of velocity pairs following a 16 byte header (Intel LSB). The 16 byte header consists of 8 zero bytes then the width and height in 2 four byte unsigned int. The values of width and height will be much less than the number of traces and samples. The data follows the header as pairs of count (four byte integer) and value (four byte float). The data runs row by row downwards.

Coordinates

The source of coordinates for a SGY line is determined by the following sources:

- SGY: The SGY file contains typically long integers at trace header bytes 73,77

- West: A supplied .west file
- Nav: A column delimited text file
- UKOOA: A UKO format file
- Ends: Manually enter the coordinates of 2 traces in the section.

For all coordinates types select the **Config** button to configure the format of the source regarding the decoding of values (decimal, DMS, Easting/Northing). The following formats of coordinate data can be decoded

- Easting / Northing
- Longitude / Latitude from decimal
- Longitude / Latitude from [D]DDMMSS
- Longitude / Latitude from [D]DDMMSSF (fraction tenths)
- Longitude / Latitude from [D]DDMMSS.FFA A=N/S or E/W

For Text Files (Nav) select the **Config…** to specify the header size as **Lines to Skip**. Also specify any comment lines by the **Comment** character.

Select the applicable **Delimited** characters (**note not consecutive**) or **fixed width** sizes between the columns (separate the sizes with commas).

In the **Header** text box, you will need to provide names for each column/field in the file, separated by a comma. The **Header** column names must include following column names:

- LINE
  The line number column (also known as Track)
- TRACE
  The trace number column (also known as Shot, for example in un-stacked data)
- X
  The source of Longitude or Easting
- Y
  The source of Latitude or Northing
- SKIP
  All other columns should be labelled SKIP

Managing Views and Appearance

In the Seismic sections tool, the **View** menu provides the following options for displaying SGY data:

- **Wiggle Trace** – Wiggle trace profiles with two way time axis
- **Time (greyscale)** – view a rasterized image of the SGY data with black representing high peaks and white representing low peaks.
- **Time (colour)** – view a rasterized image of the SGY data, low and high peak colours are set under File>Project Settings>Appearance>Trace Colour.
- **Depth (greyscale)** – view rasterized SGY data depth corrected based on assigned velocities, with black representing high peaks and white representing low peaks.
- **Depth (colour)** – view rasterized SGY data depth corrected based on assigned velocities, low and high peak colours are set under File>Project Settings>Appearance>Trace Colour.
- **Plan** – view the project's section lines in a plan map, and export to MapInfo Professional.

For each view, the High Peak and Low peaks can be reversed, by toggling **View>Invert**.

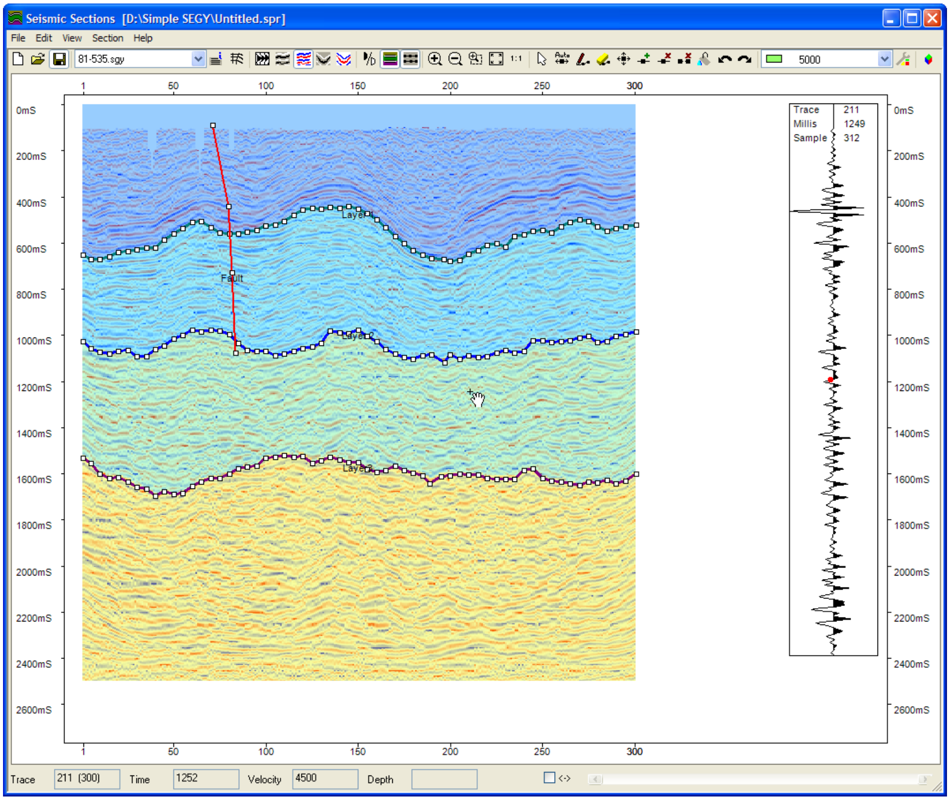
For each view type, a velocity "tint" background colour can be applied by selecting **View>Velocity Tint**. The Transparency of this layer can be adjusted under **File>Project Settings>Appearance>Velocity Opacity**.

To view the SGY data in close detail, a trace profile can also be displayed (see *Viewing Profiles*).

To enhance the display of the seismic data, increase the contrast or AGC (automatic gain control) under **File>Project Settings>Appearance**. This will aid in interpreting weaker or noisy data.

Digitizing Horizons

You can create horizons by drawing and editing polylines with the editing tools. You can draw the lines manually and colour fill between them with a velocity value/color. Make sure horizons are drawn past the edges of the data or cut through another horizon so that they define an enclosed region. You can cut lines, and you can join lines by moving the end node (pick) of one line onto the end node of another line. In auto pick mode you can click on the section and have a horizon generated.



Note You can digitize faults by simply using the Pen tool to manually draw a sub-vertical line. Then horizons can terminate where they intersect the fault line.

- **Autopick**: Click on a trace sample to autopick a horizon. Clean up autopick's attempt by using the following tools to edit, remove, cut and join lines. The sensitivity of Autopick can be adjusted under **File>Project Settings>Auto Pick**.

- **Pen**: Click to start drawing a horizon (polyline). Each click adds a pick to the section appending to the current horizon that was last clicked. Right click to end the current polyline.
    - Starting on the end of an existing horizon appends to it.
    - Ending on the end of an existing horizon joins to it.
- **Erase**: Click on an existing horizon to erase the entire polyline, or click anywhere to reset to default velocity.
- **Move Pick**: Move a pick. Moving the end of a horizon onto the end of another horizon joins them as one continuous horizon.
- **Insert Pick**: Click on a horizon line section to insert a new point within the existing line.
- **Remove Pick**: Click on a point to remove.
- **Line Break**: Click on a horizon's line section to break the horizon into two.
- **Fill**: Click between boundaries to colour fill the region with a velocity (as currently selected in the toolbar velocity combobox).
- **Edit>Undo**: Undo any edit operation above.
- **Edit>Redo**: Redo any edit operation above.

Before digitizing, it is recommended to setup your velocities and colours under **File>Project Settings>Velocities**. Each Velocity can be given a text name, colour and velocity value (m/s). These are saved in the project file.

Managing Horizons

After digitizing your horizons, change the line style and labels by hovering the cursor on a horizon and double left clicking.

Alternatively, review the entire set of horizons in a project by selecting **Edit>Horizon List**. You can change the labels, description and style here as well.

If the location of a Section changes , such as updated navigation data, the horizons may become mismatched. To rectify this, they can be shifted by selecting **Section>Properties**, and select the **Edit** button. Alternatively, you can remove all horizons from the section using **Section>Remove all horizons**.

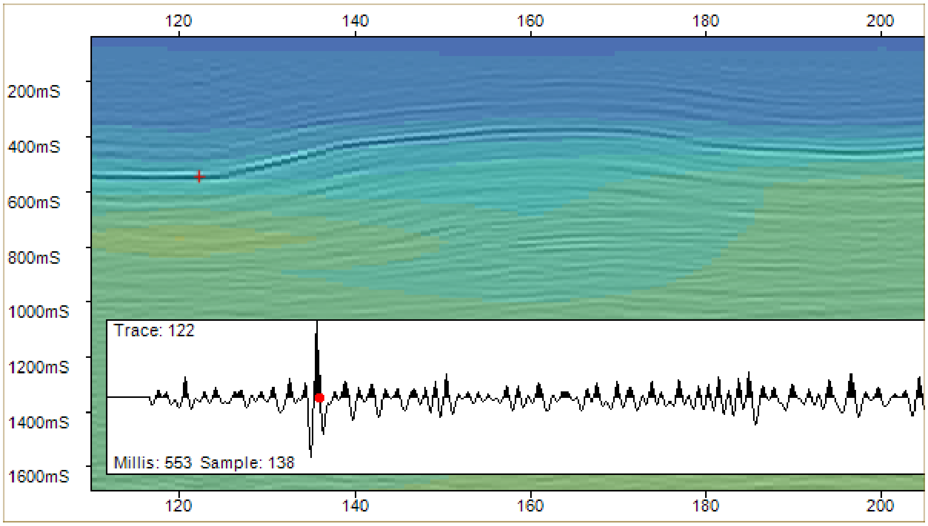
Viewing Profiles

Use the **View>Profile Vertical** or **View>Profile Horizontal** to display a dynamic wiggle trace profile, when viewing the section in time mode.

The profile will automatically change the trace (shot) profile as you move the cursor horizontally across the section. The current cursor position is represented by a red dot on the trace.

It displays the Trace (shot) number from the X axis and the two-way-time from the Y axis. It also displays the sample (hydrophone/band/station) number.

Note The profile is always displayed at 1:1 samples to pixel ratio. Increase the window size to view the entire trace.



Dialog Help

- *Project Settings Dialog Box*
- *Section Properties Dialog Box*
- *Digitizing Horizons*

Project Settings Dialog Box

The **File>Project Settings** dialog contains tab to control setting that apply across all lines in the project. The settings are saved in the .SPR xml file.

Appearance

- **Trace Width**: The width of space in between each wiggle trace, at 1:1 zoom (when one SEGY sample (receiver) is one pixel).
- **Wiggle Scale**: The width of the maximum amplitude wiggle trace at 1:1 zoom.
- **Velocity Opacity**: The transparency of the velocity tint over over the data. 100% produces a solid (opaque) colour over the seismic data.
- **Default Velocity**: The default velocity when no velocity is supplied and no manual edits have been made.
- **Depth X**: Is the maximum height of the depth display where:

(Max time) * (default velocity) * 0.5 (two-way time) * 1.5 (Depth X) = 1200m

Nav

- **Line Pattern**: Define the alphanumeric pattern for Lines based on SGY file names:

  A = alphabetical character

  N = numerical character

  * = skip character

  - = hyphen

  All other characters will be ignored. Spaces are not allowed.
- **Project Navigation**: Define a single delimited text file that defines navigation for all lines in a project.
- **Projection Override**: Define a fixed projection for all lines in the project.

Layer Table

The layer table is a place to make a layer cake of velocities that will be used by SEGY lines that don’t yet have an associated velocity file (.west) or (.vel) from user edits. The layer cake is a good starting point for providing velocities before the user starts full editing. The layer cake is better than the single default velocity. Specify the thickness of each layer in two way time, and assign any velocity value. The depth of each layer is automatically calculated.

These are stored in the project's .SPR XML file.

Velocities

The velocity table holds the discrete velocities and their associated colors that the project can use to render velocity tints. All table cells are editable by clicking. The name cell can be edited to a geological name like “Granite”. Use the generate function to create a range of colors suitable to the geology of the project’s dataset. All velocities are interval velocities.

These are stored in the project's .spr XML file.

Auto Pick

The auto pick settings are used to control the sensitivity of the autopick function.

- **Detect**: Snap the user’s click to the maxima or minima (note you can also invert the data's phase by selecting **View>Invert**)
- **Window**: The number of samples used to correlate one trace with its neighbors.
- **Likeness**: The sensitivity of the neighbor correlation, on a range of 0 (off) to 100 (exactly identical). If set to 0 the autopick will continue to the end of the section.
- **Look Ahead**: The maximum number of bad traces that the autopick can jump over to continue.
- **Trace Span**: The ideal spacing between picks on the horizon.

Section Properties Dialog Box

The Section Properties dialog controls how velocity and navigation data is sourced for a SEGY file. The velocity is needed for viewing time data with velocity tint and for time to depth conversion. Coordinate data is required for the generation of 3D view data.

Time

- **SGY File**: Name of the seismic data file
- **Line Name**: Override the line name automatically determined from the file name, based on File>Project Settings>Nav>Line Pattern
- **First Trace**: Used to adjust the position of the SGY segment file along the shot line.
- **Info** (button): Display header of SGY, containing useful metadata.

Velocity

- **West**: The velocity is provided by a .west file. Specify whether the source contains Average (from surface to the bottom horizon) or Interval (within the current horizons) velocities.
- **Layers**: Use the project's layer cake for velocity.
- **Editable**: The velocity is editable and stored in a .vel file. This information can be changed by digitizing horizons and assigning different velocities, see *Digitizing Horizons*. When switching to Editable mode the user may choose how the editable data is initialized. It can be initialized from -
    - Default Velocity - the single default velocity for **File>Project Settings>Project**
    - Copy from Layer Table - the Layers defined for **File>Project Settings>Layers**
    - Copy from West - the West file information defined under the specified West option
    - Load Edit File - the current .vel file

Horizon

- **Boundary**: Edit the collection of horizons stored in the section Boundary file.
- **Coordinates**: Use either the Project navigation file, or define navigation data for the section. See {Coordinates} for more information.

Plan View Dialog Box

The plan view shows the cruise/track plan of the SEGY lines and you can make a TAB file of the plan for viewing in MapInfo Professional. You will not see any plan if your SGY data is not yet associated with navigation data. Use the table at the top of the dialog to select sections and edit them to associate navigation data.

- **View coordinates as text** to help verify that coordinates are being decoded correctly from Navigation data.
- **Make Tab file** button will generate a TAB file of the plan for viewing in MapInfo Professional. It will be given the Project name and saved in the project folder.

Generate 3D Dialog Box

Use the toolbar button or select Section>Generate 3D to generate 3D vector, feature and image output of the sections, picks and velocity.

- **Current**: Generate output for the currently viewed section.
- **All**: Generate output for all sections in the project.
- **Style**: Select the display style for the output which corresponds to the section display Seismic Sections. This will affect the Z axis for every output (time or depth), and the colouring of the **Section Images** output.

  In addition to the style, you can also apply the velocity tint layer (using the same transparency as in the main window).
- **Projection**: Override the project's navigation/coordinate projection. This is useful for lat/lon data, as it is recommended to use only projected data in Discover3D.
- **Section Navigation**:
  - **Strict**: The navigation file must provide coordinates for exactly the first and last trace of the section.
  - **Truncate**: Remove traces from the start and end of the section before the first navigation coordinates and after the last navigation coordinate.
  - **Extrapolate**: Automatically generate coordinates for the first and last trace, based on the bearing of the first/last segment in the navigation file.

- **Curtain Segmented every**: Define the gap width between inflexion points in the Line. This defines the resolution for the bends of the seismic lines. For long or straight limes, this can be increased.
- **Image**: Define the resolution for the **Section Image** file.
- **Output Type**: Select the output file type, as either -
    - **Horizon Polylines**: 3D Polylines vectors for the digitized horizons and picks.
    - **Velocity Mesh**: 3D triangular mesh for the regions enclosed by the horizons. Coloured by their assigned velocity colour.
    - **Section Image**: Registered raster image in 3D, optionally define a DEM Grid to "hang" the image off.

16 Digitizing and Managing 3D Features

The Discover 3D Features menu provides access to a powerful set of utilities for creation, storage, manipulation and display of 3D Features.

A **3D Feature** is a three dimensional geometric entity which consists of one or more 3D point objects. Features can be:

- Simple geometric entities such points and line objects.
- More complex 3D objects such as polylines and polygons.
- Triangulated irregular network (TIN) surfaces, both open surfaces (e.g. fault planes) and closed polyhedral surfaces (i.e. solids/volumes).

Features can be easily used to represent real world entities such as geological boundaries, orebody grade shells, structural/fault surfaces and intersections or any other general three dimensional feature of interest. Features can be created in many ways:

- Digitised directly into Discover 3D (e..g snapping to drillhole intercepts) as points, polylines and polygons
- Created from vector data in MapInfo/Discover, such as drillhole section boundaries
- Extruded from 2D vector objects to create open and closed triangulated surfaces such as fault planes and buildings
- By wireframing (with tieline controls) between features in 3D to create complex triangulated surfaces and volumes such as watertable levels and orebody models
- Importing existing triangulated models/objects from other applications such as AutoCAD DXF, Datamine wireframe and Vucan triangulation files
- Importing and converting gridded surfaces (as created with Discover's Surfaces module) such as topographic grids into triangulated surfaces

Features are stored in a Feature Database. A Feature Database is a special type of relational database that supports complex three dimensional geometric objects and associated attributes. Each Feature Database contains one or more features, e.g. supergene ore boundaries, interpreted fault lines, etc. Features can contain both geometric information and accompanying attribute information which is stored in related fields within the database. Feature Databases can be queried using a simple SQL based query language to extract subsets of the database using a combination of attributes and/or geometric properties.

Aside from viewing in Discover 3D, features can be used with the 3D Solid Generator (see *Modelling Triangulated Surfaces and Solids*). The 3D Solid Generator joins a number of features together to produce a three-dimensional enclosed or planar surface; i.e. a solid body to represent an ore body zone or fault plane. Volumes can be calculated for enclosed 3D Solid bodies.

Features can also be exported into Discover Drillhole Sections (as MapInfo Professional TAB files), as well as a range of other formats.

Features are managed from the **Features** menu in the 3D interface, from which you can:

- Open and close an existing feature database (see *Opening an Existing Feature Database*).
- Create a new feature database (see *Creating a New Feature Database*).
- Modify the structure of any existing open feature database. For example, adding new fields or changing field names or types (see *Modifying the Structure of a Feature Database*).
- Import data in a range of vector and grid file formats (see *Importing Data into a Feature Database*).
- Export features to a range of industry-standard 2D and 3D file formats (see *Exporting Feature Databases*).
- Query a feature database using SQL expressions (see *Select by SQL Query*).
- Combine, Intersect, Erase, Merge, Triangulate, Aggregate, Disaggregate Resize, Delete, Cut, Copy and Paste features (see *Editing Features*).
- Set default fill and selection colours (see *Feature Options*).
- Manage features in the cosmetic layer (see *Saving Cosmetic Features to a Feature Database*).

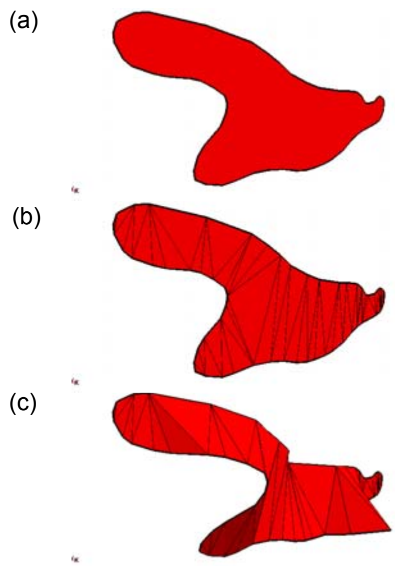
Feature Object Types

Discover 3D can create, import, export, edit and manipulates five types of feature geometric objects. These are explicitly attributed in the **Type** field of the Feature Data Window, which also automatically and dynamically reports geometric information such as object centroid (and node) XYZ coordinates, length, area and volume. The five object types are:

- **Points** - Individual nodes with solely an XYZ location. These can be digitised in 3D with the **Create New Point Feature** tool, or imported from a vector file.

- **Polylines** - Single or multi-segment line (linking nodes), These can be digitized in 3D with the **Create New Line Feature** tool, or imported from a vector file

- **Polygons** - Planar region/area consisting of multiple exterior nodes joined by external edges, with an internal planar fill. These can be digitized with the **Create New Polygon Feature** tool, or imported from a vector file.



(a) An example of a Polygon
(b) Triangulating a polygon to form a surface
(c) Moving nodes to form a non-planar surface

- **Surfaces** - A vast range of shapes are handled as surfaces, from semi-planar fault planes and unconformities, to shapes with holes and unclosed regions (e.g. 'open' cylinders). Essentially, any object consisting of a network of triangular faces (facets) that does not fully enclose a volume.

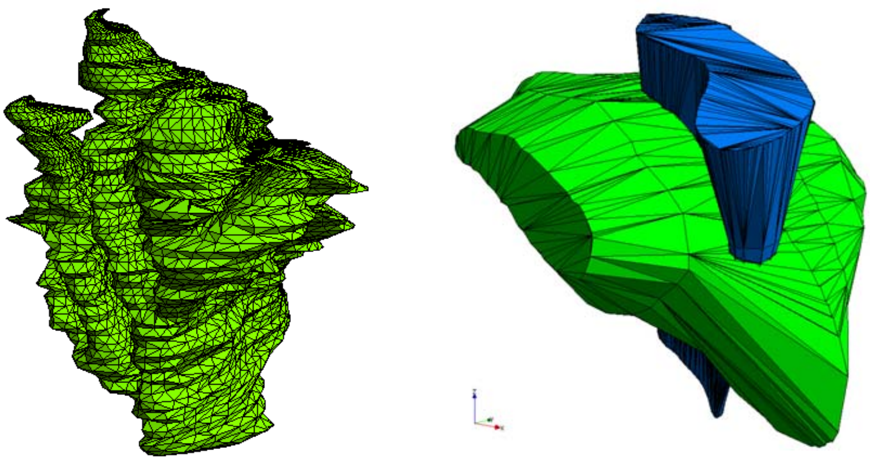
Surfaces can be created by:

- Triangulating a polygon (see *Feature Editing Toolbar*).
- Importing a gridded surface.
- Wireframing (solid generation) from polylines or polygons/surfaces (unclosed).
- Importing from a vector file.

- **Solids** (or polyhedron volumes) - An object fully enclosing a volume, with a calculable volume, such a an aquifier volume or orebody.

  Solids can be generated by:

  - Wireframing (solid generation) from polygons/surfaces (closed).
  - Importing from a vector file.



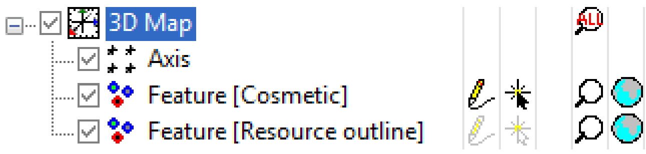
Examples of solids/polyhedron volumes

Opening an Existing Feature Database

Features are stored and maintained in a **Feature Database**. A **Feature Database** is a relational database that supports complex 3D objects and associated attributes. A Feature database can be identified by its .fdb file extension. Discover 3D has the ability to open and maintain multiple feature databases concurrently.

To open a **Feature Database** select the **Features>Open** menu option. The **Open** dialog will be displayed:

Browse to the location of an existing Feature database and click **Open**. A new **Features** branch is added to the Workspace Tree.



The **Feature Database** will be opened and displayed as a new **Feature** item in the Workspace Tree.

When an existing Features Database is opened into the 3D window the **Feature Properties** dialog should be automatically displayed to enable various parameters to be set for the display of the features including symbol, line and fill styles and label options. If the Feature Properties dialog does not automatically appear you may need to enable this feature on the View tab of the general Options dialog (see *Customising 3D Interface Settings*). Also see *Changing Feature Display Properties* for more information.

New **Feature** items are always displayed below the **Feature [Cosmetic]** entry in the Workspace Tree. The **Feature [Cosmetic]** item is always present in the Workspace Tree and it operates in a similar manner to the Cosmetic layer in MapInfo Professional as it enables 3D features to be easily created at any time and stored in a temporary feature database. Features that are created in the Cosmetic Feature database will only persist while the 3D session is open. If you wish to keep any features that have been created in the cosmetic layer you will need to save them to a permanent feature database for later use.

Interacting with Features

The display, selection and editing properties of a Feature database are configured using the Workspace tree.

The icons to the left and right of each **Feature Database** item in the Workspace Tree control the following properties:

Visibility - toggles the visibility of the features in the 3D window on and off.

Editability – enables editing for the selected Feature database. With editing mode enabled new features can be added to the database and existing features can be edited or deleted from the Feature database. Only one Feature database at a time can be made editable.

Selectability – enables features within a selected Feature Dataset to be interactively selected using the mouse. Selectability can be used to limit which Feature database(s) can be interrogated when multiple complex feature databases are open in the 3D view.

Zoom Extents – zoom the 3D view to the extents of the features in the selected Feature database.

Closes a selected **Feature Database.** The **Features>Close** menu option displays the **Close Feature Databases** dialog which provides the ability to select individual feature databases to close when the **Close** button is clicked (shown left).

Creating a New Feature Database

A new **Feature Database** can be created by any of the following methods:

- **Features>Create** menu option – creates a new empty Feature Database using the *Feature Database Creation Wizard*.
- **Features>Import** menu option **-** imports a variety of supported vector and grid file formats from external third party software programs into a new Feature Database (see *Importing Data into a Feature Database*).
- Interactive Drawing - allows new features to be directly drawn onto the cursor plane in 3D and stored in any open Feature database that is set to be editable, including he Cosmetic Feature branch in the Workspace Tree (see *Creating and Editing Features*). For information on saving cosmetic features, see *Saving Cosmetic Features to a Feature Database*.
- Use the 3D Solid Generator (see *Modelling Triangulated Surfaces and Solids*) to wireframe multiple features to build open TIN surfaces (e.g. fault planes) or closed polyhedral solids (e.g. zones of mineralisation). The solid generator can use polyline, polygon and TIN surface features as input, and the output model can be refined using user-created tie-lines. These triangulated features can be output directly to an existing or new Feature database.

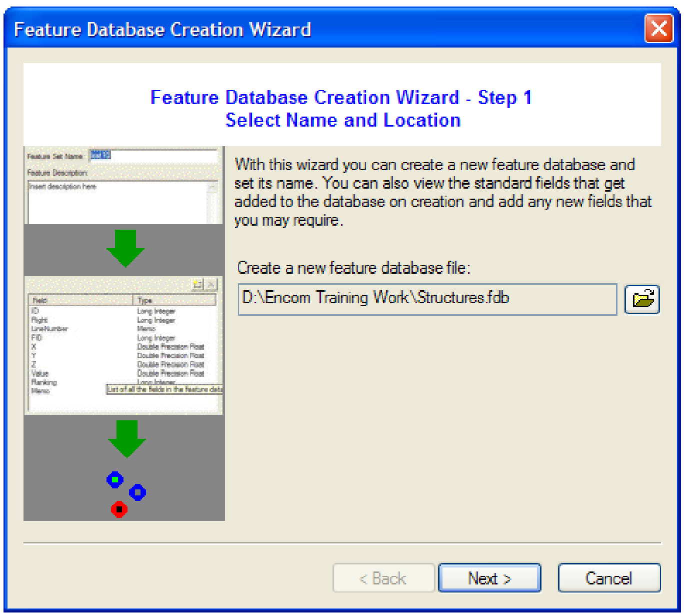
- Use the Extrusion Wizard (see *Extruding Models from Points, Lines and Polygons*) to extrude 2D or 3D features, including points, lines, polylines and polygons into triangulated 3D mesh features. The output features from the Extrusion Wizard can be directed into a new or an existing Feature database.

Feature Database Creation Wizard

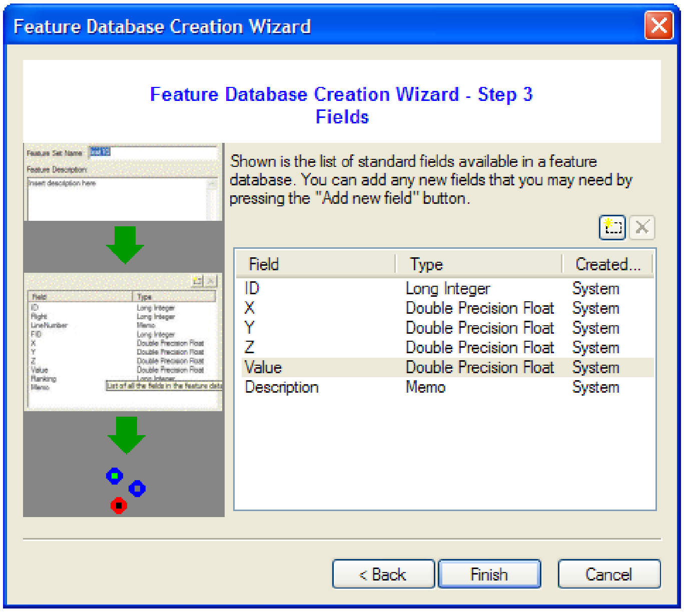
The **Feature Database Creation Wizard** creates a new empty Feature Database.

To create a new Feature Database:

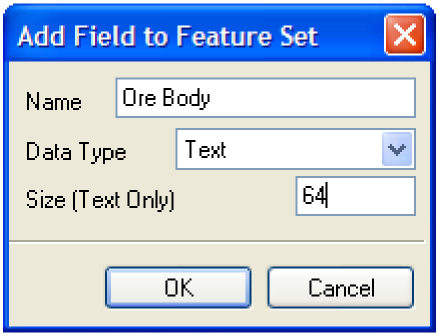
1. Select the **Features>Create** (shown left) menu option. The Feature Database Creation **Wizard** dialog is displayed at **Step 1**:



2. Use the browse button (shown left) to navigate to a folder to save the file and enter a name for the new Feature database. Click **Next**.



3. Each new feature database is created with the following set of default fields : ID, X, Y Z, Value and Description. All Feature databases must contain these default fields so their name and type properties cannot be modified. Any number of additional user-defined fields can be added to a new feature database using the creation wizard. To add a new field to the database click the new field button shown left.



4. Enter a name for the new field and select an appropriate **Data Type for the information that you wish to store**. A number of data types are available.

- Boolean – can contain binary information only (i.e. Yes/No).
- Byte – can contain byte values, ranging between 0 and 255.

- Short Integer – can contain small integer values ranging in value from -32768 to 32767.
- Long Integer – can contain large integer values ranging in value from -2,147,483,648 to 2,147,438,647 .
- Currency – can contain numeric data in currency format to enable accurate fixed-point calculations to be performed where fractional accuracy is important.
- Single Precision Float – can contain numeric values in single-precision 32-bit floating point format ranging in value from -3.402823E38 to 3.402823E38.
- Double Precision Float – can contain numeric values in double-precision 64-bit floating point format ranging in value from approximately -1.797E308 to -4.940E324 (negative), from 4.94E-324 to 1.797E308 (positive), and zero.
- Date – can contain date information .
- Text – can contain text to a maximum of 255 characters.
- Memo – can contain text to a maximum of 65,535 characters.

If a text or memo data type is selected enter an appropriate size: this property relates to the maximum number of characters that can be stored by the field. Click **OK** to create the new field.

5. Click **Finish** to create the new Feature Database. A new Features item is added to the Workspace Tree. See *Creating Features* for information on how to add features to a database.

Importing Data into a Feature Database

The **Import File Wizard** allows a range of Vector file formats and Grid file formats to be imported into either an existing or new Feature Database. This wizard is accessed via the **Features>Import** menu option.

- *Supported Import Formats*
- *Using the Import Feature Database Wizard*

Supported Import Formats

Vector formats:

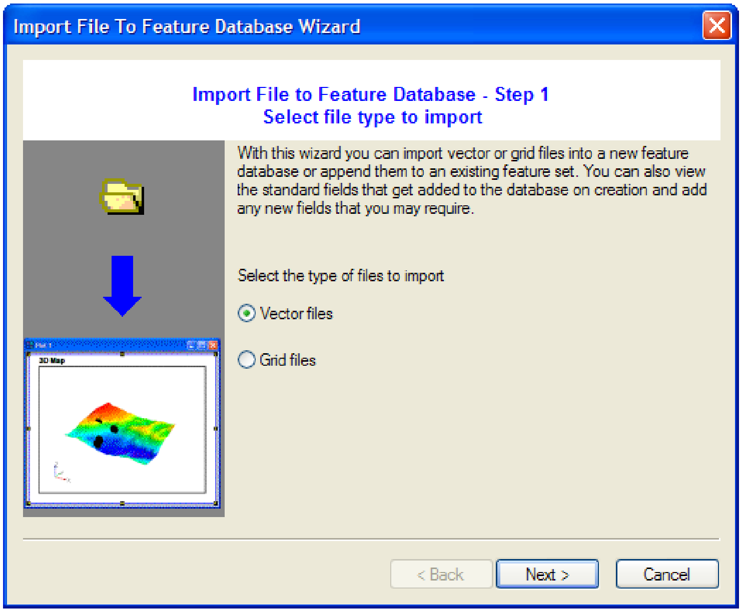
3D Studio .3DS
AutoCAD .DXF
Datamine wireframe .DM
Encom .TKM
ESRI .SHP and .TIN
Gemcom .BT2
GoCAD .TS, .PL and .VS
MapInfo .TAB and .MIF
Surpac .DTM and .STR
Vulcan triangulation .00T

Grid formats:

ASEG GXF (.GXF)
Band Interleaved by Line (.BIL including .HDR)
Encom grid (.GRD)
ERMapper (.ERS)
ESRI/Arc Binary (.ADF)
ESRI/Arc ASCII (.ASC)
Geopak (.GRD)
Geosoft (.GRD)
Landmark (.GRD)
MapInfo grid (.MIG)
Minex (.XYZ)
Surfer ASCII (.GRD)
Surfer Binary (.GRD)
USGS (.USG)
USGS DEM (.DEM)
USGS SDTS (.TAR)
Vertical Mapper (.GRD)

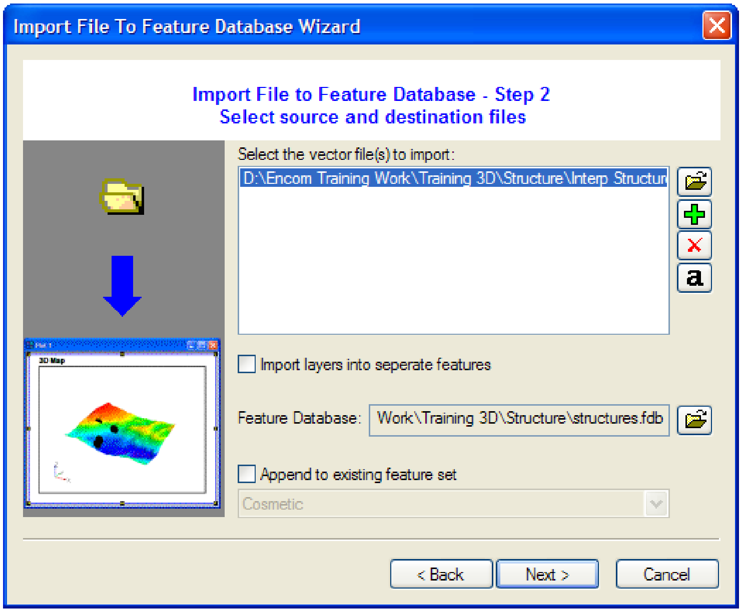
Using the Import Feature Database Wizard

1. Select the **Features>Import** (shown left) menu option. The Feature Database Import **Wizard** dialog is displayed at **Step 1**:

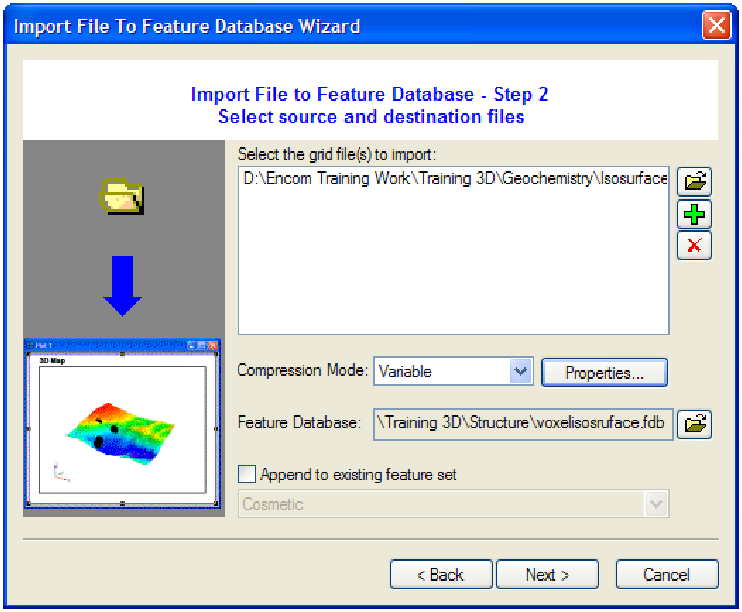


The Import File to Feature Database Wizard

2. Select the type of files to import. Two options are provided, **Vector files** or **Grid files**. See *Supported Import Formats* for a complete list of supported formats. Press **Next**.



Importing a DXF file into a new feature database called ‘Structures’



Importing a grid file into a new feature database called 'Voxel Isosurface'

3. For either the Vector or Grid formats, the second dialog allows selection of the source file using the **Browse** button.

- Multiple files can be imported simultaneously by selecting them consecutively with the **Add Files** button (shown left).

- If multiple attributed Vector files are being imported, only one file can be used to define any extra custom fields. Highlight this file in the import list, and click the **Attribute** button (shown left); the selected file will be moved to the top of the list.

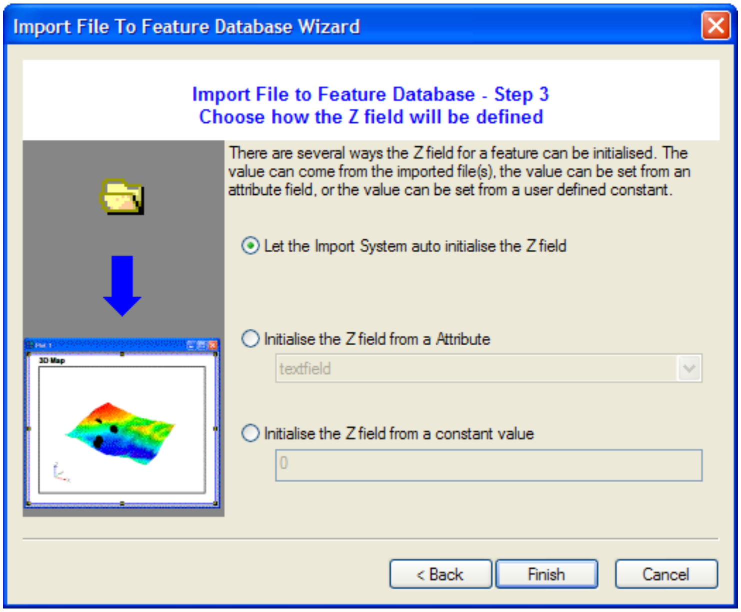
If importing a Vector file, you can **import** any contained **layers into separate features** by enabling this option

If importing a Grid file, a number of **Compression Modes** are available to facilitate efficient importing and handling of larger grids. See the *Grid Compression* section of the Gridded Surfaces in 3D chapter for a full explanation of the available options.

The imported files can either be:

- Appended to an existing open feature database by enabling this option and selecting the target database from the pull-down list or

- Created in a new feature database (or existing feature database can be overwritten) using the Browse button adjacent to the **Feature Database** option. Click **Next** to advance the wizard.



The final dialog of the Import wizard, controlling the Z value assignment.

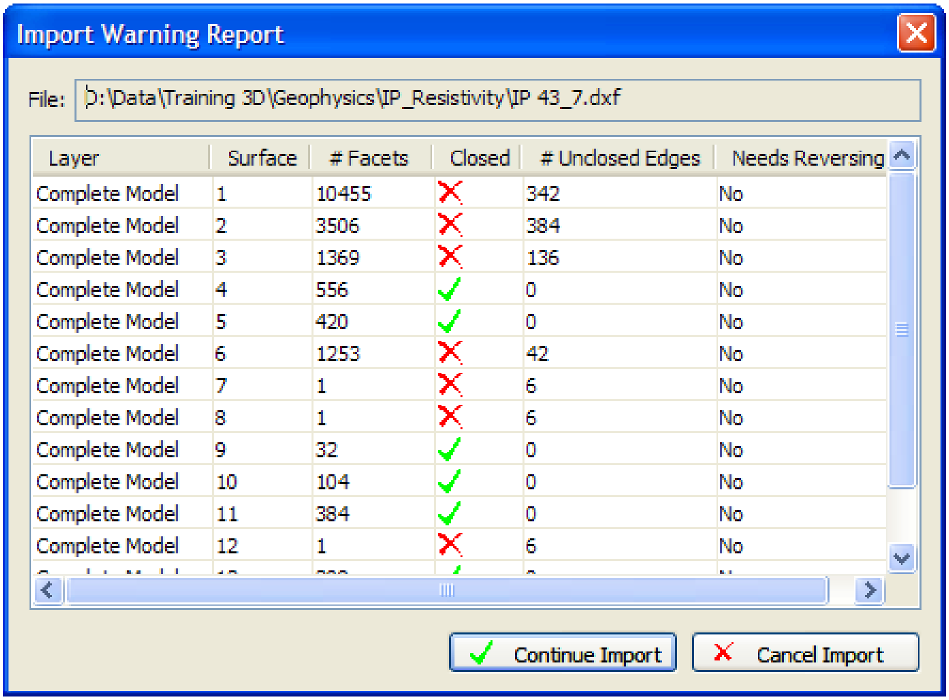
4. The final dialog of the wizard (Step 3) allows the Z field handling to be controlled. Generally this is best left as the default option (**auto initialise the Z field**), capturing this information automatically from the input file. However, you can instead specify either an attribute field or constant value to use for Z information.

5. Click **Finish** to start the import process. If a new feature database was created, it will be added to the workspace tree.

If importing a vector file composed of 3D facets (e.g. planar or volume solids), Discover 3D will automatically perform a check of each surface, examining whether all facets are closed, and whether the order of vertices needs to be reversed. If any issues are found, the user will be presented with the **Import Warning Report** dialog (pictured below). The user can either:

- **Continue the import** – all surfaces will be displayed in Discover 3D, but using some of the feature editing tools (see *Editing Features*), such as Combine, Intersect, Erase, etc., on any surfaces with unclosed facets will result in errors.

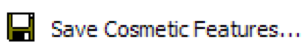
- **Cancel the Import**. Then use the *Topology Checker* utility (in the **Utilities** menu) to export from the source vector file only the surfaces whose facets are closed, and reverse the order of vertices as necessary. Import this new vector file using the Feature Import tool.



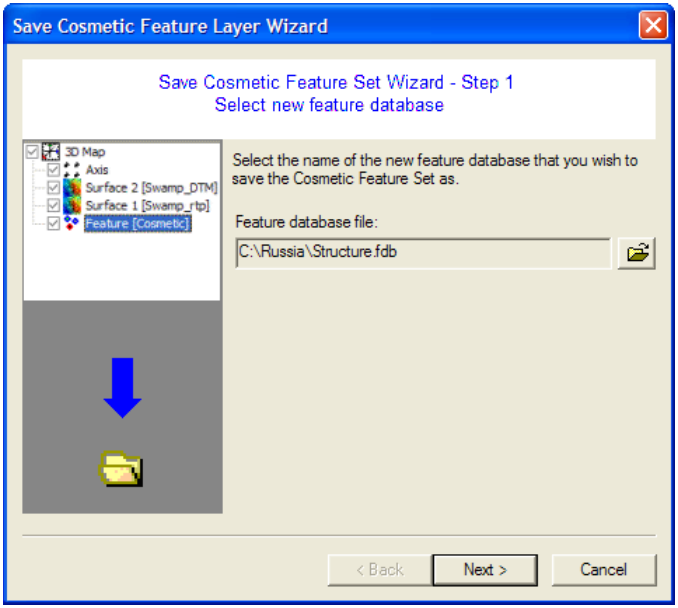
Layer	Surface	# Facets	Closed	# Unclosed Edges	Needs Reversing
Complete Model	1	10455	✗	342	No
Complete Model	2	3506	✗	384	No
Complete Model	3	1369	✗	136	No
Complete Model	4	556	✓	0	No
Complete Model	5	420	✓	0	No
Complete Model	6	1253	✗	42	No
Complete Model	7	1	✗	6	No
Complete Model	8	1	✗	6	No
Complete Model	9	32	✓	0	No
Complete Model	10	104	✓	0	No
Complete Model	11	384	✓	0	No
Complete Model	12	1	✗	6	No

Result of the automatic vector check performed by the Feature Import tool.

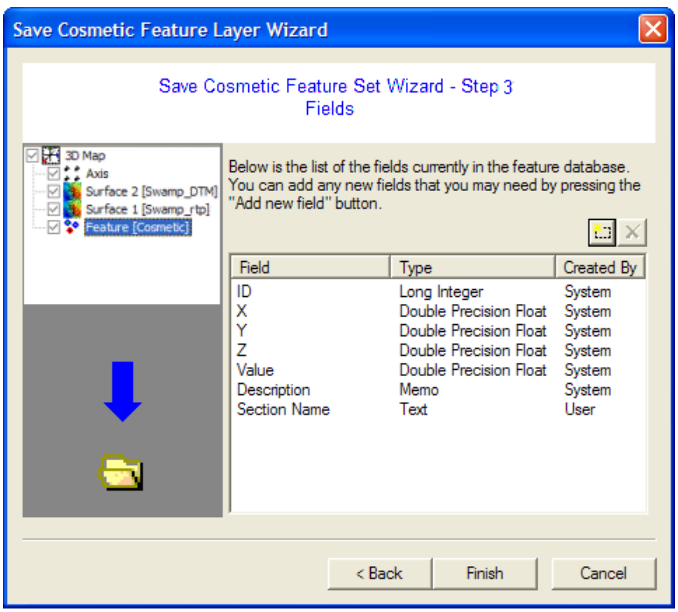
Saving Cosmetic Features to a Feature Database



If features are created in the Feature [Cosmetic] database they can be saved off to a permanent Feature Database (or overwrite an existing one) by selecting the **Features>Save Cosmetic Features** menu option. The Save Cosmetic Feature Set Wizard - Step 1 dialog is displayed:



Use the browse button to navigate to a folder to save the new Feature database and enter a name. Click **Next**.



When the new **Features Database** is opened into the 3D window the **Feature Properties** dialog is automatically displayed to enable various parameters to be set for the display of the features including symbol, line and fill styles and label options. See *Changing Feature Display Properties* for more information.

The structure of a new feature database created from the cosmetic layer can also be modified later using the Modify Feature Database tool (see *Modifying the Structure of a Feature Database*).

Modifying the Structure of a Feature Database

Feature databases can be modified in a number of ways:

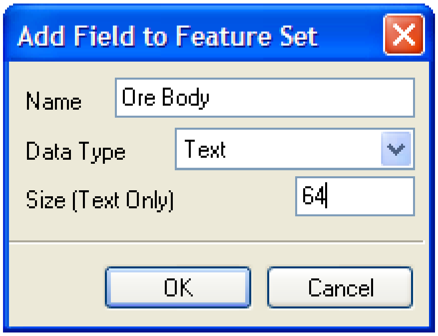
- Feature databases can be renamed.
- The name and data type of any existing fields can be altered.
- New fields can be added.

Note The cosmetic feature database cannot be modified.

To modify an existing feature database:

1. Ensure that the feature database is open in Discover 3D
2. Select the **Features>Modify** menu option
3. From the **Select Feature Set** dialog, select the database to modify and press OK
4. To rename the database, enter the new name in the **Feature set name** field on the **Name and Description** tab. If desired, description metadata can be added at the bottom of this tab.
5. Fields can be added and modified in the **Fields** tab. See steps 3 and 4 of the How to create a new Feature Database: section above [Tim either that or copy these sections ot this location. Note the default fields cannot be modified)

6. Each new feature database is created with the following set of default fields : ID, X, Y Z, Value and Description. All Feature databases must contain these default fields so their name and type properties cannot be modified. Any number of additional user-defined fields can be added to a new feature database using the creation wizard. To add a new field to the database click the new field button shown left.



7. Enter a name for the new field and select an appropriate **Data Type for the information that you wish to store**. A number of data types are available.

- Boolean – can contain binary information only (i.e. Yes/No).
- Byte – can contain byte values, ranging between 0 and 255.
- Short Integer – can contain small integer values ranging in value from -32768 to 32767.
- Long Integer – can contain large integer values ranging in value from -2,147,483,648 to 2,147,438,647 .
- Currency – can contain numeric data in currency format to enable accurate fixed-point calculations to be performed where fractional accuracy is important.
- Single Precision Float – can contain numeric values in single-precision 32-bit floating point format ranging in value from -3.402823E38 to 3.402823E38.
- Double Precision Float – can contain numeric values in double-precision 64-bit floating point format ranging in value from approximately -1.797E308 to -4.940E324 (negative), from 4.94E-324 to 1.797E308 (positive), and zero.
- Date – can contain date information .
- Text – can contain text to a maximum of 255 characters.
- Memo – can contain text to a maximum of 65,535 characters.

If a text or memo data type is selected enter an appropriate size: this property relates to the maximum number of characters that can be stored by the field. Click **OK** to create the new field.

8. Press OK to complete the changes.

Exporting Feature Databases

The **Feature Exporter** (accessed via the **Features>Export** menu option) allows a selected Feature Database to be exported as one of the following formats:

ArcView .SHP

ASCII .CSV

AutoCAD .DXF

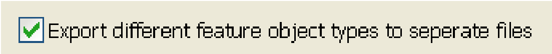
GoCAD .TS, .PL and .VS

MapInfo Professional .TAB

MapInfo Professional .MIF

The output format is set with the Browse button in the **Output Files** section

If a feature database comprises multiple object types (such as points, polygons and solids), these can be exported to a separate file for each object type:



The **Output Colour** of the exported objects can be set to the following:

- **From input feature database** - uses the colour of the feature objects in Discover 3D. This does not support features colour-modulated by a legend.

- **Fixed** - select a colour from the adjacent palette

- **Modulated by field** - set a field and a colour table/LUT to colour modulate the exported objects with.

Changing Feature Display Properties

The display style and labelling options of all features in a Feature Database can be modified in the **Feature Properties** dialog (double-click on the appropriate Feature item in the Workspace Tree to open, or right click on the Feature layer and choose the **Properties** menu option).

Feature Appearance

The **Appearance** tab of the Feature Properties dialog provides controls for point, polyline and polygon style control, including:

- **Symbol** type, size and colour
- **Fill** colour and transparency
- **Line** colour, weight and pattern.

Important Any style applied will affect all features in the database.

Polygon colour fill can be disabled by deselecting the **Fill** tick box.

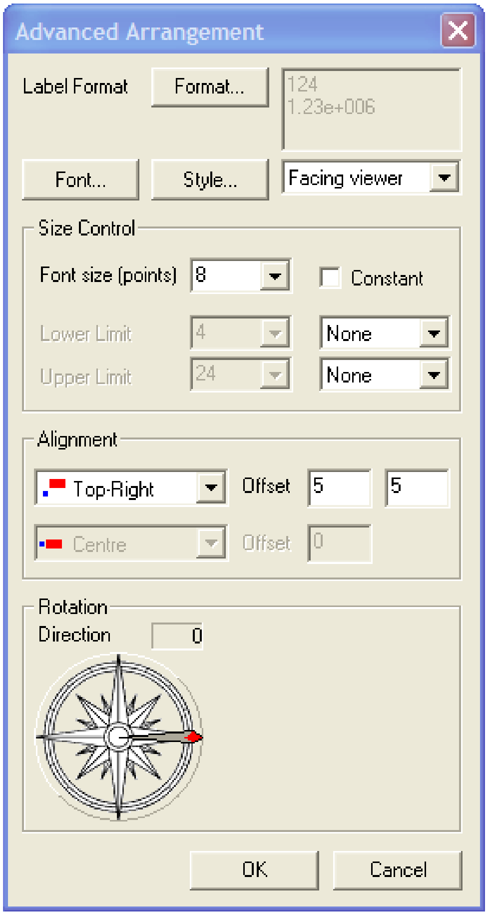
Alternatively, all features in the database can be **colour modulated:**

1. Specify a numeric field to colour modulate the features by using the **Colour** control under the **Data** tab.

2. Use the **Colour Table Edit** button under the **Appearance** tab to either set a Look-Up Table or custom Legend, or customise and apply an RBG or HSL interpolation. See the *Colour Table* selection section of Voxel Models for a detailed description of this dialog.
3. Click **Apply** (or enable the Auto Apply option) to display the changes.

Labelling Features

Feature **Labels** can be displayed by specifying an appropriate data field in the **Data** tab of the Feature Properties dialog. A **Label Skip Factor** can also be set under the **Appearance** tab. The **Advanced Arrangement** dialog (accessed via the **Appearance** button under the **Appearance** tab) provides powerful controls for modifying and controlling labels.



The Advanced Arrangement dialog for line labels

The **Format** button provides numeric formatting options (scientific, various DMS formats, general, etc), decimal place allocation as well as suffix and prefix specification

Standard **Font** controls are provided, as well as the following range of orientation options in the adjacent pull-down list:

Facing Viewer (default) – Labels are aligned parallel to the viewing/screen plane, so that they always face the viewer.

- **Fixed 3D** – Labels are fixed in the 3D environment. The pull-down list in the Rotation panel at the bottom of the dialog controls the initial orientation: Flat (the XY plane) or Upright (XZ plane).

- **Fast 3D** – identical to the Fixed 3D except that no Style controls are available; this is a very fast and memory-efficient labelling option.

The **Style** button is only available for the **Facing Viewer** and **Fixed 3D** orientation options. It enables labels to be extruded either as filled Polygons or Line Segments (wireframe) using the **Format** pull-down list. The depth of the label is set using the **Extrusion** control, expressed as a percentage of the label size. To display a flat label, set the **Extrusion** to 0%. The label is extruded perpendicular to its display plane.

Note Rendering labels as extrusions is memory intensive and will affect 3D performance. It is not recommended for large numbers of labels.

A range of **Size Controls** are available. For the **Fast 3D** orientation option, only the **Font Size** control is available (in points). Enabling the **Constant** checkbox (for the other orientation options) will keep the labels at the specified size (relative to the screen) regardless of zoom level. If the Constant option is disabled, **Lower** and **Upper Limits** can instead be set:

- **Block:** labels will disappear when the applied zoom level takes the label past the specified limit,
- **Clamp:** labels will be locked to the specified limit when the applied zoom level takes the label past the specified limit.

The label position relative to the data (collar) location can be altered using the **Alignment** controls (either preset or manual positioning). The first row of controls concern label positioning in the label plane (i.e. the relative XY components), whilst the second row controls the vertical height of the label with respect to its initial plane (i.e. the relative Z component).

The angle of the labels can also be set by moving the red-tipped arrow on the compass in the **Rotation** panel at the bottom of the dialog. The **Fixed** and **Fast 3D** labelling options also provide a **Dip** control (half-compass) in this panel.

Selecting Features

- *Graphical Selection*
- *Select by SQL Query*
- *Browser Selection*

Graphical Selection

Feature editing and SQL query both require one or more features to be selected. A feature database must be **Selectable** before features within it can be selected; this is enabled in the *Workspace Tree*. This control is useful when multiple dense databases exist in the same area; you can disable the selectability of all databases except the target database.

A number of feature selection methods are available:

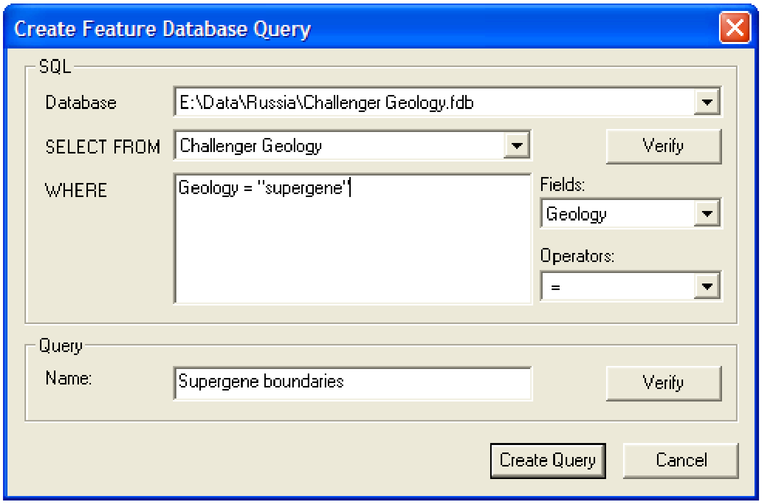
Enable the **Select** button on the *Zoom Controls Toolbar*, and click on the feature to edit. When selected, a feature will be highlighted in red (unselected features are displayed in a default blue colour). The Cursor Plane will snap to the feature's plane. To select multiple features, hold down the keyboard CTRL key whilst using the **Select** button; the Cursor Plane will snap to the last selected feature.

The **Polygonal Selection** button allows you to draw a selection region in the 3D window; all features that lie entirely within this region will be selected. The selection region is independent of the Cursor Plane; it is actually drawn within the screen plane, and selects all features behind this plane, regardless of feature orientation.

Alternatively, one or more features can be selected from the **Features** tab of the Information window by selecting their row entries (multiple selections are possible using the SHIFT or CTRL keyboard keys).

Select by SQL Query

The **Create Feature Database Query** dialog (accessed via the **Features>Query** menu option) allows features to be queried by attribute using basic SQL syntax. The output of a query is a new Feature Database as specified in the **Name** option of the dialog. This is a useful way to separate a multi-attributed database (e.g. that pictured under the *Feature Database Window* section) into a number of databases each populated with features of a single attribute.



Pull-down **Field** and **Operator** selection dialogs to the right of the Expression pane allow SQL syntax creation.

SQL syntax examples

```
Rock = "sand"
```

Returns entries in the Rock field where the only text is “sand”.

```
Rock Like "sand*"
```

Returns any entries in the Rock field prefixed with “sand”; for example, sandstone, sand and sandy cobbles.

```
Rock Like "*stone*"
```

Returns any entries in the Rock field with “stone” anywhere in the text string; for example, sandstone, siltstone, stoney cobbles and stone.

Note Discover 3D uses the asterix (*) as a wildcard, unlike MapInfo's % wildcard. The * wildcard is only activated by using the LIKE operator.

Browser Selection

Features can be selected from the **Feature Spreadsheet**; see the *Feature Selection* section below.

Creating and Editing Features

- *Creating Features*
- *Snapping Features to Other Data*
- *Editing Features*
- *Advanced Editing Functions*

Creating Features

Creation of features requires an **Editable** Feature Database (enabled via the pencil icon in *Workspace Tree*); this can be either the cosmetic or a user-created or imported database.

When a feature database is made editable, both the Cursor Plane and Feature toolbars are enabled, and the Cursor Plane is displayed. The Cursor Plane is used as a drawing plane onto which features are digitized: for detailed information on controlling the Cursor Plane, see the *Cursor Plane* section.

The **Feature Toolbar** provides the tools for the creation and editing of features, including **Points**, **Polylines** and **Polygons**. A detailed description of each buttons' functionality is presented in the *Features Toolbar* section.

To draw point, polyline or polygon features in 3D:

1. Make the target feature database **Editable**.
2. Use the *Cursor Plane* controls to position and orientate the Cursor Plane in the required location.
3. Select the appropriate feature button form the Feature Toolbar, and click on the point location or polyline/polygon start point in the Cursor Plane.
4. For polylines/polygons, click on consecutive points to outline the feature, or hold down the left mouse key whilst moving the cursor to create a continuous series of nodes.
    - Press the Esc key at anytime to cancel the feature creation.
    - Press the Backspace key at anytime to delete the last added node.
5. Complete polyline features by pressing the Enter key or double-clicking; polylines will end at the current cursor location

6. Complete polygon features by pressing the ENTER key or double-clicking; polygons will close back to the start point whilst using the current cursor location as an intermediary node/vertice. Alternatively, pressing the CTRL+ENTER key combination will complete the polygon and ignore the current cursor location.

7. Enabling the **Confirm Pick** button prior to feature creation will display a **Confirm Feature Description** dialog. This allows the feature's default fields (and custom fields if any) to be attributed with either text or values. For instance, the Description field could be attributed with the feature's geology, e.g. south_fault, supergene or zone_1, allowing a single Feature Database to hold multiple geological features.

8. Upon completion of a feature, it is highlighted in red as the current selected feature (see *Selecting Features*), allowing editing (see *Editing Features*).

The **Ruler tool** on the *Main Toolbar* can be a useful guide for distances, bearings and dip angles between features on the cursor plane. Also, the Grid option on the Cursor Plane Properties dialog (see *Changing Cursor Plane Properties*) can also be a very useful visual guide to distances/sizes.

Snapping – Allows the current feature's node (new feature or editing an existing feature) to be snapped to any dataset set as **Selectable** in the workspace tree. See *Snapping Features to Other Data*.

To create a feature surface:

Polygons are planar objects; the external nodes cannot be moved out of the objects plane. Polygons can however be converted into a feature Surface using by triangulating the internal area, forming a continuous network of triangular faces.

1. Select a polygon, polyline, or group of points in an editable feature database.

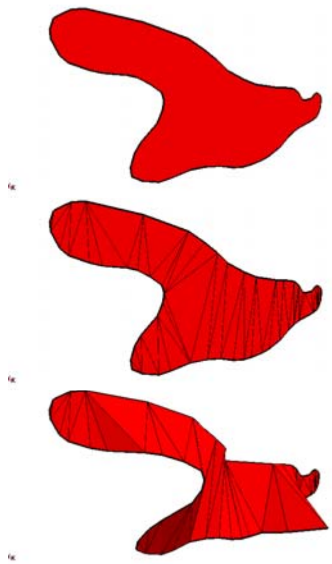
2. Press the **Triangulate** button on the Feature Editing toolbar.

3. Make the triangulated surface reshapable, and holding down the CTRL key, select a node and move it outside the orignal polygon plane.

(a)

(b)

(c)



(a) An example of a Polygon
(b) Triangulating a polygon to form a surface
(c) Moving nodes to form a non-planar Surface

See *Triangulate* below for futher information.

Snapping Features to Other Data

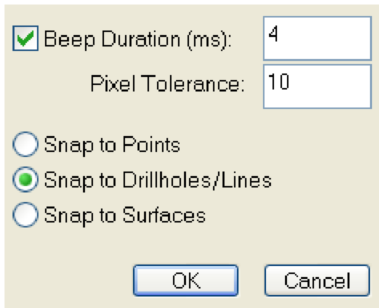
Use the **Snapping** tool to snap the current feature's node (new feature or editing an existing feature) to any dataset that is set as **Selectable** in the workspace tree. The following data types can be snapped to:

- Drillholes/Trenches
- Feature databases (surfaces, polyhedral solids, polygons, etc)
- Images
- 3D Points
- 3D Lines

Upon activation, the cursor will change to a circle in the middle of the cross-hairs. When the cursor is close to an existing node, it will change to the standard cross-hair: left-click to snap to this node.

To the right of the **Snap** toolbar button is a pull-down options dialog. This allows the **Pixel Tolerance** to be set (i.e. how close the cursor needs to be to an existing node before it can be snapped to that node). A high-pitched audible **Beep** (‘*dee*’) can also be enabled for when the cursor is within the set pixel tolerance: when the cursor moves outside the tolerance, a low pitched ‘*doh*’ will sound. The **Duration** of these beeps can be set in milliseconds.

This dialog also provides options for snapping to nodes/points, lines/drillholes/ edges or surfaces.

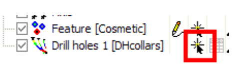


To snap nodes to drillholes, images, points, lines, and surfaces:

1. Enable snapping. In the adjacent drop down dialog, enable the appropriate snap option:

   - **Snap to Points** for 3D points or nodes in polylines, polygons or solids.
   - **Snap to Drillholes/Lines** to snap anywhere along drillhole traces or 3D lines, or to snap to the edges of solids/polygons.
   - **Snap to Surfaces** to snap anywhere on interior faces/surfaces of a polygon/solid.

   This setting does not affect snapping to images.



2. Make the relevant item (Lines, Points, Drillholes, Feature database, or Image) in the Workspace Tree **Selectable**.

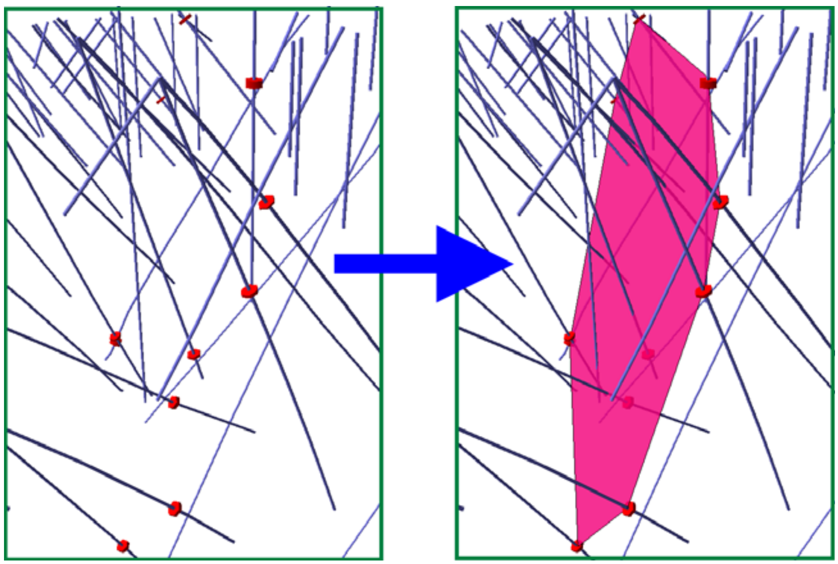
   In complex 3D sessions, it is recommended to turn the selectability OFF for all other datasets, to ensure the correct dataset is snapped to. It is also recommended to turn on the spreadsheet/browser view for the item, to dynamically visualise the attributes.

3. Ensure the cursor plane is on by toggling the cursor plane button.

4. Optionally, click the **Bond** button to fix the cursor plane to the selected item. Note that the cursor plane is not visible in this mode, but you can still digitize and snap.

5. Make a feature database layer editable and enable the required drawing tool. Move the cursor over an object location: the red crosshairs will snap to that location.

6. This is particularly powerful for 3D interpretation of downhole data; for instance, digitizing a series of points, polylines or polygons outlining mineralization or geological features for use in the 3D Solid Generator (see *Modelling Triangulated Surfaces and Solids*) or surface gridding tools (see *Creating Gridded Surfaces*).



Some digitization tips which are controlled via the **Cursor Plane** tab of the **Cursor Plane Properties** dialog:

- Use the PGUP and PGDN keys to offset the Cursor Plane perpendicularly from its existing position, in order to keep digitized features parallel. Use the **User-defined step** option to set the distance the PGUP/PGDN keys move the Cursor Plane.

- The exact location of the Cursor Plane can be set via the **Origin** (X, Y and Z), **Inclination** and **Azimuth** controls.

- To limit the amount of existing data visible either side of the Cursor Plane (i.e. simulate a 2D section envelope), enable either the **Envelope** or **Slice Clipping** options. These enable only a user-defined **Slice Width** of data to be viewed either side (envelope) or behind (slice) the Cursor Plane. This is an excellent way to ensure that interpreted boundaries are based only on data within, for example, 50 m of the Cursor Plane.

- If digitizing onto georeferenced images (e.g. geophysical profiles), use the **Bond to Image** option on the *Cursor Plane*.

Note Using a 3DConnexion SpaceNavigator (see *Using the 3DConnexion SpaceNavigator™*) can help to precisely identify and snap to the intercept location in more complex 3D environments, and is essential when digitising polylines.

Editing Features

Feature can be duplicated, repositioned, resized and deleted. Individual or multiple nodes within a feature can be added, moved or deleted. Feature attributes can also be edited.

The following basic editing functionality is available:

Editing an entire feature:

- Moving features within the curosr plane
- Moving features perpendicular to the cursor plane
- Resizing features
- Deleting features
- Cut, copy and pasting features

Editing individual nodes/vertices:

- Repositioning nodes within the cursor plane
- Repositioning nodes perpendicular to the cursor plane
- Add nodes
- Delete nodes

- Elastic movement of nodes/regions
- Extending polylines from end nodes

To enable editing of a feature or its nodes:

1. Feature editing requires one or more features to be selected in an **Editable** feature database (ensure the **Editable icon** shown left is enabled in the Workspace Tree).

2. Enable the **Edit mode** button on the *Features Toolbar*; all nodes/vertices on the selected features will be displayed. Double clicking on a editable feature will also enable the **Edit mode** for the feature:

Editing an entire feature:

The following **Editing** functionality is available for an editable feature, whether points, polylines, polygons or triangulated surfaces.

Moving a feature within the Cursor Plane – press the SHIFT key and left click on the feature, then drag it to its new location. The feature will be moved within the Cursor Plane. If more than one feature is selected, all features will be moved parallel to the plane of the last selected feature (i.e. the feature to which the Cursor Plane has snapped).

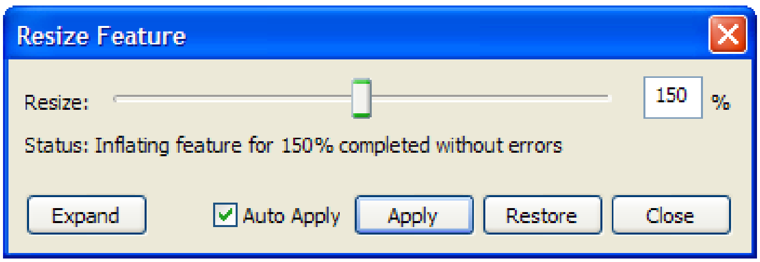
Offsetting a feature with respect to its Cursor Plane – press the CTRL+SHIFT keys and left click on the feature, then drag it to its new location.. The feature will be positioned and remain parallel to its initial orientation, i.e. it will be offset. If multiple features are selected, they will be offset to the plane of the last selected feature.

Deleting a feature – select a feature object or objects and, press the keyboard DEL key to remove a feature. Ensure you do not select a node (this will delete the node instead). Alternatively, right click on the feature (not a node) and select the **Delete** option from the shortcut menu.

Object Cut, Copy, Paste and **Delete** commands are also available from the *Features Menu*.

Resizing a feature – with a single feature selected and editable, select the **Resize** command from the **Features>Edit** menu.

Note Resizing can only be performed on polylines and polygons, but not open or closed polyhedral surfaces.



The Resize Feature dialog provides both a slider bar and manual entry to specify the new feature size, as a percentage of its current size. Press **Apply** to visualise the new size (this can be dynamic if the Auto Apply option is enabled). The original feature size can be restored (if the dialog has not been closed) by pressing the **Restore** button. A number of statistics and parameters can be examined by selecting the **Expand** button.

Editing individual nodes/vertices:

The following controls apply to nodes or vertices in the following feature types; polylines, polygons and open and closed triangulated surfaces. These tools give the user the ability to precisely experiment with and refine the shape of their models, either intricately (on a node by node basis) or regionally (using the elasticity option).

These functions require the target feature to be selected and have **Reshape** enabled

Repositioning a node within the cursor plane – placing the cursor over a node will change the cursor to the **Move** cursor (shown left). Click and drag the node to its new location within the Cursor Plane.

Repositioning a node perpendicular to the cursor plane – placing the cursor over a node will change the cursor to the **Move** cursor (shown left). Press the CTRL key then move the node as required. This functionality will not operate on polygons: use the **Triangulate** button to convert the polygon into a triangulated surface, then move the node as desired

Add a node – Ensure that the **Add Mode** button is enabled in the Feature Editing toolbar. Place the cursor over a segment (between two nodes) on a polyline or over an edge of a polygon or triangulated surface will change to cursor to an **Add Node** cursor. Click to add a node at this location; this node can then be repositioned as above. Alternatively, right-click when the cursor is over the segment, and choose **Insert Point** from the shortcut menu.

Triangulated feature surfaces will be automatically re-triangulated to accommodate the new vertice.

Delete a node – either:

- Select a node by clicking on it (the Cursor Plane cross-hairs will converge on the selected node) and press the keyboard DEL key or
- Right click on the node and choose the **Delete Point** option from the shortcut menu or
- Select the **Delete Node** button from the Features Editing toolbar, and click on the node to delete. this option allows multiple nodes to be deleted quickly. Note that whilst in Delete mode, new nodes cannot be added, nor existing nodes moved

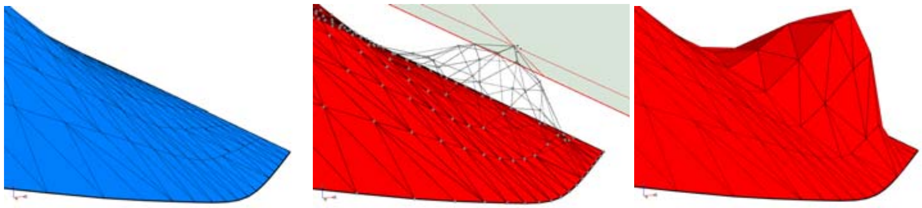
Upon deleting a vertice in a feature surface, the surface will be automatically re-triangulated

Elastic movement of nodes – Moving a node whilst the **Elasticity** button is enabled will move its neighbouring nodes. This is a powerful way of reshaping entire regions of features, whether modifying the edges of a polygon or an area within a triangulated surface. Elasticity can be applied to node movement both within or perpendicular to the cursor plane.

The Radius of this effect is set via the options button adjacent to the **Elasticity** button. This dialog also contains options controlling how the affected nodes are moved: this effect is represented graphically next to each option (Gaussian, Square, etc).

Append - An existing polyline can be extended/continued by selecting an end node, right clicking, and choosing the **Append** option. Simply start drawing the polyline extension required, and finish the polyline when required.

This can be a useful way to edit an existing line in combination with the **Break** function (see *Advanced Editing Functions*).



Elastically reshaping a region of a triangulated surface using the Sine distribution applied over a user-specified radius of effect."]

The **Ruler** tool on the Main toolbar can be a useful guide to distances, bearings and dip angles between objects on the cursor plane.

Editing node XYZ coordinates manually - The node coordinates can be edited within the **Information Sheet>Features** tab. Select the feature to edit, and in the spreadsheet, right click on it and select **Start Edit** from the pop-up menu. Select the node to edit either from the spreadsheet list or graphically, and alter the X, Y or Z coordinates as desired. Note that as feature polygons and polylines must be planar, there is a tolerance limit on how far nodes can be moved via this method: Discover 3D will perform a check to ensure that the new node position retains the features planar state. If it does not, the newly entered value will be ignored.

Advanced Editing Functions

A range of powerful multiple feature processing tools are available on the **Features Editing** toolbar.

By combining these advanced feature maniplulations with Discover 3D’s fast automated wireframing (see *Modelling Triangulated Surfaces and Solids*) and extrusion (see *Extruding Models from Points, Lines and Polygons*) capabilities you can easily experiment with creating multiple, complex geological interpretations to test your theories or evaluate your information in as much, or as little, detail as you need.

This following advanced editing and processing tools are available:

- *Triangulate* - triangulates selected objects to form TIN surfaces.

- *Combine* - combines or fuse multiple selected features into a single feature; the original features' geometries are lost.
- *Intersect* - outputs the intersection of any selected features (except points).
- *Cut* - cuts the first selected feature along the intersections with any other selected features.
- *Aggregate* - combines multiple selected features into one feature, but preserves each source feature's spatial geometry.
- *Disaggregate* - ungroup or explode aggregated features into individual features. Also detects any disconnected parts of an feature (created using the Break tool) and creates individual feature objects for each part.
- *Break Mode* - breaks a polyline into multiple parts at the selected node, as well as select multiple edges along a surface to preform a cut operation.
- *Consolidate* - recombines a selected multi-part feature object

Triangulate

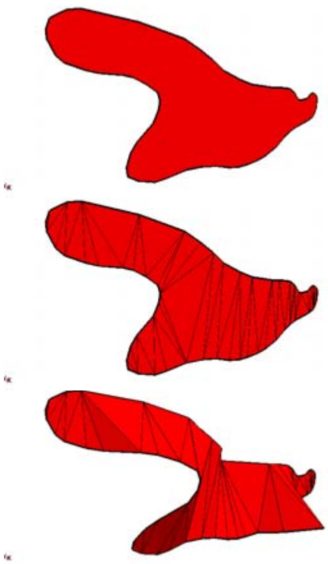
Triangulate will convert any selected feature objects, such as 2D polygons, polylines or points; into a 3D triangulated surface (a Triangulated wireframe mesh or TIN). This function is an essential tool to:

- Convert 2D planar polygons into a 3D Polygon equivalent (a Surface). This enables moving nodes outside of the polygon plane and inserting internal nodes.
- Convert 3D polylines into a surface.
- Convert a point cloud of elevation measurements into a surface (TIN).

The last two functions can also be achieved using surface gridding (see *Creating Gridded Surfaces*), however, this produces a gridded surface file which then needs to be imported into a Feature Database. Instead, the Triangulate option provides a direct vector wireframe surface result.

One prime example of the use of this function is the 3D modification of imported drillhole section boundary polygons from a 2D drillhole project. These polygon features cannot be modified outside of the polygon plane, but the geoscientist may wish to snap these to the actual drillhole locations in 3D.

(a)
(b)
(c)



A feature polygon: (a) triangulated into a feature surface (b) Individual nodes or even regions of the feature surface can then be moved outside the original polygons plane (c) using the vertice/node editing tools and/or the elasticity controls.

Converting these polygons using the triangulate tool allows the user to start accurately snapping not only the edge nodes to drillhole intervals, but also to add internal nodes and snap these to drillholes. Both node snapping operations can be performed elastically allowing a specified region around each node to be very easily and realistically shaped to conform to the new node posiiton.

Combine

Combine will combine any selected features (points, polylines, polygons, surfaces and volumes) into a single feature. This is a useful for managing more complex feature databases e.g. combining features with the same attributes into a single feature, allowing easy graphical selection of a particular attribute type.

This function fuses the selected features so that overlapping boundaries and regions are lost; and the original features' spatial geometries are lost.

To combine features:

1. Make the feature database containing the features editable.

2. Using the **Select** tool, select the first feature.

3. While pressing the keyboard CTRL key, select the additional features.

4. Press the **Combine** button.

5. In the Operation Options dialog, select the output feature database to create the single combined feature in. Ensure that the **Delete original features** option is disabled if you want to preserve the input features. Press OK.

Intersect

Intersect creates new features representing the shared (or common) areas of the selected features. This can be the intersection of any selected feature polyhedrons (surfaces and volumes) or polygons. Some examples of use:

- The portion of a fault plane that intersects an orebody.
- The lineation intersection of a quartz vein with a fault surface.
- The surface trace of a fault plane (interpreted from drilling) with a topographic surface (a DEM grid imported into a feature database, see *Importing Data into a Feature Database*).

To display the intersection of one TIN feature surface with another:

1. Make the feature database containing the intersecting features editable

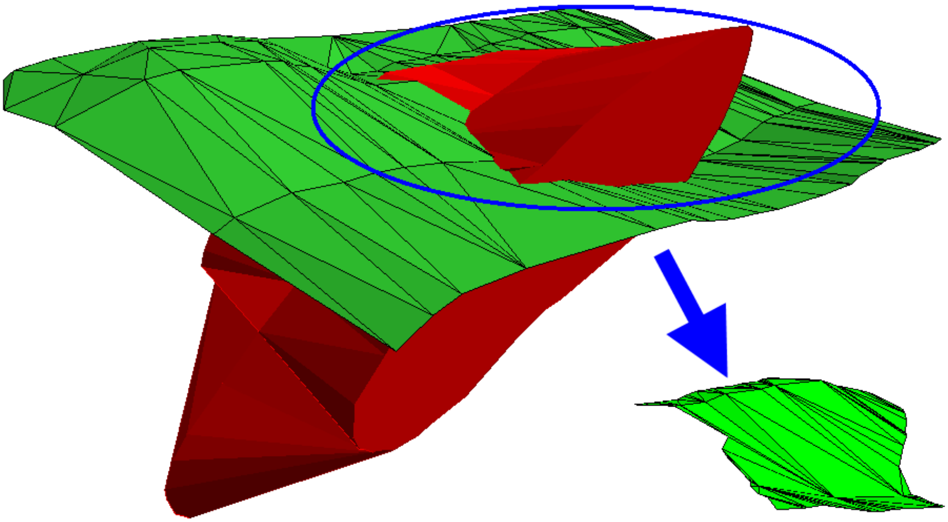
2. Using the **Select** tool, select the first feature.

3. While pressing the keyboard CTRL key, select the additional features.

4. Press the **Intersect** button.

5. In the Operation Options dialog, select the output feature database to create the intersection feature in. Ensure that the **Delete original features** option is disabled if you want to preserve the input features. Press OK.

It is recommended that any bodies being intersected are enclosed bodies (i.e. if created with the 3D Solid Generator (see *Modelling Triangulated Surfaces and Solids*), ensure an end capping is applied such as ‘flat’). This will result in the output intersection being displayed as a triangulated surface, rather than a closed polyline.



An example of an Intersection operation

Cut

Cuts the first selected feature along intersections with any other selected features. All feature types except points are supported. Cut can be used for any of the following operations.

Selected features in the same plane:

- Cut (or remove) part of a polygon feature where it overlaps with other polygon feature(s)
- Cut a polygon feature along the intersection with other polyline feature(s)

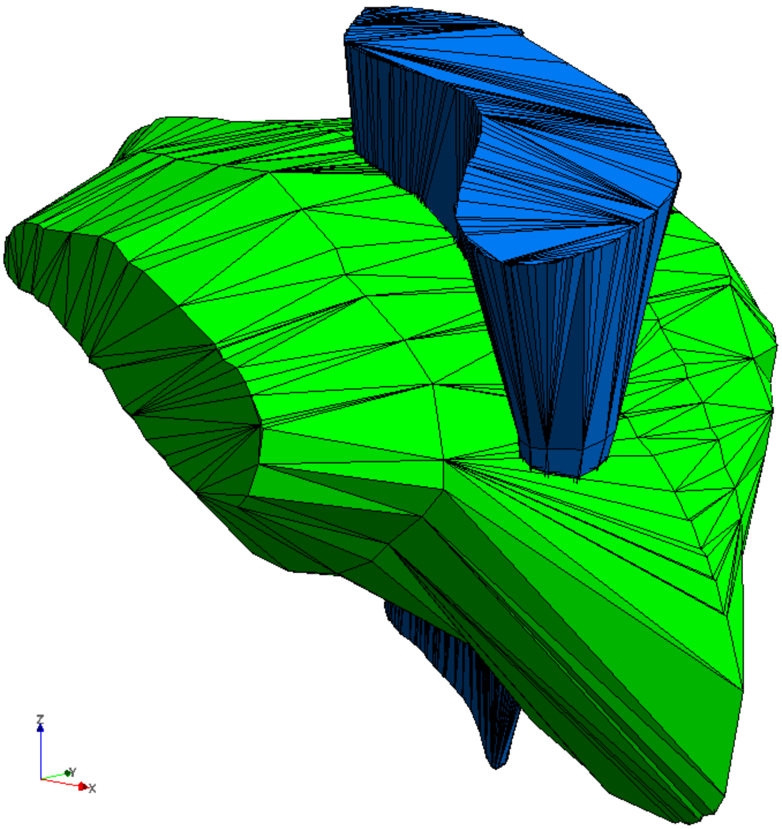
Selected features in different planes:

- Cut a polygon feature along intersections with other polygon feature(s)
- Cut a TIN surface along intersections with other TIN surface(s)

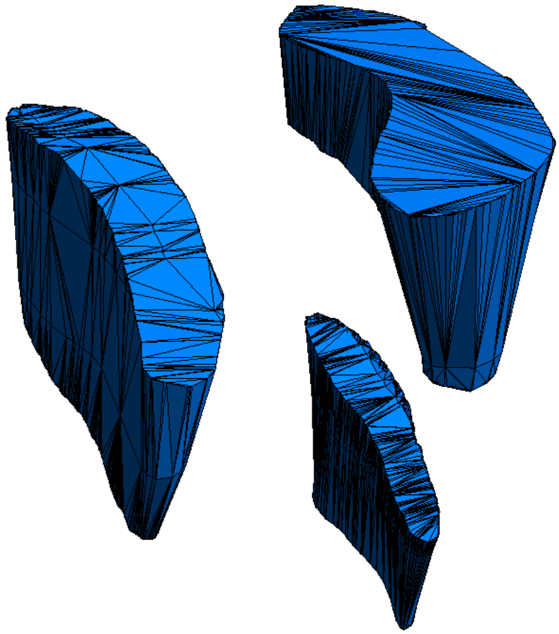
Some examples of the powerful modelling uses of this function:

- Viewing the portion of an orebody volume solid (created with the *Modelling Triangulated Surfaces and Solids*) above a proposed open cut surface (a DEM grid of the open cut imported into a feature database, see *Importing Data into a Feature Database*).

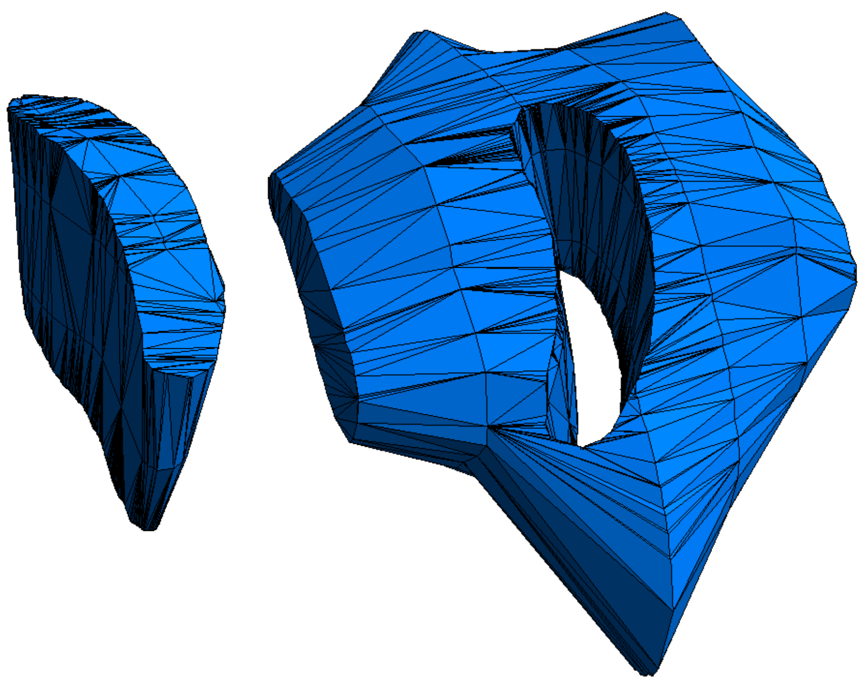
- Viewing the portion of an orebody volume solid between two fault planes, then calculating the volume of this region.
- Cutting a older fault plane with a more recent fault plane (e.g. extruded from surface traces using the *Extruding Models from Points, Lines and Polygons*).



Some examples of cut operations with the two solids shown above follow:



Result of cutting the blue solid with the green solid.



Result of cutting the green solid with the blue solid.

To cut one feature with another:

1. Make the feature database containing the features editable (the features must all be in the same feature database).

2. Using the **Select** tool, select the target feature to be cut.

3. While pressing the keyboard CTRL key, select the second feature (i.e. the cutting surface).

4. Click the **Cut** button (or use **Features>Edit>Cut**).

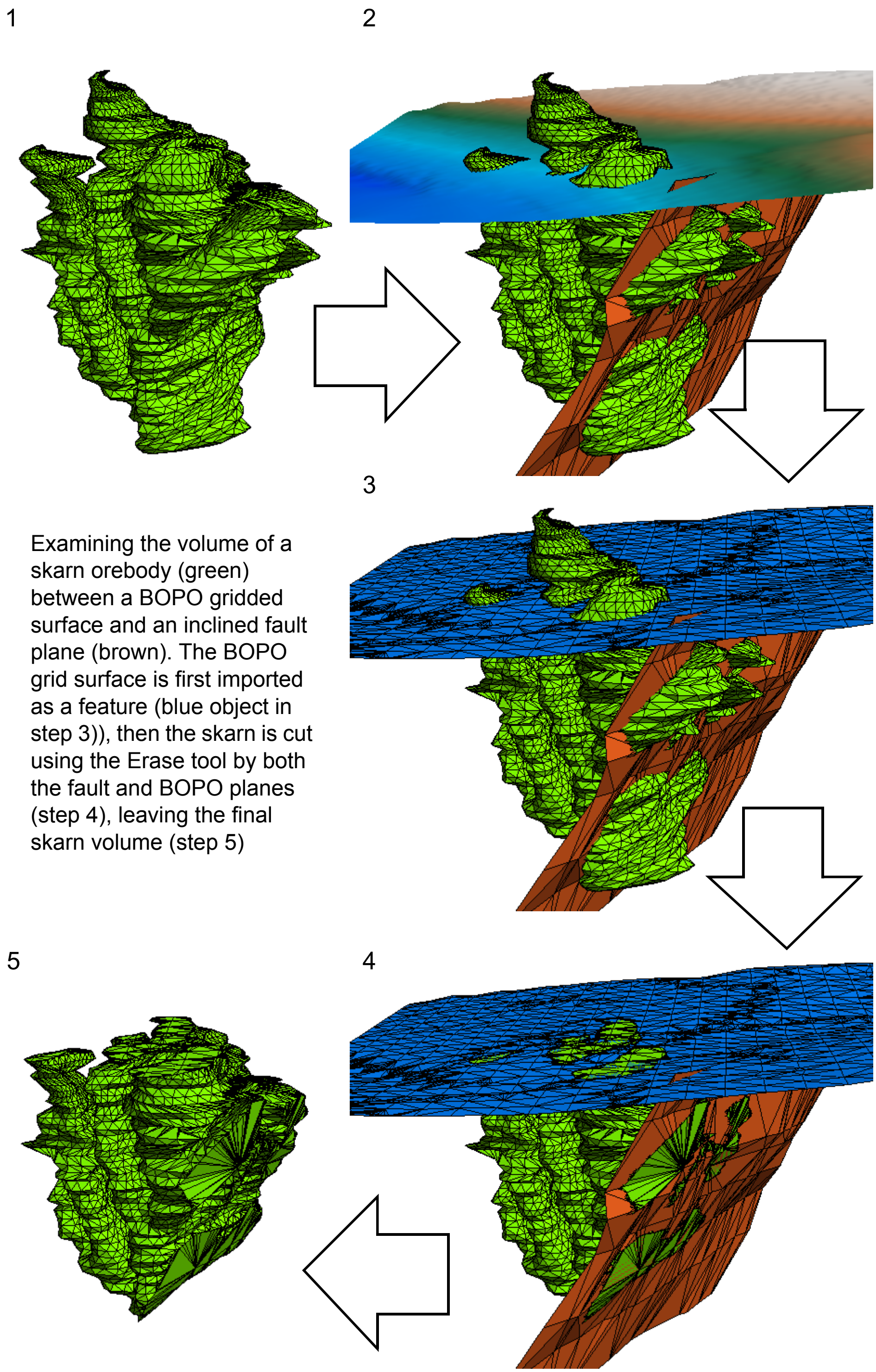
5. In the **Operation Options** dialog, select the output feature database to create the Cut feature in.

   Ensure that the **Delete original features** option is disabled if you want to preserve the input features.

6. Press OK.

Note The output feature database will likely have two or more features (the target feature cut into sub-portions). Select and Delete the unwanted features.

Note It is recommended that any bodies being cut are enclosed bodies (i.e. if created with the 3D Solid Generator (see *Modelling Triangulated Surfaces and Solids*), ensure an end capping is applied such as ‘flat’). This will result in the output body being displayed as a closed volume, rather than a open/hollow surface; a volume can then be calculated for a closed volume using the *Topology Checker*.



Examining the volume of a skarn orebody (green) between a BOPO gridded surface and an inclined fault plane (brown). The BOPO grid surface is first imported as a feature (blue object in step 3)), then the skarn is cut using the Erase tool by both the fault and BOPO planes (step 4), leaving the final skarn volume (step 5)

Aggregate

Aggregate groups multiple selected features into a single feature, whilst preserving the spatial geometry of all the original features.

To aggregate multiple features:

1. Make the feature database containing the features editable.
2. Using the Select tool, select the first object.
3. While pressing the keyboard CTRL key, select the additional features.
4. Press the **Aggregate** button.
5. In the Operation Options dialog, select the output feature database to create the single resulting feature in. Ensure that the **Delete original features** option is disabled if you want to preserve the input objects. Press OK.

Disaggregate

Disaggregate ungroup or explode aggregated features into individual features. Also detects any disconnected parts of an feature (created using the Break tool) and creates individual feature objects for each part.

To disaggregate a feature:

1. Make the feature database containing the target features editable.
2. Using the Select tool, select the target feature.
3. While pressing the keyboard CTRL key, select any additional features if desired.
4. Press the **Disaggregate** button.
5. In the Operation Options dialog, select the output feature database to create the resulting multiple features in. Ensure that the **Delete original features** option is disabled if you want to preserve the input objects. Press OK.

Break Mode

There are two modes of operation:

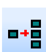
- Break a polyline into multiple parts at the selected node
- Manually cut a triangulated feature surface along mutiple contiguous selected edges/segments.

To break a feature polyline:

1. Make the feature database containing the feature polyline editable.

2. Using the Select tool, select the target feature and make it **Reshapable**.

3. Hover the cursor over the node at which you wish to break the polyline: if no node exists, first insert a node. The cursor will change to a knife symbol when over a node. Click the mouse. This can be repeated for multiple nodes if desired



4. Select the **Disaggregate** button.

5. In the Operation Options dialog, select the output feature database to create the resulting multiple feature polylines in. Ensure that the **Delete original features** option is disabled if you want to preserve the input objects. Press OK.

Optional steps:

- If necessary, delete the unrequired/surplus feature polylines
- If required, the endpoint of one of the new features can now be extended from, by right clicking on it at choosing the Append option

To break a triangulated feature surface:

1. Make the feature database containing the feature surface editable.

2. Using the Select tool, select the target feature and make it **Reshapable**.

3. Enable the **Break mode** on the Feature Editing toolbar.

4. Hover the cursor over the first interior edge delineating the desired cut line; the cursor will change to a knife symbol. Click. The edge will be bolded to indicate its selection.

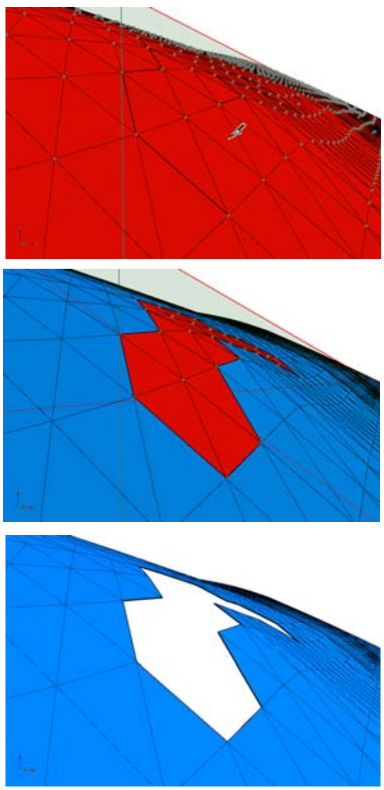
5. Repeat step for for each additional interior edge until a continuous line from one side of the surface to another has been traced, or an internal region outlined (see example below).

6. Select the **Disaggregate** button.

7. In the Operation Options dialog, select the output feature database to create the resulting multiple feature polylines in. Ensure that the **Delete original features** option is disabled if you want to preserve the input objects. Press OK.

Optional step:

- If necessary, delete any unrequired/surplus feature polygons/surfaces



Example of a triangulated surface having an internal region clipped using the Break function

Consolidate

Consolidate is primarily used to recombine features that have been broken into multiple parts with the Break tool.

To recombine a multi-part feature:

1. Make the feature database containing the feature editable.
2. Using the Select tool, select the multi-part feature (as created in *Break Mode*).
3. Select the **Consolidate** option from the **Features>Edit** menu

Feature Attributes

Feature attributes can be examined via either the Feature or Data Windows; individual feature attributes can also be edited in the Feature Database Window.

Feature attributes will include the following default fields, plus any additional user-created fields (and attributes). These default fields are automatically updated after any object edit/change:

- **XYZ** coordinates of the object's centroid.
- A user-defined **Description** field.
- **Type** of object (see *Feature Object Types*).
- **Length** of the object (polyline length, perimeter length for polygons and surfaces).
- **Area** of the object (one side of polygons and surfaces only, exterior surface area of solid/polyhedron volumes).
- **Volume** of the object (closed solid/polyhedron volumes only).

These windows are accessed via the icons on the Main Toolbar and the View menu. More information on accessing and positioning these windows can be found in *Docking and Undocking Toolbars and Windows*.

Data Window

The **Data** window displays the attributes of only the selected feature (the Feature Database must therefore be **Selectable** in the Workspace Tree; if the database is **Editable**, multiple features can be selected and displayed). It does not require you to choose the database within the tab to display; therefore multiple databases can be made selectable and can be easily and rapidly interrogated. However it does not allow attribute editing.

Alternatively, the **Feature Information** button (on the *Features Toolbar*) can be used to select features regardless of whether their database is selectable or not.

Feature Database Window

The **Feature Database** window displays a spreadsheet view of the attributes of every feature in a selected Feature Database. This allows multiple feature attributes to be examined and edited after creation; alternatively, individual features can be attributed during creation by enabling the **Confirm Pick** button on the *Features Toolbar*.

Information Sheet

Feature Set: Challenger Geology   Show selected features only

	ID	X	Y	Z	Value	Description	Type	Geology
1	1	311607.23	6400667.9	-34.241321	-34.241321		Polygon	Supergene
2	2	311562.62	6400656.1	-75.323087	-75.323087		Polygon	Supergene
3	3	311510.14	6400604	-13.147517	-13.147517		Polygon	Supergene
4	4	311451.94	6400560.3	44.472317	44.472317		Polygon	Supergene
5	5	311502.26	6400485.7	-15.308019	-15.308019		Polygon	Mineralisation 1
6	6	311584.99	6400519	-75.228429	-75.228429		Polygon	Mineralisation 1
7	7	311644.08	6400561.3	-151.25756	-151.25756		Polygon	Mineralisation 1
8	8	311811.26	6400677.2	-377.21512	-377.21512		Polygon	Mineralisation 1
9	9	311729.83	6400798	-314.12463	-314.12463		Polygon	Mineralisation 2
10	10	311680.03	6400741.9	-315.45219	-315.45219		Polygon	Mineralisation 2
11	11	311551.81	6400698.1	-167.7191	-167.7191		Polygon	Mineralisation 2

If a group of cells is highlighted in the feature spreadsheet, right-clicking will display the **Copy selected cells to clipboard** menu option. This can be useful for extracting data into Excel for data anlaysis or other uses.

Feature Selection

Select the required Feature Database from the pull-down list at the top of the **Feature Database** window. The attributes for all default and custom fields will be displayed. With **Select** mode enabled (shown left), click within a row to highlight (in blue) a record; its corresponding feature will be highlighted in the 3D window. Multiple features can be chosen using the SHIFT and CTRL keys.

If the selected Feature Database is **Editable** in the Workspace Tree, selecting a feature within the 3D display will highlight the corresponding records in the Feature Spreadsheet tab. If the **Show selected features only** option is enabled at the top of the Feature Spreadsheet view, only records for the currently selected features will be displayed in the spreadsheet.

Attribute Editing

Individual attributes of an **Editable** Feature Database can be edited directly within **Feature Database** window cells (including the X, Y and Z values). Right clicking within the attribute spreadsheet displays a shortcut menu, providing options for automatic heading and/or data spacing operations: note that these spacings can also be set manually by selecting and moving the header column and row widths. Right-clicking on a column header provides additional ascending or descending sort operations for the selected column.

Selected rows can be deleted by either pressing the keyboard DELETE button, or by right-clicking on the row header and choosing the **Delete Selection** option. This shortcut menu also contains options to **Cut**, **Copy** and **Paste** the current selection, allowing feature duplication (duplicated features could then be offset using the controls discussed in *Editing Features*).

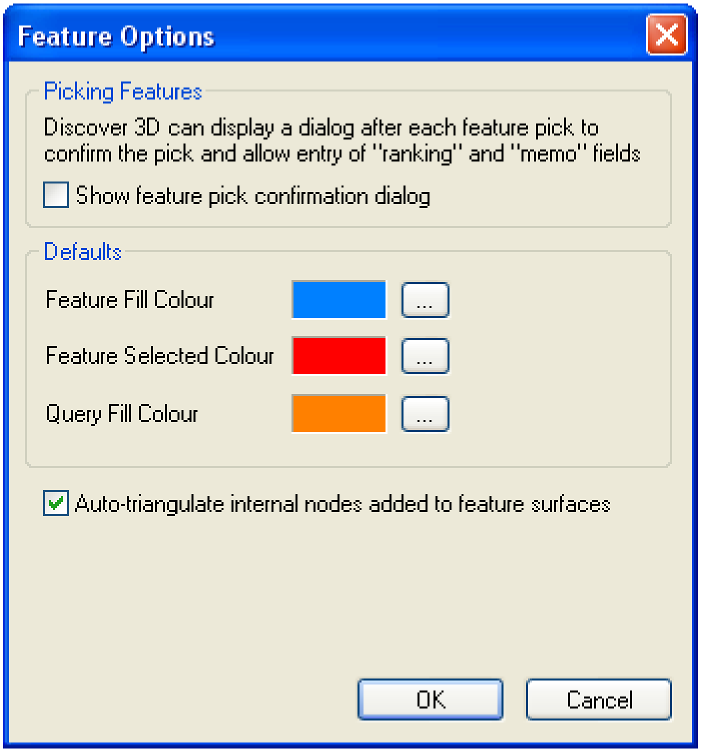
Node Editing

The X,Y and Z values in the attribute spreadsheet only represent the centroid locations for polyline and polygon features: to edit individual node co-ordinates, right click within a selected row, and select the **Start Edit** option from the shortcut menu. An editable list of the co-ordinates for every node comprising the feature will be displayed: selecting a row entry will snap the crosshairs in the 3D display to corresponding node. Conversely, selecting a node within the 3D display will highlight the corresponding row entry.

Right-clicking within this mode will display a shortcut menu with three mode-specific options:

- **Insert Node**: inserts a new node below the currently selected node, populated with the currently selected nodes coordinates,
- **Delete Point**: deletes the selected node, or
- **Stop Edit**: finishes node editing and returns the view to the attribute spreadsheet.

Feature Options



Feature Options dialog

The **Features>Features Options** menu item enables the following default options to be set:

1. Turn on/off the feature pick confirmation dialog which is displayed at the end of the creation of each feature. This dialog enables the entry of data into the default Description field and any other user-defined fields in the Feature Database. This is identical to the **Confirm Pick** button in the *Features Toolbar*.

2. Set default colour for feature polygon **Fill**.

3. Set default colour for **Queried** feature.

4. Control whether feature surfaces are auto-triangulated when nodes are added to internal edges (Default on).

Note It is recommended to have this option enabled; disabling it is only recommended for advanced users who wish to add a number of internal nodes and then perform a triangulation incorporating all new nodes simultaneously, resulting in a subtly different triangulation geometry (compared with triangulation performed after each node is added).

17 Displaying Voxel Models

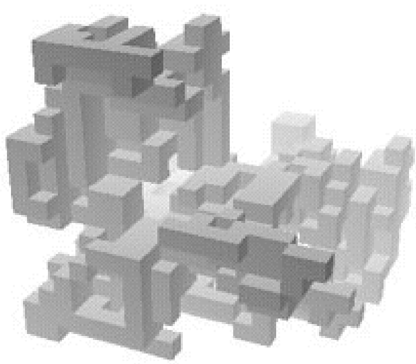
Discover 3D allows voxel models (also known as 3D meshes, block models or 3D grids) to be displayed with a variety of options, such as slices, threshold range or isosurfaces. Voxel models can be imported from third-party software such as mine simulation or inversion packages, or created in the Discover 3D (see *Creating and Manipulating Voxel Models*).

- *What are Voxel Models?*
- *Supported Voxel Model Formats*
- *Creating a 3D Voxel Model Display*
- *Importing and Exporting Voxel Models*
- *Viewing Voxel Model Information*
- *Changing Voxel Model Display Properties*
- *Displaying a Floating Colour Legend*
- *Displaying Multiple Voxel Models*
- *Integrating Voxel Model Displays with Other Objects*

What are Voxel Models?

The term voxel refers to a volume element and is the three-dimensional equivalent of the two-dimensional grid cell. As used in geological or geophysical modelling, voxel models represent volumes of the earth which are subdivided in a regular way into sub-volumes, or cells. Each cell, created from six sides, contains an earth volume of uniform property attribute. Such properties as magnetic susceptibility, density, conductivity or IP property (eg chargeability or phase) can be used. Certain software products (such as Vulcan, Gemcom, Surpac) and geophysical software from inversion applications (eg UBC) can create 3D voxel files.

Note Discover 3D can also create 3D voxel model outputs, for example for a 3D drillhole dataset (see *Creating and Manipulating Voxel Models*).



Example of complex voxel shape

Supported Voxel Model Formats

The following exclusions apply to all supported voxel model formats:

- Rotated models are not supported, except for native Encom 3D grid format. The rows/columns of the model need to be parallel to the X and Y axis. If you wish to import a rotated model, an ASCII (CSV) version of the file must be obtained, and then converted to an Encom3D grid using the **Generic ASCII Import** (*Importing ASCII Voxel Models*).
- The model must be continuous with no missing cells, and include all null values. This includes cells and padding around the edge of the data. For example if the model is 10 rows by 10 columns by 10 planes, then there must be 1000 cells defined. If you wish to import a model which is missing null values, an ASCII (CSV) version of the file must be obtained, and then converted to an Encom3D grid using the **Generic ASCII Import** (*Importing ASCII Voxel Models*).
- Terrain or DEM offset is not supported. The planes must be parallel.

Discover 3D currently has direct read support (see *Importing and Exporting Voxel Models*) for five voxel/block model formats:

- UBC (University of British Columbia)
- CEMI (Consortium of Electromagnetic Modelling and Inversion – University of Utah)
- ASCII XYZ Simple (centred and uniform mesh only)

- Encom3D Grid (native format)
- Noddy (developed under AMIRA grant by Dr Mark Jessell at Monash University) formats

Note Generic ASCII model formats that contain variable sized cells or non-centred coordinates or are non-continuous or are rotated can be imported and converted to the native Encom3D format using the **Grids>Voxel Manager**. See *Importing ASCII Voxel Models*.

Additionally, Discover 3D has limited support for the following voxel formats:

- Datamine – single precision only.
- Gemcom – Gemcom cell data types are supported: 16-bit integer, 32-bit integer and 32-bit floating point.
- Geosoft Voxel.
- GoCAD Voxel.
- Micromine block model - ASCII file and single precision only. Sub-blocks are not supported.
- Surpac – version 3 supported, with preliminary support for versions 1 and 2. The following Surpac cell data types are supported: 32-bit integer, 64-bit real and string.
- Vulcan – single and multi-parameter support.

The files used as input to the Voxel Model are the result of modelling and inversion simulation software programs. In some cases the Voxel Model can be used to display the input models of the research programs or, alternatively, the output inversion results of the programs. The supported formats for UBC, CEMI and Noddy are ASCII with relatively simple specifications. For more inforamtion about these formats, see:

- *UBC Model Format*
- *CEMI Models*
- *Noddy Models*
- *ASCII XYZ Models*

UBC Model Format

The UBC model requires two separate ASCII files. These are:

- Mesh file – 3D mesh defining the discretisation of the 3D model region, and
- Property file – the assignment of the property (eg magnetic susceptibility, density etc) for each cell of the model.

Mesh File

This file contains the 3D mesh which defines the model region. **Mesh** has the following structure:

```
NE NN NV
Eo No Vo
ΔE1 ΔE2 … ΔENE
ΔN1 ΔN2 … ΔNNE
ΔV1 ΔV2 … ΔVNE
```

where:

`NE` Number of cells in the East direction
`NN` Number of cells in the North direction
`NV` Number of cells in the vertical direction

`Eo No Vo` Coordinates, in metres, of the southwest top corner, specified in Easting, Northing and Elevation. The elevation can be relative, but it needs to be consistent with the elevation used to specify the observation position in additional location files (called OBS.DAT and OBS.LOC).

`ΔEn` Cell widths in the easting direction (from West to East)
`ΔNn` Cell widths in the northing direction (from South to North)
`ΔVn` Cell depths (top to bottom)

The mesh can be designed in accordance with the area of interest and the spacing of the data available in the area. In general, the mesh consists of a core region which is directly beneath the area of available data, and a padding zone surrounding this core mesh. Within the core mesh, the size of the cells should be comparable with the spacing of the data. There is no restriction on the relative position of data location and nodal points in the horizontal direction.

An example of a mesh file where each cell is 50m by 50m by 50m in size:

```
10 10 5
0 0 0
50.0 50.0 50.0 50.0 50.0 50.0 50.0 50.0 50.0 50.0
50.0 50.0 50.0 50.0 50.0 50.0 50.0 50.0 50.0 50.0
50.0 50.0 50.0 50.0 50.0
```

Property File

To illustrate and describe the properties of a model, this example uses Magnetic Susceptibility, but all UBC property files are of this format irrespective of the property being displayed. The following details the file structure:

$sus_{1,1,1}$
$sus_{1,1,2}$
$sus_{1,1,NV}$
$sus_{1,2,1}$
$sus_{i,j,k}$
$sus_{NN,NE,NV}$

where:

$sus_{i,j,k}$ is susceptibility at location i,j,k

[i j k] = [1 1 1] is defined as the cell at the top south-west corner of the model. The total number of lines in this file should equal NN NE NV, where NN is the number of cells in the North direction, NE is the number of cells in the East direction, and NV is the number of cells in the vertical direction. The lines must be ordered so that `k` changes the quickest (from 1 to NV), followed by `j` (from 1 to NE), then followed by `i` (from 1 to NN). If the surface topography (Topo.DAT) file is supplied, the values above the surface will be ignored. These values should be assigned –1.0 (nulls) to avoid confusion with the other model elements.

CEMI Models

Voxel models used by the Consortium of EM Inversion (University of Utah) are similar in format to those of UBC. The CEMI format uses a separate header file to describe inverse model parameters. Separate values of the key parameters are used to specify the required spatial and model properties. The key parameter begins with the "#" character in the first position of the line, and the values follow till the next "#" character. The example below includes the required parameters (dimensions and steps in the X, Y, and Z directions). The values in the model file may be resistivity, density or magnetic susceptibility and so on.

Example of MYMODEL.CEM header:

```
!CEMI Model Header - 27th Feb 2003 (Software Version 2.3)
#xdim
    50
#ydim
    40
#zdim
    12

#xcell          ! X cell widths
    50
#ycell          ! Y cell widths
    50
#zcell          ! Z cell widths (variable)
50    50    100    100    100    100    100    100    100    100
100   100
#invpar
    stg = 1:5;    % List of stages to execute
    wordy = 2;    % Messages: 0-silence, 2-very wordy
    mfit = 0.03;    % level of fit in the inversion
    srcpar{1} = 1    % source parameters
    sig0 = [1/305 1/283 1/452 1/461 1/607 1/452 1/556]; % conductivity
of layers of the normal section
    hh0 = [100 200 200 100 100 200]; %  of layers of the normal section
(m)
    an0 = [1 1 1 1 1 1 1];   % vector of background layer anisotropies
    x = 1000:100:6500;  % Cell center x-coordinates of the inverted
area (m)
    y = 3200:100:8100;  % Cell center y-coordinates of the inverted
area (m)
    z = [25 75 150 250 350 450 550 650 750 850 950 1050];  % Cell
center z-coordinates of the inverted area (m)
    dz = [50 50 100 100 100 100 100 100 100 100 100 100];  % Vertical
cell sizes (m)
    iflag = 2    % Inversion flag: 1-Born 2-DQA 3-QA
    ma = 0    % A priori model
    Nit = [50 20 10]    % Number of iteration at each stage of focusing
    uconst = 10000    % Upper constraint of anomalous conductivity
    lconst = 0.001;    % lower constraint of anomalous conductivity
    Pfsteep = 0.01    % Parametric functional steepness
    qalpha = 0.5    % Updating multiplier for regularization
parameter
    amlt = 0.1    % Updating multiplier for regularization parameter
    qmlt = 0.95    % Updating multiplier for regularization parameter
    keyq=0
```

```
    KEYZ=1
#comment
    iterstage=3
    Conductivity
    Example inversion results
```

The MYMODEL data file has the structure of a set of four columns defining an order of data in any line of X, Y, Z, Value

```
3.2000000e+003  1.0000000e+003  2.5000000e+001  3.0490700e+002
3.2000000e+003  1.1000000e+003  2.5000000e+001  3.0490700e+002
3.2000000e+003  1.2000000e+003  2.5000000e+001  3.0490700e+002
3.2000000e+003  1.3000000e+003  2.5000000e+001  3.0490700e+002
3.2000000e+003  1.4000000e+003  2.5000000e+001  3.0490700e+002
```

Usually geographical coordinates are used consistent with the input inversion source data, but coordinates could be a local system, even rotated with respect to the geographical coordinates.

Noddy Models

Noddy models can be viewed in the Voxel Modelling facility. Two files are necessary. These are a mesh file (G00) and a property and lithology file (Gnn). The property file reflects the number of different lithological units within the model (eg G12 etc).

An example of a .G00 file is shown below:

```
VERSION = 7.11
FILE PREFIX = Noddy Block Format
DATE = 01/01/05
TIME = 12:00:00
UPPER SW CORNER (X Y Z) = -900.0 -900.0 5001.0
LOWER NE CORNER (X Y Z) = 10800.0 7800.0 201.0
NUMBER OF LAYERS = 16
   LAYER 1 DIMENSIONS (X Y) = 39 29
   LAYER 2 DIMENSIONS (X Y) = 39 29
   LAYER 3 DIMENSIONS (X Y) = 39 29
   LAYER 4 DIMENSIONS (X Y) = 39 29
NUMBER OF CUBE SIZES = 16
   CUBE SIZE FOR LAYER 1 = 300
   CUBE SIZE FOR LAYER 2 = 300
   CUBE SIZE FOR LAYER 3 = 300
   CUBE SIZE FOR LAYER 4 = 300
CALCULATION RANGE = 3
INCLINATION OF EARTH MAG FIELD = -67.00
```

```
INTENSITY OF EARTH MAG FIELD = 63000.00
DECLINATION OF VOL. WRT. MAG NORTH = 0.00
DENSITY CALCULATED = Yes
SUSCEPTIBILITY CALCULATED = Yes
REMANENCE CALCULATED = No
ANISOTROPY CALCULATED = No
INDEXED DATA FORMAT = Yes
NUM ROCK TYES = 8
ROCK DEFINITION = 1
   Density = 2.000000
   Sus = 0.001000
ROCK DEFINITION = 2
   Density = 2.200000
   Sus = 0.001100
```

The file PROPERTY.ROX details the lithology in the Noddy model. An example of its format is:

```
Version = 7.100000
Number of Rocks = 19
   Unit Name   = Amphibolite
   Height   = 0
   Apply Alterations   = ON
   Density   = 2.96e+000
   Anisotropic Field   = 0
   MagSusX   = 6.00e-005
   MagSusY   = 0.00e+000
   MagSusZ   = 0.00e+000
   MagSus Dip   = 0.00e+000
   MagSus DipDir   = 0.00e+000
   MagSus Pitch   = 0.00e+000
   Remanent Magnetization   = 0
   Inclination   =   0.00
   Angle with the Magn. North   =   0.00
   Strength   = 0.00e+000
   Color Name   = Dodger Blue
   Red   = 30
   Green   = 144
   Blue   = 255

   Unit Name   = Andesite
   Height   = 0
   Apply Alterations   = ON
   Density   = 2.61e+000
etc….
```

ASCII XYZ Models

This format is a simple text representation of a voxel model which allows data to be imported from and exported to other software packages. The file starts with a header line containing the names X Y Z followed by the names of the attributes. The format for each subsequent line is X Y Z Attribute1 Attribute2 Attribute3 etc… The X, Y, and Z coordinates are the centre positions of each cell, while the attributes are the numeric data values for each cell. The fields may be separated by either spaces or commas. Every cell in the rectangular prism model must be stored – even null cells. The mesh must be regular – i.e. cell widths, heights and depths are uniform across the model. The data should be stored row by row and plane by plane. (i.e. X is the fastest changing field, followed by Y, and then Z). Planes can be stored in either an upwards or downwards sequence.

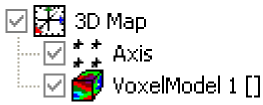
Note Generic ASCII model formats that contain variable sized cells or non-centred coordinates can be imported and converted to a standard format using the **Grids>Voxel Manager**. See *Importing ASCII Voxel Models*.

An example is repesented below:

```
X Y Z susceptibility
-287.5 -287.5 -387.5 0.0001272899972
-262.5 -287.5 -387.5 0.0001027179969
-237.5 -287.5 -387.5 9.764270362e-005
-212.5 -287.5 -387.5 7.809209637e-005
-187.5 -287.5 -387.5 4.203460048e-005
-162.5 -287.5 -387.5 2.547840086e-005
-137.5 -287.5 -387.5 2.742360084e-005
-112.5 -287.5 -387.5 2.43577997e-005
-87.5 -287.5 -387.5 2.702029997e-005
-62.5 -287.5 -387.5 2.904669964e-005
```

Creating a 3D Voxel Model Display

To add a **Voxel Model** to Discover 3D you can either click the **Display Voxel Model** button or select the **Display>Voxel Model** menu item. Both of these operations add a **Voxel Model** branch to the Workspace Tree.

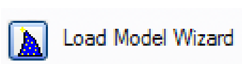


Voxel Models are also produced by the Discover **3D Voxel Gridding Toolkit** (see *Creating and Manipulating Voxel Models*), for example from gridding dirllhole's donwhole assay data.

Importing and Exporting Voxel Models

- *Using the Load Model Wizard*
- *Using the Export Model Wizard*
- *Import and Export Voxel Models with the Grid Management Tool*

Using the Load Model Wizard

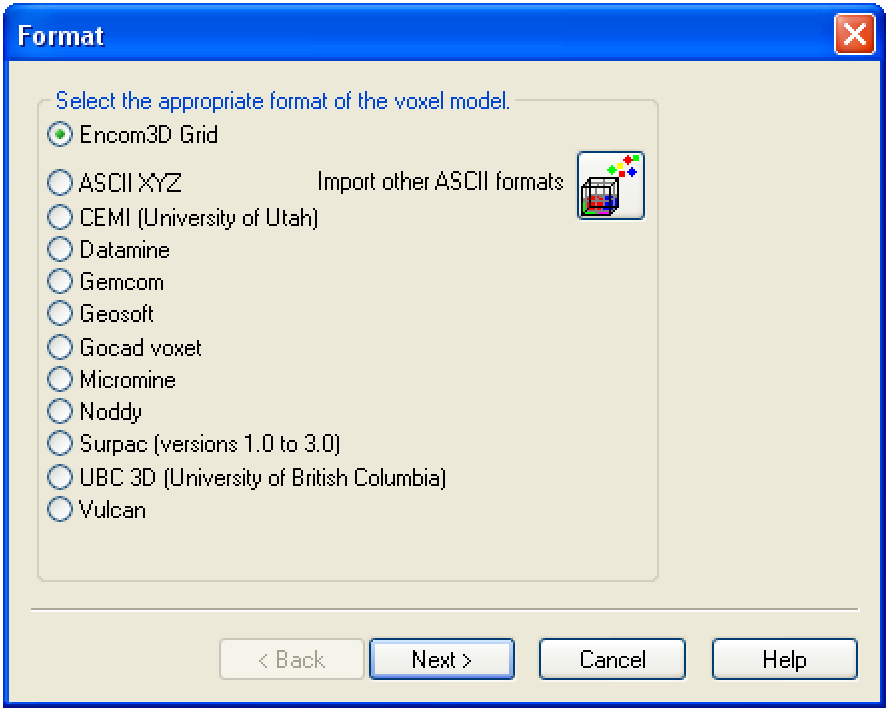


The **Data** tab of the **Voxel Model Properties** dialog contains the **Load Model Wizard** button . This wizard allows the selection of all model types, except for generic ASCII files.

Click **Import Other ASCII Formats** to open the **Grid Manager** tool to convert generic ASCII files into an Encom Voxel model. See *Importing ASCII Voxel Models* for more information.

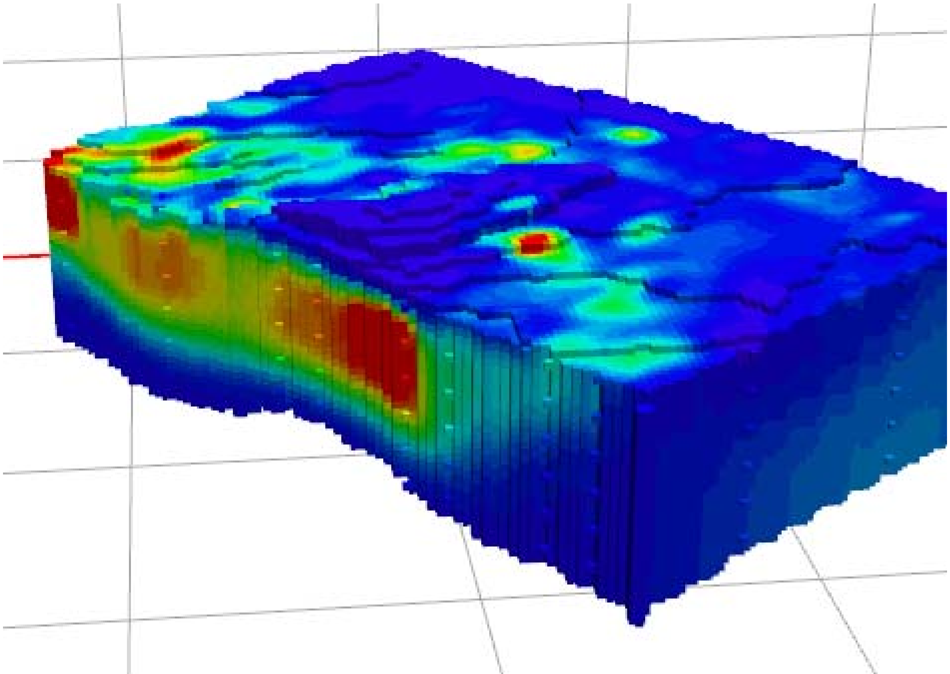
Important Rotated models are not support for any format other than Encom3D Grid. The bounds of the model need to be parallel with the X and Y axis. If you wish to import a rotated model, you will need to obtain an ASCII (CSV) version of it. Refer to *Importing ASCII Voxel Models* to import and convert this rotated ASCII model to an Encom3D Grid.

Note Generic ASCII model formats that contain variable sized cells or non-centred coordinates or is non-continuous can be imported and converted to a standard format using the **Grids>Voxel Manager**.



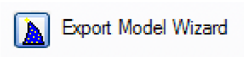
Using the Voxel Model wizard to select the model type and relevant model files

Note that only a single model can be specified at one time and this is listed in the **Data** tab after being specified. If the **Auto-Apply** option (at the base of the dialog) is enabled, upon reading the model file, Discover 3D automatically displays the model.

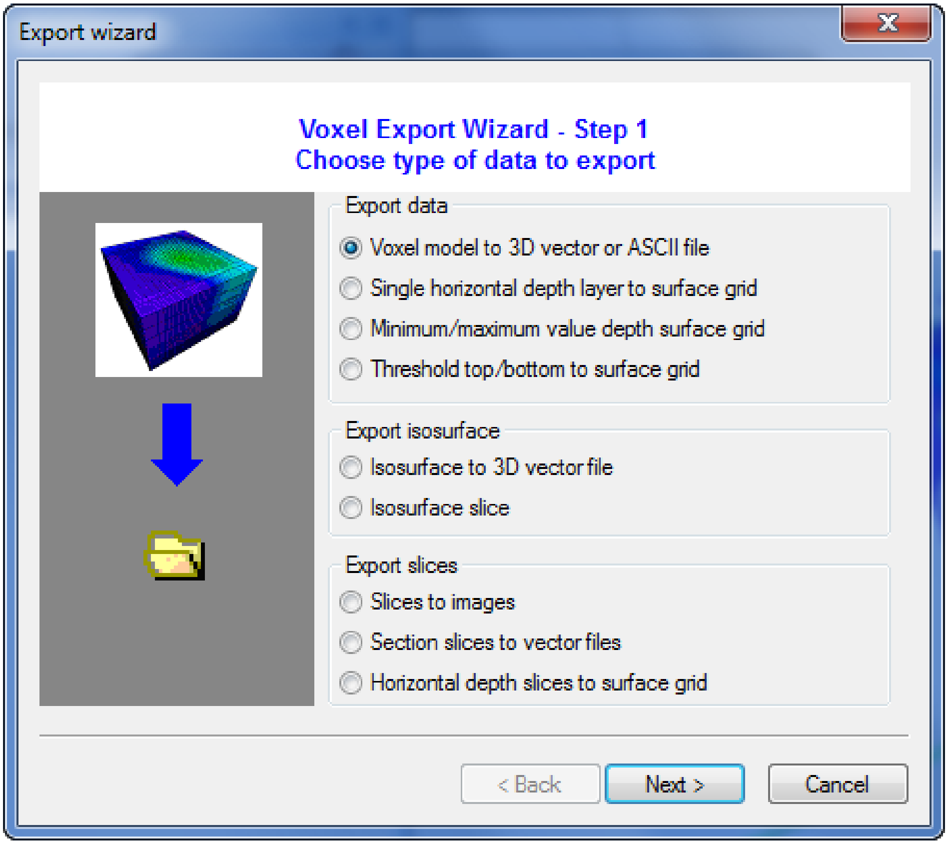


Initial display of the Voxel Model

Using the Export Model Wizard



The **Data** tab also allows the export of **Voxel Models** in the UBC format. The **Export** button (shown left) will open the **Voxel Export Wizard.**



Voxel Model Export Wizard

The export options are:

- **Voxel model to 3D vector or ASCII file** - export the thresholded model to either a DXF vector of individual cells, or a UBC or XYZ model format.
- **Single Horizontal depth layer to surface grid** - export a horizontal (Z) slice as a 2D surface grid
- **Minimum/Maximum value depth surface grid** - This option produces a 2D band surface grid with one band representing the minimum or maximum value vertically in the voxel model, and the second band indicates the Z depth that this occurs in the voxel model.
- **Threshold top/bottom to surface grid** - produces a 2 banded 2D surface grid, with a "top" and "bottom" band defining the Z value of the top and bottom extents of the thresholded model.
- **Isosurface to 3D vector file** - Export an Isosurface to a DXF vector.

- **Isosurface slice** - Export the intersection of an isosurface with defined X/ Y/Z slice planes.

- **Slices to Images** - Export the current slices to an 3D registered EGB images.

- **Section Slices to vector files** - Export the current Discover section slices to vector TAB files which are opened in the 2D Discover sections.

Note Discover polyline sections are not supported by this option. Vertically exaggerated section are not supported.

- **Horizontal depth slices to surface grid** - creates a multi-banded grid which has one band for each current Horizontal (Z) slice.

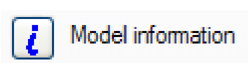
The Isosurface and Horizontal Depth Slice export options are only available if these rendering options have been applied to the voxel model.

Import and Export Voxel Models with the Grid Management Tool

The *Grid Management* tool in the Voxel Toolkit can also be used to import and export voxel models. It includes a powerful ASCII import wizard (see *Importing ASCII Voxel Models*) that loads delimited ASCII data and generates multi-banded 3D grids.

The tool can also be used to export grids in UBC, Encom 3D grid, Encom ModelVision TKM, and ASCII formats.

Viewing Voxel Model Information



The **Model Information** button opens a text report which can be saved for the model geometery and properties.

In addition to voxel geometric statistics such as the cell size, volume and data mean/range values, information report also contains the following calculations, useful for more advanced voxel analytics:

- sum*volume (clipped) = sum[value(i)*volume(i)]
  where i is every cell unclipped

- thresholded sum*volume (clipped) = sum[value(i)*volume(i)]
  where i is every cell unclipped and in the threshold range

- thresholded mean(by cell) = [sum[value(i)]]/count(i)
  where i is every cell unclipped and in the threshold range

- thresholded mean(by volume) = [sum[value(i)*volume(i)]]/[sum(volume(i))]
  where i is every cell unclipped and in the threshold range

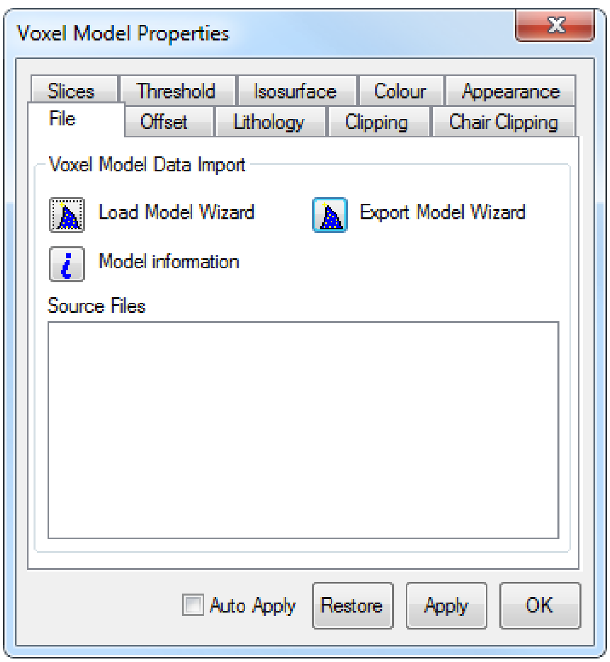
This is also a useful tool for seeing the lithology layers included in the model. For more details on lithology layers, see *Lithology Control*.

The text report can be saved using the **Save Report** button.

Detailed statistical information can be viewed about the model by clicking the **Statistics Explorer** button (see *Statistics Explorer Tool*).

Changing Voxel Model Display Properties

The display of volume elements (voxels) in the **Voxel Model** is controlled by the properties of the **Voxel Model** branch in the Discover 3D **Workspace Tree.** Once a **Voxel Model** branch exists in the tree, double clicking it or highlighting it and right-clicking allows you to display the **Properties** dialog.



Voxel Model Properties dialog

The **Properties** dialog uses different tabs to control various aspects of the voxel model display. These are:

- **File Tab** – enables import and export of the supported format models and viewing model information.
- **Offset Tab** – allows the voxel model to be offset and scaled using a Z offset and surface grid.
- **Lithology Tab** – specify the lithology legend table containing the lithology string names and colour patterns for the voxel model.
- **Clipping Tab** – externally clip the model in each of the three principal axes.
- **Chair Clipping Tab** – controls the primary axis slicing to reveal internal attribute distribution in three dimensions.
- **Slices Tab** – allows multiple slices of the model to be displayed at intervals along any of the three principal axes or on any user-defined plane.
- **Threshold Tab** – specifies an upper and lower range of attribute values that control which voxel cells are displayed.
- **Isosurface Tab** – specify one or more data values to visualise as 3D surfaces through the voxel model. This is the 3D equivalent of contour lines.
- **Colour Tab** – controls the colour and transparency of the voxels.
- **Appearance Tab** – toggles the display of thresholding, slices, colour fill, isosurfaces and wireframing.

Use the controls on these tabs to do these tasks:

- *Turning Model Rendering Options On and Off*
- *Offsetting and Exaggerating*
- *Clipping Along Principal Axes*
- *Chair Clipping*
- *Colour and Transparency*
- *Lithology Control*

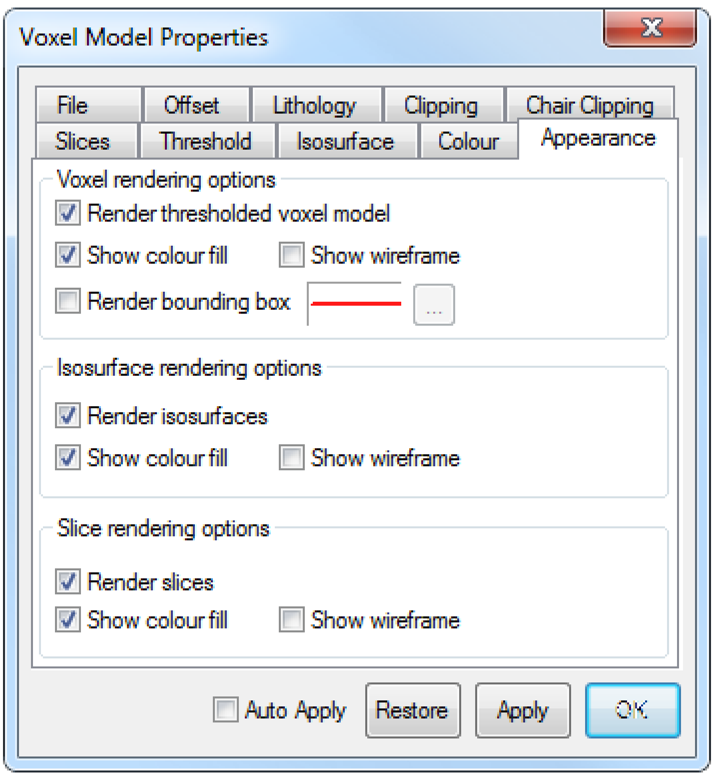
- *Thresholding*
- *Isosurfaces*
- *Slice View*

Turning Model Rendering Options On and Off

The **Appearance** tab of the **Voxel Model Properties** dialog functions as the primary display control for the various voxel rendering types:

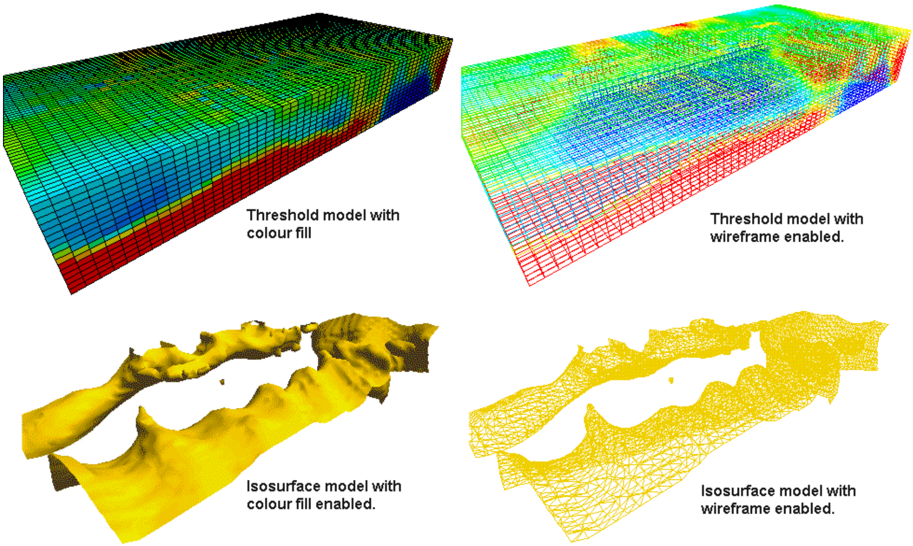
- Thresholded view (see *Thresholding*)
- Isosurface view (see *Isosurfaces*)
- Slice view (see *Slice View*)

These display types can be toggled on/off from this tab, whilst still preserving their tab-specific display configurations. Each option can additionally be toggled with colour fill and/or wireframe views.



Appearance tab of Voxel Model Properties dialog

Examples of the above options are shown below:



The various Voxel Model appearance settings

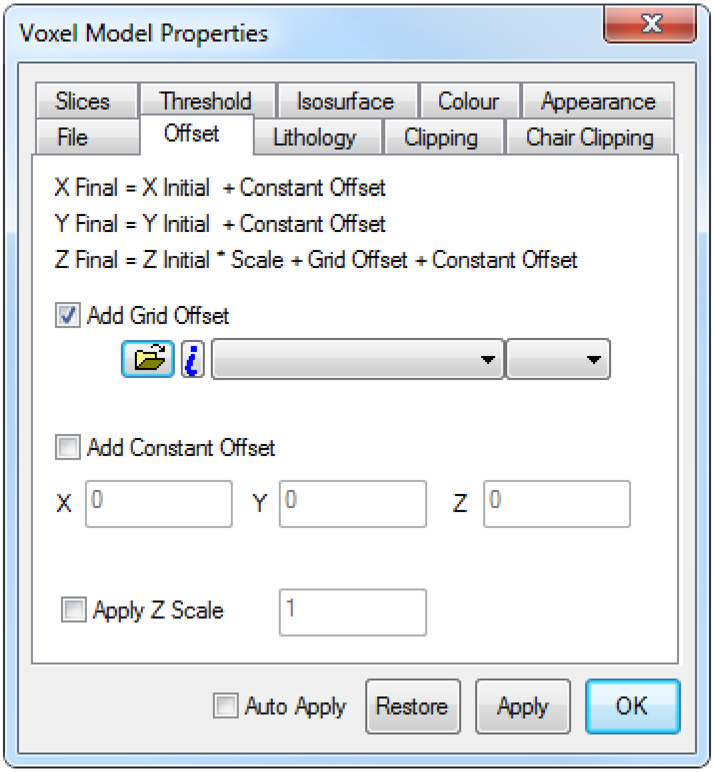
Offsetting and Exaggerating

Some voxel models have no information that determines their upper surface shape. Instead, they are composed of horizontal layers meaning that their top surface is flat.

Note You can also clip a voxel model cells above or below a specified surface grid using the **Grids>Voxel Utilities>Clip to DEM** tool.

The **Offset** tab provides two options to control the offset of the voxel model:

- **Add Grid offset** – add Z offset values based on a specified grid, or
- **Add Contstant Offset** – allows constant offsets to be specified separately for the X, Y and Z values.

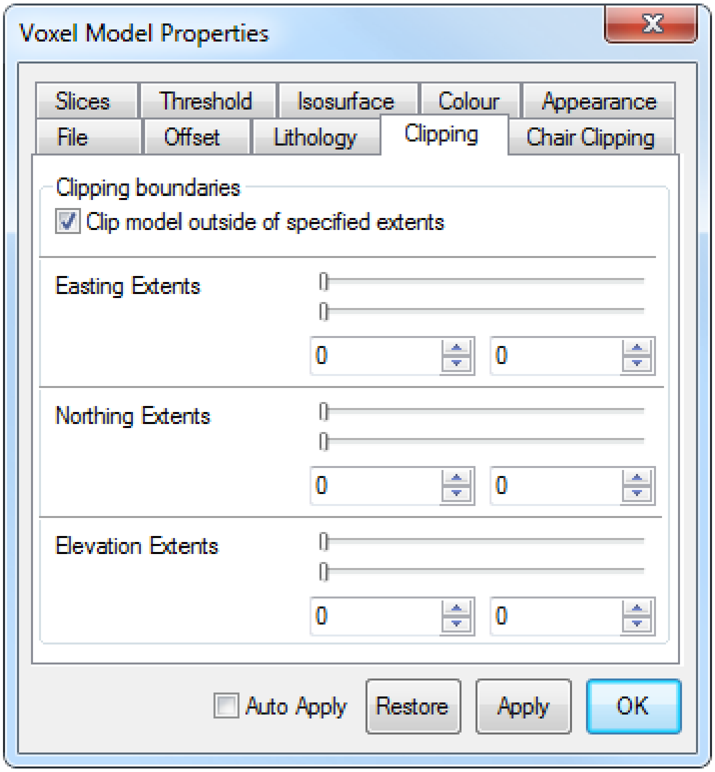


The Offset tab and defining a surface to alter the top surface

As well as offsetting the voxel model, you can also apply a Scaling factor that can be used to exaggerate the vertical size of the voxel model.

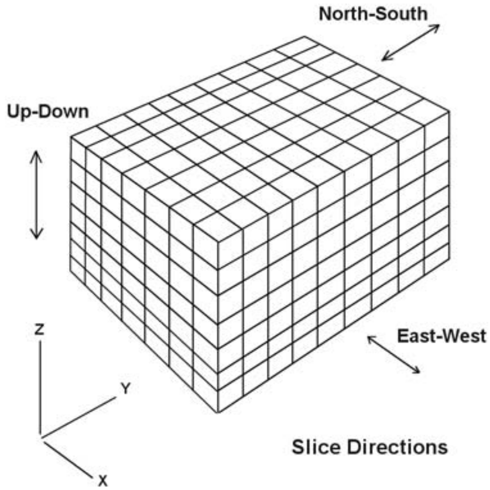
Clipping Along Principal Axes

The **Voxel Model** display allows you to interactively clip the rows, columns and layers of the displayed volume. You can remove individual voxel slices in any of north-south, east-west or top-bottom directions. The clipping controls are available from the **Clipping** tab.



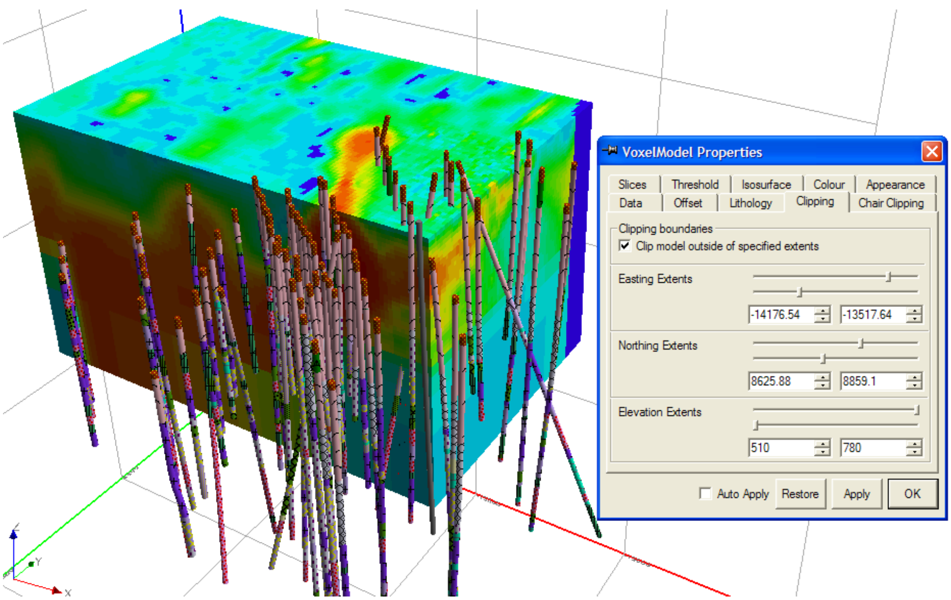
Clipping tab of the Voxel Model Properties dialog

Enable the clipping process by selecting the **Clip model** option in the **Clipping** tab. Two slider bars are provided for each of the **Easting**, **Northing** and **Elevation** directions. So long as the **Auto-Apply** option is checked, changes to any of the slider bars removes or makes volumes of the model appear or disappear.



Slice the mesh of the voxel model in any of the three principal coordinate directions

The entire model is displayed if the slider bars for each direction are on opposite sides of their extent. Specific positions of slicing can be entered into the various entry fields if desired.

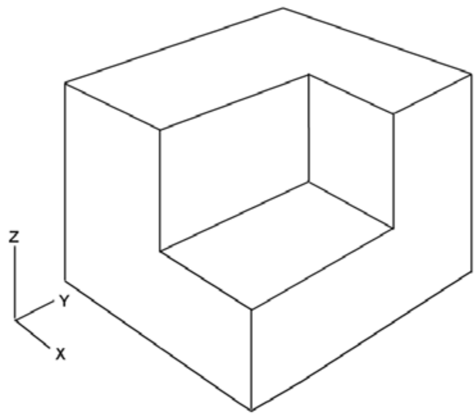


Clipping layers from the model reveals internal structure

The clipping operation either reveals or hides the various rows, columns or layers. No gradation of display is provided. Each row, column or layer is either displayed or not.

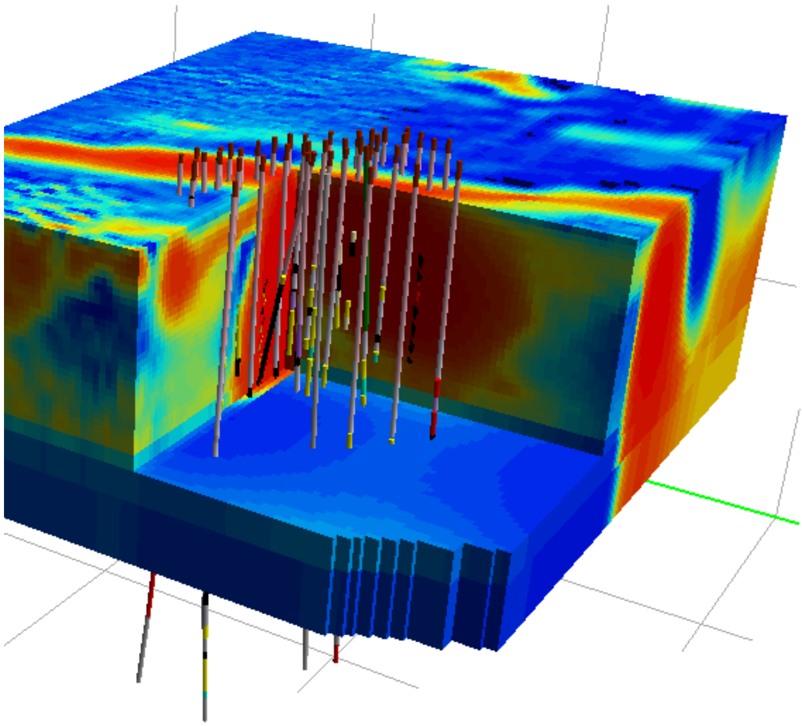
Chair Clipping

The **Chair Clipping** tab provided in **Voxel Model Properties** dialog is an extension of the logic used for **Clipping.** Chair Clipping enables clipping in any of the three primary coordinate directions (East, North and Vertical – X, Y and Z) resulting in a rectangular prism block being removed from the overall model shape.



Chair Clipping showing a portion clipped in each coordinate direction

Chair Clipping provides a method of identifying patterns of the displayed attribute within the voxel model and visualising how these trends migrate through the volume. The various clipping in any of the three coordinate directions can be controlled from slider bars and updated automatically if the **Auto-Apply** option is enabled in the **Chair Clipping** tab of the **Properties** dialog.



Chair Clipping of the model to reveal internal structure

The standard chair clips can be easily visualized and controlled using the button at the top right of the **Chair Clipping** tab. In the following dialog, the arrow buttons will sequentially highlight the standard cuts. Enable Auto Apply to visualise the cut dynamically, or press OK to close the dialog and visualise the cut.

Chair clipping can be used in combination with model clipping (see *Clipping Along Principal Axes*) to further add interactive viewing control.

Colour and Transparency

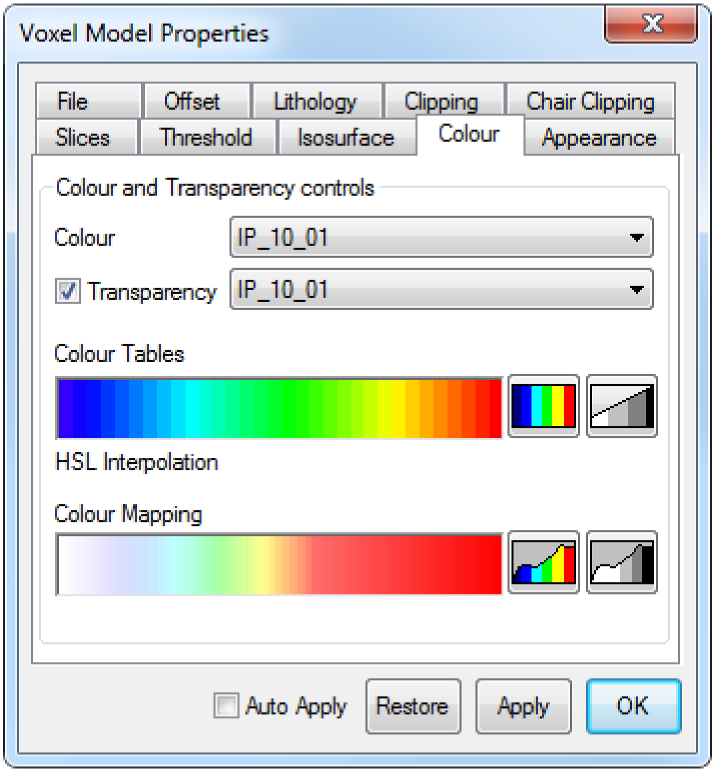
The **Colour** tab allows the colour and transparency of a voxel model to be modified.

Select the data field to **Colour** modulate from the top pull-down list; this can either be a numeric field (such as an assay value or geophysical property) or the **Lithology mode** when handling discretised voxel models (i.e. voxel models comprising a number of unique lithological codes - see *Discretised Gridding Method*).

If transparency modulation is required, enable the **Transparency** check box and select a data field from the adjacent pull-down list. (Transparency modulation is not available when in lithology mode.)

Two tools are provided:

- A choice of colour or opacity tables.
- Transformation of the selected table via colour or transparency mapping.



The Colour Properties tab to control the colour or opacity table and transform applied to a model

Colour Table

The **Colour Scale** button to the right of the Tables colour bar opens the **Colour Scale** dialog which allows the creation and/or loading of colour tables

Four methods of colour scale definition are available:

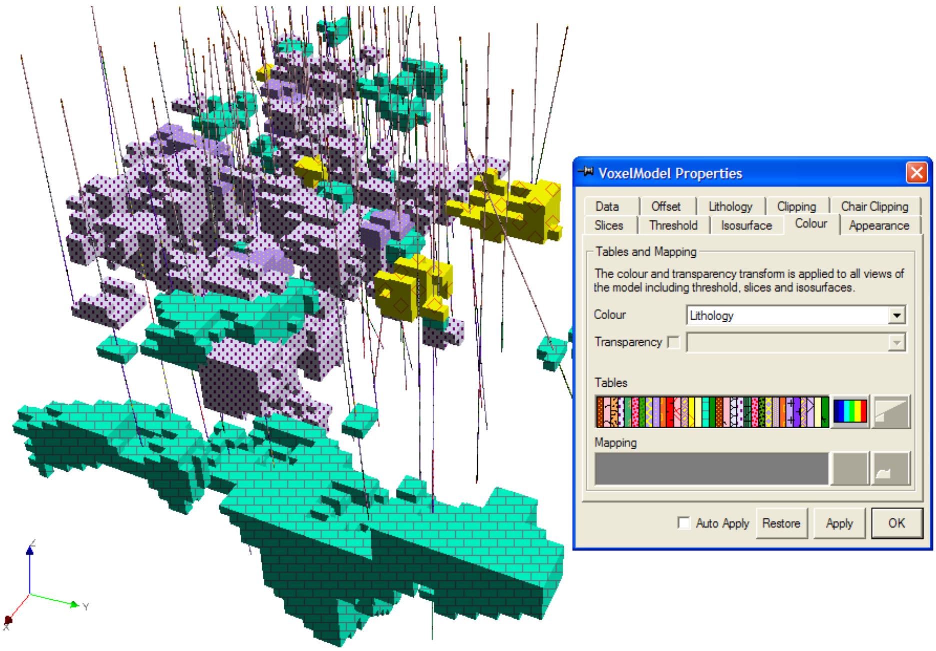
- **RGB Interpolation -** interpolates between two colours in Red:Green:Blue colour space.

- **HSL Interpolation -** interpolates between two colours in Hue:Saturation:Luminosity colour space.

Set the first and last colours of the colour scale by selecting the **Colour Browse** buttons at the bottom or top of the colour bar. When clicked, a standard Windows colour selection dialog is displayed allowing colour specification. These can be reset by clicking the **Set Default Colours** button.

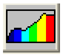
- **Look Up Tables** - the standard look-up table formats are supported and are installed as part of your Discover 3D installation. These can be created or edited using the Colour Look-Up Table Editor (see *Using the Colour Look-Up Table Editor*).

- A custom **Legend** created using the Legend Editor (see *Using the Legend Editor*) in Discover 3D or the Legend Editor in the Drillhole module of Discover. This is of particular use for colouring discretised voxel models (see *Discretised Gridding Method*) created using, for example, a series of lithology/rock codes.



Thresholded lithology voxels coloured with by applying a custom legend table

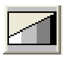
Colour Mapping



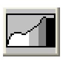
The **Colour Mapping** button to the right of the **Mapping** colour bar opens the **Colour Mapping** dialog. This enables a range of **Data Transforms** to be applied to the previously selected colour table. For more information, see *Advanced Colour Mapping*.

Note The **Colour Mapping** dialog is unavailable for voxel models in **Lithology mode**.

Transparency Tables and Opacity Mapping



The **Transparency Table** button to the right of the **Tables** colour bar opens the **Opacity Table** dialog (this button is only active when the Transparency check-box is enabled). Use this dialog to create a simple opacity table defined by one to four control points. A number of simple predefined tables are available which should cover the requirements of most users. The table is displayed graphically on a data Percentage vs. Opacity graph. This graph can be modified either by selecting and dragging the graph vertices (red crosses) in the graph, or manually altering the appropriate Percent and Opacity values to the right of the graph.

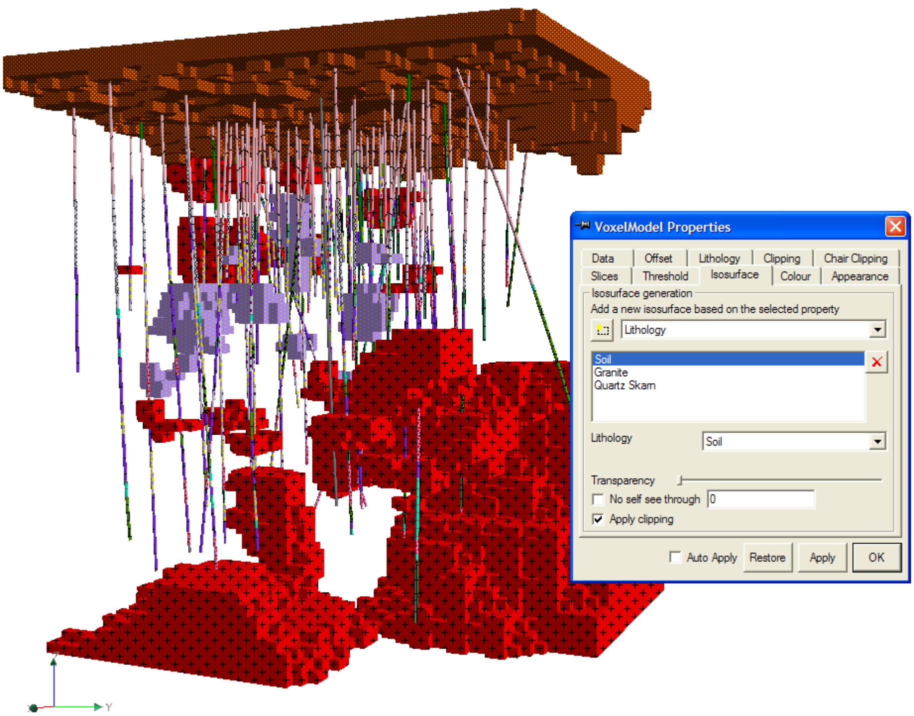


The **Transparency Mapping** button to the right of the **Mapping** colour bar opens the **Opacity Mapping** dialog (this button is only active when the Transparency check-box is enabled). The dialog is identical in content and controls to the Colour Mapping dialog above but is applied instead to the transparency option selected previously in the **Opacity Table** dialog. For more information, see *Advanced Colour Mapping*.

Lithology Control

Discretised voxel models (where a series of unique attributes, e.g. rock or alteration codes, have been used to create the block model - see *Discretised Gridding Method*) can be coloured using an appropriate **Colour Legend** via the Colour Scale Selection dialog under the **Colour** tab. It is also possible to control and display individual attributed components via either the **Isosurface** (see *Isosurfaces*) or **Threshold** (see *Thresholding*) tabs, utilising the appropriate **Lithology** mode controls.

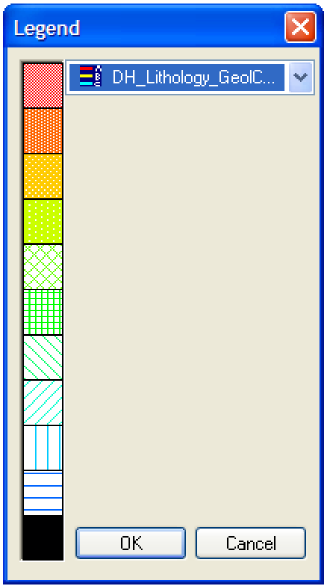
Note If no lithological field is present in the voxel model file, a default set of approximately 32-36 lithology bands will be constructed. For a single band/ attribute voxel model, these will simply be the bins from a histogram equalized distribution of the data. For multi-band models, it will be divided into subsections that reflect each combination possible. For example, in a two-band model 36 combinations with 6 bins in each band will be generated. I.e. Band1(Bin1) + Band2(Bin1); Band1(Bin1)+Band2(Bin2).. etc. These bins values can be checked within the Value, Min and Max boxes for each Lithology and band combination.



Displaying multiple lithology layers from a discretised voxel model as isosurfaces, with a Colour Legend applied

In order to allow the **Threshold** and **Isosurface** tabs to display the attribute codes for individual layer control in Lithology mode (as pictured above), the appropriate **Colour Legend** must be loaded via the **Browse** button (shown left) under the **Lithology** tab. If a colour legend is not set, these dialogs will simply display a list of index codes (i.e. Index 1, Index 2, etc) instead of the appropriate rock/alteration codes.

Note After Colour Legend selection, click the **Apply** button to ensure the changes are applied to the threshold/isosurface tabs.



Colour Legend selection dialog accessed via the Lithology tab

Thresholding

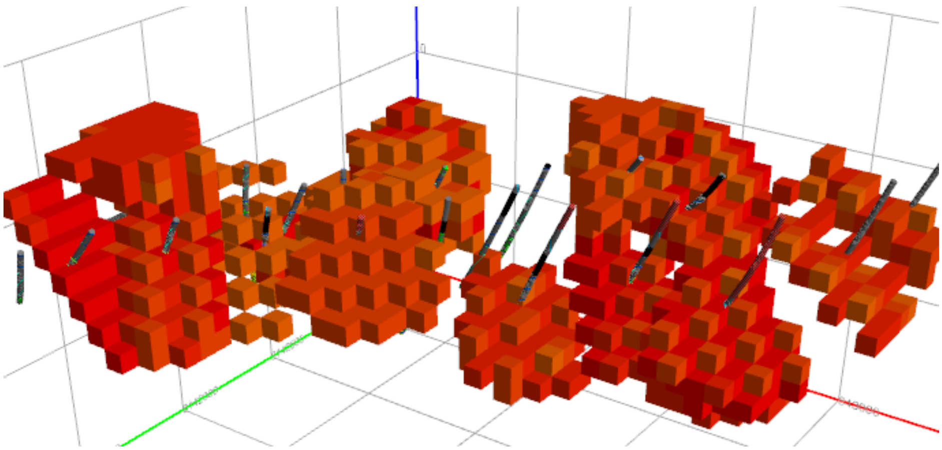
Thresholding of voxel models allows voxels (cells) to be displayed only if their specified attribute lies within the chosen data range. The threshold feature operates in two modes:

- **Interactive mode** – use slider bars to control the displayed voxels by dynamically altering the upper and lower data range.
- **Lithology mode** – display voxels with only predetermined data ranges/ unique attributes i.e. voxel models created with the *Discretised Gridding Method*.

Interactive Mode

The interactive mode of thresholding enables you to dynamically specify an upper and lower range of data attributes and their corresponding voxels for display.

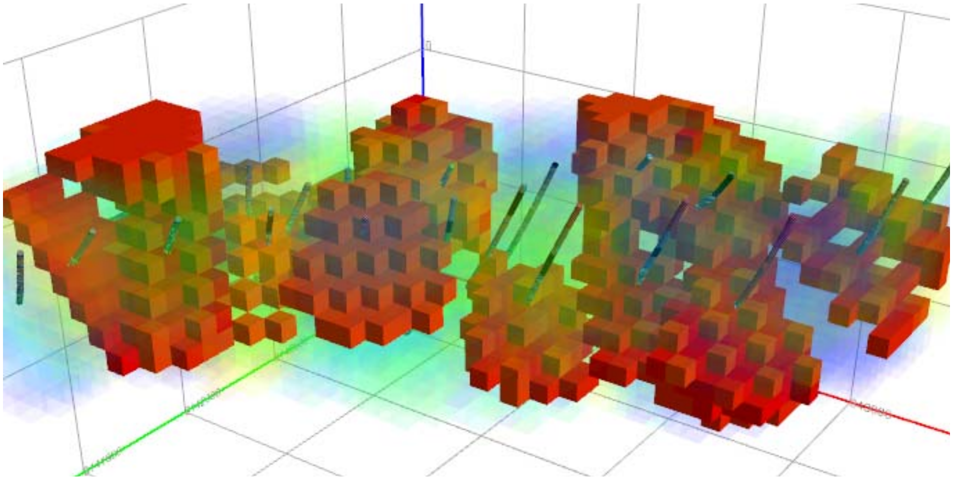
To use this form of thresholding, select the attribute from the list and enable the **Threshold** checkbox for the attribute.. When selected, two slider bars and two entry fields become available. With the **Auto-Apply** option on, if you move either of the upper range or lower range slider bars, the voxels displayed automatically vary. Operation of the slider bars also adjusts the upper and lower data values of the two entry fields. Conversely, you can enter specific values in these entry fields. By default a histogram equalized data range is used across the slider control. Enabling the **Linear Slider** checkbox will change this to a linear slider across the data range.



Interactive voxel thresholding with normal opacity

Two **Opacity** modes are available:

- **Normal** – only the Accepted Voxel Range is displayed, with its transparency governed by the adjacent transparency slider bar, or
- **Cloud** – the Accepted Voxel Range is displayed as solid or opaque, whilst the Rejected Voxel Range is displayed with the level of transparency set by the adjacent slider bar.

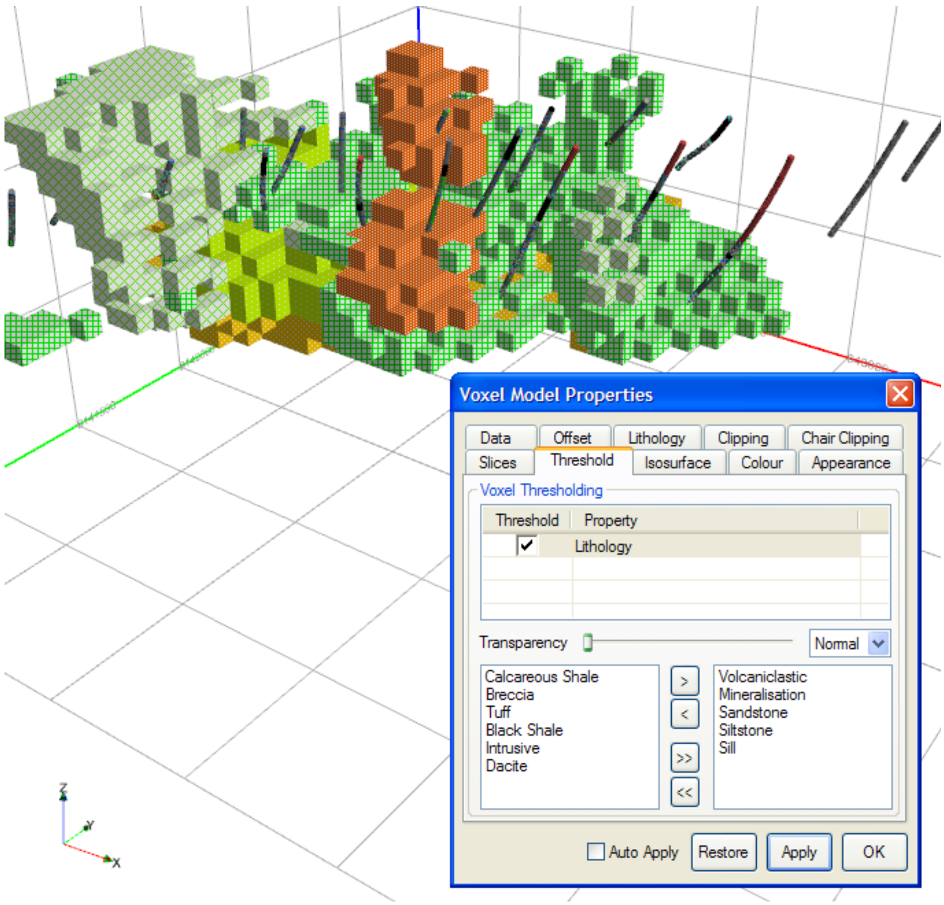


Interactive voxel thresholding with cloud opacity

Lithology Mode

The Lithology mode allows a selected number of predetermined data ranges or attributes to be displayed. These may represent rock types or attribute ranges relating to conductivity, magnetic susceptibility etc. To effectively utilise this mode, ensure that an appropriate **Colour Legend** has been assigned in the Lithology tab.

To access this mode, select the **Lithology** option from the top pull-down list in the **Threshold** tab, and tick the **Threshold by** option.



Lithology control of voxel thresholding using cloud opacity

From the **Lithology** table listing provided, choose the items to be displayed and move these into the right hand list area with the arrow buttons. If the **Auto-Apply** option is enabled, the items with the specified attribute ranges should immediately display.

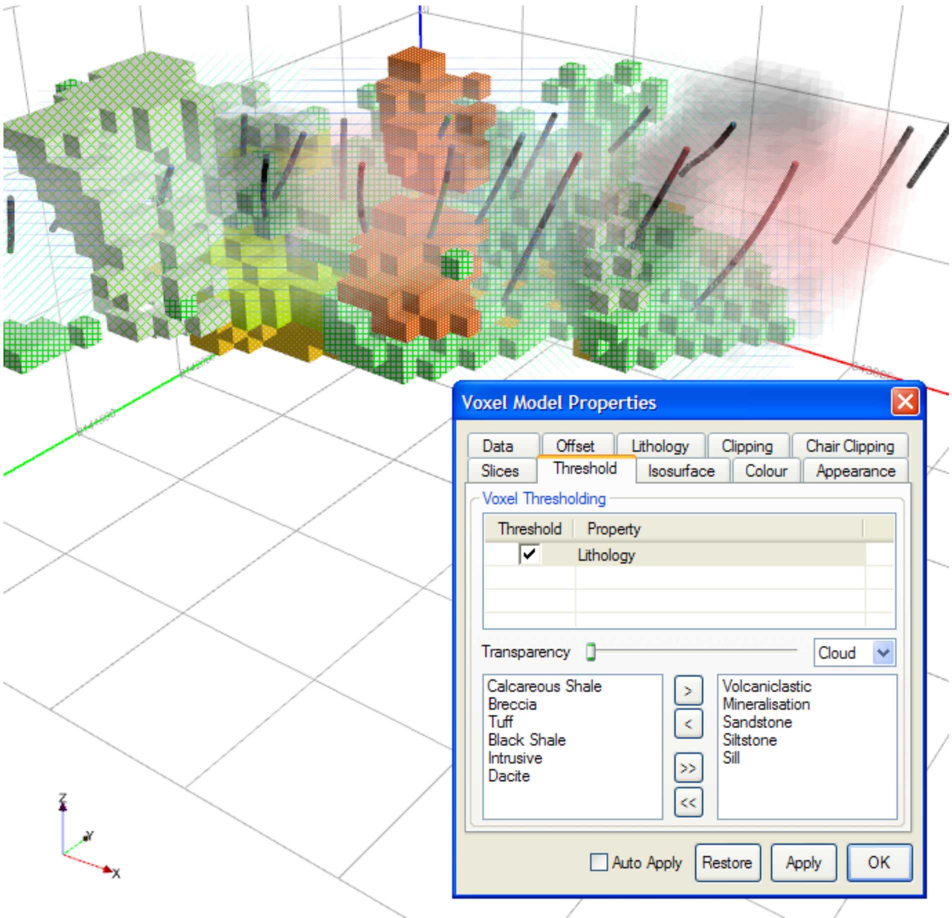
Two **Opacity** modes are available:

- **Normal** – only the selected Lithologies are displayed; their grouped transparency is governed by the adjacent slider bar.

Note Using the Normal mode when applying lithology thresholds produces identical results to displaying lithologies as *Isosurfaces*. However lithological isosurfaces allow individual transparency control for each lithological layer and is thus the recommended option.

- **Cloud** – the selected Lithologies are displayed opaque, whilst the unselected Lithologies are displayed with the level of transparency set by the adjacent slider bar.

Note Applying a Colour Legend as a *Colour Table* is not recommended when using Cloud mode to display lithology thresholds, as it is very processor intensive (particularly colour legends incorporating patterns). Instead it is recommended to colour the voxel model using a standard LUT or RGB/HSL interpolation.



Lithology control of voxel thresholding using cloud opacity

Isosurfaces

Similar to threshold settings, the **Voxel Model** can be made to display a surface of a single attribute data value. The isosurface generator positions the surface similarly to a contour map by examining the enclosing voxels and determining the correct location for each intersection point of the surface throughout the volume. It then triangulates these points and computes the surface.

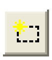
As for thresholding, the isosurfaces can be created using the:

- Lithology data - the isosurface is mapped by the data ranges of the lithological bands. Note that the created isosurface uses only the lower of the data range values. Lithology Isosurfaces cannot be exported as an isosurface DXF from the **Export** tab.

- Interactive mode - this mode allows a particular attribute value to be used to create the isosurface.

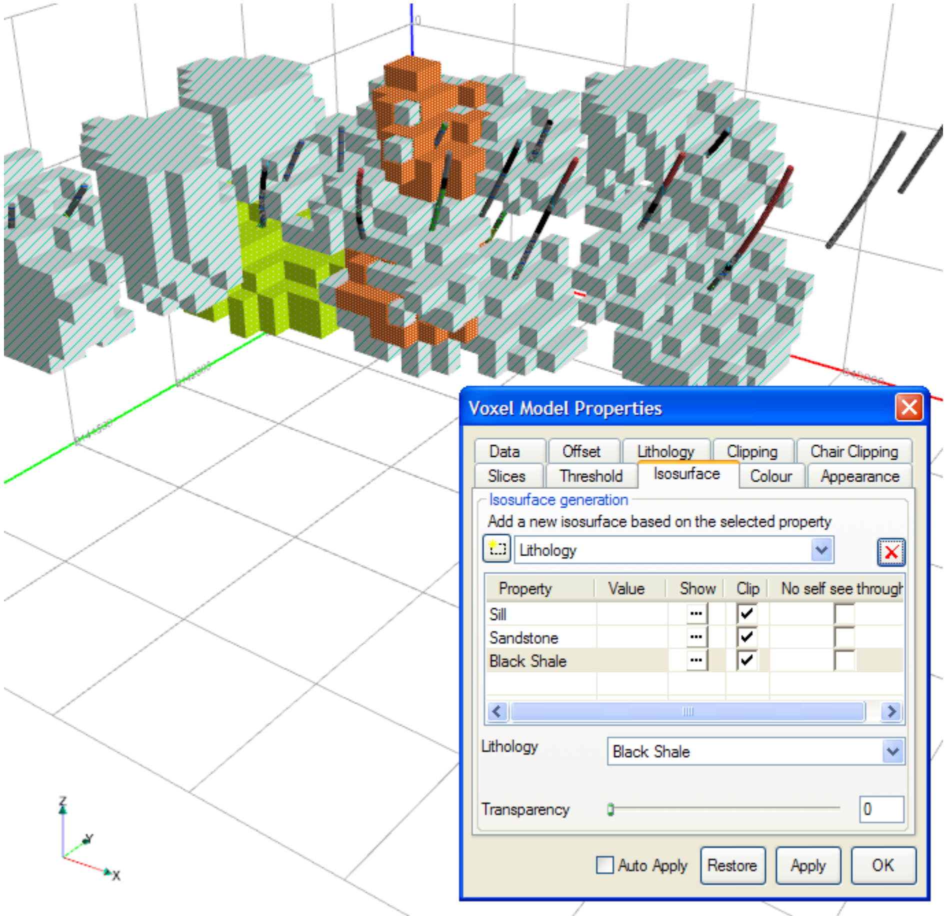
Note Ensure the **Render thresholded voxel model** option is disabled in the **Appearance tab** otherwise you may not be able to see the results of creating the isosurface as it may be buried within the thresholded or full model display.

Lithology Mode



To effectively display lithologies as individual isosurfaces, ensure that an appropriate **Colour Legend** has been assigned in the Lithology tab. Also setting this Colour Legend in the Colour Table button under the **Colour tab** allows the preset colours/patterns to be used. In the **Isosurface** tab select the ‘Lithology’ option from the top pull-down list and add as layer by clicking the **Add** button (shown left). To alter the layer, highlight it and select the required lithology entry from the **Lithology** pull-down in the middle of the dialog.

Note Multiple lithology table entries can be added (as pictured below); the transparency of individual layers can be controlled via the **Transparency** slider bar.



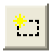
Displaying multiple lithology layers from a discretised voxel model as isosurfaces

You can remove a selected layer by using the **Delete** button. The Isosurface slider bar and entry field are inactive when in **Lithology** mode.

Note Displaying lithologies (e.g. rock types) as isosurfaces is identical to displaying them as thresholds (see *Thresholding*), except that you can control each layers transparency individually as isosurfaces.

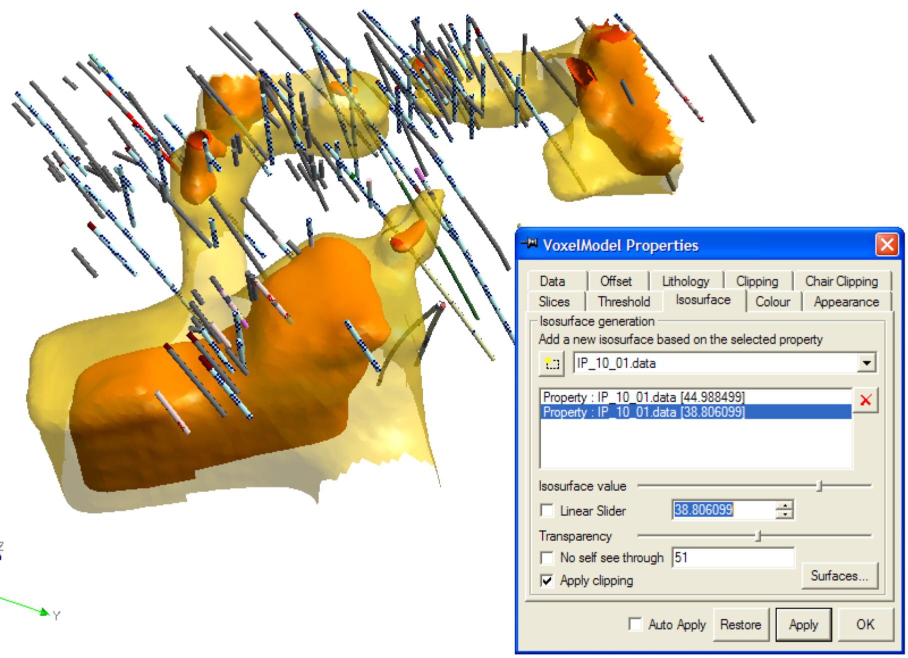
Interactive Mode

The **Interactive mode** allows you to specify the attribute value of the isosurface. The created isosurface is also triangulated to produce a smooth surface of constant attribute value. Colouring of the surface is controlled by the **Colour** tab.



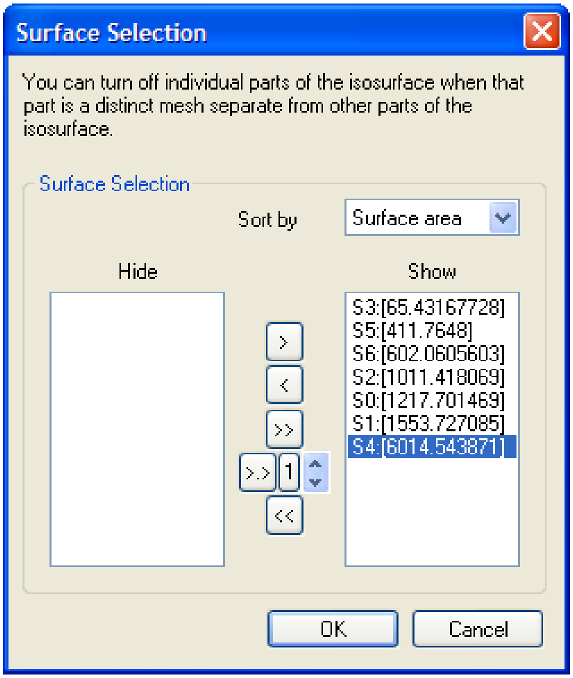
To create an **Isosurface**, select the required voxel model data (or properties) from the pull-down list. Click the **Add** button (shown left) and a default attribute value is added to the list. Additional values can be added if more than one isosurface is to be created. Once a layer is available, the **Voxel Model** displays a surface corresponding to the data value shown. You can use the slider bar, or enter a specific data value if desired. The display automatically updates if the **Auto-Apply** option is enabled.

Note Numeric isosurfaces cannot be coloured by a legend. They can be coloured by Look-up tables, or alternatively exported to a DXF and the colour changed in the DXF vector layer.



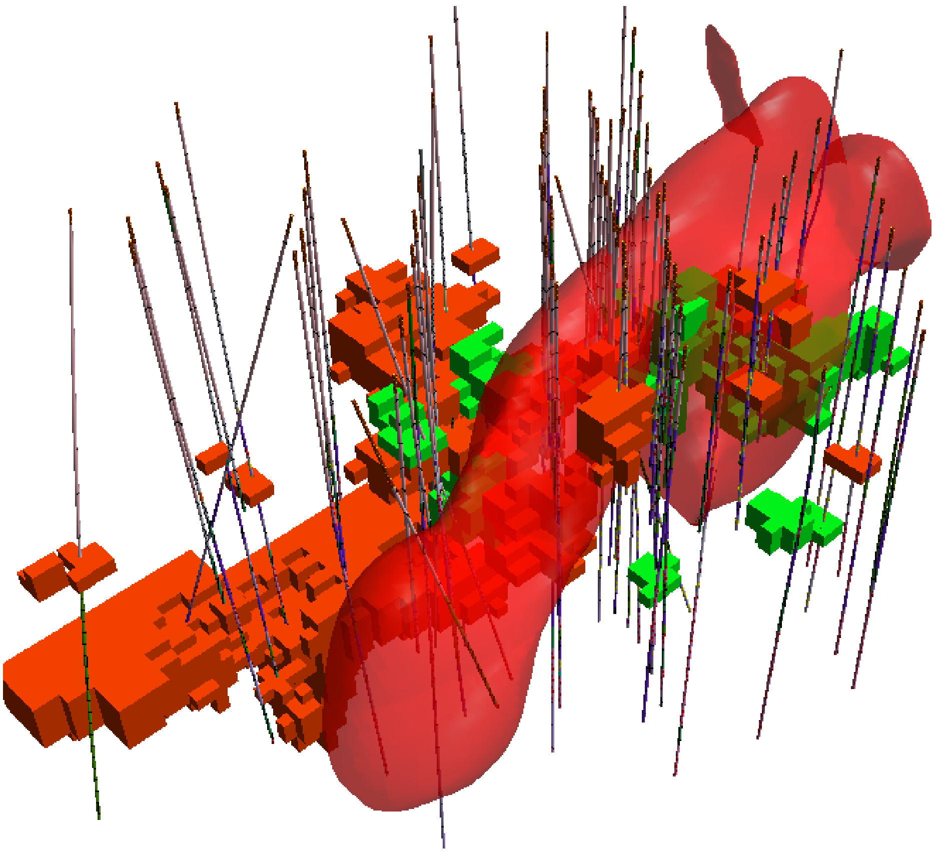
Displaying multiple Property isosurfaces in Interactive mode

The **Surfaces** button allows individual parts of a Property isosurface to be displayed or hidden, when these parts are a distinct mesh separate from other parts of the isosurface.



The isosurface Surface Selection dialog, allowing display control of individual components of an isosurface

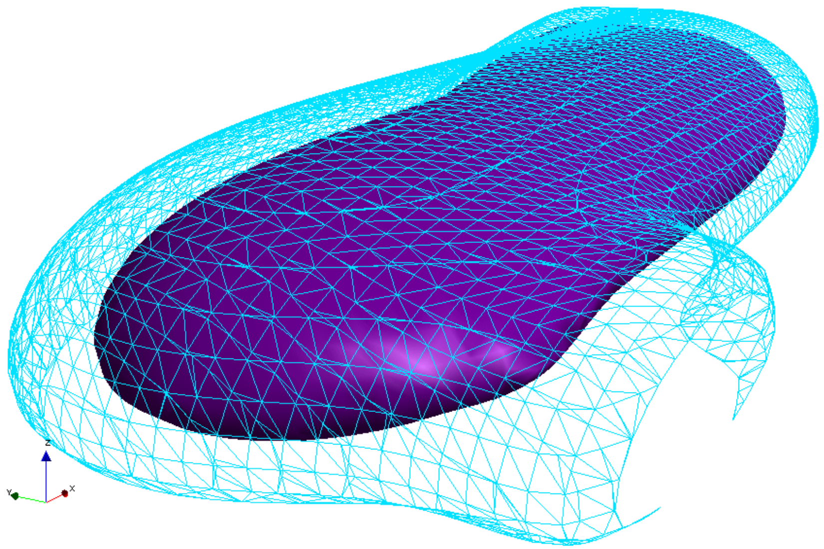
It is also possible to combine both **Lithology** and **Interactive** modes by adding isosurface layers from both the Lithology band and other data bands. Separate numerical and lithology voxels can be merged using **Grids>Voxel Utilities** menu option.



A combination Property/Lithology isosurface display

Using Wireframes

Occasionally it is useful to use wireframes instead of isosurfaces alone. **Wireframing** provides a means of seeing through one surface to a second that would otherwise be obscured. An example is shown below:



Two isosurfaces used with the outer one represented only as wireframed, the inner solid

The wireframes also provide information about the control points used in the triangulation process to create the isosurfaces.

To apply wireframes to **Isosurfaces**, enable the **Show wireframe** option under the **Isosurface rendering options** of the **Appearance** tab.

Slice View

Voxel models can be visualized as either individual slices or a series of slices though the model (i.e. a series of sections through the dataset). This can be an effective method of visualizing the interior of larger or more complex block models coincident with other datasets such as drill holes, solid models, etc.

Slice location, orientation and spacing is controlled with the **Slices** tab of the **Voxel Model Properties** dialog.

- *Slice Orientation*
- *Slice Display Settings*
- *Viewing Slices as Georeferenced Images (EGB)*

Slice Orientation

There are two options for specifying slice orientation: parallel to a principal plane (XY, YZ, ZX) or user defined.

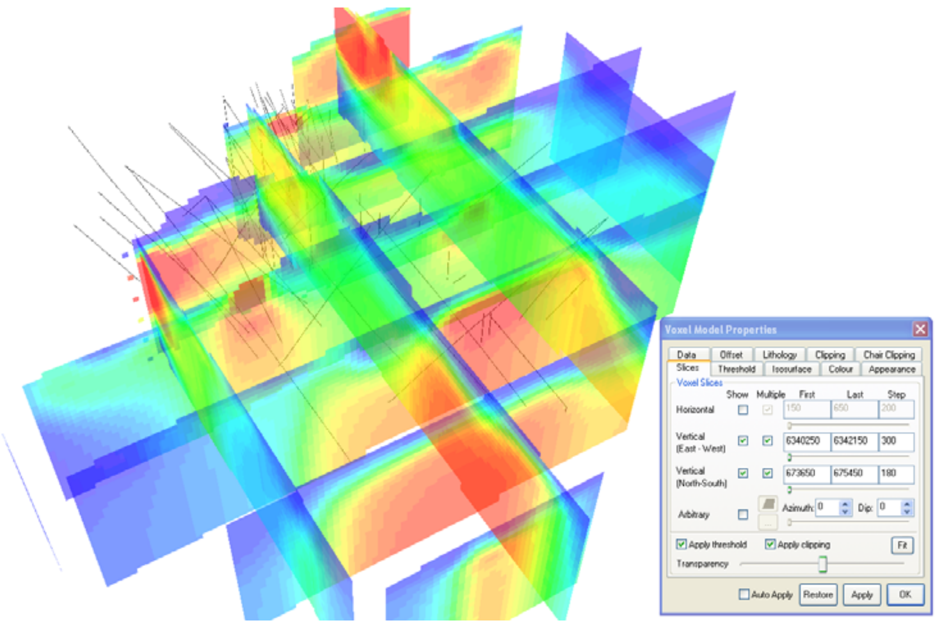
To slice parallel to a principal plane:

1. On the **Slices** tab, enable the **Show** option for the orientation required – **Horizontal** or **Vertical** (E-W or N-S). Note that more than one orientation can be enabled simultaneously.

2. If multiple slices are required in the same orientation, enable the **Multiple** tick box

3. Edit the **First** and **Last** coordinates for the orientation if desired. If the **Multiple** option is disabled for the slice orientation, the coordinates value for the single slice can be adjusted using the slider bar.

4. If multiple sections in one orientation are being displayed, the distance between slices can be adjusted using the **Step** value.



Example of multiple slices displayed in both vertical orientations in tandem with a drillhole project.

To slice on the cursor plane:

1. On the **Slices** tab, select the **Arbitrary** option.

2. Select **Cursor Plane**.

3. Enable and orientate the cursor plane to the desired slice orientation (see *Cursor Plane*).

4. Click the **Synchronize** button (far left) to capture the cursor plane parameters.

   - Permanently Synchronising the cursor plane updates the slice position continuously as the cursor plane is moved. This mode does not allow Multiple Slices to be rendered

   - Choosing to a non-permanent synchronisation updates the slice position once only, and allows multiple slices to be visualized.

5. Click **Apply** to display the slice.

To slice on Discover drillhole sections:

1. Open the Discover drillhole project and the sections in MapInfo/Discover 2D.

2. On the **Slices** tab, select the **Arbitrary** option.

3. Select **Discover Section**.

4. Select from the list the Discover sections to use.

5. Click **OK** and **Apply** to display the slices.

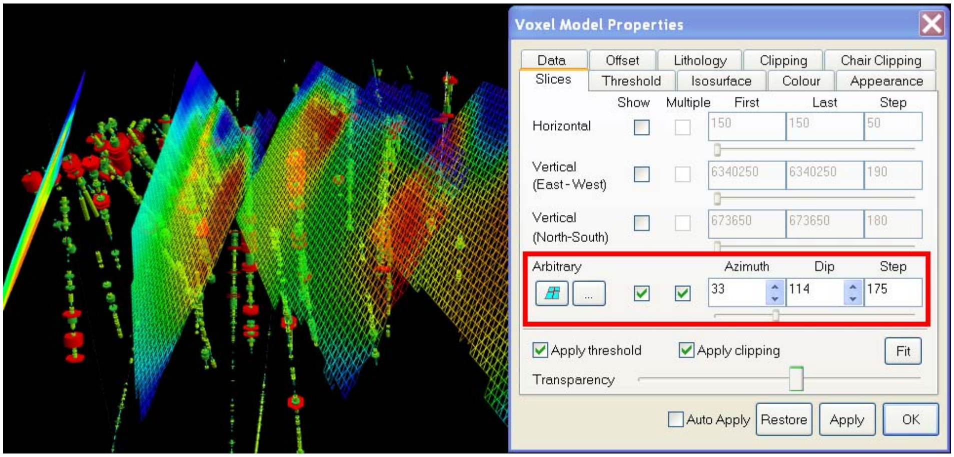
6. To edit the sections shown/displayed in 3D, repeat steps 1–5.

To slice on a user-defined plane (manual):

1. On the **Slices** tab, select the **Arbitrary** option.

2. Select **Manual**.

3. Use the dip, azimuth and slider controls in the **Slice** tab, or click the **Configuration** button. The configuration dialog allows more explicit definition of the arbitrary slice's X, Y, Z, dip and azimuth parameters, including a dynamic preview window of the planes orientation



Example of an arbitrary slices synchronized to the orientation of the cursor plane. The slices are displayed as a wireframe with no fill colour (see Slice Display Settings)

Slice Display Settings

The display properties of slices are controlled from the **Slices** and **Appearance** tabs of the Voxel Model Properties dialog.

To turn on and off slices:

- Voxel model slices can be turned off without disabling any options within the Slice tab itself by toggling the **Render Slices** option in the **Appearance** tab.

To change the slice transparency:

- The **Transparency** of slices can be adjusted using the slider at the bottom of the **Slices** tab.

To change the cell outline and fill:

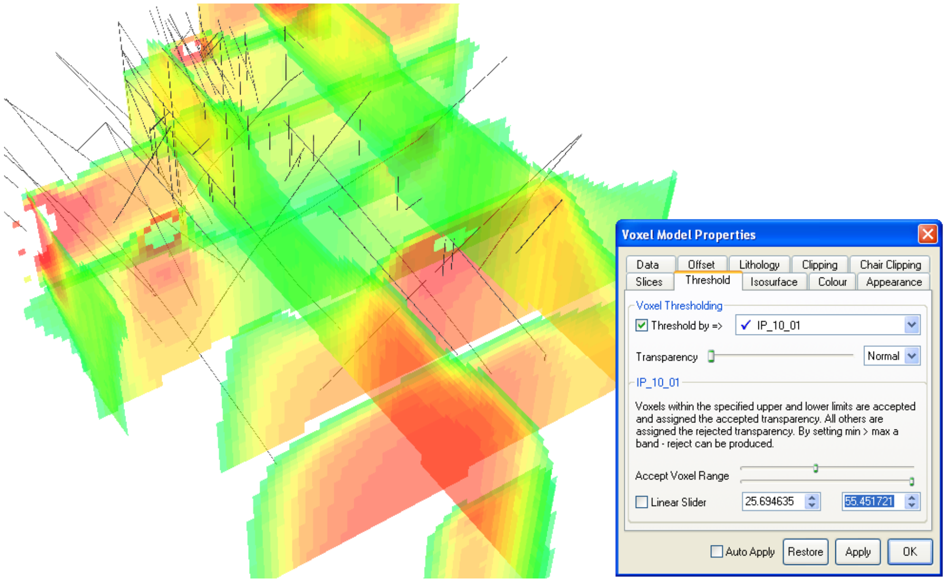
- Individual cell outlines and fill can be turned on and off with the **Show Wireframe** and **Show colour fill** options under **Slice rendering options** of the **Appearance** tab.

To limit the cells displayed to a target data range:

1. Enable the **Apply threshold** option at the bottom of the **Slices** tab

2. In the **Threshold tab**, set the appropriate dataset and enable the **Threshold by** option. Use the sliders to constrain the data range

3. In the **Appearance** tab, disable the **Render thresholded voxel model** option, and press **Apply**.



Example of multiple slices presented with thresholded data range highlighting regions of interest.

Viewing Slices as Georeferenced Images (EGB)

Any voxel sliced view configured in the **Slices** tab can be converted into a georeferenced EGB image series. This allows large memory-intensive voxel models to be handled as much more efficient rasterised cross-sectional images

1. Configure and display the desired voxel slice view in the **Slices** tab.

2. Go to the **File** tab of the **Voxel Properties** dialog, and select the **Export Model Wizard**.

3. In step 1 of the wizard, select the **Slices to Images** option and press **Next**.

4. In step 2 of the wizard, change the output EGB file location and name if desired. The output image X and Y dimensions (quality) can also be altered, as well as the image format. Press **Finish** when complete.

5. Add an Image branch to Discover 3D window, and browse to and open the output EGB file.

6. Turn off the visibility of the source voxel model.

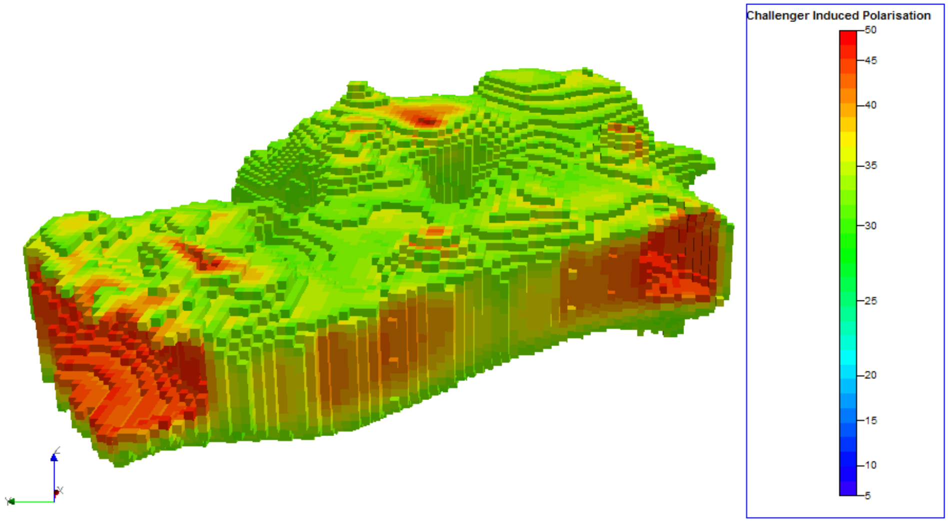
Viewing Slices on 2D Discover Drillhole Cross-Sections

A voxel model can be sliced to opened Discover cross seciton and also exported as TAB vector files onto these sections.

1. Configure and display the voxel model.
2. Open the Discover Drillhole project.
3. On the **Slices** tab, select **Show Arbitrary**.
4. Click the **Populate Sections** icon.
5. Click **Apply** and visualise the slices in 3D.
6. On the **File** tab, select **Export Model Wizard**.
7. Select **Export section slices to vector files**.
8. Click **OK** and the output 2D vector files will be opened in Mapinfo in the corresponding section windows.

Displaying a Floating Colour Legend

A *Floating Colour Bar* can be added to the 3D window and referenced to a voxel model, allowing easily visualisation of the relationship between colours and numeric data ranges.



An example of a Floating Colour Bar displayed for a thresholded voxel model

Displaying Multiple Voxel Models

The **Isosurface** section (see *Isosurfaces*) described multiple voxel displays showing the distribution of two or three voxel layers. In this case, the multiple displays are derived from the same voxel model.

Using the **Voxel Model** within Discover 3D you can also have voxel displays from multiple model sources. This is particularly useful if you have results from different methods that need to be compared. For example, if a survey area has results from a gravity survey inverted and for the same area, a magnetic survey has been used to derive an inversion model, the model output results can be compared directly in the one display.

Similarly, if a model is prepared as a seed model for an inversion, this technique provides a method of directly comparing the seed model with the results computed from an inversion process using the seed model.

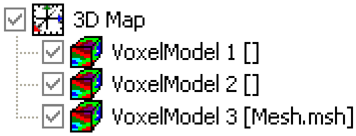
In the section *Creating a 3D Voxel Model Display* it was shown how to display a 3D Map and insert a layer for a **Voxel Model**. To create additional **Voxel Model** layers and associate them with multiple models, the process is the same but extra **Voxel Model** layers are added to the Workspace Tree.

The steps involved are:

1. Create a Voxel Model in the 3D Map as described in *Creating a 3D Voxel Model Display*.

2. Select the appropriate model as described in *Using the Load Model Wizard*.

3. Repeat steps 1 and 2 to add extra voxel model branches. The Workspace Tree should appear similar to that below:

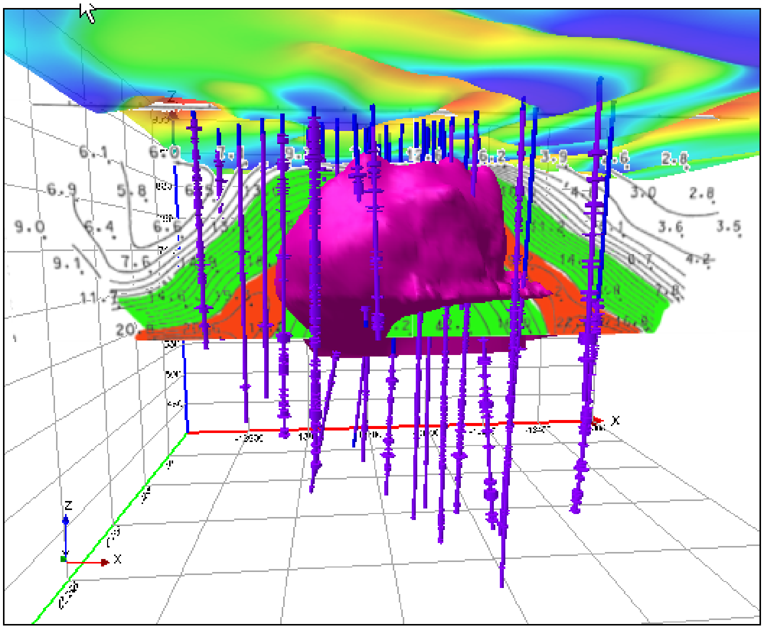


Integrating Voxel Model Displays with Other Objects

Once the **Voxel Model** display is used for importing and displaying models as described, you can take advantage of Discover 3D's ability to create powerful interpretational displays by integrating other display objects. Other object types supported include:

- Gridded surfaces.
- Drillholes.
- External 3D graphics files such as DXF format.
- Located bitmaps derived from scanned or captured imagery.
- GIS layers derived from MapInfo Professional.
- Models derived from third-party packages such as ModelVision Pro, EM Gui etc.

An example of the complexity of display that is possible with integration in the **Voxel Model** and Discover 3D is shown here:



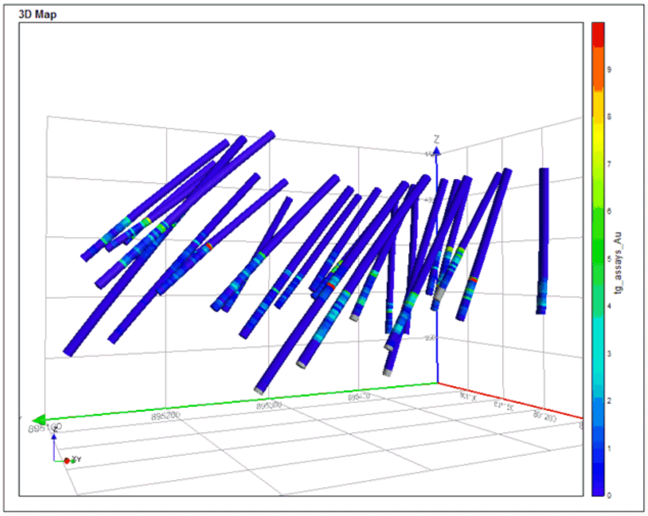
Display of drillholes, located bitmap, model and isosurface using the Voxel Model

For information on importing and displaying any of these additional objects, refer to the relevant sections of this guide.

18 Creating and Manipulating Voxel Models

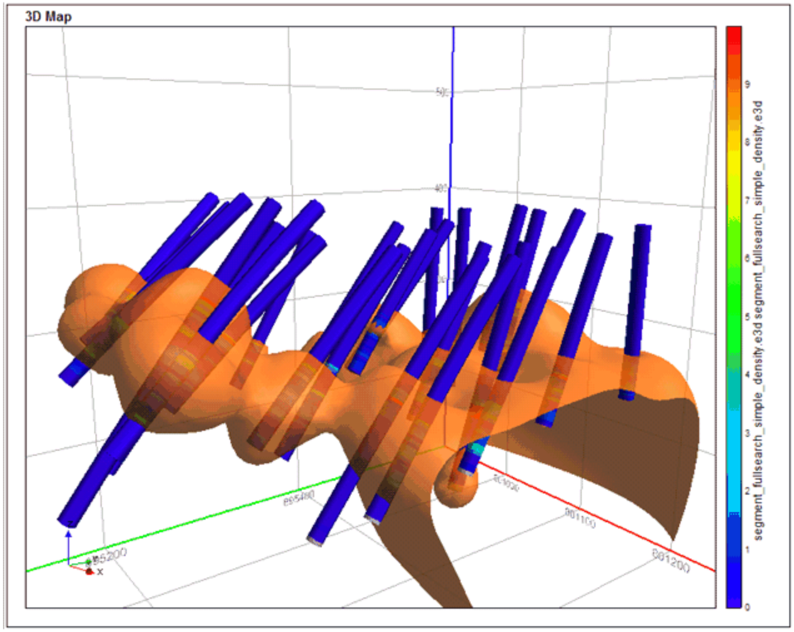
The voxel tools on the **Grids** menu provide the ability to create and process voxel models (often referred to as 3D meshes, block models or 3D grids) from drillhole or 3D point data. Gridding interpolates between the existing data points/intervals, creating a continuous 3D mesh of cells, each of which is assigned a value (either numeric e.g. geochemical assay or character e.g. lithology). The 3D interpolation process allows dispersed source data to be visualised as a continuous dataset (e.g. a voxel model isosurface), providing the geoscientist with a powerful analytical tool.

For example, the image below shows a small collection of drillholes with colour modulation based on gold assay data.



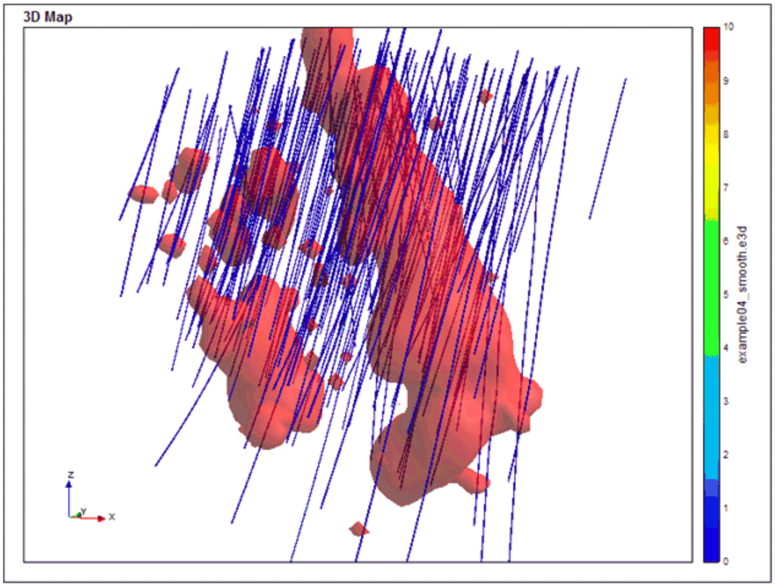
Example of drillholes to be gridded using gold assay intervals

The dataset was gridded with the **Voxel Gridding** tool, producing a 3D grid/voxel model. Discover 3D can display voxel models as thresholded (see *Thresholding*) block models or as *Isosurfaces* (amongst other visualisation options). In this case (below) an isosurface was extracted from the voxel model to show the distribution of the gold above a certain threshold. The isosurface is semi-transparent to enable the drillholes within to be visualised.



Isosurface of gridded gold values from sampling of the drillholes

Another example (below) is produced from a larger dataset and shows the continuity of the gold mineralization across the drillholes.

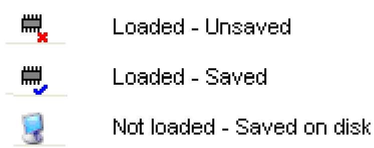


Example of the continuity of gold assays from the drillhole dataset

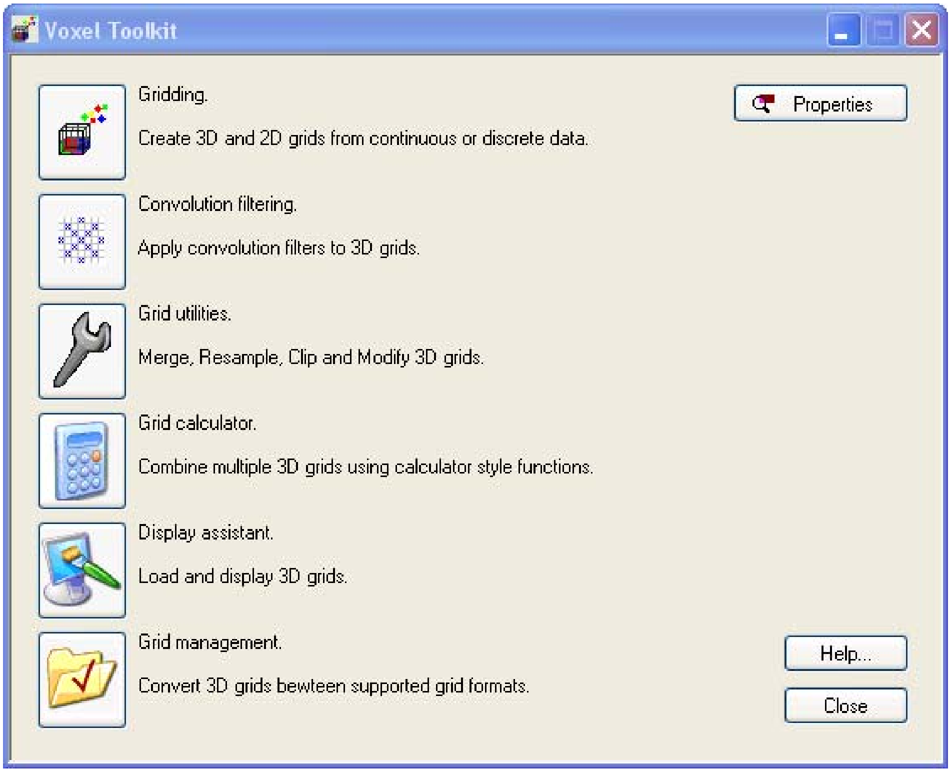
The voxel tools available on the **Grids** menu are:

- **Grids>Voxel Gridding** – create 2D grids (surfaces) or 3D grids (voxel models) from 3D point or segment data, such as a drillhole database. The use of the Voxel Gridding tool is described in:
    - *Creating a Voxel Grid*
    - *3D Gridding Methods*
    - *3D Gridding Parameters*
- *Grids>Voxel Filter* – apply convolution filtering to Voxel Models.
- *Grids>Voxel Utilities* – A powerful range of utilities for advanced vector/ voxel analytics, voxel resampling and merging and clipping voxels to vector surfaces/volumes.
- *Grids>Voxel Calculator* – conduct mathematical operations on Voxel Models.
- *Grids>Display Assistant* – load and display Voxel Models in Discover 3D.
- *Grids>Voxel Manager* – import and export Voxel Models in a variety of formats.

Voxel tools will automatically load voxel models that are open in Discover 3D. The list of grids presented in the various function dialogs will display an icon indicating whether a grid is on-disk, in memory or in memory and unsaved. Grid information can be viewed for any file by selecting the information icon in the list. Grids listed in the current tool are available to all other voxel tools.



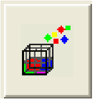
On closing a voxel tool, the Voxel Toolkit dialog will open, providing direct access to the other voxel tools."



The **Properties** button accesses advanced settings for the Grid calculator and Gridding tools. For more information, see *Advanced Gridding Properties*.

Creating a Voxel Grid

Grids>Voxel Gridding



The gridding operations in the Voxel Toolkit interpolates 3D spatial data into a regular 3D grid (voxel model or block model – terms commonly used in the mining industry). The 2D gridding operations generate horizontal surfaces (in the X, Y plane) at a user-specified RL level. These grids can then be exported as ER Mapper format files.

The input data may be either a collection of three dimensional points or drillhole intervals. An interval must be defined by two 3D points – the beginning and end of the interval. The value of the sampled interval is assumed to be constant between its end points.

Note Voxel Gridding also provides the option for 2D grid, however it is recommended to use the **Grids>Surface Gridding** which contains more powerful interpolation methods options as well as an dynamic preview window.

To simplify the process of creating 2D or 3D grids the gridding operations are presented with a wizard interface. The gridding methods and controls available are described in *3D Gridding Methods*.

Creating a voxel model from drillhole or sample data:

The following steps guides the user through process required to apply 3D gridding to a drillhole projects. This will apply an interpolation (in this case Inverse Distance Weighting) to a downhole data field (e.g. an assay field), the creating a voxel model. This can then be used to visualise high assay value correlation between holes (see *Changing Voxel Model Display Properties*).

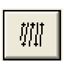
1. Ensure that the drillhole project is displayed in 3D (see *Displaying Drillhole and Trench Data in 3D*).

2. In the 3D window, select **Grids>Voxel Gridding**.

3. Select the **Continuously Variable** technique. Set this to **Inverse distance weighting**. Press OK.

4. In the **Input Data** tab, select the Dataset to grid, and check that the XYZ field assignments are correct. Select the **Data Field** to grid, such as an assay or geophysical field. Enable the **From-To segments** option (will handle drillhole intervals as segments rather than as points).

Note If you wish to grid different fields in the dataset using a different method, such as lithology with Discrete, these separate voxel models can then be merged using **Grids>Voxel Utilities**.



Optionally, prior to gridding, you can reduce the input drillholes to just those with good assay sampling and are closely spaced. Selecting a subset of drillholes will improve the speed of the gridding process.

5. On the **Conditioning** tab use the *Data Conditioning* button to remove abnormal or invalid data, such as negative values representing missing samples and spurious outliers, as well as providing Null value assignment and handling. It can also be utilised to query out and display specific portions of a dataset, for example all downhole gold assays between 2g/t and 5g/t, or only intervals with a QBX or QV lithological code.

6. In the **Size** tab, use the **Auto** button to set the cell sizes to **Low resolution** (i.e. larger grid cell sizes). Higher resolutions (smaller grid cell sizes) should only be set after the gridding parameters have been extensively refined- they will create a much larger block model (memory consumption) as well as significantly increase the processing time.

7. In the **Size** tab, click **Detect Rotation**. This will set a default rotation angles to match the trend in the input dataset. Click the **Preview** button to visualise this mesh.

Note If custom cell sizes are entered, ensure that the **Auto>Fit to extents** option is then applied to recalculate the number of cells to cover the input dataset.

---

8. In the **Search** tab, select **Elliptical Search.**

    - Leave the default Strike, Plunge and Dip orientations of 0.

    - Set the **Major** and **Minor axis** search values at approximately a third to half the average spacing between the holes.

    - Set a **Depth axis** value of approximately half of the XY axes, which corresponds to the much higher sampling downhole.

    - Set a **Search Expansion** of 3. This will result in the gridding algorithm using the initial search radius defined; if it cannot find any data points or meet the sample selection criteria within this radius, it will then try a double size radius and finally a triple sized radius. If it still cannot calculate a cell value with data points within this volume, the cell value will be set to Null.

    - Also set **Grid passes** to 3. This is useful for uneven data as initial ‘rough’ pass is performed at 1/3 the resolution to determine areas where to perform subsequent half and full resolution passes will be calculated.

9. Still in the **Search tab**, set the **Sample Selection** parameters to 2 Z sectors, and 2 points for both minimum and maximum. These set the required number and distribution of data points in the search area before a cell will be calculated. The more stringent these rules, the more likely the output model will reflect the input data.

    The **Sample Selection** criteria can be relaxed by enabling the **Gridding Rule** option.

10. The **Preview** button at the top of the **Search** tab allows the user to visualise the resulting search volume with an overlay of the input data points in a rotatable 3D view. The search distance and number of search expansions can be dynamically altered from within this view. Press Next to advance to the final dialog.

11. The **Method** tab provides a number of **Weight Models**; leave the default **Power model** selected. These control how the data points within the search volume are weighted in the final cell value calculation, based on their distance from the cell centre. Press Finish to start the gridding process.

12. Upon gridding completion, a **Save** grid dialog will appear, prompting for a file name and location. The output grid is saved as an Encom 3D format (.E3D). Upon closing this dialog, the new grid will be displayed in the 3D window and also the *Display Assistant* dialog.

For more information about inverse distance weighting and other gridding methods available, see *3D Gridding Methods*.

3D Gridding Methods

- *Continuously Variable Gridding Methods*
- *Discretised Gridding Method*
- *Distance Gridding Method*
- *Density Gridding Method*

Continuously Variable Gridding Methods

This technique is used to grid variably spaced numeric data such as drillhole assay values. It smoothly interpolates values at each grid node based on a weighted function of the surrounding data. Two weighting methods are available:

- *Inverse Distance Weighting*
- *Kriging*

Inverse Distance Weighting

Inverse Distance Weighting (IDW) is a universal technique that can be applied to a wide range of spatial data. IDW uses weighted averages to estimate new grid cell values and can be used as either an exact or a smoothing interpolator. The value assigned to each grid cell in an output grid is calculated using a distance weighted average of all data point values that fall within a specified search radius if the grid cell.

The IDW method is optimal when the input data is uniformly distributed throughout the volume to be gridded, and some degree of smoothing is required in the output grid.

IDW gridding provides the following wizard dialogs:

1. *Input Data*
2. *Data Conditioning*
3. *Grid Size*
4. *Search Parameters*
5. *Method*

Kriging

Kriging is a geostatistical gridding method originating from resource geostatistics, which is used due to its flexibility and data-driven approach to surface interpolation. Kriging is an advanced technique which is based on the assumption that the spatial variability in a measured property is neither due to totally random or deterministic constraints. The main advantage of Kriging over simpler interpolation techniques such as IDW is that it is data driven and uses a weighting model which is adaptive to the inherent trends in a data set rather than imposing a set of fixed conditions upon them. Using Kriging in interpolation can be a complex process as it requires an intimate knowledge of the structure and variability in the data set in order to choose an appropriate sample model and set of gridding properties.

Over the past several decades Kriging has become a fundamental tool in the field of geostatistics. The method of interpolating a surface using Kriging is generally performed as a two stage process:

Stage 1

The input data is analysed to establish the spatial predictability of the measured values in the study area. This analysis generally focuses on the spatially correlated component of the data by means of determining the degree of spatial dependence among the sample points. The average degree of spatial dependence among variables is summarised in a plot known as the semi-variogram. The semi-variogram is a concise means of representing the average intersample variation according to sample separation distance and direction. In order to use the sample variance as part of the interpolator in the Kriging process it is necessary to model the semi-variogram in order to define a mathematical function which optimally describes the underlying structure in the data. This process is known as variogram modelling which in itself can be a very involved and complex task. Once an appropriate model has been chosen it can then be used to estimate the semivariance or weighting at any given sample distance.

Stage 2

New values are interpolated or estimated at locations which have not been adequately sampled. This process is known as interpolation 'Kriging'. The simplest technique, known as "ordinary kriging" uses a weighted average of the neighbouring samples to estimate the unknown value at a given grid node. The weights are optimised for each node using the variogram model, the distance to the surrounding samples and the inter-sample variance.

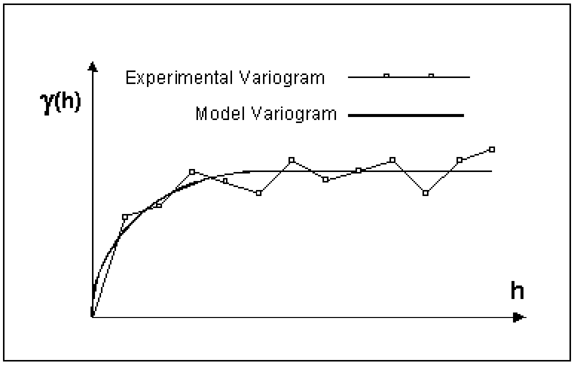
The first step in kriging is to construct a variogram (or semivariogram) from the input data which describes the spatial correlation between the sample points. A variogram generally consists of two parts:

- The experimental (or sample) variogram, and
- The model variogram (a descriptive function which mathematically models the experimental variogram).

The degree of spatial dependence among sample points is measured by the average semivariance:

$$\gamma(h) = \frac{1}{2n}\sum_{i=1}^{n}[z(x_i) - z(x_i + h)]^2$$

Where *h* is the distance or *lag* between sample points, *n* is the number of samples separated by *h* and *z* is the attribute value of interest. The computation of *γ(h)* is performed in two steps. First pairs of sample points are grouped together by distance. For example, if the distance interval (or lag) is 1000m then pairs of points separated by less than 1000m are grouped together into a 0-1000m lag, samples separated by a distance of between 1000 and 2000m would be grouped into a lag of 1000-2000m and so on. Next the average distance *h* and the average semivariance *γ(h)* is calculated for each group. If spatial dependence exists among the samples, then pairs of points closer together will have more similar values than pairs that are further apart. The semivariogram is a plot which has the average semivariance *γ(h)* along the y-axis and the separation distance *h* along the x-axis.



Experimental and Model Variogram used in kriging

The semivariogram can be broken down into thee main components; the **Nugget**, **Sill** and **Range**.

Nugget – is the semivariance at a distance of zero and represents the degree of sample repeatability or spatially uncorrelated noise.

Range – is the spatially correlated portion of the semivariogram that exhibits an increase in the semivariance with distance. Towards the limit of the range the semivariance levels off such that with additional increases in distance it is indistinguishable from one point to the next. This point of flattening is called the sill.

Sill – is the point at which the semivariance (range) levels off to a relatively constant value.

Once an experimental variogram has been computed, the next step is to define a model variogram. A model variogram is a mathematical function that models the trend in the experimental variogram. Once the model variogram is constructed, it is used to compute the weights which are used in the Kriging interpolator. The basic equation used in ordinary kriging can be described as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i$$

Where n is the number of points in the data set, f_i are the attribute values of these points, and w_i are weights assigned to each point. This equation is essentially the same as the equation used for inverse distance weighted interpolation except that rather than using weights based on an arbitrary function of distance, the weights used in Kriging are based on the model variogram. The creation and analysis of the sample and model variogram is done using the **Variogram** dialog on the *Statistics Explorer Tool* , which is accessed from the **Kriging** dialog (see *Computing a Sample Variogram* and *Defining a Model Variogram*).

The gridding wizard dialogs utilised by Kriging are virtually identical to IDW:

1. *Input Data*
2. *Data Conditioning*
3. *Grid Size*
4. *Search Parameters*
5. *Kriging Estimation Method*

The main difference is the final **Method** tab, which provides Kriging-specific options.

Discretised Gridding Method

This method uses **Inverse Distance Weighting** to interpolate discrete or classified data. An example would be to classify geological rock codes recorded for drillhole samples and assign these to a voxel model. Both string and numeric data can be classified by this method. Once the data has been classified it is then interpolated into a grid using similar techniques to the continuous interpolation method. The main difference being the value assigned at each grid node represents a specific classification code rather than a weighted value. The classification code which is assigned to each node is the one with the highest probability of occurring at the interpolation point.

The method produces a legend file as one of the products of the gridding process. This legend file can define surface and line colour and pattern for each classification. It can be used as a rendering guide when the grid is displayed.

Discretised gridding presents the following gridding dialogs:

1. *Input Data*
2. *Data Conditioning*
3. *Classification*
4. *Grid Size*
5. *Search Parameters*
6. *Method*

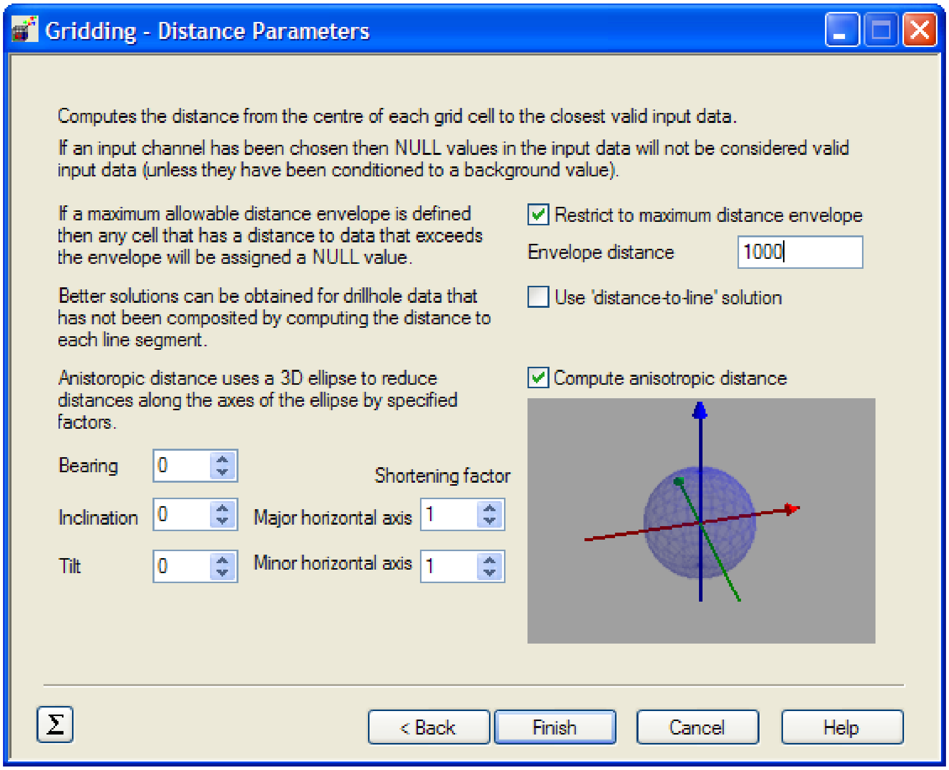
Distance Gridding Method

This produces a grid that records the distance from the centre of each grid cell to the nearest valid data point. It is useful for identifying gaps in data coverage. It is also used as an input to some filtering processes because it can be used to define nodes in the output grid that are ‘near’ to input data points.

Distance gridding utilises the following wizard dialogs:

1. *Input Data*
2. *Data Conditioning*
3. *Grid Size*

In addition, the final dialog is the **Distance Parameters** dialog. In this, a **maximum distance envelope** can be specified to clip the grid to from the nearest input data. Any grid node that is outside this envelope is assigned a null value.



Anistropic distance options can be defined which works in a similar way as discrete search parameters. See *Anisotropic Search* for more detail.

If you have checked **From-to segments** in the *Input Data*, a more accurate solution can be obtained by also considering the perpendicular distance to each segment as well as the end points (the **distance-to-line enhancement** option). This has the unfortunate side-effect of severely reducing the speed of operation. If the segments have been discretised (see *Discretised Gridding Method*) sufficiently then it is usually sufficient to use just the point data. Note that the performance of the algorithm can be improved by specifying an envelope distance.

Density Gridding Method

The **Density gridding** method produces a grid which records a measure of the point density at each grid cell.

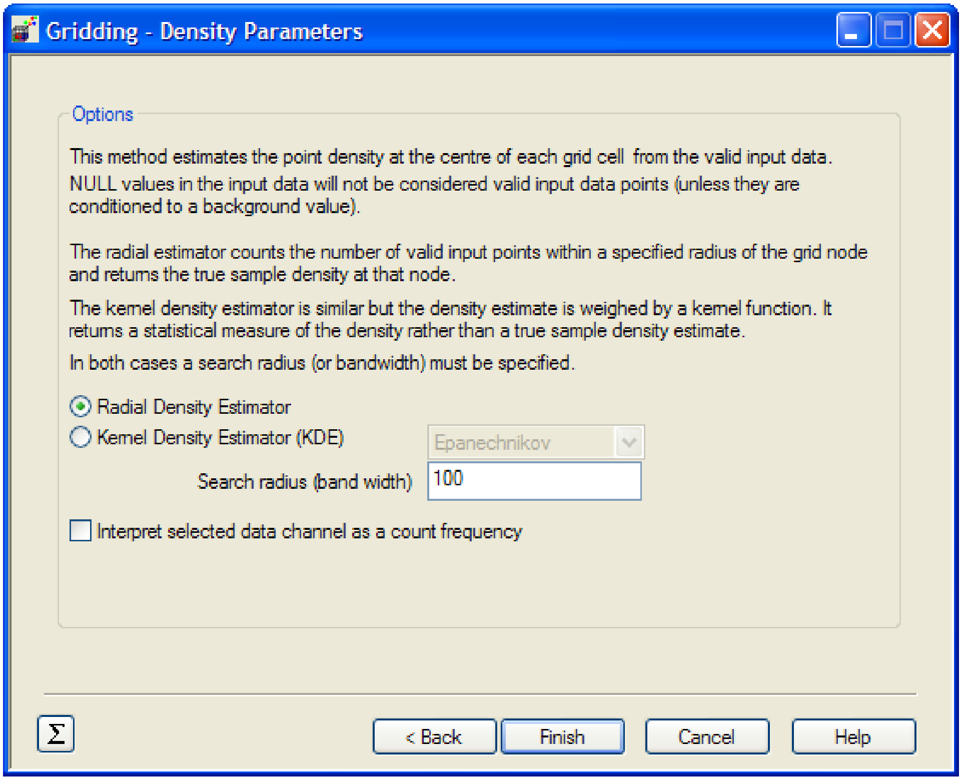
As with Distance gridding, it utilises the following gridding wizard dialogs:

1. *Input Data*

2. *Data Conditioning*

3. *Grid Size*

Additionally a **Density Parameters** dialog provides two estimator function options for the independent calculation for each grid node: the **Radial density estimator** and the **Kernel density estimator** (KDE).



The **Radial method** returns a true measure of the point density at each grid cell (measured as the number of samples per cubic volume unit – usually metres). It is a simple method that counts the number of input samples within a specified radius of the grid cell position and then normalises that count by the area of the search.

The **Kernel density estimator** is a non-parametric density estimator. It uses a similar approach but it weights the input samples by a kernel function that is normally a function of the normalised distance of the sample to the grid cell. To achieve a good result with the KDE function it is more important to choose an appropriate search radius – sometimes referred to as the bandwidth – than to choose an appropriate kernel function. If the bandwidth is too small the density will be under-smoothed whereas if the bandwidth is too large the density will be over-smoothed and lacking in resolution.

Given a kernel function *K* and a search radius (or bandwidth) *h,* the estimated density at any point x is given by:

$$\hat{f}(x) = \frac{1}{n}\sum_{i=1}^{n} K\left(\frac{x - x(i)}{h}\right)$$

Where *n* is the number of samples.

The following kernel functions are supported:

Kernel	**K(u)**
Uniform	$\frac{1}{2}I(\mid u \mid\leq 1)$
Triangle	$(1- \mid u \mid I(\mid u \mid\leq 1)$
Epanechnikov	$\frac{3}{4}(1 - u^2)I(\mid u \mid\leq 1)$
Quartic	$\frac{15}{16}(1 - u^2)^2 I(\mid u \mid\leq 1)$
Triweight	$\frac{35}{32}(1 - u^2)^3 I(\mid u \mid\leq 1)$
Gaussian	$\frac{1}{\sqrt{2\pi}}exp(-\frac{1}{2}u^2)$
Cosinus	$\frac{\pi}{4}cos(\frac{\pi}{2}u)I(\mid u \mid\leq 1)$

If you have taken multiple samples at each input data location and this information is recorded in the input source data you can use this information to bias the density estimation.

Often a density grid will contain a large number of cells with zero density. These can be converted to NULL values via the grid calculator using an expression like NEW = if (OLD == 0, null, OLD) which will make visualising the grid easier.

Note that this method only uses point samples, so to ensure that segment sampled data is accurately represented, use the *Advanced Gridding Properties* options to set a suitable **Segment break length**.

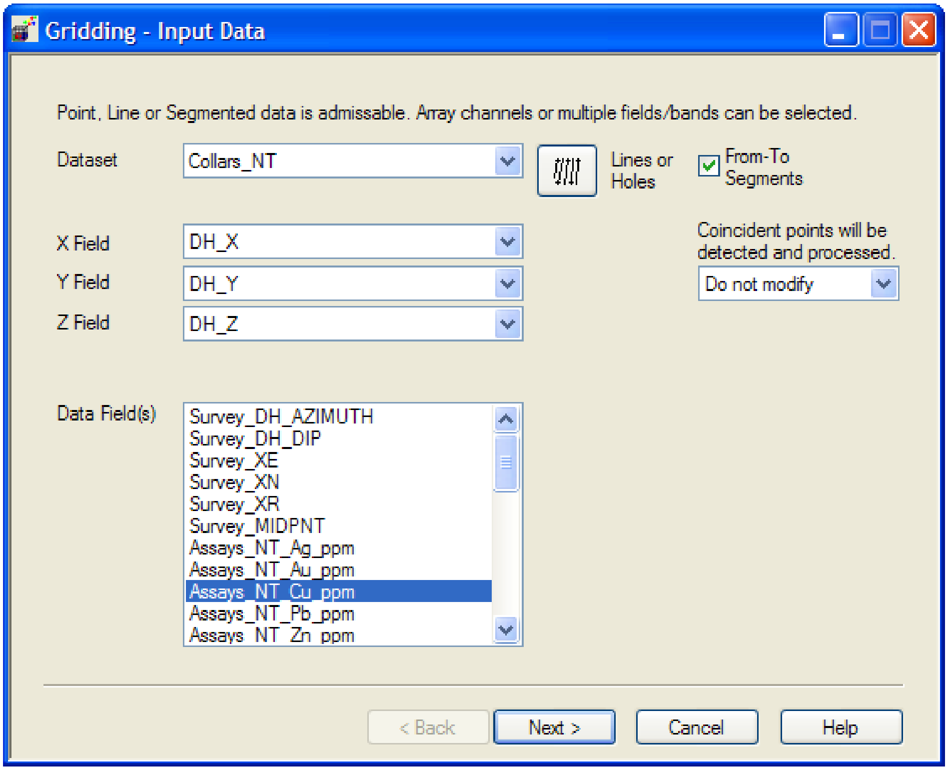
3D Gridding Parameters

The parameters used in voxel gridding are controlled from a series of dialog boxes displayed by the Gridding Wizard. Some dialogs are common to several methods and some are specific to one or another method (see *Continuously Variable Gridding Methods*, *Discretised Gridding Method*, *Distance Gridding Method*, and *Density Gridding Method*):

- *Input Data*
- *Data Conditioning*
- *Classification*
- *Grid Size*
- *Search Parameters*
- *Method*
- *Kriging Estimation Method*

Input Data

The second dialog (**Input Data**) of the Gridding Wizard requires a source dataset to be selected from those currently loaded in Discover 3D. Both drillhole, 3D point and 3D line datasets can be used. The input lines or drillholes can be specified manually or via a graphical selection dialog by clicking the **Line Selection** button.



Specify the input dataset to be gridded plus the data fields

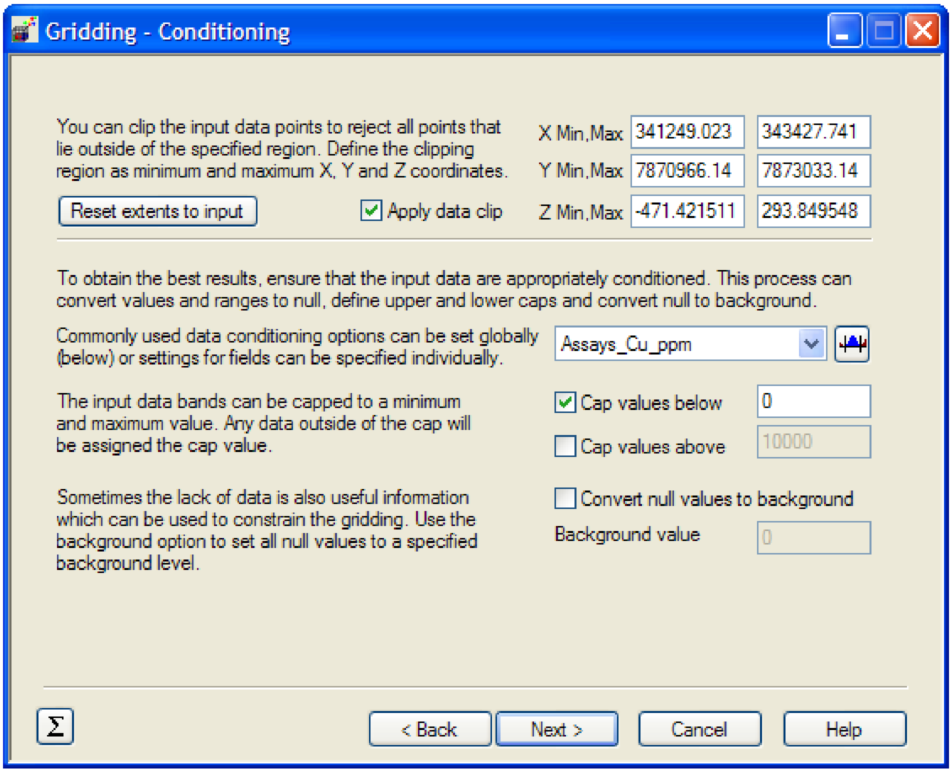
By default the input data is assumed to be 3D point source data. Alternatively, you can specify that the input data are **From-To Segments**, i.e. drillhole intervals. In this case, each sample is defined by a beginning and end point location, and the value of the data over the segment is assumed to be constant. This results in more accurate solutions as the distance from the interpolation point to the sample may be less than the distance to either of the sample end points.

X, **Y** and **Z** (RL) **fields** need to be specified from the appropriate pull-down lists, as well as a **Data field** (e.g. an assay field for Continuously Variable gridding, or a lithology field for Discretised gridding). Alternatively a multi-banded grid can be created by selecting multiple data fields (use the CTRL or SHIFT keys for multiple selections); the output grid will have multiple independent properties stored at each grid cell. Note that not all export formats support multi-banded grids.

A range of **Coincident point** handling options are provided in a pull-down list (average, minimum etc). If using drillhole interval data with the **From-To Segments** option enabled, it is recommended that this is set to **Do not modify**. Using any other option in this instance may result in modification of the value of the assay along the entire interval, as the first point of each interval is likely to be coincident with the last point of the previous interval.

Data Conditioning

The **Conditioning** (third) dialog of the Gridding wizard allows the source dataset to be **clipped** to specified X, Y and Z extents by enabling the **Apply data clip** option. These values can be reset to the initial dataset extents by clicking the **Reset extents to input** button. All source data points outside the defined region are ignored and do not contribute to the gridding. The **Line Selection** button (see *Input Data*) can be also used to remove lines and holes that are not to be used in the gridding process.



Conditioning of data to be gridding using the 3D Toolkit

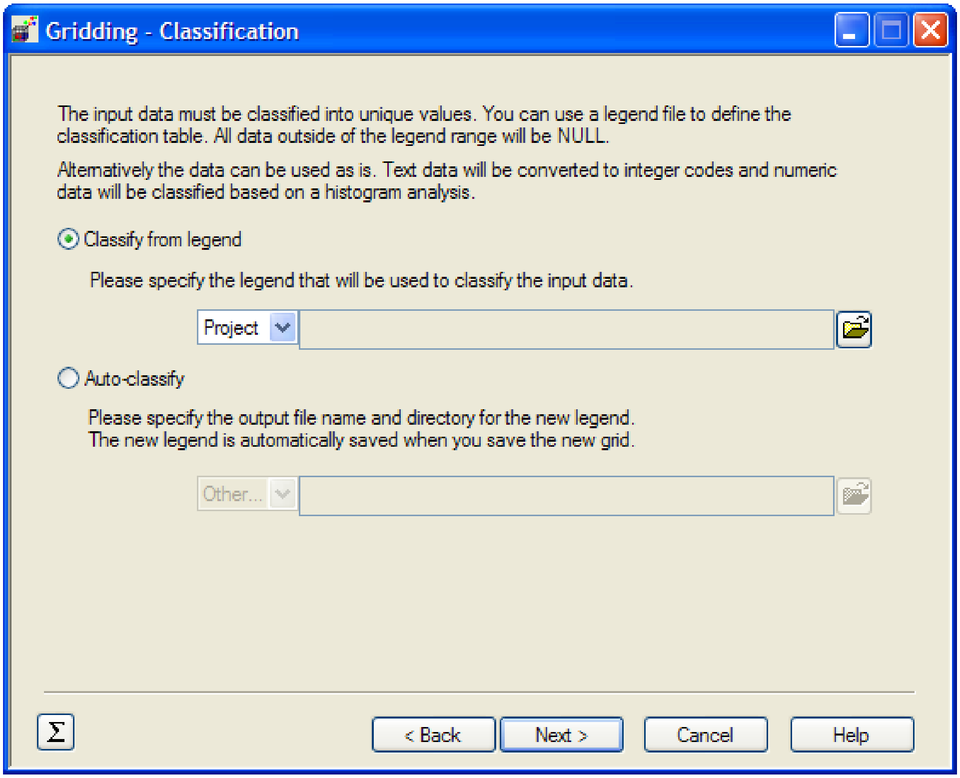
This dialog also provides a range of options for **data conditioning** prior to 3D gridding. This process is essential to eliminate poorly defined data from the input dataset, such as negative values representing missing samples (e.g. Sample Not Received (SNR), Below Detection limit (BDL), etc). It also provides data capping options (e.g. capping gold assays in mineralization systems prone to nugget effects), as well as conversion of Null values to a user-defined background value. Failure to remove such artefacts can result in meaningless output grids.

Setting a **Cap values below** or **above** value will cap source data outside the set limit to the limiting value. For example, with **Cap values above** set to 500, a gold assay on 725ppm will be handled during 3D gridding as a 500ppm value. It is also possible to **Convert null values to a background** value specified by the user, in order to constrain the gridding. For example, if gridding drillhole geochemical assays, much of the hole may not have been sampled and in these areas the assay result may be assumed to be equal to the background value. This helps prevent anomalies ‘ballooning’ into areas with no source data coverage.

These conditioning options presented in the main dialog are global settings: if multiple data fields have been selected in the **Input Data** dialog (i.e. in order to create a multi-banded grid), these settings will affect all fields equally.

Further conditioning controls are available via the **Advanced Settings** button (shown left) to the right of the listed data field. This opens the Field Data Conditioning dialog (see *Data Conditioning*). If multiple data fields have been selected in the Input Data dialog (see *Input Data*), the **Advanced Settings** button can be used to set individual capping and background values for each field by selecting the required field from the pull-down list.

Classification



The **Classification** dialog provides two options for classifying unique values:

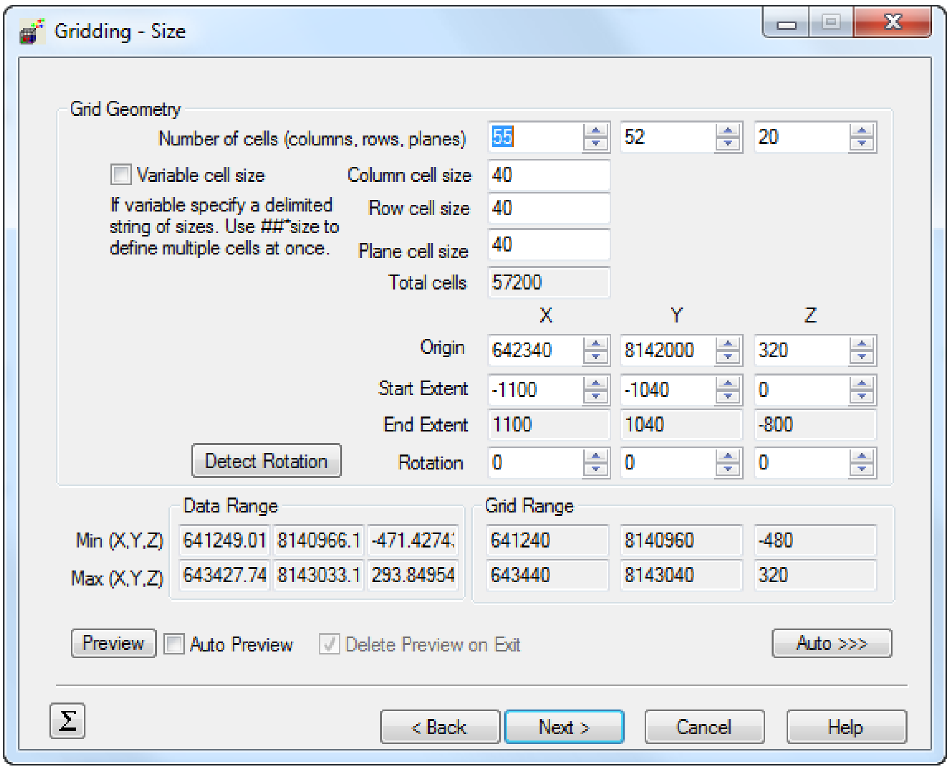
- **Classify from Legend**: use an existing .leg file to classify the input data. Existing legend files which have been created (for example to colour downhole lithologies in a drillhole project) can be used. For the location, ensure you select **System** and define a Legend name. They will then be available in the Legend Editor and other dialogs. Any data which does not match a record contained in the legend file will be treated as a NULL value in the output.

- **Auto-classify**: creates a new legend using the entire input dataset. All input data will be classified unless it has been set as **Null** in the **Field Data Conditioning** dialog. For the location, ensure you select **System** and define a Legend name. They will then be available in the Legend Editor and other dialogs.

Grid Size

The Gridding Wizard **Size dialog** allows the dimensions and extents of the output 3D grid to be specified. Required parameters are:

- Basic grid cell size (individual X (column), Y (row) and Z (plane) dimensions).
- Number of cells in the X, Y and Z orientations.
- The grid origin: by default this is set to the centre of the model.
- Start and End extent: the relative distance of the first and last cell centre-point in the model.
- Rotation of X/Y/Z axis about the Origin point.



3D grid size specified using the Auto Med Res option

The simplest option for setting these parameters is by using the **Auto** button (shown left). It has three settings (High, Medium and Low Res) for calculating the appropriate origin, cell size and cell count to cover the source data extents at the specified resolution.

In addition to the Auto size options, there is also an option to **Detect Rotation** of the input dataset. This optimizes the coverage of the grid mesh to cover the dataset.

Once parameters are set, the wireframe mesh can be previewed at any time by selecting the **Preview** button. Or enable the **Auto Preview** option to dynamically update the preview any changes are made to the Size dialog.

Alternatively the basic grid **cell size** can be set manually by setting the desired **Column**, **Row** and **Plane cell sizes** and then using the **Auto** button's **Fit to Extents** option to calculate the corresponding origin and number of cells in the three dimensions to cover the source data appropriately.

The cell size can be fixed or variable. For example, a larger cell size can be specified on the edges of the 3D grid and a finer cell size in the centre. A variable cell size can be set by checking the **Variable cell size** option and defining the cell sizes in all three dimensions using a count*size notation. For example 5*200 10*50 5*200 specifies a total of 5 cells with width 200 metres on both outside surfaces and 10 cells of 50 metres width in the middle.

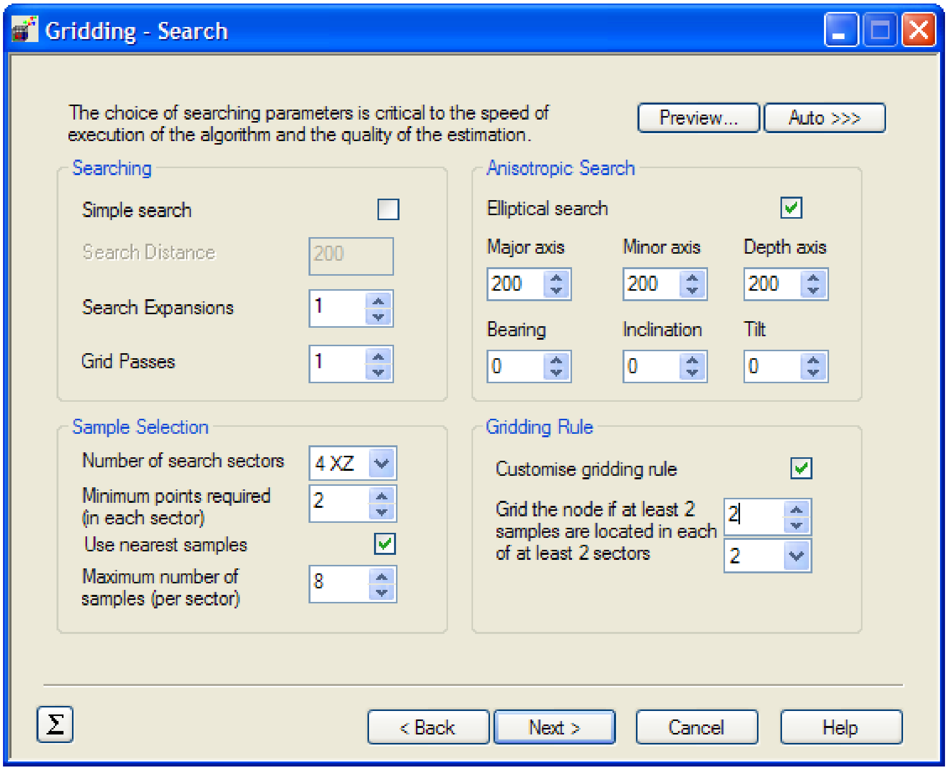
It is also possible to manually set the **Number of cells** and/or **Origin** coordinates in each of the three dimensions. Note that manually altering the **Number of cells**, **Origin** coords or applying **Variable cell sizes** can easily result in an inappropriately sized output grid: for example by gridding only a small portion of the source data extents (if any at all) or alternatively creating a grid far larger than the source data extents. Comparison of the **Grid Range** extents to the **Valid Data Range** extents is essential when altering these parameters manually.

The **Load** and **Save** options under the **Auto** button retrieve or save settings to a grid Extents file (.E3X). The **Match Grid** option allows you to browse to an existing grid on disk and retrieve the grid size from that.

The definition of 2D grids is similar except that variable cell sizes are not supported and only a single plane can be generated.

Search Parameters

The **Search Parameters** dialog is displayed for the *Continuously Variable Gridding Methods* (Kriging or IDW) and *Discretised Gridding Method*. It provides a plethora of options (similar to those in the 2D gridding tool in Discover) for refining the search algorithm, allowing the speed of grid creation to be balanced against the quality of the output grid.



Specifying the Inverse Distance Weighting parameters using a Simple Search

The interpolation algorithm proceeds by examining each grid cell in turn. It finds the distance from the grid cell centre to all source data points. Using one of a variety of weighting schemes, it interpolates a value at the node by summing the contribution from all data points, weighted according to the distance between the grid cell and the data point. The weighting scheme ensures that data close to the grid cell contributes more to the final value at that cell than data far from the cell.

This is a simple technique that suffers from one major problem – it is slow and runs in a time proportional to the number of grid nodes multiplied by the number of input data points. In many cases, it is simply too slow and so a variety of techniques are used to improve performance. Unfortunately all of these techniques degrade the quality of the solutions. The key to generating a quality grid is to balance speed requirements against quality requirements and experimentation is almost always necessary. For this reason it is recommended to experiment with a low resolution grid first and then generate the final grid at the required resolution once the appropriate gridding parameters have been determined.

Searching

The first option designed to improve gridding performance is to introduce a search **Distance**. The algorithm will only search for contributing data points within this specified distance of the grid cell. Data points outside of this distance will be ignored.

However, this can result in a node failing to be gridded (i.e. assigned a null value) if insufficient data points are found within the search distance to satisfy the algorithm. Two additional options can help resolve this:

- A **Search Expansion** (up to 5 expansions) can be specified. If an insufficient number of data points are found within the search distance, the search distance is increased (by the initial radius) and the search is repeated. This occurs up to ‘n’ times – as specified by the user. The final search distance therefore is the number of expansions multiplied by the initial search distance: i.e. specifying a search distance of 15m with three search expansions would result in three consecutive searches of 15m, 30m and 45m.

- By setting the number of **Grid Passes** to greater than one (5 maximum), the gridding algorithm will conduct the specified number of passes over the dataset, adding the grid nodes populated in the first pass as input data for the next pass. With each successive pass, the gridding resolution will increase: the search radius for any pass is equal to the pass number (which is decreasing e.g. 3, 2, 1) multiplied by the specified search distance. Thus with a search distance of 25m and three grid passes, the first search pass would utilise a 75m search radius, the second 50m (incorporating any nodes gridded in the first pass as additional input data), and the last 25m (again with the output nodes of the second pass as additional input data). This feature is useful if the input data points are unevenly distributed as it fills in the gaps in coverage more efficiently.

The 3D Gridding also includes a **Simple** mode that does not restrict the search radius and allows all input data to contribute to each solution. This may, however, take a long time to run, but the grid generated is often quite smooth and may serve as a guide for further gridding attempts.

Anisotropic Search

When gridding data that is sampled sparsely in one or two dimensions and densely in another, it is advisable to use an **Elliptical search**. This can also be a powerful tool when a regional trend is interpreted to be influencing data distribution, e.g. a regional NE-trending fault/fluid conduit system may affect the soil sample geochemical trends. You may wish to weight samples along strike of a grid node more than those parallel to it.

Elliptical searches use the search **Distance** as the default ellipsoid axes lengths (ie spherical search).

Note The orientation of the Anistropic search ellipse is independent of the Grid Size (mesh wireframe) orientation. For example if Z is 0 degrees, then the search axis will be vertical, regardless of the grid's rotation.

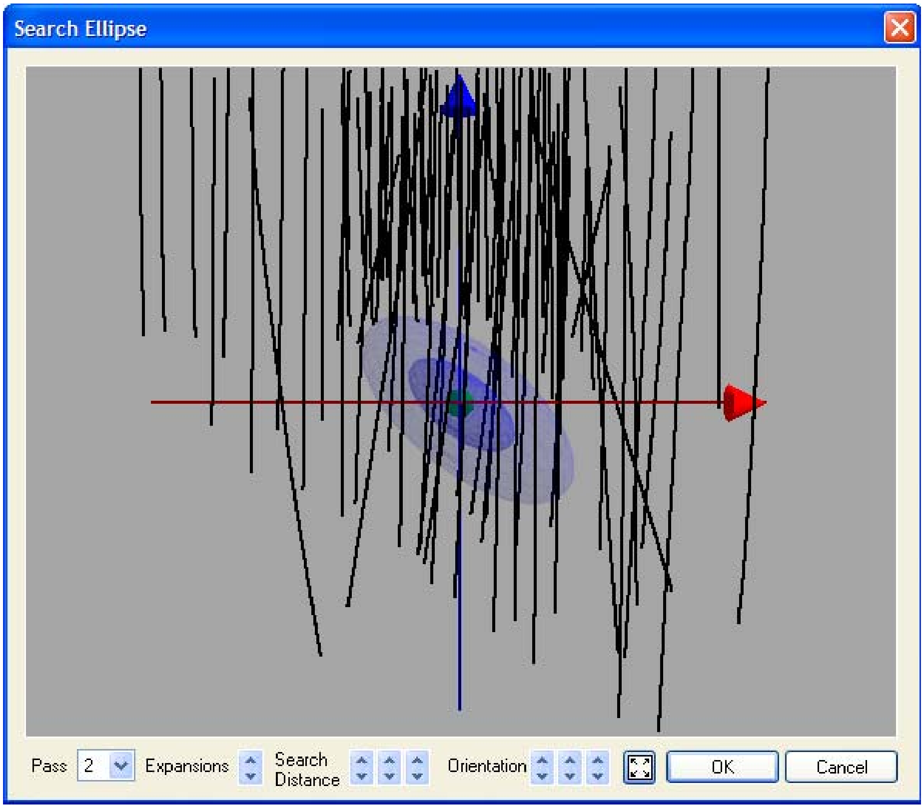
The orientation of the ellipse from the XYZ axes is denoted by the **Bearing** (orientation or azimuth of the major axis), **Inclination** (the dip or plunge of the major axis, which is the angle above the horizontal plane), and **Tilt** (the dip or plunge of the minor axis, which is the angle above the horizontal plane) of the elliptical search area.

You can specify a **Major axis** length (or search distance), a **Minor axis** length and a **Depth axis** length. For the default orientations of 0 degrees, these correspond to the X, Y and Z axis respectively.

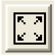
Note The convention for ellipses is that the Major Axis>Minor Axis>Depth Axis. You can use the orientation controls to place these axes in the desired position for your data.

Preview

The current search spheroid or ellipsoid can be viewed in a simple 3D preview window with the source dataset by clicking the **Preview** button at the top right of the **Search** dialog.



The controls at the base of this dialog link directly back to those in the **Search** dialog. You can therefore modify ‘on-the-fly’ the search distance, ellipsoid axis lengths and orientation, as well as the number of grid passes and expansions, visualising these changes immediately with respect to the source data distribution.



The **View All** button will reset the preview screen to the extents of the entire dataset.

Search parameters can be saved and reloaded as a grid Parameters file (.e3g) using the **Load** and **Save** options under the **Auto** button.

Sample Selection

Using a search distance can result in the contributing data points not being evenly distributed about the grid cell, resulting in poor solutions. **Search sectors** are a powerful way to ensure that input data is evenly spatially distributed about the grid cell. 1, 2, 4 or 8 sectors can be specified; for 2 or 4 sectors the axis (or axes) which separate the sectors is also specified. The higher the number of sectors specified, the more evenly distributed the source data must be about the grid node, but the more likely the gridding will fail for the node. This criteria can be made even more rigorous by specifying the **Minimum points required in each sector**. In densely distributed datasets with multiple points in each sector, the **Maximum number of samples per sector** to be used in the grid node calculation can also be controlled.

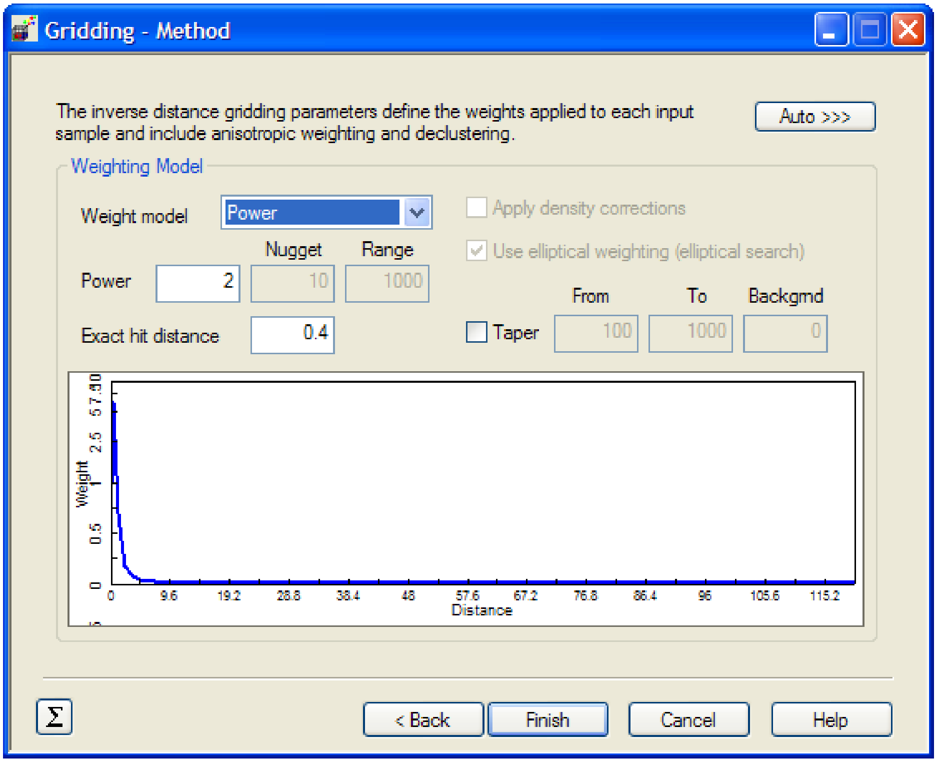
Gridding Rule

In cases where rigorous search sector parameters result in too few nodes being gridded, it is possible to relax these rules using the **Customised gridding rule**. This option allows the node to be gridded if at least *n* samples are located in each of at least *m* sectors, where either *n* or *m* (or both) can be less than the values specified in the *Sample Selection* section of the dialog.

Method

The final **Method** dialog is presented only for the *Inverse Distance Weighting* and *Discretised Gridding Method* techniques. This dialog provides a choice of weighting schemes and associated parameters which are applied to the input data after the *Search Parameters* have been applied. Weighting models include: **Linear, Exponential, Power law** and **Gaussian**. The default option is a **Power Law**.

The specific weighting schemes are described below. A graph depicting the variation in **Weight** with **Distance** is provided at the bottom of the dialog for visual reference. The graph is dynamically updated whenever the method changes.



The Method tab for the IDW technique

Weighting Models

Linear

The weight assigned to each data point which is used to interpolate the value at a grid node is proportional to its Euclidian distance from the node being interpolated. If a simple linear (to cut-off) weight model is selected, the **Nugget** and **Range** parameters are enabled. These parameters can be adjusted to vary the weighting assignments that are applied within a **Range** and **Nugget** distance. At distances less than the **Nugget** distance, the maximum weight is applied to the input points (i.e. all data values will contribute equally to the averaged node value). The **Range** defines an outer distance threshold for the weight model. Any data points which exceed the Range distance but are less than the **Search Distance** (see *Search Parameters*) will be proportionally weighted by their distance from the grid node such that:

Distance < Range: Weight = $Nugget - \dfrac{Dist}{Range}$

Distance >= Range: Weight = 1

Exponential

The weight assigned to each data point is proportional to its distance from the grid node being interpolated raised to a specified power. Increasing the power value will result in smaller weights being assigned to closer points and more distant points being assigned equal but large weights. Increasing the power value will cause each interpolated grid node to more closely approximate the value of the sample which is closest to it. As with the **Linear** model the **Nugget** and **Range** properties can be modified to constrain the distance over which the exponential weight model is applied.

Distance < Range: $\quad \text{Weight} = \exp\left(-Dist \times \frac{Power}{Range}\right)$

Distance >= Range: $\quad \text{Weight} = 0$

Power

The weight assigned to each data point is proportional to the inverse of its distance to the grid node, raised to a specified **Power**. Increasing the power value reduces the amount of influence distant points have on the calculated value of each grid node. Like the exponential weighting model, large power values cause grid cell values to approximate the value of the nearest data point. Smaller power values will result in a higher degree of averaging such that their values will be more evenly distributed among neighbouring grid nodes. The default weighting value is 2 (i.e. the weight of any data point is inversely proportional to the square of its distance from the grid cell). This is appropriate for most situations. If required, the weighting value can be altered to any positive value.

$$\text{Weight} = \frac{1}{Dist^{Power}}$$

Gaussian

The weight assigned to each data value is determined according to a 1D Gaussian function centred on the grid node. The shape and standard deviation of the Gaussian function is proportional to the **Range** with larger values producing a flatter function and a smoother grid.

$$\text{Weight} = \frac{1}{2^{\left(\frac{Dist}{\left(\frac{Range}{2}\right)^{2}}\right)}}$$

Elliptical weighting is available when the **Elliptical Search** option is enabled in the *Search Parameters* dialog. It adjusts the distance weighting function for data points within the search ellipse depending on their relative position with respect to the elliptical shell defined by the search ellipse. Points located on the same elliptical shell will be assigned equivalent weights, even though their distance from the ellipse centroid may be different.

Tapering (only available for the *Inverse Distance Weighting* technique) allows interpolated values to be reduced to a background value at large distances from a grid node. It can be used to modify the weighting function in situations where the search distance is greater than the **Range** but there is considered to be little correlation with the source data:

- Between a distance of zero and the **From** distance the taper function is assigned a constant value of 1 (i.e. no modification is made to the grid node value).
- Between the **From** and **To** distance the taper function is applied as a linear weighting between the grid node value and the background value.
- Beyond the **To** distance grid nodes are assigned the background value only.

Apply Density corrections dynamically adjusts the search algorithm to optimise grid cell interpolation in areas of data clustering. Activating density corrections can help to enhance detail in datasets where sample points are unevenly spaced (e.g. drillhole geochemical assays) and may, in some cases, produce a smoother or more representative grid. The density correction modifies the weights for each contributing point based on the sample density at that point.

Note Enabling the **Use nearest neighbours** option (in the *Search Parameters* dialog) is a simple form of declustering. If the **Use nearest neighbours** option is enabled the **Apply density corrections** option will be unavailable.

Exact hit distance is a distance tolerance which is used for assigning real data values to grid nodes. As the inverse distance gridding technique attempts to interpolate a continuous function through the data, a certain number of grid nodes will coincide exactly with the position of some of the input data points. Where grid nodes and data points coincide, the distance between them is zero, so by default the data value is assigned a weighting of 1.0 and all other data points in the search radius are given a weight value of zero. This means that grid nodes that are coincident with input data points are assigned the value of the coincident data point rather than an interpolated (averaged) value derived from the data points surrounding it. The **Exact hit distance** defines the distance below which a grid node and data point are considered to be coincident.

The process of assigning the input data value to grid a nodes can produce significant ‘spotting‘ in the output grid, particularly if the data value of the coincident point/grid node deviates significantly from the points surrounding it. By adjusting the exact hit distance it is possible to increase the tolerance distance for which input data values are assigned to grid nodes. Assigning this value to a high number can produce unacceptable spotting or concentric banding in the output grid, while reducing the value below 1 has little or no effect.

Search parameters can be saved to and reloaded from a Parameter file (parameter files are assigned the file extension .e3g) using the **Load** and **Save** options under the **Auto** button.

Kriging Estimation Method

Two types of Kriging estimation are provided; **Point** and **Block**. Point Kriging estimates a point value at each grid node. Block Kriging on the other hand estimates the average value from all points which fall within a rectangular block centred on each of the grid node. Because Block Kriging estimates the average point value for each block it tends to generate smoother grids. Because Block Kriging uses an average of the input sample points in a given region, it is considered to be an imperfect interpolator even when an input sample falls exactly on a grid node. The Block Kriging method provides controls to decimate (or break up) the blocks into a smaller mesh of sub cells, defined by the X, Y and Z increment values. When a decimation factor is applied in Block Kriging, the value assigned to each block is determined as the average of the sub cells inside the block rather than the average of the entire block.

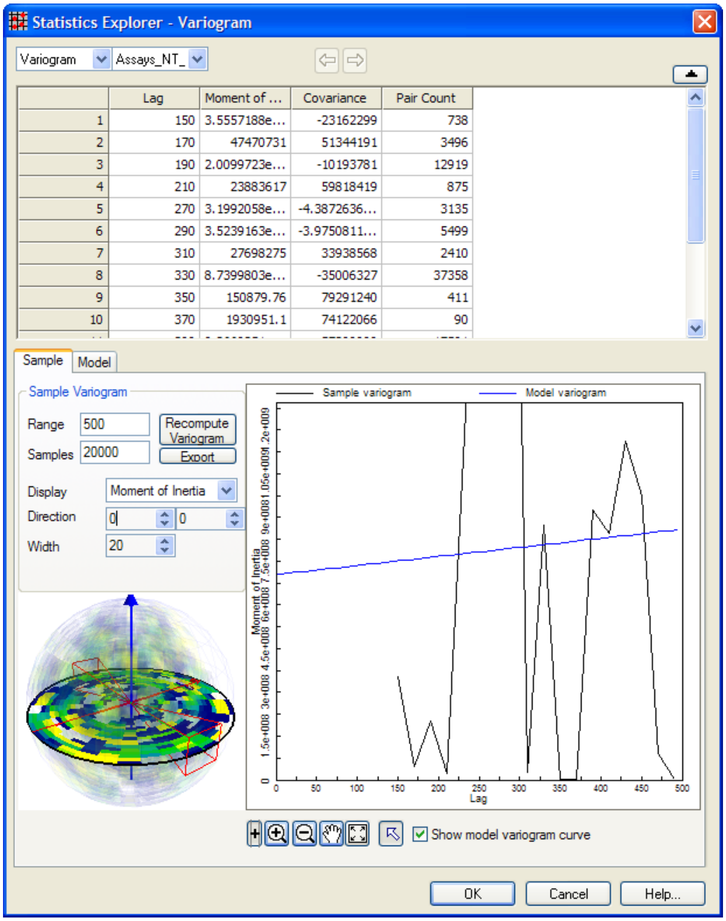
In addition to the point and block estimation types the 3D gridding tool supports two Kriging methods; **Ordinary** and **Simple**. Ordinary Kriging focuses on the spatial correlation component between the measured values but ignores drift (drift is a regional trend in the data which exists is addition to spatial correlation between samples). **Simple** Kriging is similar to Ordinary Kriging with the exception that the weights used do not sum to unity and the average of the entire data set is used in the interpolation of each grid node rather than the local average of points that fall within the **Search Distance** of the node. Consequently, Simple Kriging can be less precise than Ordinary Kriging and generally produces smoother grids.

Computing a Sample Variogram

Selecting the **Variogram** button from the **Kriging** method tab opens the **Variogram** dialog on the *Statistics Explorer Tool*. This utilises the assigned data field in tandem with the spatial coordinates to compute a **Sample variogram**. A variogram shows the degree of correlation between data in a spatial dataset in different directions and at different distances.

Computing a sample variogram is a laborious task and can take a very long time as the number of samples increases. In order to construct a variogram it is first necessary to compute the distances between every possible pair of points in the dataset. In most cases it will be impractical to do this unless the dataset is relatively small (< 30,000 samples). In order to speed up sample variogram calculations, several restrictions are imposed. These are:

- Capping the maximum **range** (distance between points) that will be considered.
- Capping the number of input **samples** that will be considered.



Display of the calculated variogram

A default variogram is computed using parameters that should ensure that the computation time is of the order of a few seconds. Thereafter, the range and maximum number of samples being considered can be modified and the variogram recomputed by clicking the **Recompute Variogram** button. Note that if the maximum number of samples is set to negative one (-1), then all samples in the dataset will be used to compute the sample variogram.

It is not advisable to include all samples in the data set if it exceeds 30,000 samples as the computation time required to build the variogram could be extremely long. If the number of samples is restricted, the algorithm will look at a sub-set of samples that are evenly distributed spatially within the dataset. In some cases it may be desirable to increase the range and increase the maximum number of samples to improve the statistical reliability of the variogram.

Once computed, the sample variogram is plotted in the left lower corner of the **Statistics Explorers Sample** dialog as a radial grid. The default display is the **Moment of Inertia** (or semivariance). The moment of inertia is defined as half the average squared difference between the x and y coordinates of each pair of points. This is equivalent to the spread (separation distance) of the lagged points from the 45 degree line (1 to 1) on a h-scatter plot. A directional variogram is extracted from this grid and is plotted in graphical format on the right of the dialog. The source data is shown in a spreadsheet at the top of the dialog. The red wedges represent the area of the grid that is used to extract the directional variogram. This volume is controlled via **Direction** (0 to 360 degree clockwise from North), **Plunge** (-90 to 90 degrees via the control adjacent to the Direction control) and **Width** (degrees of arc) controls. If the width is set to 180 degrees then direction is ignored and an omni-directional variogram is extracted. Otherwise a direction-dependent variogram is extracted. The variogram records several parameters including the moment of inertia (semi-variogram), covariance and pair count.

The 3D preview is a spatial representation of the statistical measures of the dataset (it's a 3D semivariogram grid). It has X (red), Y (green) and vertical (blue) axes indicators to show orientation in Cartesian space. In the 3D preview you can choose to display 'moment of inertia', 'covariance' or 'sample count'. The colours represent the values of the chosen statistic in 3D space. It uses a linear colour stretch over a HSL Blue-Red colour scale. The radius of the sphere corresponds to the range. The colour at any point in space represents the chosen statistical measure value of data pairs that have that separation in that direction. For many datasets you will often see much of the sphere is blank as there are no data pairs in that direction at that range. The 3D view is intended to help you identify the dominant directions of correlation in your dataset. Depending on the dataset it may not be particularly helpful and you may only want to look at the isotropic variogram. However for other datasets the anisotropy of the correlation can be clearly seen in the 3D sphere.

The extraction plane is the intersecting statistic values from the 3D sphere displayed opaquely on the plane. All other value are displayed semi-transparently. It is designed to help you see the dominant directions of correlation in your dataset. The 2D graph of the semi-variogram is derived from the variogram grid by extracting data in a particular direction over an angular range. This is shown as the pair of red tetrahedrons in the 3D preview. The data that falls within the tetrahedrons is averaged by range and displayed as a graph. If you choose an angular range of 180 degrees then you get the isotropic semi-variogram which averages all the data over the range.

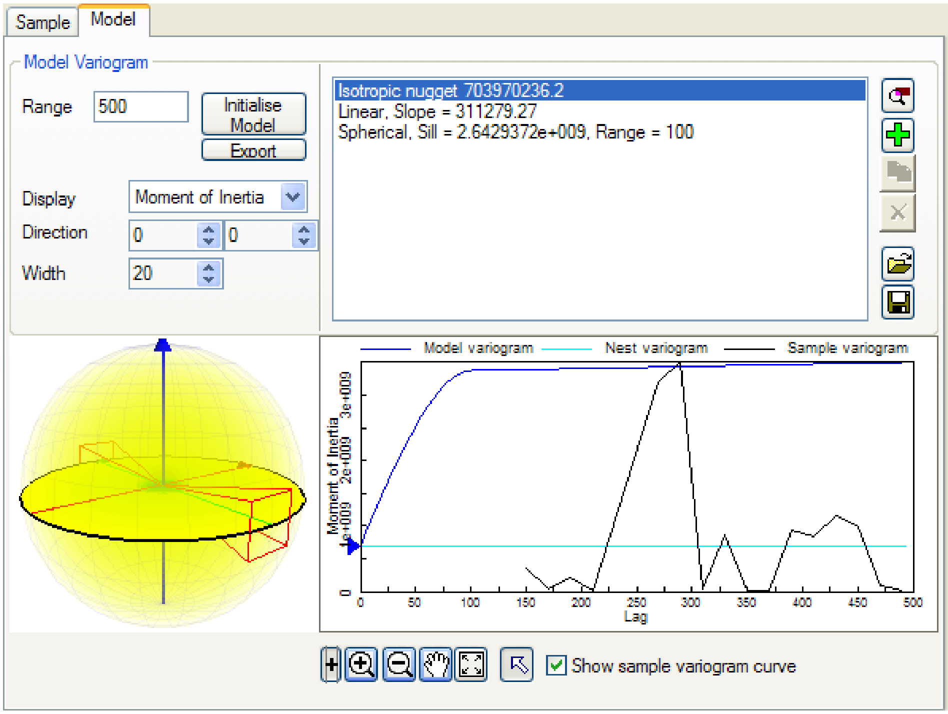
Defining a Model Variogram

Once the Sample variogram has been computed a **Model Variogram** can be constructed or edited using the **Model** tab on the **Statistics Explorers Variogram** dialog. The model variogram will be automatically initialised to a default linear model with a nugget to fit the data. You can click the **Initialise Model** button at any time to reset the model to this default.

The model is plotted to a range controlled via the **Range** edit parameter. By default this is equal to the sample variogram range.

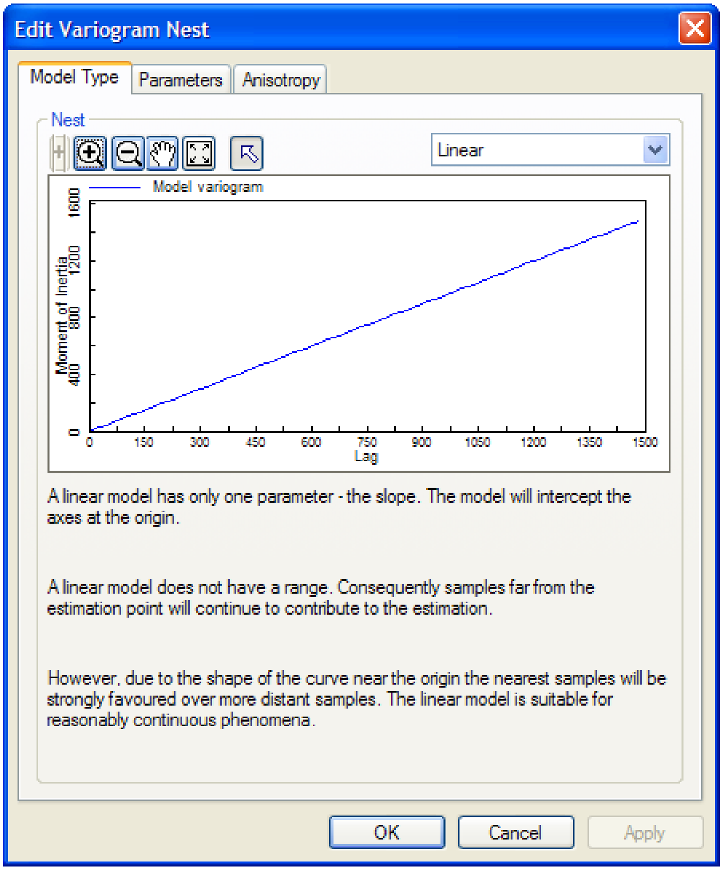
The model variogram is displayed as a grid and also as a directional extraction – just like the sample variogram. The sample variogram curve can be plotted for comparison. In the upper right corner of the **Model** dialog the model variogram nests are displayed. Each nest corresponds to an individual component of the model. Individual nests can be edited, added, cloned and deleted via the buttons on the right or by double clicking on a nest in the list. The final variogram model is generated from the accumulated nests.

The basic parameters of each nest can be altered directly from the ‘model’ graph. To do so select the appropriate nest from the list of models at the top of the dialog and then adjust the parameters by moving the blue edit (triangle) tags on the variogram graph.



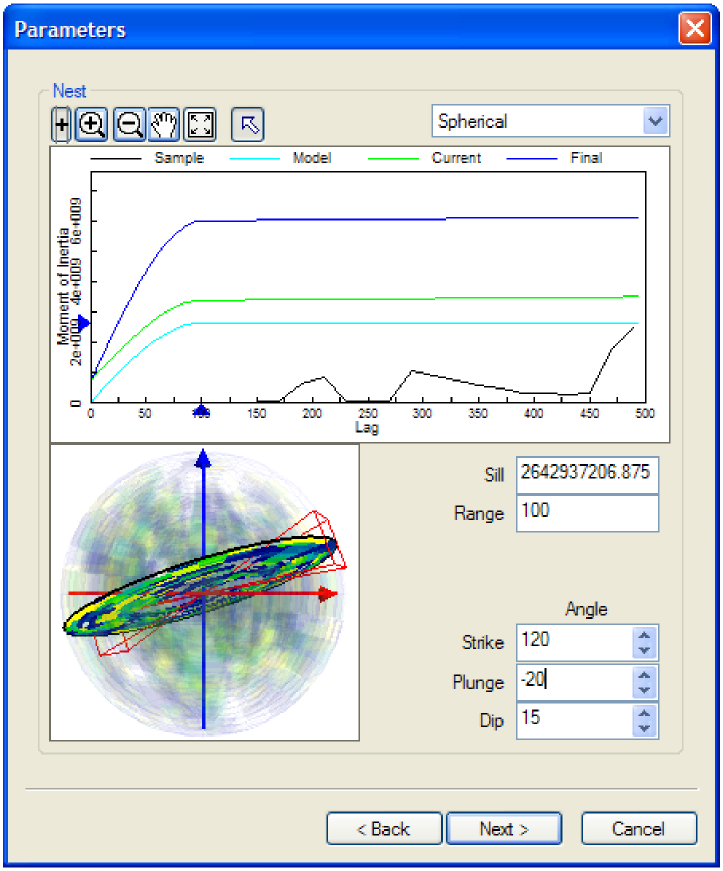
The Model dialog of the Statistics Explorer showing three individual nests of a model variogram. The top portion of the dialog lists the type and parameters of each nest and the corresponding plot at the bottom shows the selected nest graphically.

Editing a nest in the model presents the following **Edit Variogram Nest** dialog. If a new nest is added the dialog is presented in a wizard mode rather than a tabbed mode.



Edit Variogram Nest dialog showing the various Model Types that are available

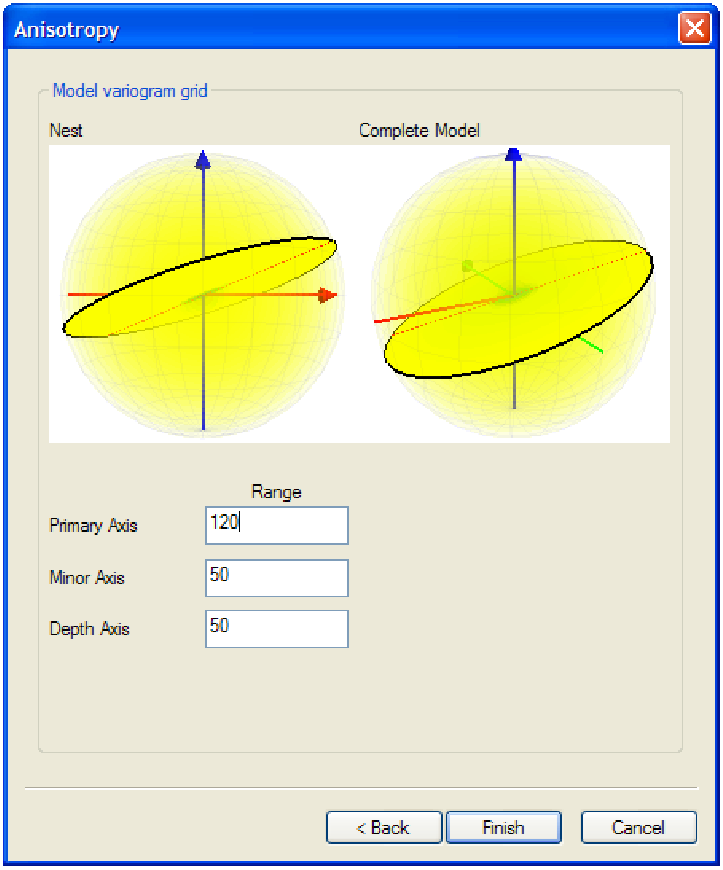
The **Model Type** tab allows you to select an appropriate theoretical model to use as the basis of the nest. A graph is displayed on the Model Type tab to show the selected function. The coordinates used in the example graph are theoretical and are not representative of the actual model coordinates.



Controlling the parameters of the variogram

The **Parameters** tab provides controls for editing the model parameters. At the top right of the dialog is a selection control for switching between the available model types. The graph below it shows the sample variogram, the model variogram *excluding* the current nest you are editing or adding, the currently selected nest and the final model variogram *including* the current nest. The model parameters are presented below the graph and can be edited directly. Most of the model parameters can also be adjusted graphically by dragging the blue triangular handles that are presented on the graph axes. For example, when a spherical model is selected the left vertical (Y) axes handle allows the editing of the sill and the bottom horizontal (X) axis handle allows the range to be edited.

The sample variogram grid is displayed in the bottom left corner of the dialog. The view direction for the grid can be changed by clicking on the plot and dragging the cursor to spin the view in 3D. Changing the view direction of model variogram grid does not change the direction parameters used to extract the model variogram. The grid graph rotation and spin controls are provided for visualisation purposes only. Note that the width of the extraction is equal to the width of the extraction on the main **Sample** tab and cannot be modified here.

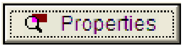


Editing the variogram nest

The **Anisotropy page** controls the range of the model along three axes – the **Primary**, **Minor** and **Depth** axes. A preview of the currently selected nest is shown on the left, whilst the final model preview is shown on the right of the dialog.

The default **Primary Axis** range is equal to the range established on the **Parameters** tab. The **Minor** and **Depth Axis** ranges are used to introduce anisotropy into the model. If the minor and depth axes ranges are set equal to the primary axis range then the model is isotropic.

Advanced Gridding Properties



Advanced **Gridding** properties can be accessed via the **Properties** button on the **Voxel Toolkit** main window.

Load entire hole length: when disabled, Discover 3D will only use that portion of the hole that has valid data. As NULL values can be converted to background and used to constrain the gridding (see *Data Conditioning*), this option is enabled by default.

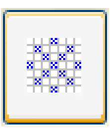
Segment break length: used when loading drillhole data and the **From-to** segment option is enabled (see *Input Data Specification*). Each segment is sub-sampled so that there is a point location along the segment every specified unit of distance. This can be used to speed up the distance to data gridding method and to return more accurate point density grids.

Default numeric null: allows user-definition of the global null value for the input X, Y, Z and data fields.

Default text null: allows user-definition of the global null string for input string/ character fields; only applicable when using **Discretised Gridding**.

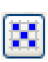
Convolution Filtering

Grids>Voxel Filter



The **Convolution filtering** function on the **Voxel Toolkit** main menu allows one or more 3D filters to be applied to a 3D grid.

The tool enables you to load/import and unload grids from the top grid manager (using the **Load and Unload Grid** buttons (shown left)).



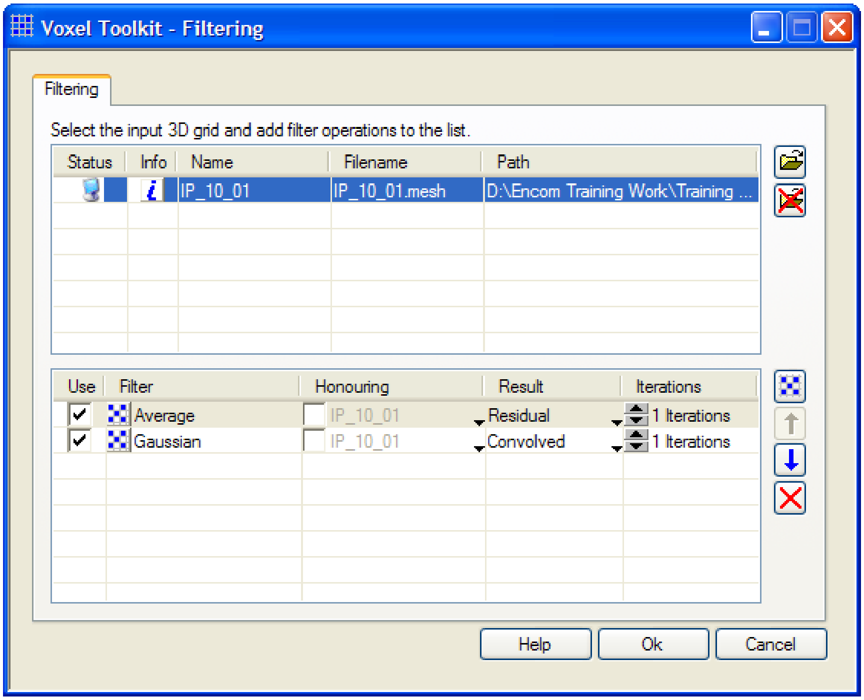
Filters can then be applied by clicking the **Add filter** button. Multiple filters can be applied to the grid in a user-specified order. The filter properties are accessible via the **Filter** button in the **Filter** column. Ensure the target grid is highlighted prior to clicking **OK.**

Basic filter properties include:

- **Data honouring -** When used, grid nodes that are 'close' to a known data point will not be modified by the filter operation. Currently a distance grid is used to measure the 'closeness' of a node to the input data. The distance grid can easily be generated using the 3D gridding tool.

- **Result -** Either the convolved or residual result can be output.

- **Iterations -** You can apply the filter one or more times.



Convolution filtering applied to a selected 3D grid

Voxel Utilities

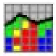
Grids>Voxel Utilities

Discover 3D provides a range of powerful Voxel Utilities, from simple voxel cell size resampling through to advanced analytics and calculations between voxels and polyhedron volumes.

This functionality can aid in determining the average voxel cell value (e.g. gold grade) within a user-modelled polyhedron volume, allowing simple resource calculations; clip a voxel to a mineralisation volume's extents or interpolated unconformity surface; or convert lithological vector volumes into a voxel model of densities.

These utilities are accessible via the **Grids>Voxel Utilities** menu option in the 3D application:

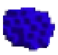
- *Assign Values to Features* – Attribute multiple feature objects with any coincident voxel cell values.



- *Clip to Surface* – Clip a voxel model with a surface, producing two output voxel models.

- *Clip to Vector Volume* – Clip a voxel model to one or more vector volumes, producing two output voxel models.

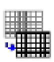
- *Convert Nulls* – This utility converts the NULL value in the input voxel model to a specified value.



- *Convert Vector to Voxel* – Convert one or more polyhedron volumes into a voxel model.

- *Create Mesh* – Create a blank voxel mesh, either from an existing voxel model or via user-specified parameters.

- *Merge* – Merge two or more voxel models into a single multi-banded voxel model.



- *Resample* – Resample and/or resize an existing voxel model into a new voxel mode

Assign Values to Features

Attribute multiple feature objects with the statistical aggregation of any coincident (for feature points) or contained (for polyhedorn volumes) voxel cells. The following statistical calculations are available for polyhedrons: Mean, Minimum, Maximum and Sum.

For example, this can be used for calculating the average gold grade of a number of wireframed supergene volumes from their contained cell values of an encompassing voxel model.

Requirements:

- A voxel model (preferably open in Discover 3D).
- A feature database open in Discover 3D containing either point or solid (polyhedron volumes) objects only.

To assign values from a voxel model to a features database:

1. Open the source voxel model and coincident feature database into Discover 3D.
2. From the **Grids>Voxel Utilities** menu, select **Assign Values to Features**.
3. Select the source voxel model in the **File Selection** dialog and press **Next**. If not already open, a voxel model can also be loaded into Discover 3D via the browse button at the top right of this dialog.
4. In the **Assign Values to Features** dialog, select the **Feature database** that values will be assigned to.

Note Feature datasets with mixed object types, or any invalid object types such as polylines, polygons and surfaces will be rejected.

5. Select the fields in the feature database to be attributed:
   - For multi-banded voxel models, select the **Voxel** field.
   - For polyhedron feature datasets, under **Assign variable**, select the desired fields to attribute with any of the Mean, Minimum, Maximum or Sum operations. If an appropriate field does not exist, use the adjacent **New Field** button to add and name new numeric fields.

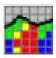
     Polyhedron volume feature datasets can be attributed from only numeric voxel models.
   - For point feature datasets, use the **Assign value** pull down to set the target field for coincident cell numeric or text attributes. If an appropriate field does not exist, use the adjacent **New Field** button to add and name a new field.

     Point feature datasets can be attributed from either numeric or lithological voxel models.
6. Select the desired **Test rule** and **Acceptance test** (if appropriate) (see *Volume Selection Method*) and press **Finish**.

When processing has finished, open the Feature Data Window, and select the target feature database selected in step 4. Note the selected fields have now been statistically attributed.

Note Attributable feature database fields (such as MaxCu, MeanCu or Lithology) can be pre-added to the database (use the **Features>Modify** tool) or added from within this utility.

Clip to Surface



Cut a voxel model with a surface, producing two output voxel models (one either side of the surface). The clipping surface can be either a triangulated feature surface (e.g. a fault plane), or a 2D gridded surface (e.g. DEM topographic grid).

This could be used to restrict an interpolated downhole-copper voxel model to the volume below an unconformity surface.

Requirements:

- A voxel model (preferably open in Discover 3D).
- A gridded surface or a feature database (preferably open in Discover 3D) containing a single surface.
- The extents of the surface (perpendicular to the intersection direction) must fully encompass the extents of the voxel model in the direction
- The surface must fully truncate the voxel model
- If using a feature database, it must contain only a single surface object. Feature databases with any other feature type, or multiple objects, will be rejected.

To clip a voxel model with a feature surface:

1. Open the voxel model in Discover 3D.
2. Open the feature database in Discover 3D.
3. From the **Grids>Voxel Utilities** menu, select **Clip to Surface**.
4. Select the voxel model in the File Selection dialog and click **Next**. If not already open, a voxel model can also be loaded into Discover 3D via the browse button at the top right of this dialog.
5. In the **Clip to Surface** dialog, select the appropriate feature database from the **Use an open feature database** option.

6. Set the **Intersection direction**. This controls the 'ray' or 'hit' test direction for each voxel cell to the feature surface, with options for testing along either the X, Y or Z axis directions.

Note The intersection/hit test is performed on the cell centres only.

This should be as close to perpendicular to the plane of the feature surface as possible. For instance, a relatively horizontal unconformity surface would be best tested against with a Z axis intersection direction, whilst a near vertical fault surface orientated north-south would require an X axis intersection direction.

7. This will generate two voxel models; the locations and names of these can be altered at the bottom of the dialog. The user can elect to prevent the generation of either output grid via the check boxes. Press **Finish** to run the clipping process.

To clip a voxel model with a gridded surface:

1. Open the voxel model in Discover 3D

2. From the **Grids>Voxel Utilities** menu, select **Clip to Surface**.

3. Select the voxel model in the **File Selection** dialog and press **Next**. If not already open, a voxel model can also be loaded into Discover 3D via the browse button at the top right of this dialog.

4. In the **Clip to Surface** dialog, select the **Use DEM surface** option. Browse for the gridded surface via the adjacent **Open** button. If using a multiple-banded grid, also select the band from the **Band** pull-down list

5. Set the **Intersection** direction. This controls the 'ray' or 'hit' test direction for each voxel cell to the feature surface, with options for testing along either the X, Y or Z axis directions.

Note The intersection/hit test is performed on the cell centres only.

This should be as close to perpendicular to the plane of the feature surface as possible. For instance, a relatively horizontal unconformity surface would be best tested against with a Z axis intersection direction, whilst a near vertical fault surface orientated north-south would require an X axis intersection direction.

6. This will generate two voxel models; the locations and names of these can be altered at the bottom of the dialog. The user can elect to prevent the generation of either output grid via the check boxes. Press **Finish** to run the clipping process.

Clip to Vector Volume

Clip a voxel model to one or more vector volumes, producing two output voxel models. This functionality supports both feature database polyhedrons as well as 3rd-party vector formats such as AutoCAD (.DXF) volumes.

This is an excellent means of visualising/restricting a voxel model to only those portions within a wireframed mineralisation volume/s, perhaps as a precursor to using the *Assign Values to Features* tool.

Requirements:

- A voxel model (preferably open in Discover 3D) and
- A feature database volume or volumes open in Discover 3D or one or more 3D volume third-party vector files (eg AutoCAD DXF)

To clip a voxel model with a vector volume:

1. Open the source voxel model.
2. If using a feature volume, open the coincident feature database in Discover 3D.
3. From the **Grids>Voxel Utilities** menu, select **Clip to Vector**.
4. Select the source voxel model in the **File Selection** dialog and press **Next**. If not already open, a voxel model can also be loaded into Discover 3D via the browse button at the top right of this dialog.
5. In the **Clip to Vector** dialog, choose one of:
   - **Use a feature dataset** and then select the feature database containing the polyhedron volume/s from the pull-down list.
   - **Use a 3D vector file** and then use the adjacent Browse button to select one or more vector files.

Note Only polyhedron volumes (i.e. 'solids') will be processed; any other object types will be ignored in the input feature dataset or vector files will be ignored.

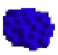
6. Select the appropriate **Test rule** and **Acceptance test** (if appropriate); see *Volume Selection Method* for more information.

7. This process will generate two output voxel models, one with valid cells within the clipping volumes, and the second with valid cells external to the clipping volumes. Change the voxel names and output location if desired, and press **Finish**.

Convert Nulls

This utility converts the NULL value in the input voxel model to a specified value (e.g. a different null value). Existing NULL values will be converted.

Convert Vector to Voxel



Convert one or more polyhedron volumes into a voxel model. This utility supports volumes in either a feature database or multiple input vector files (e.g. DXF and TKM). Output voxel values can be sourced from a feature database field, or set via a single global value.

This allows the conversion of a polyhedron volumes into voxel regions of constant values, perhaps as a precursor for more advanced voxel processing/calculations such as using the *Voxel Calculator*. For instance, convert a series of modelled lithogical wireframe volumes into a voxel model of appropriate specific gravities, allowing further processing against a gold grade voxel to calculate basic resource estimates.

The **Convert Vector to Voxel** tool also supports *Encom ModelVision TKM models*.

Requirements:

- A feature database with one or more polyhedron volumes open in Discover 3D, or
- One or more third-party 3D vector files (eg AutoCAD DXF) containing volumes to be selected by the user.

To convert vector volumes to voxels:

1. If using a feature volume, open the feature database into Discover 3D

2. From the **Grids>Voxel Utilities** menu, select **Convert Vector to Voxel**.

3. In the **Convert Vector to Voxel** dialog, choose one of:

   - **Use an open feature database** and then select the feature database containing the polyhedron volume/s from the pull-down list or

   - **Use vector files** and then use the adjacent Browse button to select one or more vector files.

4. Select the value/s to assign to the resulting output voxel:

   - **Individually per polyhedron volume** (feature objects only): Enable the **Select data fields** tick box, and select the numeric feature field to populate the voxel model from. Determination of overlapping volumes voxel cells can be handled via a range of Cell overlap rules (such as average, minimum, maximum, etc)

   - **Globally** (both feature databases and third-party 3D vector files): Set the **Global value** parameter at the bottom left of the dialog.

Note Only polyhedron volumes (i.e. 'solids') will be processed; any other object types will be ignored in the input feature dataset or vector files will be ignored.

---

5. Select the appropriate **Test rule** and **Acceptance test** (if appropriate); see *Volume Selection Method* for more information.

6. Press **Next**.

7. In the following dialog, select one of:

   - **Create a new grid with the XYZ extents of the input polyhedron volumes** (recommended)

   - **Copy the extents of an existing grid**

8. Press **Finish**.

Encom ModelVision TKM models

Selecting files of this type will ignore any global value assigned, and instead use the TKM files internal attributes (if they exist), in the following order, generating a multi-banded voxel model:

- Susceptibility

- Density
- Remanence - Q Ratio
- Remanence - Magnetization
- Remanence - Azimuth
- Remanence - Dip

Create Mesh

Create a blank voxel mesh, either from an existing voxel model or via user-specified parameters.

Merge

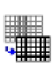
Merge two or more voxel models into a single multi-banded voxel model. All input bands are preserved and all cell boundaries are preserved without interpolation.

The output merged grid is a multi-banded voxel, with one band per input voxel. Band visualisation can be controlled via the pull-down list in the COlour tab of the Voxel Properties dialog (see *Changing Voxel Model Display Properties*).

Requirements:

- Open the utility from the Grids>Voxel Utilities menu
- Select the voxel models to merge and press Finish.

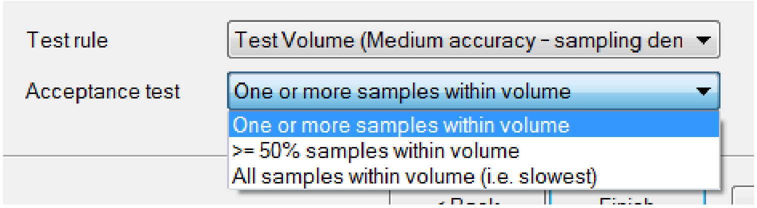
Resample



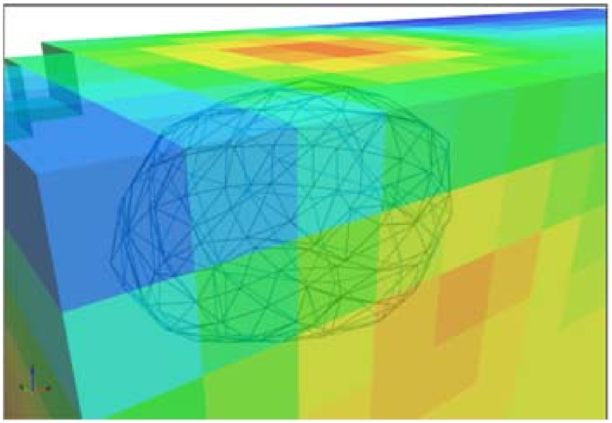
Resample and/or resize an existing voxel model into a new voxel model; a number of interpolation options are available.

Volume Selection Method

All utilities involving processing between feature objects and voxel models need to determine whether a voxel cell is within the feature volume or not. The accuracy and speed of this calculation is controlled via the *Test Rule* and related *Acceptance Test* options.



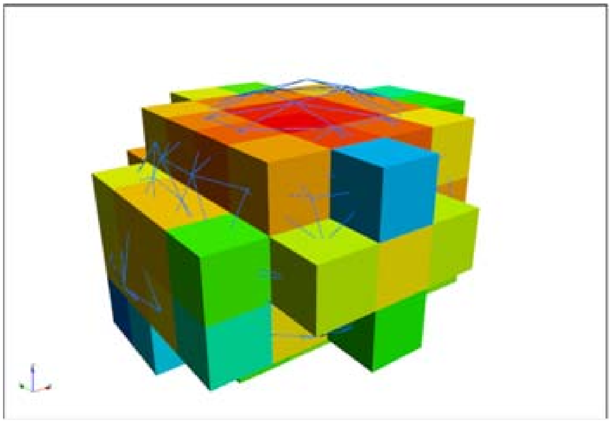
In general, the more accurate the method chosen, the more iterations/tests that need to be performed, and therefore the slower the processing time.



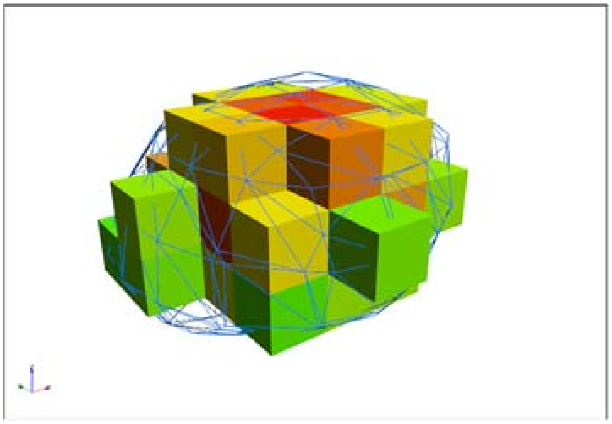
Test Rule

Four levels of density testing are available for determination of whether a voxel cell is within a polyhedron volume:

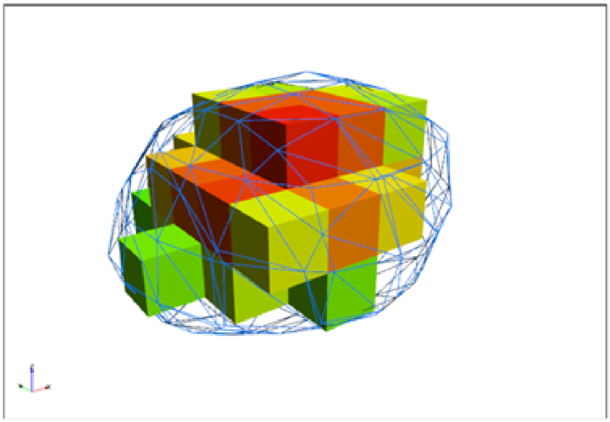
- **Test centre**: Only the voxel cell centre position is evaluated. Fastest but least accurate method.



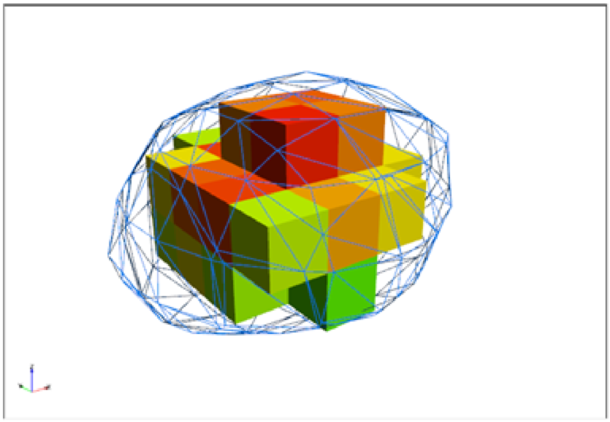
- **Test volume – low accuracy**: Each voxel cell is subdivided into 8 regions (2X x 2Y x 2Z) and the centre position of each of these is evaluated



- **Test volume – medium accuracy**: Each voxel cell is subdivided into 27 regions (3X x 3Y x 3Z) and the centre position of each of these is evaluated.



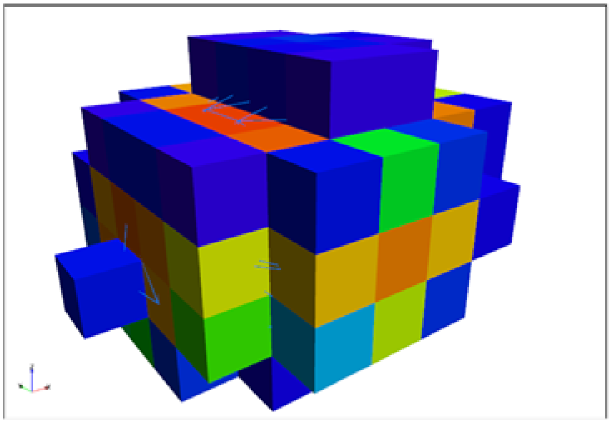
- **Test volume – High Accuracy**: Each voxel cell is subdivided into 64 regions (4X x 4Y x 4Z) and the centre position of each of these is evaluated (i.e. slowest but most accurate method)



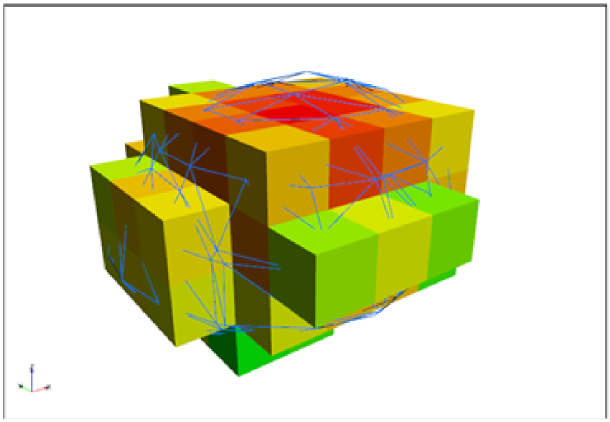
Acceptance Test

If any of the Test volume options are selected, a range of **Acceptance tests** are enabled. These allow the user to specify how many of the subdivided regions (samples) need to be within the volume for the voxel cell to be accepted:

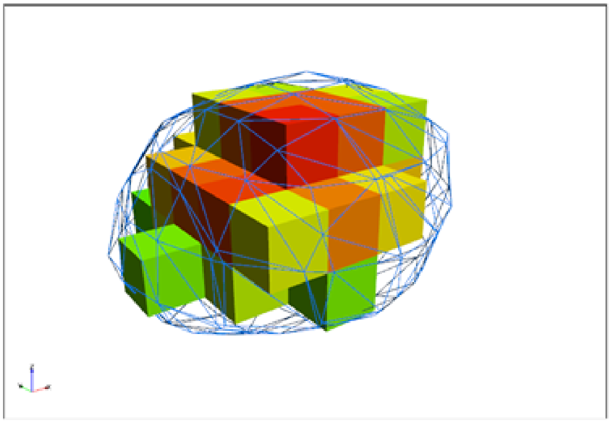
- **One or more samples within the volume**: Fastest but least accurate method.



- **>= 50% of the samples within the volume**: Medium accuracy.



- **All samples within the volume**: Slowest but most accurate method.



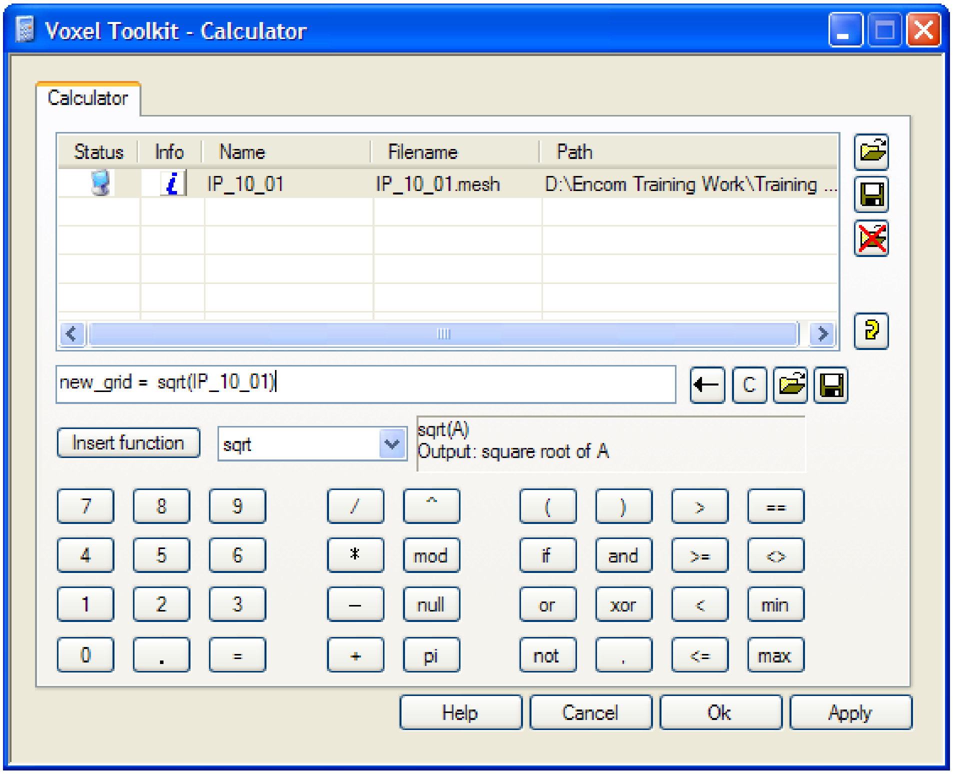
Voxel Calculator

Grids>Voxel Calculator

The **Voxel Calculator** is a powerful tool for modifying voxel models. To operate the calculator, build an expression using the following syntax:

NewName = exp<Name1>…exp<Name2>

The syntax is fully described in *Voxel Calculator Syntax*.



3D Grid Calculator dialog

The calculator uses the grid names as arguments for the expression. These grid names are shown in the grid list and are assigned a default when the grid is loaded, saved or created. To ensure that the names are easy to use and are not too verbose, you can edit the name at any time by double clicking on it and overtyping the original name. Note that this does not affect the grid file on disk and is only stored locally.

The calculator can employ a full range of mathematical, scientific and Boolean operations. Use the **Load** and **Save** buttons to the right of the expression text to retrieve and store expressions for easy reuse.

The calculator recognises NULL values in the input grids. At any cell, if any of the input values are NULL then the result of the expression will be NULL. You can test for NULL values using the Boolean operators and the ‘null’ keyword.

Click **Apply** to run the calculator on the expression. This creates an output grid that is not automatically saved to disk. You can save it manually from the calculator or when *Import and Export Voxel Models with the Grid Management Tool*.

Note When a lithology/discretized model is processed with the grid calculator, the existing discretized field is automatically converted into a numeric Band containing integer values that match the lithology classes. The lithology band in the output model will be reset to the 32-bin histogram that a model with no lithology/discretized information contains.

Click **OK** to run the calculator on the expression. This creates an output grid which you are prompted to immediately save to disk. If you choose to save the grid you are then directed to the **Display Assistant** that displays the new voxel model.

Advanced Calculator Properties

Advanced **calculator properties** are accessed via the **Properties** button on the Voxel Toolkit main menu. This allows you to control the output mesh size. There are two options:

- A super-set of all input meshes.

  This option is the default and produces an output grid that fully encompasses all the input meshes. The mesh incorporates all the mesh boundaries of the original input meshes.

- The intersection of all input meshes.

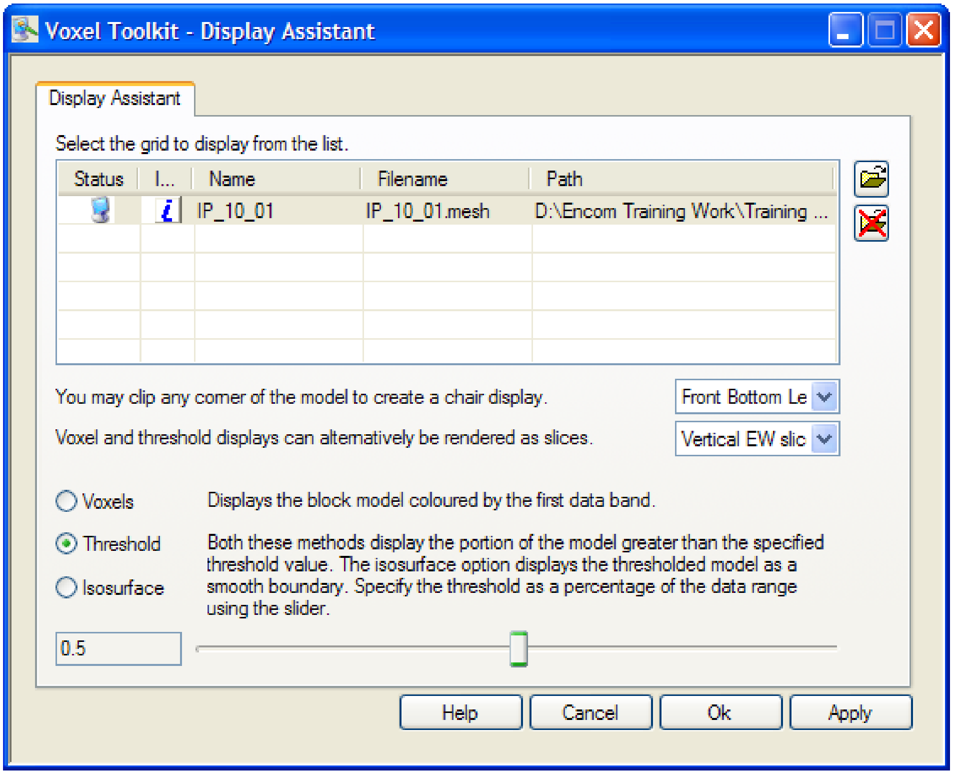
  This alternative can choose to output the area that is the common intersection of all the input meshes. In many cases this is a more logical choice since unless you implement Boolean operations to prevent it, the expression always evaluates to NULL when the target cell is outside one or more of the input meshes.

The advanced properties also allow you to control the NULL handling. By default, strict NULL handling is tuned off. This means that the outcome of simple expressions using the +, - , * and / operators will be valid if one of the operands is NULL and it will be equal to the non-NULL operand. If strict null handling is turned on, the outcome of any expression where any operand is NULL will be NULL.

Display Assistant

Grids>Display Assistant

The **Display Assistant** allows direct access to voxel model displays within Discover 3D.



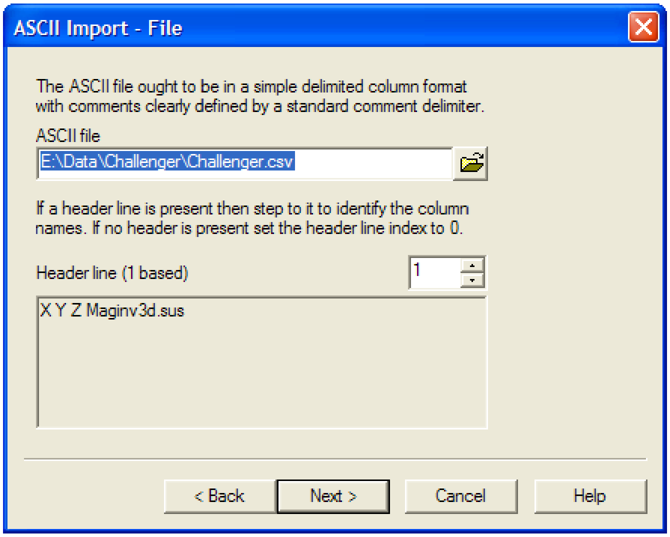
Display Assistant dialog with two voxel model files selected

The **Display Assistant** dialog allows you to do most of the basic display functions of voxels within Discover 3D. You can:

- Create a chair display by selecting the section of the model to be clipped from a drop-down menu,
- Display the voxel model with slices (not available for isosurface displays), and

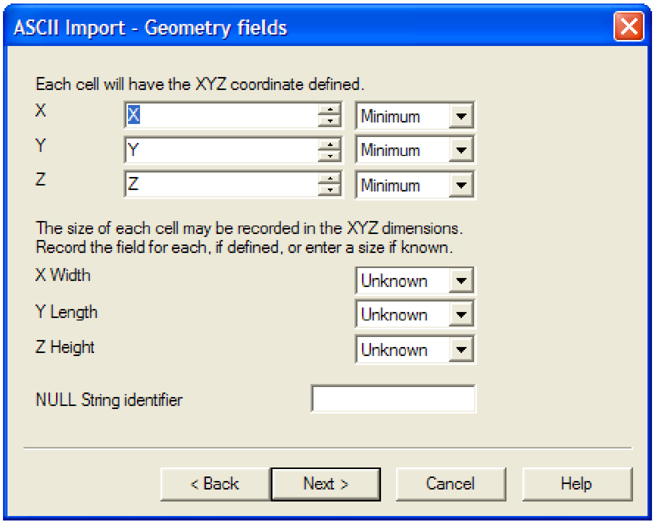
- Have the voxel model display in Voxel, Threshold or Isosurface format. The Threshold and Isosurface styles use a slider bar to control the property threshold to remove or display.

For greater and more precise control over a displayed voxel model, refer to the descriptions of the various properties in *Changing Voxel Model Display Properties*.



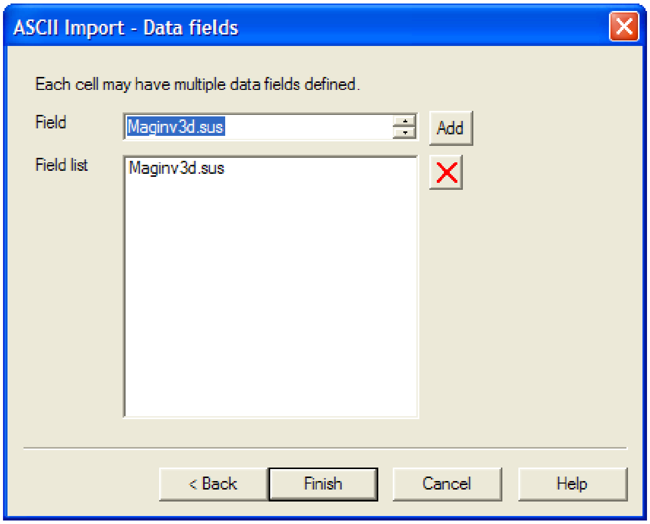
Specify the input data file

If the cell extent is specified it can also be loaded. Otherwise you can specify a constant cell size or no known cell size. In the latter case all cell boundaries are detected and the import is assumes that the cells lie between the imported boundaries.



Specify the East, North and Vertical fields plus any sizes of cells known

Finally, you can specify one or more data fields which are each saved as a band in the 3D grid. Note only the Encom 3D Grid format can support multi-banded 3D grids.

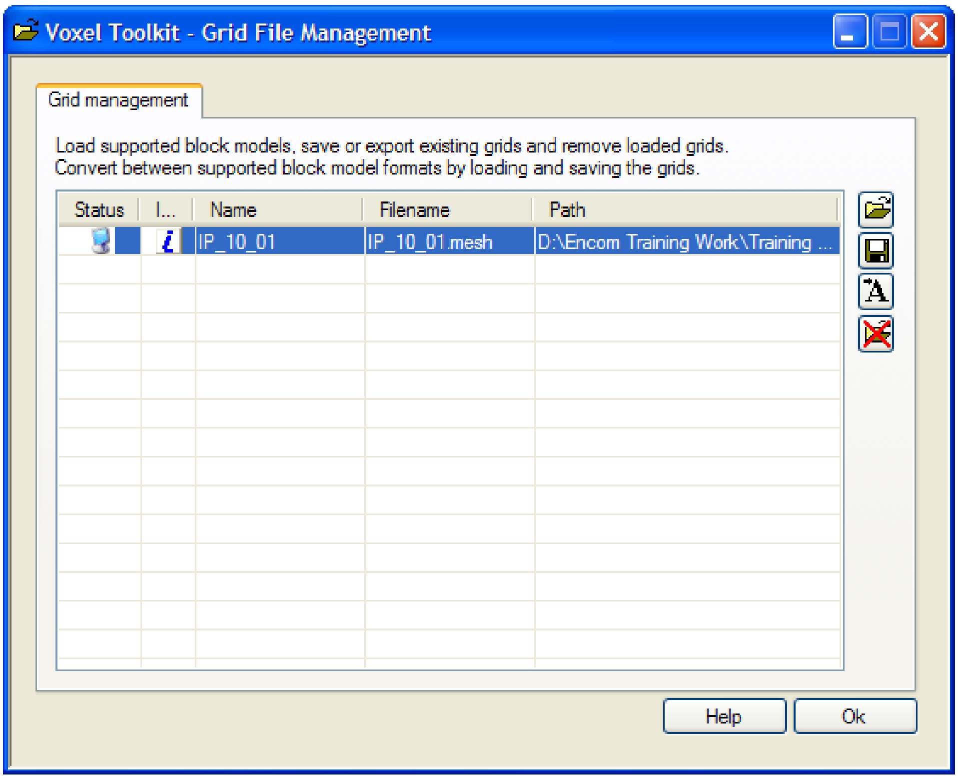


Specify the data fields to read for the gridding

Grid Management

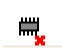
Grids>Voxel Manager

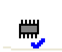
The Voxel Management tool enables you to load, import, save, export and unload voxel models.



The Grid Management tool for opening, loading and saving 3D grids

The program uses a grid management scheme that only loads grids into memory when they are required. The **Status** icon in the list indicates whether a grid is on disk, in memory (saved) or in memory (unsaved). Grids that are on disk or in memory (saved) can be safely unloaded. This does not modify the grid file on disk.

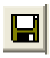

Loaded - Unsaved


Loaded - Saved


Not loaded - Saved on disk

The Voxel Manager tool can load the same grid formats as are available in the Voxel Properties dialog via the Load Model Wizard option (see *Using the Load Model Wizard*).

Note The Voxel Manager tool can import and convert any generic formatted ASCII file to a standard Encom3D grid format, which then can be loaded. See *Importing ASCII Voxel Models*.



The tool can also save or export grids. Grids can be saved in UBC and Encom 3D formats.

Grids can be exported to ASCII file in either a simple delimited XYZ Data,Data… format where the coordinate is the centre of the cell or to a XYZ NX NY NZ Data,Data… format where the coordinate is the origin of the cell and the cell dimensions are specified. You can also export to an Encom ModelVision TKM file.

The information button allows the properties of an Encom3D grid file (native voxel format) to be edited. This allows the grid's registration to be shifted, as well as a rotation applied.

Importing ASCII Voxel Models

The ASCII Import tool can import delimited ASCII block model data (e.g. comma, tab or space delimited) as a standard Encom3D voxel model. The Encom3D format supports variable cell sizes and multiple bands, non-continuous and rotated models.

ASCII file format

- The ASCII file can contain multiple header lines or have no header lines; if a header line is present it can be used to assign field names. If no header line is specified (by setting the header line to line 0) then the fields are referred to using column numbers where the first column is column 1.
- The ASCII data XYZ coordinates can be either centred or non-centred.
- Block model cell sizes can be:
    - contained in the ASCII file for any of the X, Y and Z axes
    - a user-specified constant value for each axis
    - automatically calcualted by the import tool

- Any missing cells from the ASCII file will be automatically assigned as Null values, to ensure the model is contiguous.

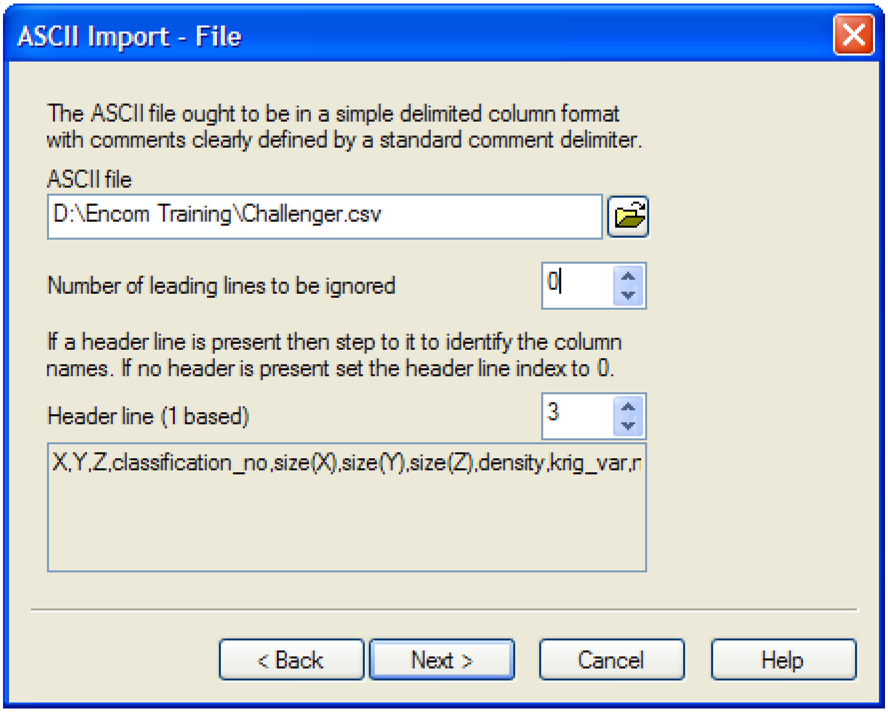
To import an ASCII Voxel Model file

1. In Discover 3D, select the **Grids>Voxel Manager** menu option.

2. Select the **Open** file button.

3. From the **Format** list, select **ASCII Import** and click **Next**.

4. Select the **Open** file button and navigate to the input ASCII file.



5. If there is a header present in the file, alter the **Header Line** number until the column titles appear in the preview area at the bottom of the dialog. If there is no header line, set this to 0. Click **Next**.

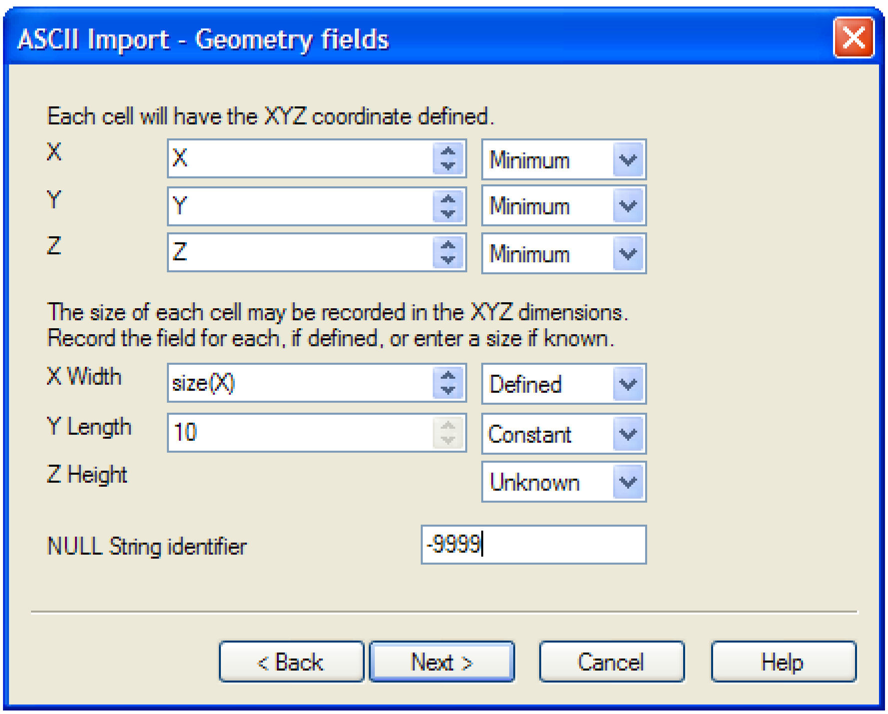
6. Check the X, Y & Z column assignments, and adjust if necessary using the Up/Down controls. Set the position type of the XYZ coordinates as appropriate. Minimum and Maximum refer to the respective XYZ corner coordinates in each cell, while centre refers to the cell centre.

Note A different definition can be set for each or the X, Y, Z axis.

7. Cell sizes for each of the X, Y & Z dimensions can be set independently as:

   - **Defined**: if the ASCII file contains X, Y or Z cell size fields for each record, set these fields as appropriate and select the Defined option
   - **Constant**: if a X, Y or Z cell sizes are a knwon constant vlaue, enster the value/s as appropriate and set to Constant
   - **Unknown**: If no cell size information is known, set the dimension to Unknown

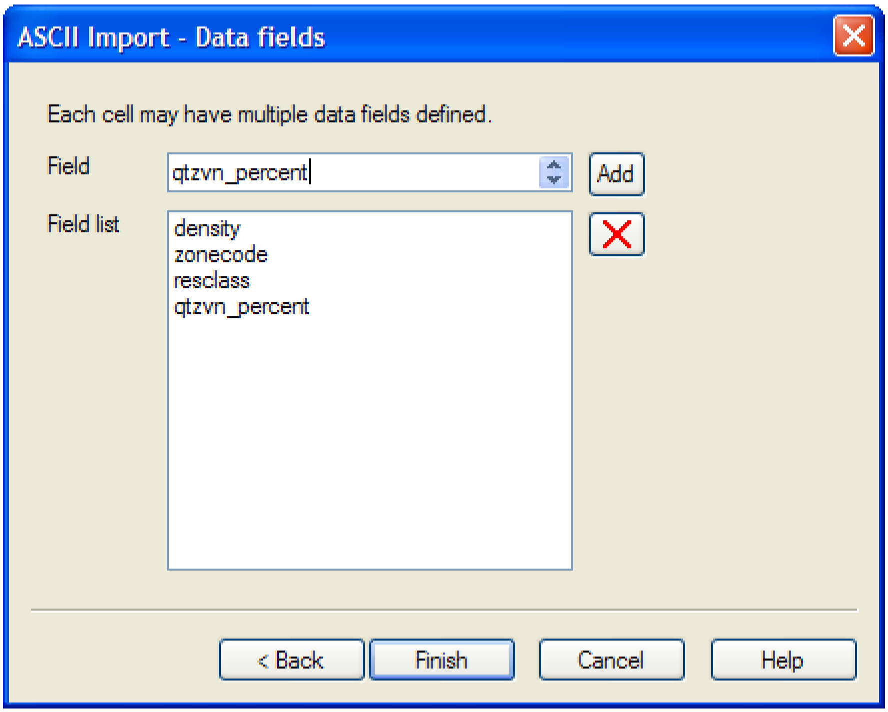
Note A different definition can be used for each axis.



8. If there is a known null data value in the ASCII data, enter this value into the **NULLString identifier** section at the bottom of the dialog. Click Next.

Note This NULL value is used in each selected data band for the model (excluding the XYZ fields). If there is a null value in one column which may conflict with the data in another data column, leave this blank or import each data band one at a time into separate voxel models.

9. Select each data column used by navigating to the field and clicking **Add**. To remove any data columns form the list, select the field and click the Delete button. It is recommended to import only one data field per model, rather than multiple fields, to reduce the output block model size and optimise 3D rendering/handling. Click Finish once the data field/s have been selected.



10. You will be prompted to save the imported grid.

11. The grid has now been saved to disk. Close the voxel tools.

12. This voxel model can now be opened into Discover 3D by drag and dropping the .E3D file, or using the **Display>Voxel Model** menu option

19 Modelling Triangulated Surfaces and Solids

Discover 3D provides the geoscientist with a powerful suite of modelling capabilities. These allow you to convert your 2D and 3D interpretations into fully three-dimensional models, such as fault planes, unconformity surfaces, water table interfaces, mineralization volumes, alteration haloes, stratigraphic volumes, etc. Additionally, these models can be extensively manipulated, refined and experimented with, in order to test concepts such as possible mineralisation extensions, examine spatial relationships between models, update models with new drilling information, etc.

Advanced tools allow analysis between voxel and vector volumes, such as assigning the average grade to a wireframed volume from the contained voxel cells values, or creating voxel models from solids. Combining these capabilities with dynamic geometric statistics for feature objects allows basic resource calculations to be performed.

- *Modelling Methods and Tools*
- *Extruding Models from Points, Lines and Polygons*
- *Wireframing Models from Polylines, Polygons and Surfaces*
- *Selecting Points Within a 3D Wireframe Model*

Modelling Methods and Tools

Extruding Models from Points, Lines and Polygons

Discover 3D's Extrusion Wizard can extrude lines, surfaces and volumes from simple objects such as :

- MapInfo vector objects (points, polylines and polygons).
- Feature objects digitized directly in 3D, such as points, polylines and polygons snapped to drillhole intervals.

For more information, see *Extruding Models from Points, Lines and Polygons*.

Wireframing Models from Polylines, Polygons and Surfaces

The Solid Generator can wireframe between feature polylines, polygons or surfaces, to create open triangulated surfaces or closed polyhedron volumes. The source feature objects can be created from:

- Importing 2D drillhole section layer (see *Viewing Section Layers*).
- Digitizing directly in 3D (in particular by snapping to drillhole intercepcts (see *Creating and Editing Features*).
- An extensive range of vector formats can be imported (see *Importing Data into a Feature Database*) as feature objects.
- Gridded surfaces can be imported (see *Importing Data into a Feature Database*) as feature objects.

For more information, see *Wireframing Models from Polylines, Polygons and Surfaces*,

Smoothly Interpolating Models as Gridded Surfaces

Point and polyline feature objects and 3D Point datasets can be smoothly interpolated to create realistic surfaces using algorithms such as triangulation, inverse distance weighting, etc. The source data may be:

- Digitizing feature objects directly in 3D (in particular by snapping to drillhole intercepts (see *Creating and Editing Features*)
- A vector format imported as feature objects (see *Importing Data into a Feature Database*)
- A MapInfo Professional 2D point dataset visualised in 3D as a 3D Point dataset (see *Create 3D Points Tool* and *View Intervals as 3D Points*), such as a mappable subset of downhole drillhole data (e.g. unconformity intercepts or water table depths).

For more information, see *Creating Gridded Surfaces*.

Create a Triangulated Mesh Surface (TIN)

All types of vector data imported or digitized in a Feature Database can be used to create a triangulated surface (TIN). This is useful for example, point clouds or counters, to perform a quick triangulation where interpolation is undesirable (see *Creating and Editing Features*).

Block or Voxel Modelling

Drillhole and feature point and polyline data can be interpolated mathematically to build block or voxel models. Components of these can then be exported for use with other models, such as isosurface DXFs.

For more information, see *Creating and Manipulating Voxel Models*)

Model Editing and Modification

Once models have been created as feature objects, they can be extensively modified and/or multi-object operations performed with Discover 3D's *Advanced Editing Functions*. Capabilities include cutting one feature surface or solid with another (e.g. truncating an orebody with a fault plane), and elastically reshaping regions of a feature surface.

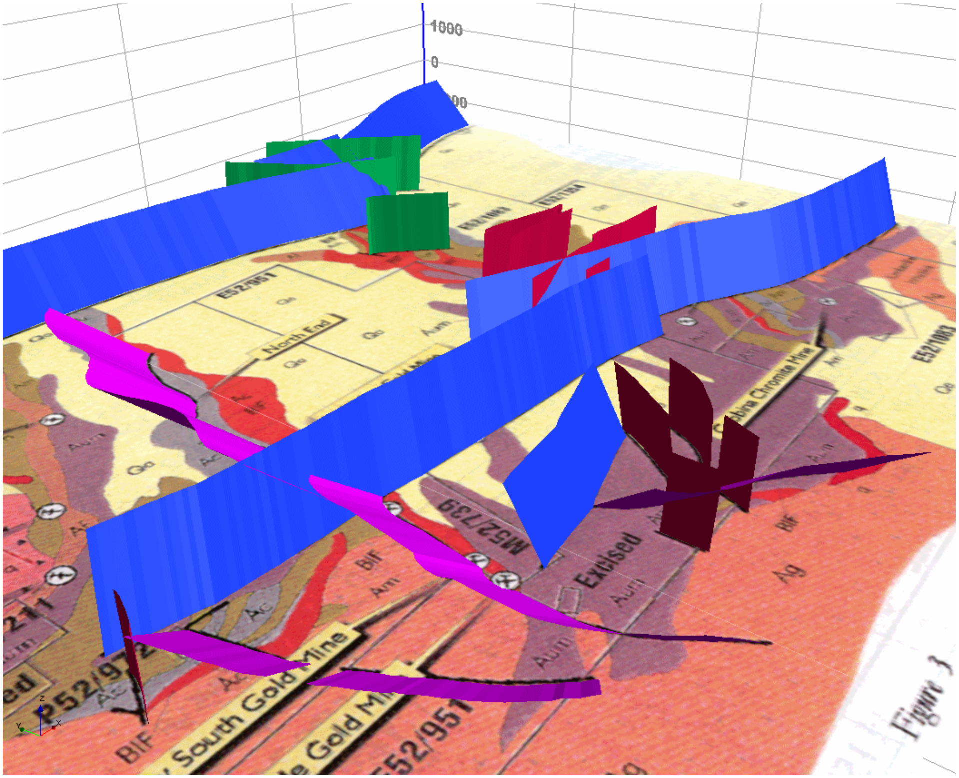
For example, to convert a 2D drillhole section layer polygon into an accurate 3D surface:

1. Import the section layer polygon *Viewing Section Layers*.
2. **Triangulate** the polygon to convert it into a feature surface (so its nodes can be moved outside the plane of the polygon.
3. Visualise the drillhole project in 3D, enabling colour or thickness modulation for the attribute related to the section boundary (eg Cu assays or lithology). Make the 3D drillholes **Selectable** in the Workspace Tree.
4. Turn on **Snapping**.
5. Make the boundary surface **Editable**.
6. Enable **Elasticity**, and change the elasticty type to **Sine**, with an appropriate radius of effect.
7. Select a vertice on the surface, and drag it to snap to an appropriate drillhole intercept. Depending on the Elasticity radius, the surrounding nodes will also be displaced to conform with the snapped nodes new location.

Extruding Models from Points, Lines and Polygons

The **Extrusion Wizard** allows you to extend (extrude) the shape of a 2D or 3D object (e.g. a polygon or polyline) from a base surface to a second or upper/lower surface. This allows meaningful and useful visualisation of various bodies such as fault surfaces, mine shafts and workings, vein systems and buildings in three dimensions.

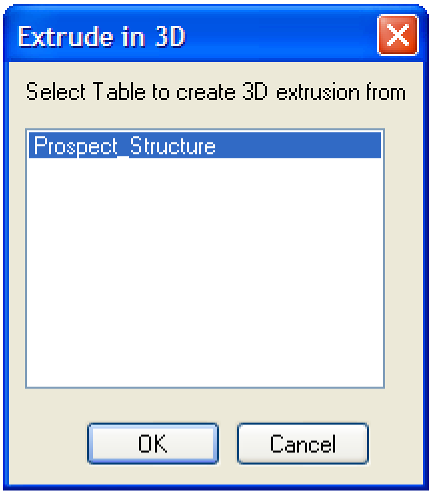
The **Extrusion Wizard** can be accessed from both the 2D and 3D windows. It can operate on objects either within a mapped MapInfo Professional table (or a map window selection) or a Feature database (see *Digitizing and Managing 3D Features*). Height information can be specified from fields within the MapInfo Professional table/Feature dataset or manually set by the user. The sides of the extruded shape can be created as polygonal walls that give the impression of an enclosed volume within the extruded outline of the source object(s).



Fault visualisation from polylines in a MapInfo table using the 3D Extrusion Wizard

To activate the Extrusion Wizard in MapInfo/Discover (2D):

- Select the **Discover 3D>3D Extrusion Wizard** in the MapInfo Professional interface. In the following dialog, select the table from which to create a 3D extrusion. This table should have suitable map object outlines such as polygons, closed polylines (eg. contours) or regular objects such as rectangles, squares or circles. If the table or selection contains point objects the 3D Extrusion Wizard can create three dimensional objects such as curtains or vertical bars using point data. Click **OK**. If not already open, 3D window will open.



Extrude in 3D dialog

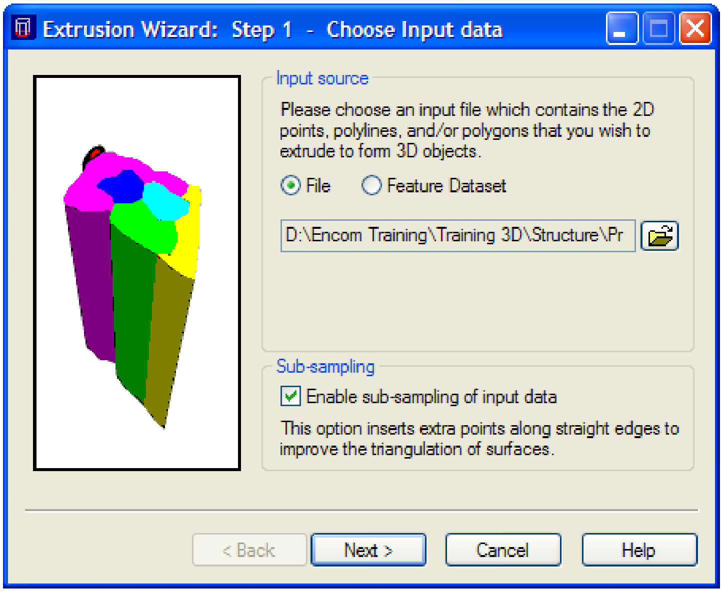
To activate the Extrusion Wizard in Discover 3D Window:

- Select the **Extrusion Wizard** option from the *3D Utilities* menu

Using the Extrusion Wizard

Step 1 – Choose Input Data

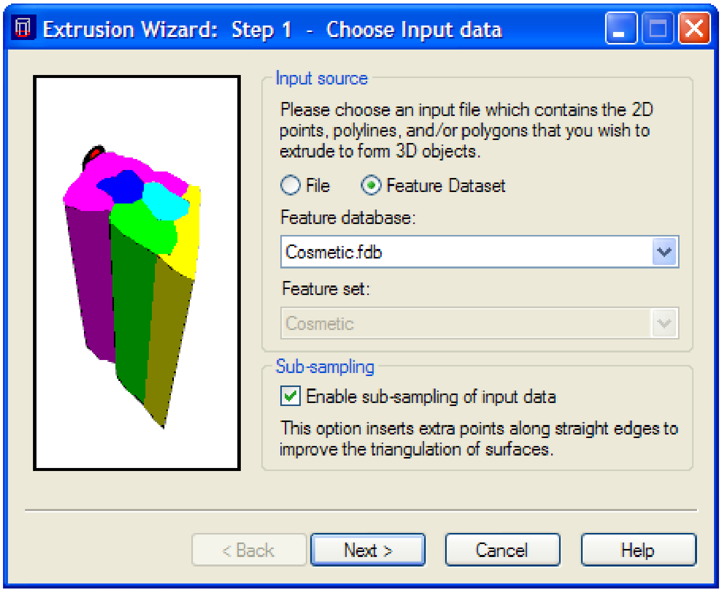
The first dialog of the **Extrusion Wizard** allows data source selection. If running the wizard from the MapInfo Professional 2D interface, this will be auto-populated by the preceding selection dialog (the **File** option is enabled by default, allowing the selection of MapInfo Professional TAB or MIF or ESRI shape files).



Step 1 of the Extrusion Wizard configured for a MapInfo Professional TAB source file

If running the wizard from the 3D interface, enable the **Feature Dataset** option to select a data source of this type. Pull-down lists will be displayed allowing the selection of the correct database and feature set.

Note The **Extrusion Wizard** can only utilise a single base RL/Z value per object (unless these values are sourced from a grid surface). Therefore, it can only correctly extrude Feature databases that are planar and in the XY plane (i.e. the horizontal plane).



Step 1 of the Extrusion Wizard configured for a 3D Feature Database source file

An **Enable sub-sampling of input data** option is available; this can improve the quality of the output extrusion by inserting extra points along any straight edges of the source object.

Step 2 - Set Primary Surface Properties

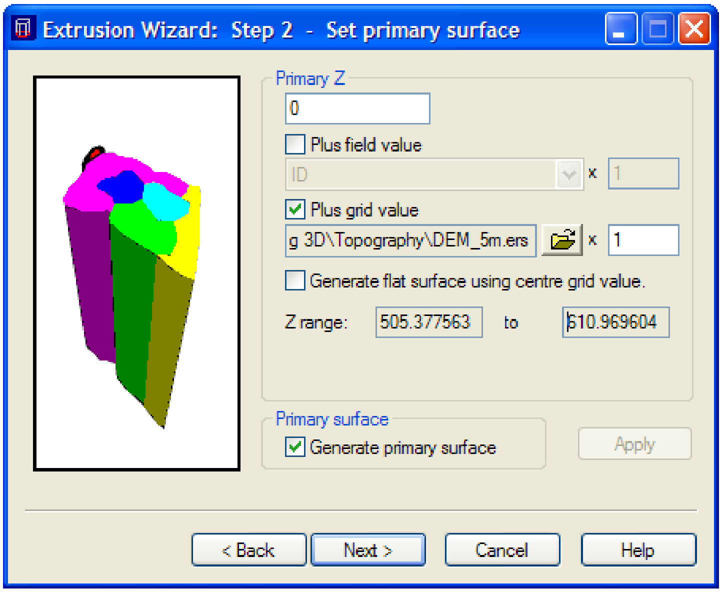
The Primary Z (or base) value of the objects to be extruded can be specified in three ways:

- **Primary Z -** enter a fixed primary Z value to be applied to the objects to be extruded. Note that the elevation uses a positive up sense of height.
- **Plus Field Value -** use a primary Z value from a field contained within the MapInfo Professional table for each map object to be extruded. For example, if building base levels were to be used, these could be stored in the selected table with a data column called bldg_base or base_RL. The values are in metres.
- **Plus Grid Value -** use a primary Z value calculated from a surface grid for each map object to be extruded. For items that you wish to lie on top of a topographic surface, use this surface as the base height. The values are in metres. Check the **Generate flat surface using centre grid line** to use a constant grid value as the primary Z value.

Note The Z value units are in the same units as the coordiante system XY units. Typically this is metres.

You can apply a scaling or add field or grid values to the data values of the base. This is useful when attempting to match the elevations of other objects you may wish to import into your three dimensional views. The **Scale** factor is multiplicative such that a value of 2 doubles the offset height of the base of the object. To use field or grid Base Z values only leave primary Z as 0.

The **Generate Primary Surface** option will create a base for the extruded object. If this option is left unchecked the extruded object will be open at the base. Click **Next** to continue.



Step 2 of the Extrusion Wizard

Step 3 - Set Extruded Surface Properties

The Extruded Z value (or upper surface value) of the objects to be extruded can be specified in one of three ways:

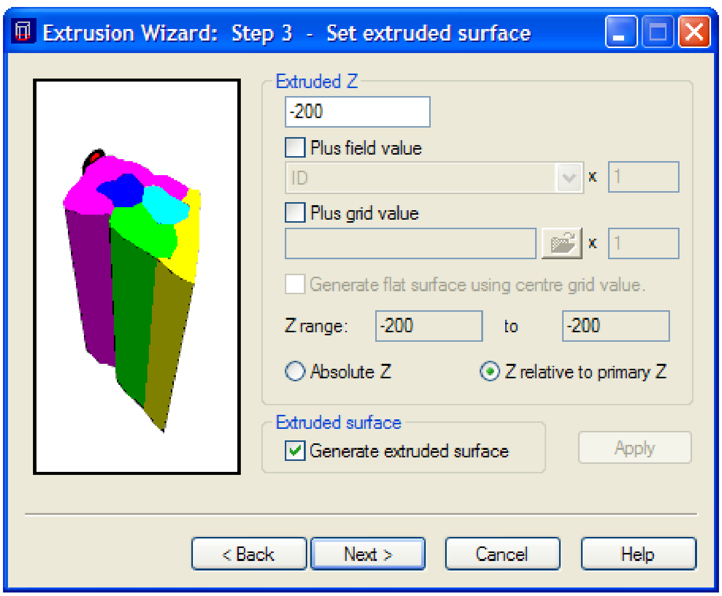
- **Extruded Z** - enter a fixed value of height to be applied to the objects to be extruded. Note that the elevation uses a positive up sense of height.
- **Plus Field Value** - use an extruded Z value from a field contained within the MapInfo Professional table for each map object to be extruded. For example, if building height levels were to be used, these could be stored in the selected table with a data column called bldg_height or height.

- **Plus Grid Value -** use an Extruded Z value calculated from a surface grid for each map object to be extruded. The values are in metres. Check the Generate flat surface using centre grid line to use a constant grid value as the Extruded Z value.

You can apply a scaling or add field or grid values to the data values of the Extruded surface. This is useful when attempting to match the elevations of other objects you may wish to import into your three dimensional views. The **Scale** factor is multiplicative such that a value of 2 doubles the offset height of the base of the object. To use field or grid Extruded Z values only leave Extruded Z as 0.

The height of the upper surface can also be chosen **Relative to primary Z** (i.e. an object height from bottom to top) or in **Absolute** terms such that the actual elevation of the top surface is defined.

The **Generate Extruded Surface** option is turned off by default. This option will create a top for the extruded object. If this option is left unchecked the extruded object will be open at the top. Click **Next** to continue.



Step 3 - Define the upper surface of the extruded objects

Step 4 – Choose Inclination

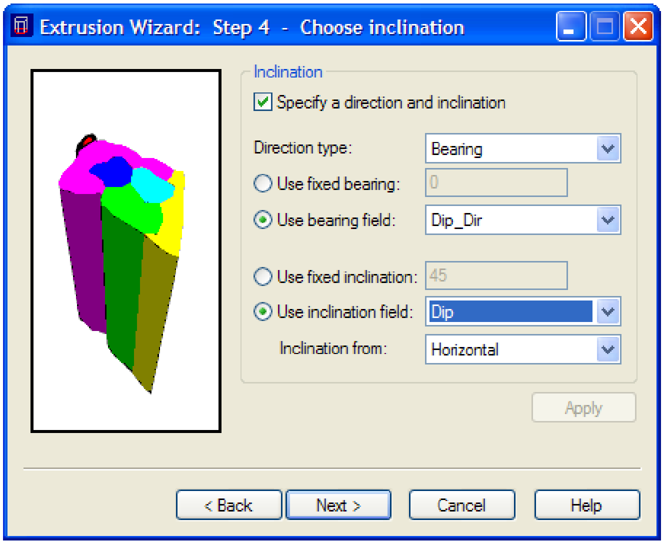
Ticking the **Specify a direction and inclination** option allows these parameters to be applied to the extruded objects. This allows faults and veins to be displayed in their correct structural orientations. Likewise buildings, shafts and pipelines can be visualised with a specified tilt angle.

Bearing and **Inclination** values can be specified either as fixed/constant values, or from appropriate fields in the object table:

1. Specify whether your direction/azimuth measurement is the dip direction, or a strike reading (i.e. using the right-hand rule: 90 degrees anti-clockwise from the dip-direction).

2. Set the direction type (as set in step 1) to be either a fixed value (manually enter the value) or a selected numeric field in the table.

3. Specify the inclination as either a constant (user-input) value, or a selected numeric field in the table (e.g. dip, plunge, etc). The inclination can be set either as measured from the horizontal (0 degrees for a horizontal or flat inclination, negative values dip downwards) or from the vertical (dips of 90 or -90 will therefore be horizontal).



Step 4 – Specify bearing and inclination parameters

Step 5 – Choose grid compression

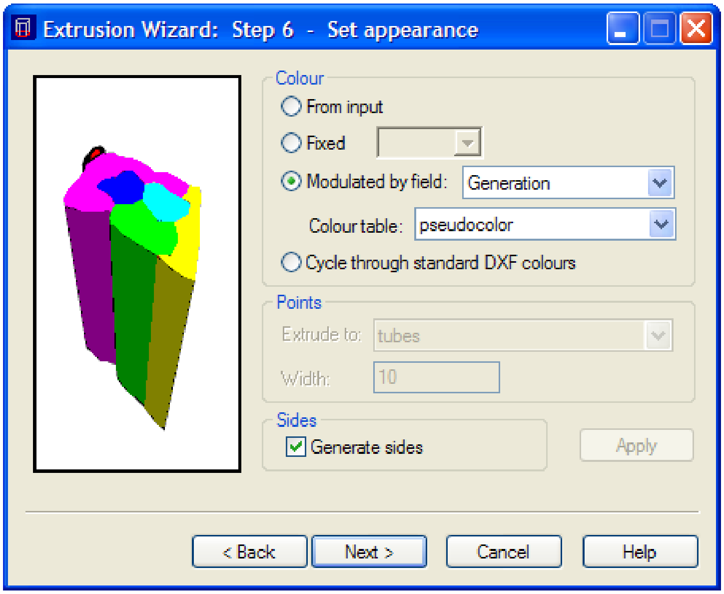
This dialog allow the Primary and Extruded grids (if specified in steps 2 and 3) to be compressed. It is generally recommended to leave the compression factors at their default levels, unless dealing with very large grids (e.g. a very large and detailed DEM grid).

Applying compression uses the Tomek algorithm, which variably adjusts to the data; this attempts to retain the boundary of the grid, including internal holes. This will decrease the processing time required to create the output extruded vector file, but may lead to discrepancies between the vector file and the original grid.

After setting a compression level using the slider bar, press the **Compute %** button to return the actual percentage compression factor to be applied.

Step 6 - Set Appearance

The colour and appearance of individual extruded objects can be controlled by the following wizard screen.



Step 6 - Specify the surface properties of the extruded objects

The extruded objects are drawn as 3D DXF objects in the 3D display. Four options are provided to allow the colour of the objects to be controlled:

1. **From input** - the individual objects obtain their colour from the default list of standard colours

2. **Fixed** - all the created objects are of the same colour. Select the required colour from the pull-down palette.

3. **Cycle through standard DXF colours** - apply colours from the 16 standard colours provided for DXF support

4. **Modulated by Field** - individual objects are coloured using the data value in the specified field and a selected look-up Colour table. A wide range of look-up tables are provided. The field selected must be a numeric field.

If the original data specified for extrusion is points rather than polygons, the point data can be extruded as **Lines**, **Tubes**, **and Square** or **Triangular prisms**.

By default the **Generate Sides** option is ticked to create an enclosed object.

Step 7 - Choose Output File

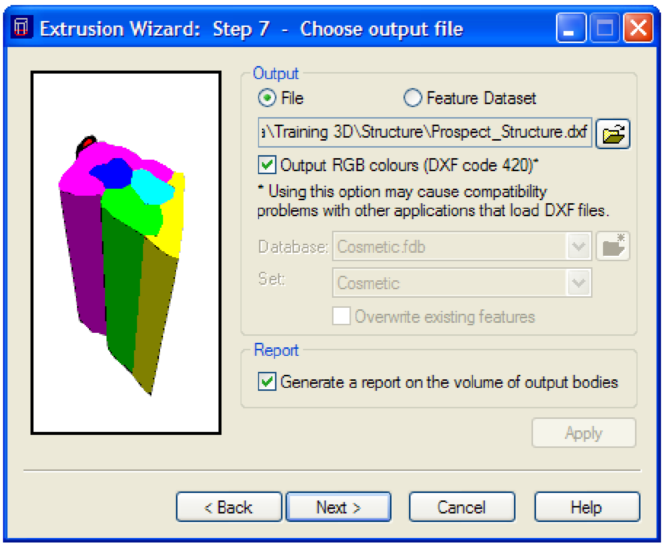
The Output File screen allows the extruded shapes to be saved as:

- *DXF vector file output*, or
- *Feature dataset output*

DXF vector file output

Enabling the File option will create an output DXF vector file utilising the same name and directory (default) as the input file. These can be altered using the Browse button.

The output file can include **Output RGB colours**. However, not all DXF-compatible programs can handle RGB colours within a DXF file.



The Output File screen configured to generate a DXF Vector file.

An option is also provided to generate a text report of each output 3D solid's extents, surface area and volume (this only applies to extruded polygons). Selecting the **Report** option in this screen allows the creation of this report as a .CSV file in the output file directory.

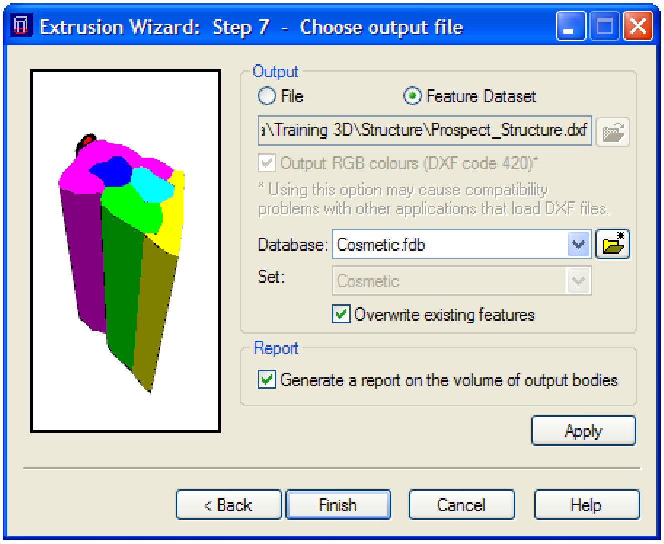
Press **Next** to progress to Step 8 (only applicable for DXF output).

Feature dataset output

Selecting the Feature Dataset option allows the extruded objects to be added to either a new or existing *Digitizing and Managing 3D Features*. These can then be edited in 3D using the feature editing tools (see *Editing Features*).

Either select a feature database already open in 3D from the pull down list, or create a new feature database using the new file button to the right.

By default, the extruded objects will **Overwrite existing objects** in an existing feature database.



The Output File screen configured to generate the extruded objects in a feature database.

If the **Finish** button is clicked, the extruded objects are created in the defined feature database and added to the Discover 3D map window. The **Extrusion Wizard** is then exited.

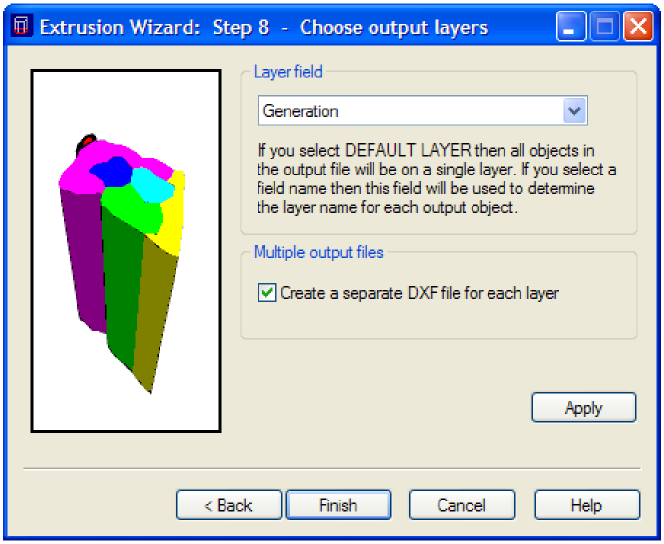
If the **Apply** button is clicked, the wizard creates the feature database and automatically displays it in the Discover 3D map window but leaves the **Extrusion Wizard** operating. It also activates additional **Apply** buttons on each of the previous wizard screens. This allows the outputted extrusion to be modified by using the **Back** and **Next** buttons to make changes. To view changes click the **Apply** buttons. The objects are then updated in the Discover 3D map window.

Step 8 - Choose Output Layers

If DXF output was selected in step 7, the extruded 3D DXF objects can be saved either:

- To single layer within the output DXF file by selecting the **Default Layer** option or

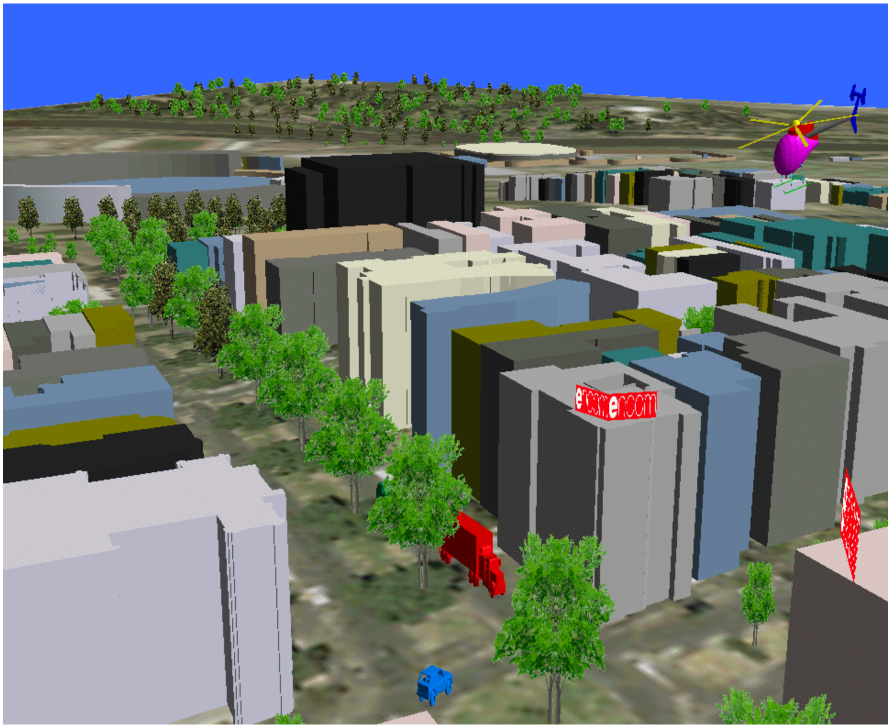
- If field is selected from the pull-down list, a separate layer will be created in the output DXF for each unique attribute in this field. It is also possible to **Create a separate DXF file for each of these layers**, resulting in an output DXF file for each unique attribute in the specified field.



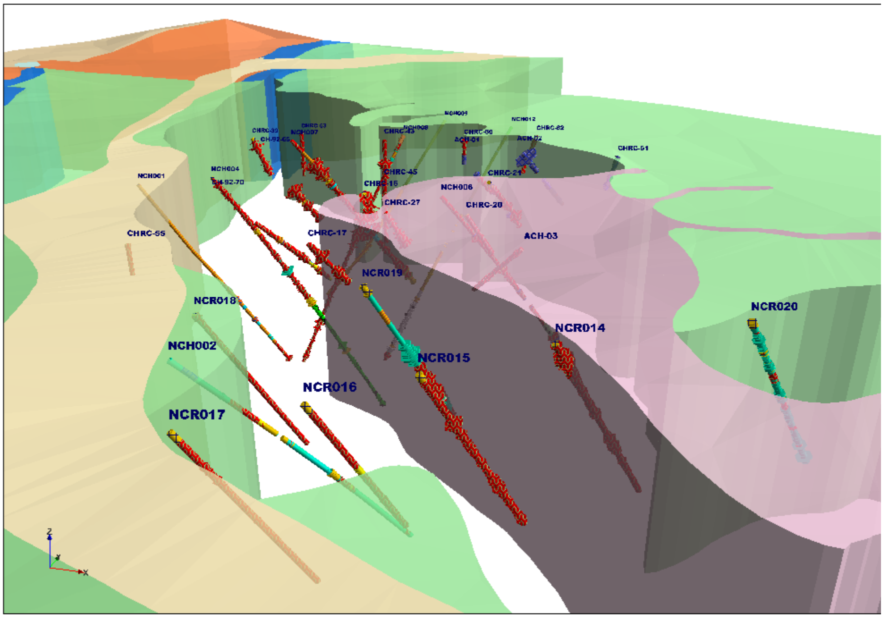
Step 8 of the Extrusion Wizard , configured to create a separate DXF file for each group of objects sharing a unique attribute in the Generation field

- If the **Finish** button is clicked, the extruded objects are created in the defined DXF file(s) and added to the Discover 3D map window. The **Extrusion Wizard** is then exited.

- If the **Apply** button is clicked, the wizard creates the DXF file and automatically displays it in the Discover 3D map window but leaves the **Extrusion Wizard** operating. It also activates additional **Apply** buttons on each of the previous wizard screens. This allows the outputted extrusion to be modified by using the **Back** and **Next** buttons to make changes. To view changes click the **Apply** buttons. The objects are then updated in the Discover 3D map window.



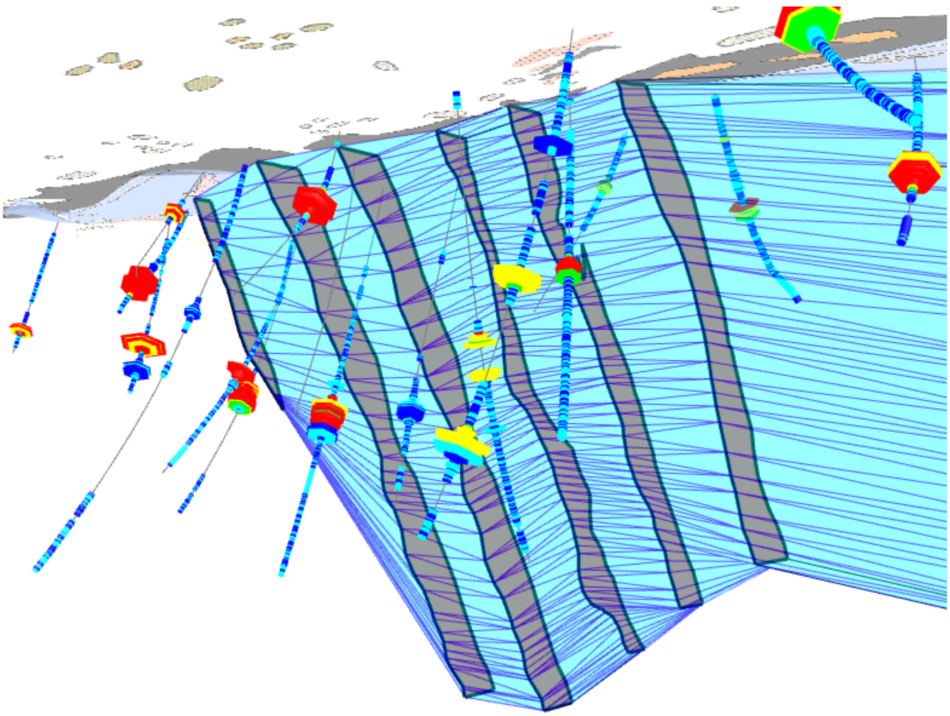
Example of a cityscape created by extruding polygons outlining buildings



Example of stratigraphic modelling using the Extrusion Wizard

Wireframing Models from Polylines, Polygons and Surfaces

The 3D Solid Generator is useful for creating triangulated 3D objects, such as an ore body, fault or other geological solids (e.g. aquifers); as well as infrastructure such as dams or open cut extents. This utility joins across areas between objects in the Feature databse (polylines, polygons or surfaces) to produce simple triangulated open 3D surfaces (e.g. faults), or closed polyhedral solids (e.g. alteration zones or ore bodies).



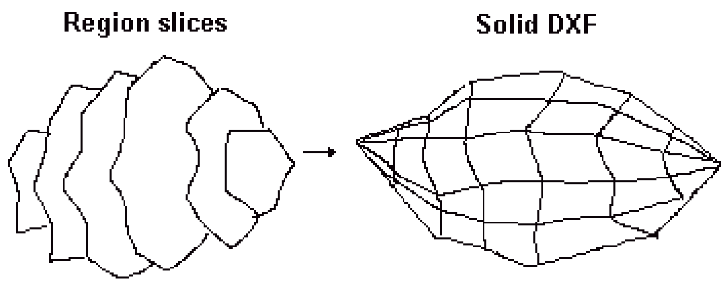
The input data for the Solid Generator consists of a variety of feature types. This commonly includes:

- **Polylines** - a series of feature polylines can be wireframed into a feature surface. For example a series of polylines snapped to drillhole fault intercepts can be used to model this fault surface in 3D.

- **Polygons** - a series of digitized feature polygons can be wireframed into a 3D solid. For example, boundaries digitized on a series of 2D drillhole cross-sections can converted into a 3D solid connecting these sections.

- **Surfaces** - a series of feature surfaces (or triangulated surfaces) can be wireframed to create a closed solid. For example, 2 or more separate grid surfaces representing the top and bottom of the water table could be imported as feature surfaces, then an enclosed volume generated from them.

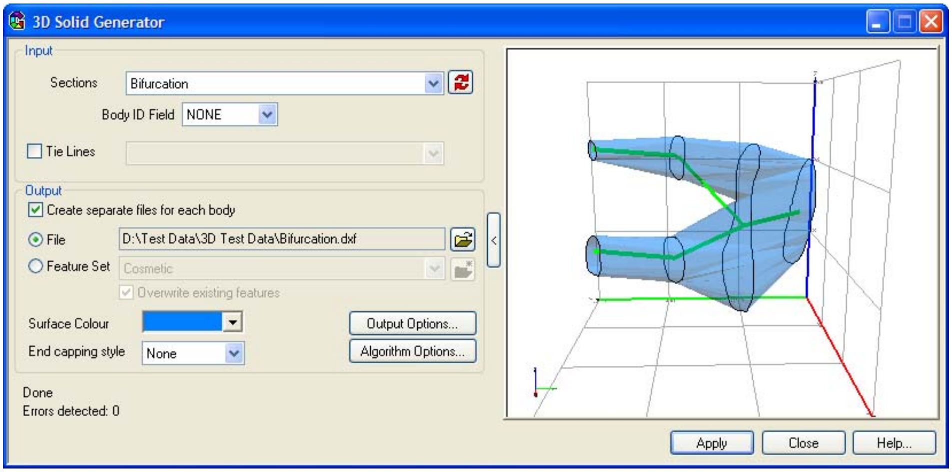
Note Polygons must be completely closed regions. Triangulated Surfaces (TINs) must be open and continuous - they cannot have internal holes or breaks.



Individual slices of interpreted regions are used to create a solid object DXF

The **3D Solid Generator** uses an improved algorithm with the following capabilities:

- Ability to handle Feature Objects digitized into a series of planes/sections with no dominant axis.
- Automatically detect body bifurcations.
- Automatically detect and build disconnected parts of the solid.
- Ability to handle multiple bodies contained in a single dataset if an attribute field is specified by the user.



The 3D Solid Generator's algorithm can process bifurcating shapes.

To create a 3D solid ore body from digitized 2D section layers:

1. Create the boundaries (see *Digitizing Section Vectors*) - this can be by digitizing in 2D and viewed in 3D, or by digitizing in 3D.

2. Use the 3D Solid Generator to create the 3D solid (see *Creating a Wireframe Model*).

3. If the output is not satisfactory, you may need to add tie lines (see *Using Tie Lines to Control Wireframing*) by digitizing in the cosmetic layer or by adjusting the wireframing parameters (see *Using Algorithm Parameters to Control Wireframing*).

To create an enclosed solid from surface grids:

1. Two surface grids may be generated from top and bottom intercept points on drillholes. These can then be digitized as points and a 2D surface grid created from these. See *Creating Gridded Surfaces* for more information.

2. Import the surface grids into a Feature database using **Features>Import** (see *Importing Data into a Feature Database*).

3. Use the 3D Solid Generator to create the 3D solid (see *Creating a Wireframe Model*).

4. If the output is not satisfactory, you may need to add tie lines (see *Using Tie Lines to Control Wireframing*) by digitizing in the cosmetic layer or by adjusting the wireframing parameters (see *Using Algorithm Parameters to Control Wireframing*).

Digitizing Section Vectors

The source Feature database containing polygons/polylines representing interpreted/inferred geological/structural/infrastructure vectors can be created in two ways:

- *Digitizing Layers in 2D* – digitizing into multiple Discover drillhole sections/ plans as **Digitized Section Layers**. These can then be exported from MapInfo/Discover as a **Feature Database** for use by the **Solid Generator**.
- *Digitizing Sections in 3D* - digitizing directly in the 3D environment into an editable **Feature Database** (either the cosmetic layer or a new database) using the tools on the *Cursor Plane*.

Digitizing Layers in 2D

For steps and information on digitizing sections in 2D and viewing in these 3D, see *Viewing Section Layers*.

Digitizing Sections in 3D

To digitize vectors directly into the 3D environment ensure that the appropriate base datasets are open in 3D (e.g. Drillholes, voxel models, etc). Ensure that the Feature database into which the boundaries are to be digitized is editable (either the cosmetic layer or a new database). Align the Cursor Plane (using the tools on the *Cursor Plane*) as required and use the Feature drawing tools to digitize the appropriate polygon and/or polyline vertices.

The *Creating and Editing Features* section of the Feature Database chapter discusses many of the tools needed for 3D digitizing. In particular, the ability to snap to drillhole intervals is a important function.

Some digitization tips which are controlled via the **Cursor Plane** tab of the Cursor Plane Properties dialog (see *Changing Cursor Plane Properties*) are:

- Use the keyboard PgUp and PgDn keys to offset the **Cursor Plane** perpendicularly from its existing position, in order to keep digitized features parallel.
- Use the **User-defined** step option to set the distance the PgUp and PgDn keys move the Cursor Plane.
- The exact location of the Cursor Plane can be set via the **Origin** (X, Y and Z), **Inclination** and **Bearing** controls.

- To limit the amount of existing 3D data visible either side of the Cursor Plane (i.e. simulate a 2D section envelope), enable either the **Envelope** or **Slice Clipping** options. These enable only a user-defined **Slice Width** of data to be viewed either side (envelope) or behind (slice) the Cursor Plane. This is an excellent way to ensure that interpreted boundaries are based only on data within, for example, 50m of the Cursor Plane.

Select the **Perpendicular** button on the Cursor Plane toolbar to orientate the view direction perpendicular to the cursor plane., thereby increasing the accuracy of your digitization.

Try enabling the Orthographic view mode (on the *Zoom Controls Toolbar*) to remove object scaling due to distance bias (as occurs in the default Perspective view). This will allow accurate object size comparison.

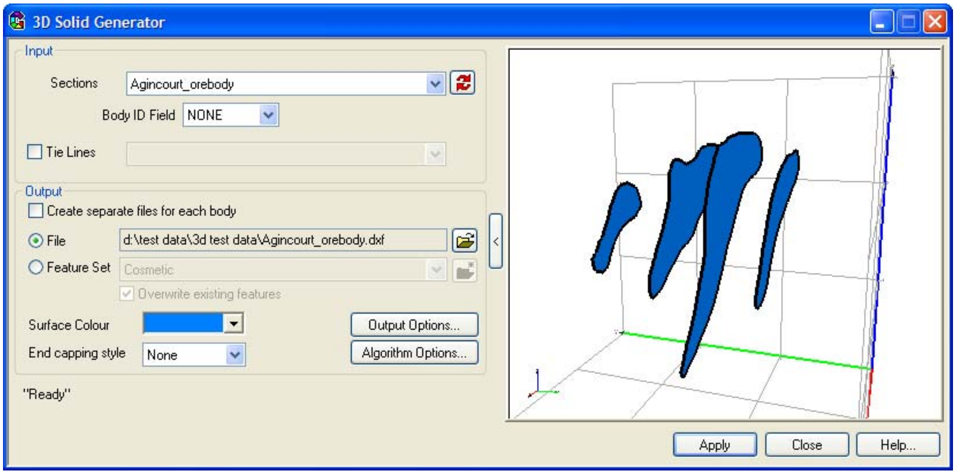
The **Ruler tool** on the *Main Toolbar* can be a useful guide to distances, bearings and dip angles between objects on the cursor plane.

- If digitizing onto georeferenced images (e.g. geophysical profiles), use the **Bond to Image** option on the *Cursor Plane*.

Note When creating very simple shapes for use by the 3D Solid Generator, it is recommended to use more than just the bare minimum on nodes/vertices to create the shape (i.e. more than say just 6 or 10 nodes to create a ‘banana’ shaped polygon). Solids created out of shapes with too few nodes generally result in some ‘ugly’ triangulations (as the 3D Solid Generator has very few nodes with which to work), particularly through bends/curves, which are very difficult to model satisfactorily without a good distributions of nodes. The **Refinement** control of the Solid Generator can help overcome this problem automatically (see the *Using Algorithm Parameters to Control Wireframing* section below)

Creating a Wireframe Model

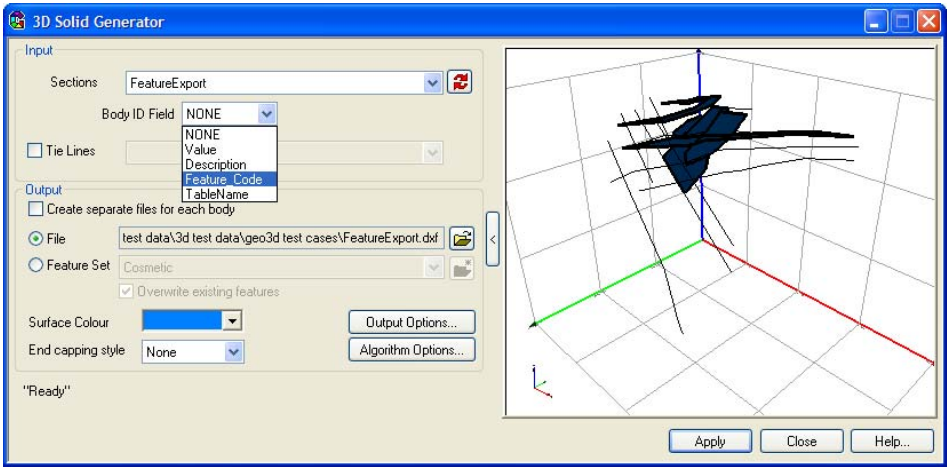
The **3D Solid Generator** is accessed via the *3D Utilities* menu within the 3D interface. It requires two or more individual Feature Objects (not in the same plane) in a single Feature Database in order to create an output model.



3D Solid Generator dialog

In the **3D Solid Generator** dialog, the top **Feature** pull-down list enables the choice of the **Feature Dataset** containing the source objects (including the cosmetic layer).

The **Body ID Field** option enables a field in the feature database to be used to separate the output into separate bodies/solids. For instance, a series of 2D drillhole project section boundaries are created and attributed using the **Feature_Code** field (see *Digitizing Layers in 2D* above) with "ore body", "supergene" and "south_fault". The boundaries are exported from 2D as a feature dataset. Within the Solid Generator utility the **Body ID Field** would be set to the **Feature_Code** field, in order to build a separate object for each of the unique attributes in this field.



The 3D Solid Generator configured to create separate output bodies (using the Body ID Field selector) for each unique attribute in the Feature_Code field of a "FeatureExport" *dataset. This dataset was created by exporting a Discover drillhole project's attributed section boundaries (Discover 3D>View Section Boundaries).*

The 3D Solid Generator supports the optional use of **Tie Lines**. These allow you to control both individual vertex-vertex joins on adjacent objects, as well as the overall join order of the source objects. To use tie lines, enable the **Tie Lines** option and specify the feature database containing the tie lines from the adjacent pulldown list. See the *Using Tie Lines to Control Wireframing* section below for further information on creating and using tie lines.

An **Output** file can be either a:

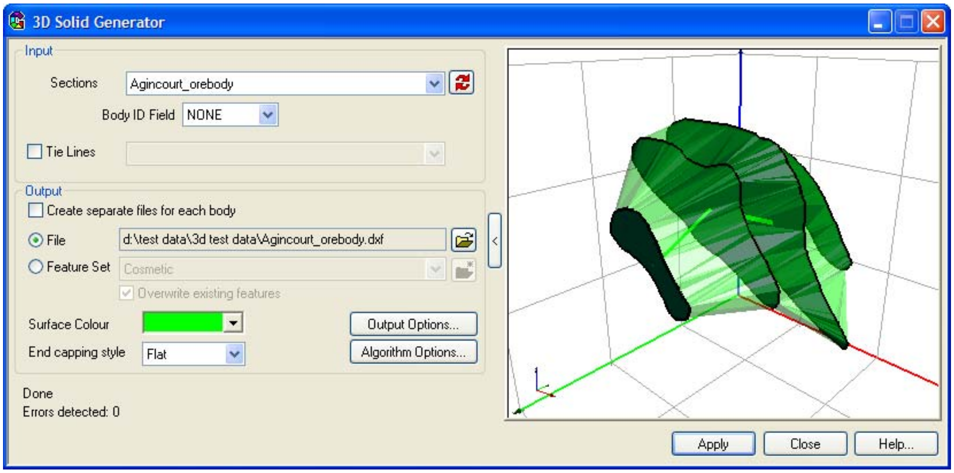
- **DXF File** by enabling the File option. By default this utilises the same name and location as the source file (the cosmetic feature layer is created in the Discover 3D temporary directory, as set via the **Options** dialog (see *Customising 2D Interface Settings*).

- **Feature Dataset**. Select the output shapes to be created in an open feature database from the pull-down list, or create a new feature database using the adjacent Browse button. If adding to an open database, the default option is to **Overwrite any existing feature** objects in the database.

If a BodyID field (e.g. Feature_Code) has been specified, the Solid Generator can create a separate output file for each unique attribute by enabling the **Create separate files for each body** option.

The **Preview** window of the dataset can be rotated by holding down the left mouse button whilst moving the mouse. Hold down the right mouse button (whilst moving the mouse) to alter the zoom level.

A **Surface Colour** can be set for output solid/s, as well as an **End capping style** (only applicable to closed polyhedral objects). The available end capping styles are None, Flat, Point or Smooth; however, selecting **None** will not produce a volume calculation in the output report generated at the bottom of the dialog. For most volume calculations a Flat end-capping style is recommended.



Completed DXF solid with a Flat end capping style previewed in the 3D Solid Generator, including surface area and volume calculations in the report screen

Once the **3D Solid Generator** dialog has been configured click the **Apply** button to generate the output solid/s. Depending on the number of Feature Objects involved, their complexity, and whether a **Body ID Field** has been specified, this may take some time. During processing, lines indicating the drawing/join order between Feature Objects are displayed and eventually the solid outlines are displayed in the preview screen.

Either a new **Feature** or **Vector** branch will also be displayed in the Workspace Tree populated with the output solid (see *Working with Vector Data in 3D* or *Changing Feature Display Properties* for controlling and altering the solid's display).

If enabled in the **Output Options**, a summary report file will also be generated in the output directory, with the suffix "_rpt.txt". This contains information about the output model extents, dimensions, area and volume is applicable. It will also indicate if any errors were encountered.

Note

A volume cannot be calculated for a solid created from a series of polylines, even if the end points of each polyline are coincidental. Volumes can only be created for solids created from a series of polygons.

The 3D Solid Generator dialog can be kept open while changes are made to a **Feature Dataset** (see *Editing Features*), allowing the shape of the input objects or tie-lines to be refined prior to reprocessing.

The **Refresh** button allows any such changes to be updated in the preview window of the dataset within the **3D Solid Generator**.

Output Options

The **Output Options** dialog controls the following components of the generated output solid and their display colour:

- **Surface** – it is recommended that this option be enabled (default) as it creates the surface planes between joined vertices,
- **Edges** – displays only the linework representing the joins between vertices,
- **Structure** – shows the connections between objects (sections) used to interpolate the solid, and
- **Diagnostics** – contains duplicates of any erroneous/self-intersecting facets.

Enable the **Create seperate file for each bod**y if a BodyID field (e.g. Feature_Code) has been specified, by default the Solid Generator can create a separate output file for each unique attribute.

An option to **use RGB colours** in an output DXF file is available. Note, however, that not all DXF-compatible programs can handle RGB colours within a DXF file.

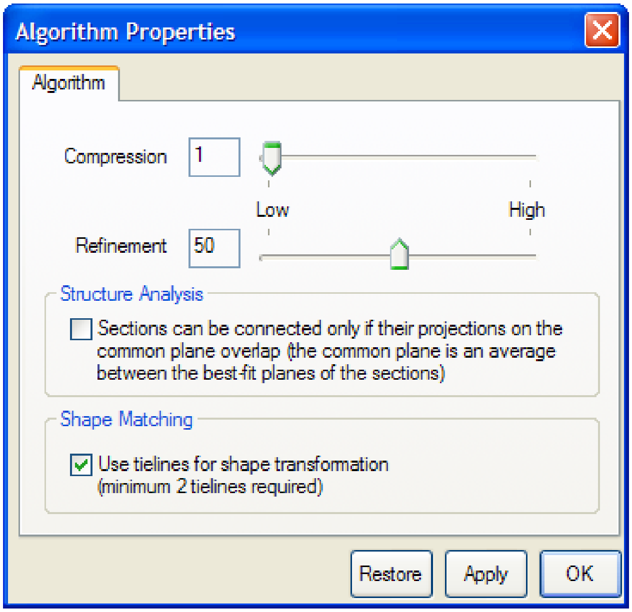
Enable the **Dynamically update view** option to update the 3D view during solid generation (once per facet).

Generate summary report will generate a report containing information about the output model extents, dimensions, area and volume is applicable. It will also indicate if any errors were encountered.

Automatically open report file will open a pop-up dialog after Apply has been pressed, showing the contents of the report.

Using Algorithm Parameters to Control Wireframing

The **Algorithm Properties** dialog contains a number of advanced functions that can help to improve the quality of the output shapes, particularly when applied in conjunction with tie-lines (see *Using Tie Lines to Control Wireframing*).



Compression

Applying compression to an object/shape database reduces the number of nodes/vertices utilised by the 3D Solid Generator. This can be useful for dealing with high density datasets (e.g. objects with 100's or 1000's of nodes/vertices) where each object is separated by relatively large distances (with respect to the distance between nodes in each object), which would result in very long and thin triangular facets being generated.

This option should be used with caution, and only applied when the dataset meets the above criteria and processing is very slow and/or generates a large number of errors. Generally using a compression value of up to 5 is sufficient: a value of 1 often solves most issues.

Note Variable compression uses the Tomek algorithm, which variably adjusts to the data. Applying compression will naturally result in not all of the input object nodes being utilised in the output triangulation, possibly creating discrepancies between the output surfaces and the input shapes. Minimise the compression factor applied to reduce this problem.

Refinement

The Refinement option allows the user to control the degree to which extra points are added to polygon edges in order to produce higher quality triangulations in connecting surfaces.

A setting of zero (0) results in no new points added to edges.

A setting of 100 results in points being added to edges to ensure that no edge is longer than 10 percent of the longest edge in the original polygon. In this case the longest edge will have approximately 10 points inserted.

The control is linear from 0 to 100 and is described in the following formula:

```
Max. Segment Length % = 100 - (Refinement * 0.9)
```

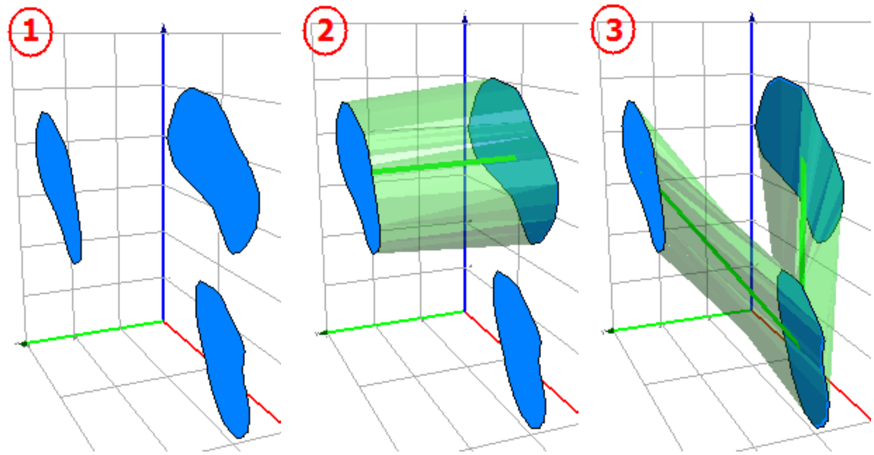
The following table shows some example settings and the resulting maximum output segment length expressed as a percentage of the maximum input segment length for that polygon:

Refinement	Maximum segment length %
0	100
20	82
40	64
60	46
80	28
100	10

Structure Analysis

The **Structure Analysis** option will only join objects when there is some degree of overlap between the objects on a common plane (default option). Where no common plane exists, an averaged plane is calculated between the various object planes involved.

Disabling this option allows the Solid Generator to search laterally (parallel to the common section plane) for objects to incorporate into the processing; this will, however, increase the processing time of the 3D Solid Generator. The images below illustrate this functionality.



The effect of the Structure Analysis option. Figure 1: three parallel polygons, with the intermediate polygon substantially offset along the common plane. Figure 2: With Structure Analysis enabled, the intermediate shape is ignored in the solid generation process, as it does not overlap with the other two polygons. Figure 3: Disabling the Structure Analysis option allows the Solid Generator to find and utilise the intermediate shape.

The lateral search radius utilised (by disabling the **Structure Analysis** option) depends on a number of factors, primarily relating to the distance between the object planes. If disabling the Structure Analysis option fails to incorporate 'outlying' objects into the final solid, you will need to create and apply tie-lines (see *Using Tie Lines to Control Wireframing*) to force the source objects to be joined in the required order.

Shape Matching

The **Shape Matching** option will only be applied (if enabled) when the 3D Solid Generator detects two or more tie-lines (see *Using Tie Lines to Control Wireframing*). It is recommended to always have this option enabled.

Shape matching is performed on 2D projections of the original objects. The algorithm constructs a warping transformation that makes the vertices of the projections corresponding to the tie lines' ends coincide. This transformation affects all vertices by moving them closer to the points of exact match and improves the edge detection procedure.

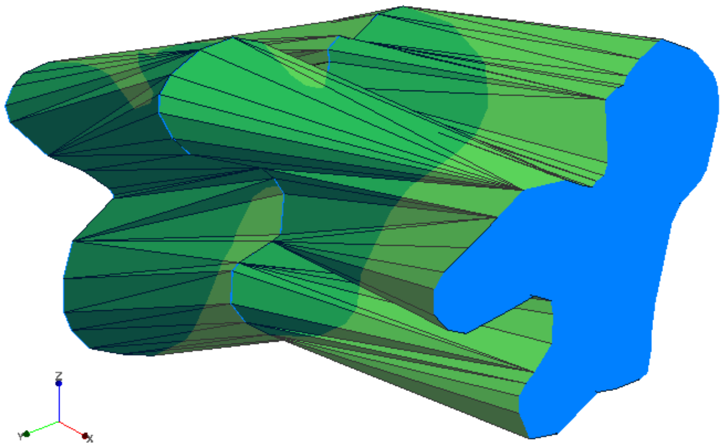


Figure 1: A wire-framed solid dxf (green) created between three polygons without using tie lines. Note the poor correlation between peaks and troughs

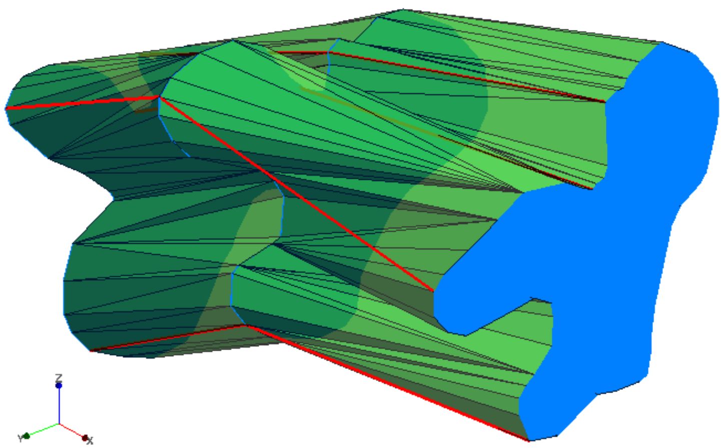


Figure 2: Tie lines (red) created to control the join of major structures

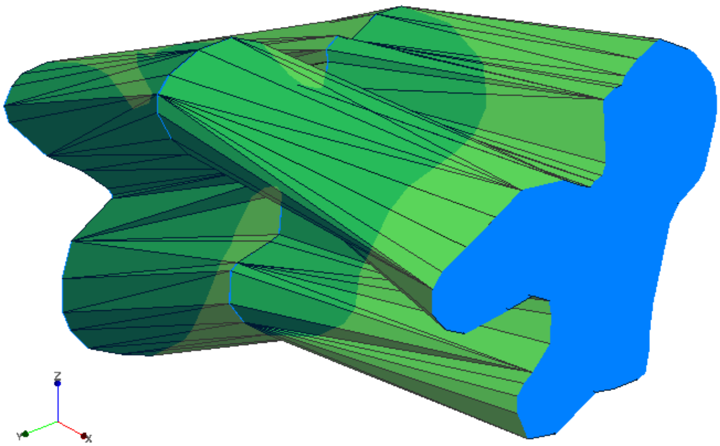


Figure 3: The output solid created using tie lines from the previous step, and Shape Matching disabled. There is a good correlation between peaks and troughs at tie line locations, but it is less robust between these locations.

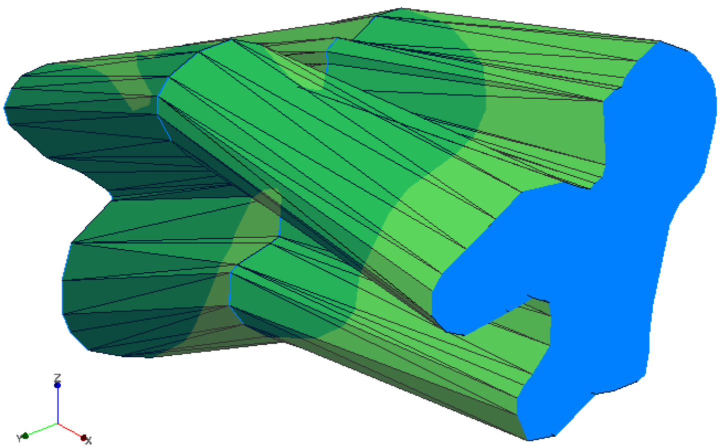


Figure 4: The output solid with Shape Matching enabled and the tie line input from the previous steps. This displays an excellent correlation between peaks and troughs, and realistic surface creation elsewhere.

Using Tie Lines to Control Wireframing

Whilst the 3D Solid Generator will generally produce satisfactory results for simpler shapes, for more complex shapes you may notice incorrect vertex joins occurring. These issues can be resolved via the use of **Tie Lines,** allowing you to:

- Manually control/specify multiple individual **vertex-vertex** joins between objects (e.g. significant inflexion points such as splay separations, related peaks and troughs, etc).

- Control the **join order of objects** in complex datasets, such as those with large distances between objects, large lateral offsets between objects, or complicated joining orders.

An example of the use of tie lines to guide solid generation between two different shapes is presented below. Each figure portrays the same dataset, viewed from different angles.

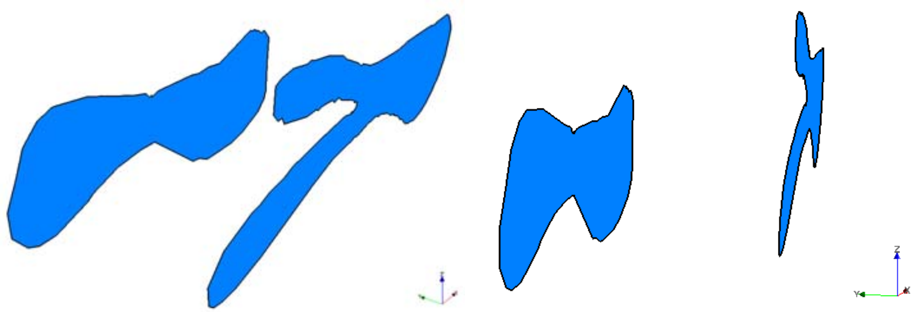


Figure 1: Two adjacent parallel polygons, the left a simple polygon, whilst the right hand shape splays into two lenses

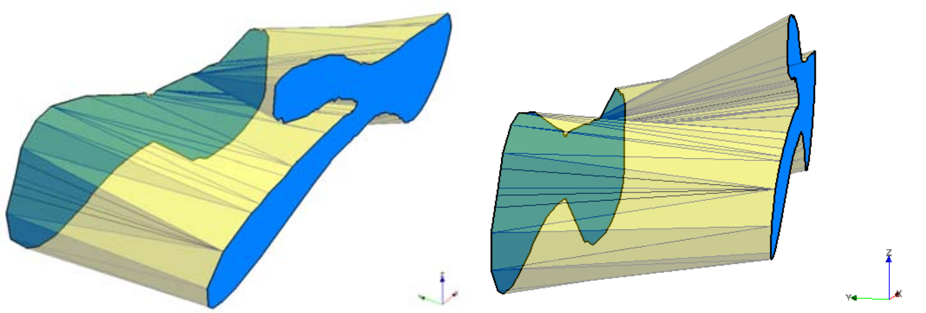
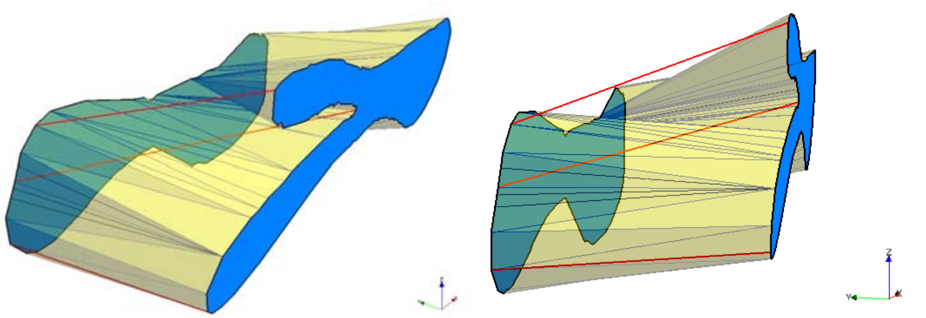


Figure 2: The resulting semi-transparent and wireframed dxf solid generated by the 3D Solid Generator without using tie lines. Whilst the bottom lens of the right-hand polygon has been joined satisfactorily to the left-hand shape, the upper lens on the right hand shape has not.



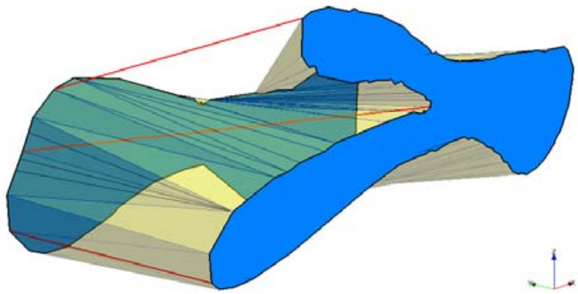


Figure 3: The preferred solid would have both these lenses joined to the near end of the left-hand shape. Red tie lines have been added indicating the user-required vertex-vertex joins between the two objects.

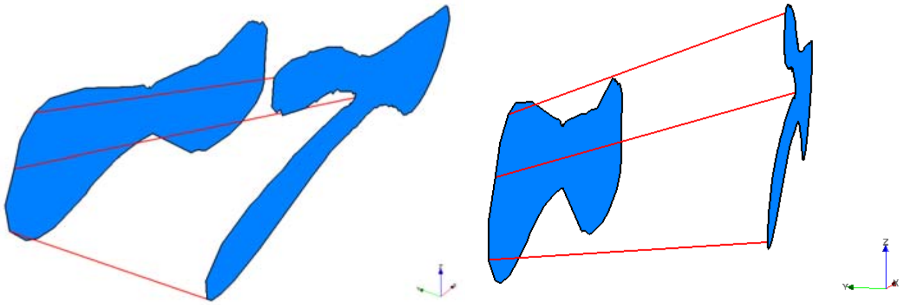
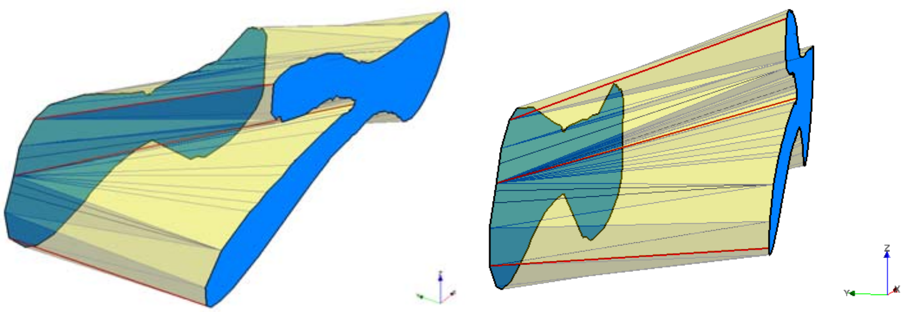


Figure 4: With the original (unsatisfactory) output solid removed, the tie line joins are clearer, linking the most prominent inflexion points (troughs and peaks) on the right-hand shape with the corresponding vertices on the left-hand polygon.



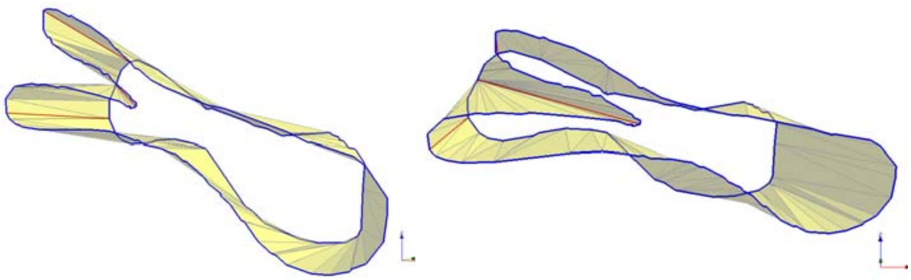


Figure 5: The resulting wireframed and semi-transparent solid incorporating the user-specified tie lines. The twin-lensed shape now clearly joins the left-hand polygon as desired. The red tie-lines are displayed for reference.

Before creating tie lines, it is recommended that the source shapes are put through the 3D Solid Generator by themselves to see what output is possible, and then utilise tie lines to refine this output. Experiment with the following options before applying tie lines:

- If the dataset contains multiple shapes relating to different attributes (e.g. different alteration haloes, fault sets, etc, ensure that an appropriate **BodyID** field is specified so that each series of shapes is handled separately
- If shapes are offset laterally, try disabling the **Structure Analysis** option under the **Algorithm Options** button.

Note Using a tie line dataset may significantly increase the processing time of the 3D Solid Generator.

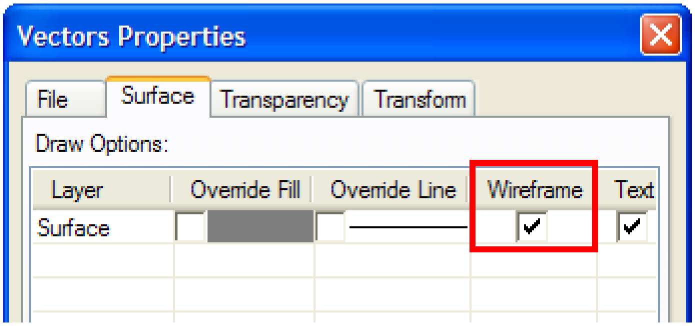
Creating Tie Lines

The following points are critical to the correct use of tie lines:

- Tie lines must be placed in a **separate feature database** to the source polygon/polyline objects being joined (commonly the **Cosmetic** layer is used for initial tie line creation, and then saved using the **Save Cosmetic Features** command).
- Each tie line **must be a line**, not a polyline (i.e. each tie line must comprise of exactly two nodes only).
- Each end of a tie line must be **snapped to a node** on a separate object.

To create tie-lines:

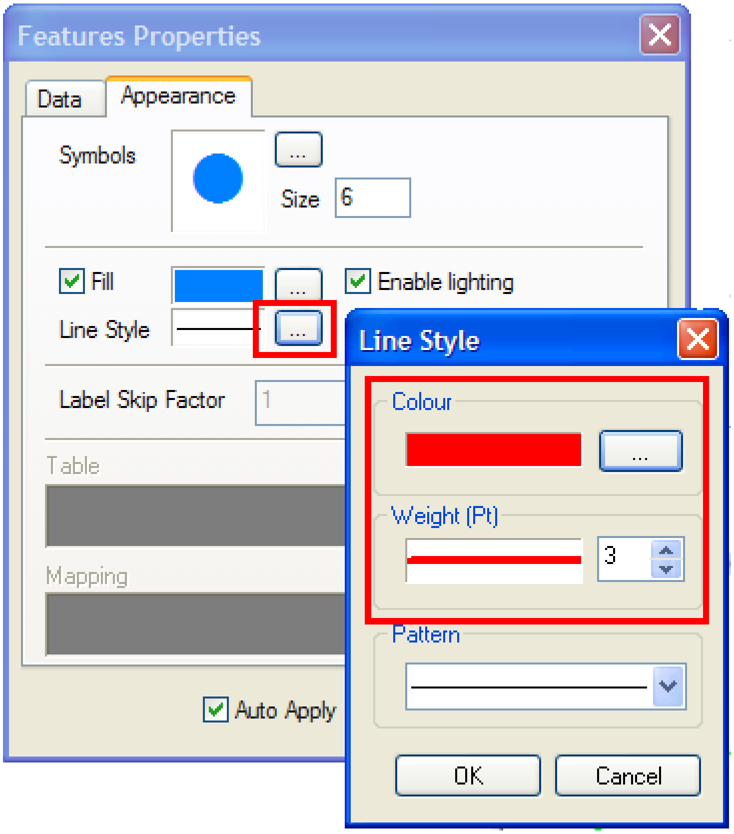
1. Run the source dataset through the **3D Solid Generator** (with and without the **Structure Analysis** option enabled) to visualise the solid output as a .DXF. In the 3D window, make this output DXF semi-transparent, and enable the wireframe view.



Enabling the wireframe view of a vector file

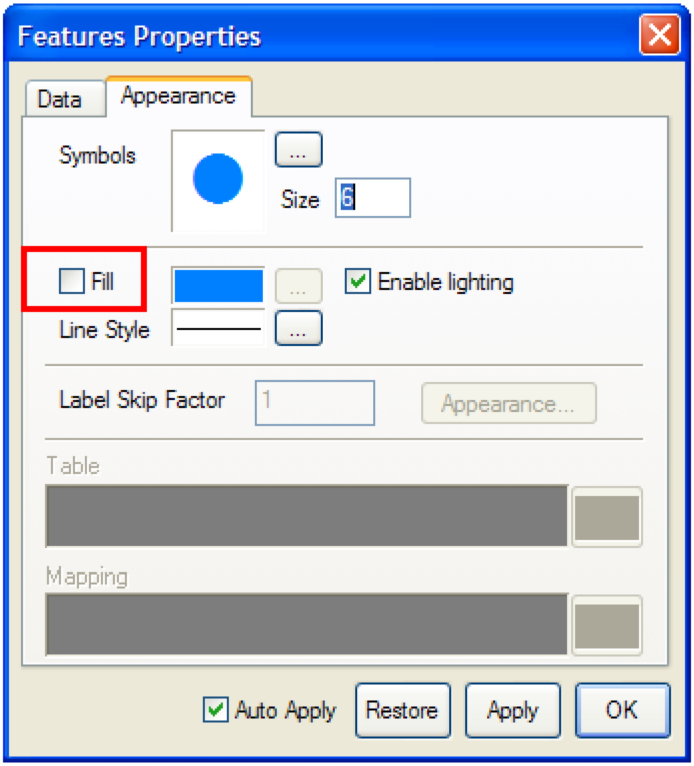
2. Examine in detail via the wireframe view the various vertex-vertex joins created by the 3D Solid Generator algorithm.

3. Make the Cosmetic Layer **editable**, and set its line colour to red, with a weight of 3 points.



Setting the line style of the tie lines in the Cosmetic Layer to stand out

4. Make the source object feature database **selectable**, and perhaps turn off the **Fill** appearance option for the source shapes to more clearly see their boundaries.



The polygon Fill tick box on the Feature Properties dialog (red circle) allows the fill of polygons to be disabled, allowing their boundaries to be more clearly visualised when a high density of polygons is involved.

5. Enable the **Snap** tool.

6. Rotate the 3D View to encompass the first two nodes (on adjacent objects) to be joined.

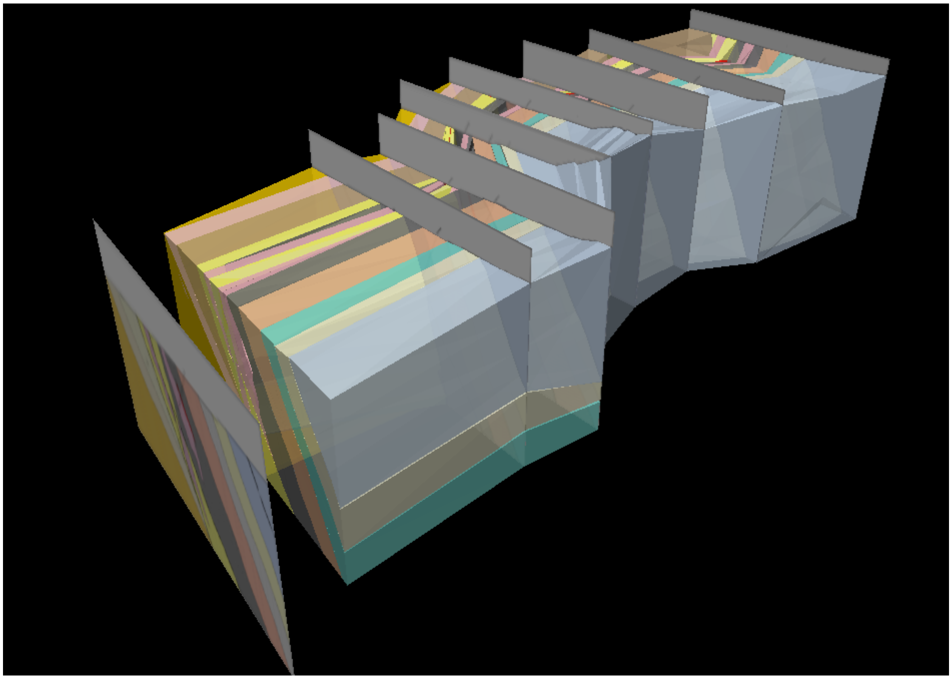
7. Enable the **Create Line** tool, and hover the cursor near the first node. When the snap beep sounds, click to snap to the point. Move the cursor to the second node, and double click when the snap beep sounds to snap to this node and finish the line.

8. Repeat steps 6 and 7 for the most prominent vertice joins required.

9. Rerun the 3D Solid Generator, enabling the **Tie Lines** option and setting the **Cosmetic Layer** as the tie line dataset.

10. Re-examine the output, noting the effect the input tie lines have had on the resulting wireframe. If necessary, repeat steps 6-9, adding further tie lines to refine the output wireframe.

Note The 3D Solid Generator can be kept open through this entire process. It will automatically refresh any changes to the updated source and tie line datasets when the **Apply** button is clicked. To refresh the preview screen in the 3D Solid Generator at any time, click the **Refresh** button.



An example of simple wireframe modelling

Selecting Points Within a 3D Wireframe Model

After you have created a 3D wireframe model that represents an enclosed solid area, you can select points from datasets that lie within the bounds of the 3D polyhedron volume.

The **Clip 3D Points to Vector Volume** tool allows you to use a Feature database or 3D Vector file to clip and select points object from a Feature Database (see *Creating and Editing Features*) or points dataset (such as one created by **Discover3D>Create 3D Points** or **Discover3D>View Intervals as 3D Points**).

To clip 3D points to vector volume:

1. Open the points feature database or points dataset.

2. Select **Tools>Clip 3D Points to Vector** volume. The **Clip 3D Points to Vector Volume** dialog is displayed.

3. Under **Points Input**, select the points feature database or points dataset.

4. Under **Vector Volume Input**, select the vector volume to clip the points to. This can be either a Feature Database or 3D Vector file.

5. Optionally, select a **Volume attribute field** from the available attributes in the selected vector volume. This will be added to the output clipped file. This is useful for when there are multiple solid objects that you are clipping to.

6. Under **Output**, select the output file containing the selected points. Choose to save as a new Feature database file or a new TAB file.

7. Click **Process** and the output will be opened in Discover3D window.

Viewing Models in 2D MapInfo/Discover

Once your 3D model is generated, imported or refined, then you able to export or import this 3D wireframe vector model, or voxel/block model, into 2D Discover sections and Maps.

This can be achieved with the following:

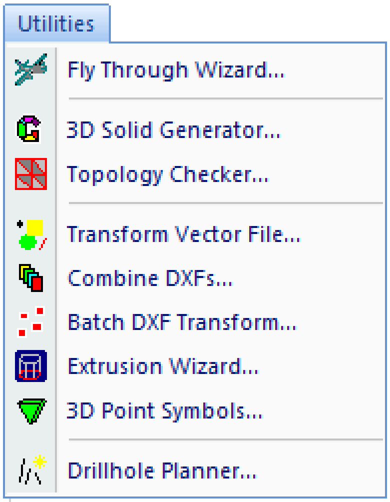
- Voxel models to 2D Discover sections. See *Using the Export Model Wizard*.

- DXF or FDB Vector models to 2D Discover sections. These can be imported by the Discover Section Manager. See the *Discover User Guide*, Chapter 18 “Working with Drillholes: Managing Sections and Plans: Display 3D Model”.

- 2D Discover section "screenshot" of 3D view. See *Creating a Drillhole Cross-section Image*.

- 2D MapInfo map "screenshot" of 3D view. See *Creating an Image Using the Cursor Plane*.

20 3D Utilities



Discover 3D Utilities menu.

The Discover 3D **Utilities** menu provides the following functionality:

- *Fly Through Wizard* – Create 3D Fly Through animations and AVI movies.
- *3D Solid Generator* – Create 3D solid objects and calculate their volume from digitized or imported feature objects (e.g. drillhole section or plan lithological/ore body boundaries).
- *Topology Checker* - Analysis tool for 3D vector models. Models with incorrect vertex orders can modified, and only layers with closed facets exported, allowing error free import into a feature database.
- *Transform Vector File* – Import and modify the origin, projection and scaling of objects in a variety of vector formats. Also allows vector file format conversion.
- *Batch DXF Transform* – Multiple external DXF files can be positioned, scaled and rotated using a control table, and combined into one output file.
- *Combine DXFs* – Combine two or more DXF files into single DXF file.
- *Extrusion Wizard* – Allows certain feature objects to be extruded vertically or using strike/dip information. An example of use is for creating 3D fault planes from surface traces.

- *3D Point Symbols* – Use the 3D Symbol Generator utility with point data to create orientated DXF symbols for display in Discover 3D.
- *Drillhole Planner* – Allows drillholes to be dynamically positioned and targeted in the 3D environment, allowing the geo-scientist to test their 3D interpretations/interpolations.

Fly Through Wizard

Discover 3D allows 3D views to be captured for replay using a Fly-Through Creation Wizard. An automatic transition between view positions allows the playback to be smooth and provide the visual appearance of a ‘movie’. The ‘flythrough’ effect can follow a predetermined track or a series of joined view points with gradual transition between each.

Refer to *Creating a Fly-Through Animation* for further details.

3D Solid Generator

The 3D Solid Generator is useful for creating interpreted 3D objects such as ore body, fault or other geological solids (e.g. aquifers), as well as infrastructure such as dams or open cut extents. This utility interpolates between objects in the Feature Database (polygons or polylines) to produce simple, open 3D surfaces (e.g. faults) or closed polyhedral objects (e.g. alteration zones).

Refer to *Modelling Triangulated Surfaces and Solids* for further details.

Topology Checker

Discover 3D can check the integrity of 3D vector models prior to import into a Feature Database. This is important if advanced feature object editing (see *Creating and Editing Features*) functionality is to be used on the model (such as the Combine, Intersect and Erase tools). These tools require a consistent method of ordering the vertices of every face and that all surfaces are closed; any gaps or incorrectly defined faces in a model will lead to errors.

The Topology Checker utility collates all faces and organises them into closed surfaces reporting any that break the rules. It can then correct errors and export the intact surfaces as a new DXF file. It has its own 3D visualiser with many features to identify and examine anything from the entire model right down to individual facets and their coordinates.

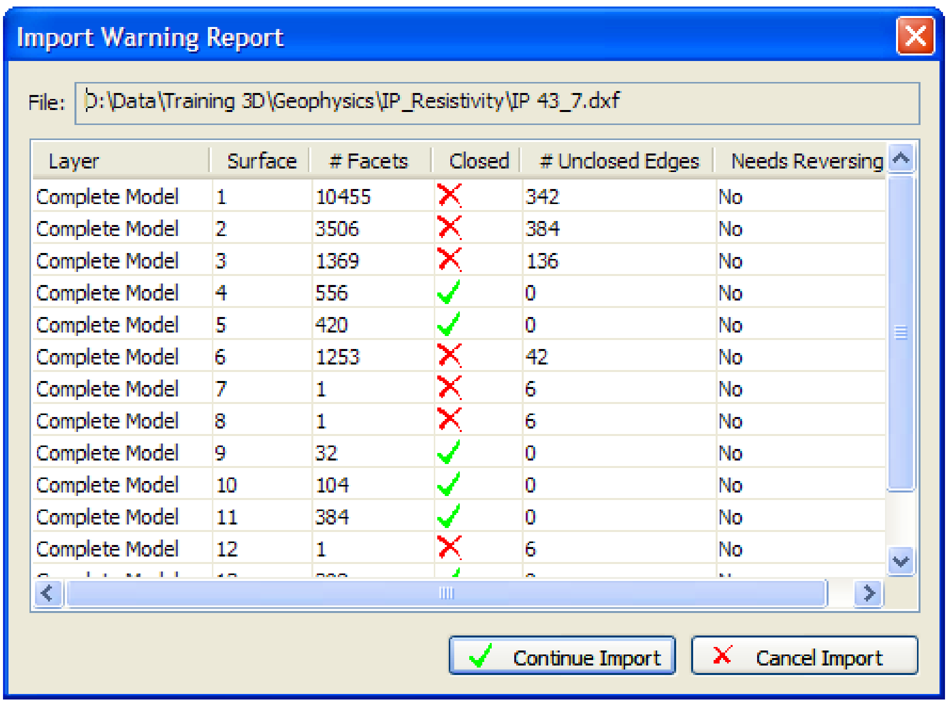
The Topology Checker can create an output file which either omits or displays incorrect object topology.

- *Using the Topology Checker*
- *Topology Checker Controls*

Using the Topology Checker

The Topology Checker will only need to be run after errors have been detected during *Importing Data into a Feature Database* (e.g. planar or volume surface) into a Feature Database. Errors in each surface will be indicated by the following **Import Warning Report** dialog, detailing which triangles or polygon edges are unclosed, and which need their vertices reversed.

Note Unclosed edges refer to when there is not an adjacent polygon/triangle facet along the edge. Reversing refers to where the order of the nodes around a polygon/triangle conflicts with the standard ordering for the file (e.g. clockwise vs. counter-clockwise).



Errors reported in the Import Warning Report.

1. If you are intending using the **Feature Editing** tools on the model, press the **Cancel Import** button, and run the **Topology Checker** from the **Utilities** menu in the 3D interface.

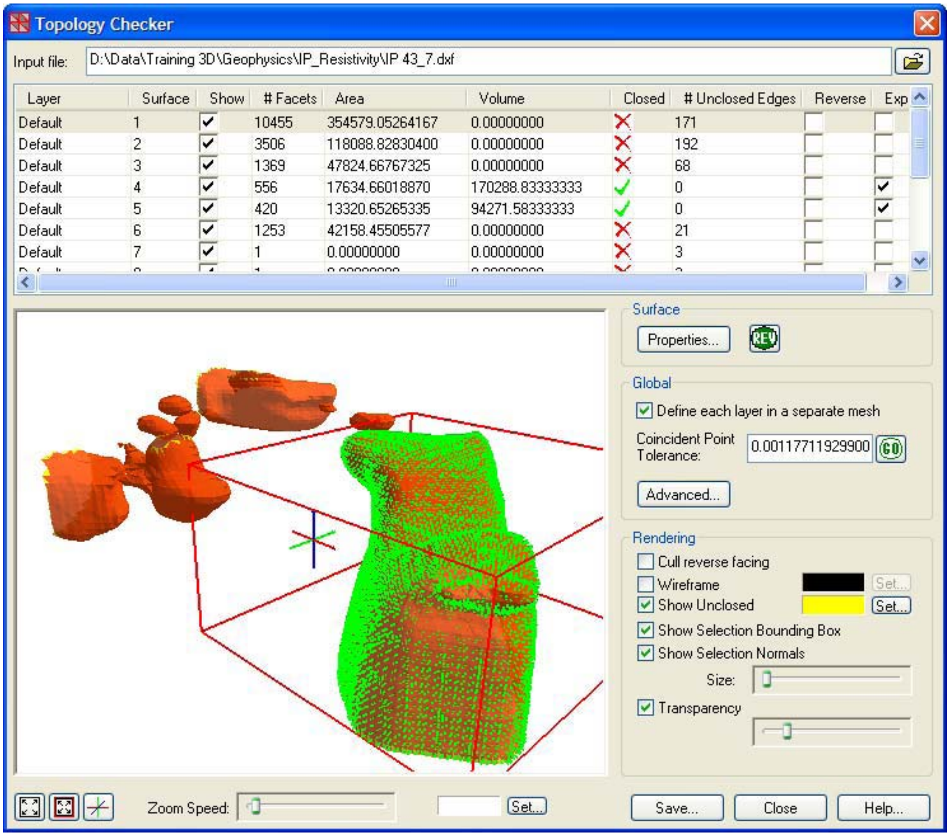
2. Use the **Browse** button at the top right of the dialog to select the source vector model file. The various layers comprising the model will be listed, and attributes detailed including whether they are closed, number of facets, volumes, areas, etc.

3. The last **Export** column allows the user to select which of the model's layers will be exported. By default all **Closed** layers will be enabled for export (i.e. unclosed layers will not be exported). This will therefore result in new vector file comprised solely of closed layers. Unclosed layers can be visualised in the preview area by enabling the **Show Unclosed** option, and setting an appropriate colour.

4. Any layers requiring the **Reversing** of their vertex order will have this applied automatically when the output file is generated.

5. Press the **Save** button to export the selected layers to a new vector file.

6. Rerun the *Feature Database Creation Wizard*, using this new vector file as the source file.



The Topology Checker dialog

Topology Checker Controls

The **Topology Checker** utility is accessed from the **Utilities** menu in the 3Dinterface. The Topology Checker dialog that appears consists of the following properties:

- **Input File** - Selects the model file to load into the Topology Checker utility.
- **List View** - The list view displays all surfaces for the input model. Right clicking on the list view will activate a context menu with various options on how to sort and select cells with the list view columns.

List View Columns

- **Layer** - Name of the layer in the input file.
- **Surface** - The surface number contained in a Layer.
- **Show** - Show/hide the selected surface.
- # **Facets** - The number of facets (triangles) composing the mesh for a surface.
- **Area** - The area of the surface.
- **Volume** - The volume of the surface.
- **Closed** - Green tick indicates that the surface closed. Blue tick indicates the surface is closed but has a negative volume. Red cross indicates that the surface is not closed. A surface is closed if all edges of the triangles in the surfaces mesh are shared by another triangle in the mesh.
- # **Unclosed Edges** - The number of edges that are not shared in the surface.

- **Reverse** - Indicates that the vertex order of the 3D faces for the selected layer is to be reversed. Press the **Rev** button to execute the reverse process for all enabled layers. Any layers requiring vertex reordering to facilitate use in Discover 3D will automatically have their vertex order reversed when the **Save** button is pressed.
- **Export** - Select which surfaces are to be exported when saving the output file.

Surface Options

- **Properties** button - The properties button will open the **Surface Properties** dialog for the currently selected surface in the list view. This dialog displays a spreadsheet containing the X, Y, and Z coordinate for each vertex in each triangle for the surface. Triangles that are not closed, that is, have an edge that does not join another triangle are displayed in a different colour.

Global Options

- **Define each layer in a separate mesh** - This control will determine whether a defined layer in the input file should be defined in its own mesh or whether all layers should be defined in a single mesh. This is done on loading the input file.

- **Coincident Point Tolerance** - Tolerance value that is used to improve the mesh by detecting coincident points and reassigning the triangles to use the first instance of any repeated point. Set this to -1 for the Topology Checker to automatically calculate this value.

- **Advanced** button - Opens **Advanced Options** dialog, the option **Surface Triangles are Counter Clockwise** can be toggled. This option will reverse the direction a triangles composing a surface will be drawn. The reversing of this can be observed by checking the **Show Selection Normals option**. The direction of the Normals arrows indicate direction.

Rendering Options

- **Cull reverse facing** - Turn on/off display of any back facing triangles in the mesh.

- **Wireframe** - Turn on/off display of the wireframe mesh for the model. The colour of the wireframe mesh can be chosen by using the **Set** button.

- **Show Unclosed** - If this option is turned on the colour of any triangle faces that are not closed, will be overridden with the set colour. The override colour can be set using the appropriate **Set** button.

- **Show Selection Bounding Box** - Turn on/off the bounding box for the model. The bounding box is defined for the currently selected surface in the list view.

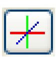
- **Show Selection Normals** - Turn on/off the display of normals for the model. The normals are defined for the currently selected surface in the list view only. The size of the normals can be modified using the size slider control. A normal is defined by the centroid of a triangle in a mesh and the perpendicular angle to this facet.

- **Transparency** - Turn on/off transparency for the model. The percentage of transparency can be modified using the appropriate slider control.

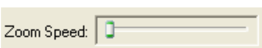
3D View Controls

Fit view to window - Zooms the view so that the entire model can be seen.

Fit view to bounding box of selected surface - zooms the view so that the selected surface is the focus of the view.



Advanced settings for the axes scale - Allows the user to apply individual scales to the X, Y and Z axes, as well as defining the position where the user wants the view to look at.



Zoom speed - Changes the speed at the which the user can zoom in/out to/from the model.

Background colour - Sets the colour of the background window for the 3D view.

Transform Vector File

The **Transform Vector File** utility can transform, reproject, and convert a wide range of vector file formats. The following input file formats are supported:

3D Studio .3DS

AutoCAD .DXF

Datamine wireframe (point and triangle) .DM

ER Mapper vector .ERV

ESRI shapefiles .SHP

ESRI TIN .ADF

Gemcom .BT2

GOCAD vector .TS, .PL, .VS

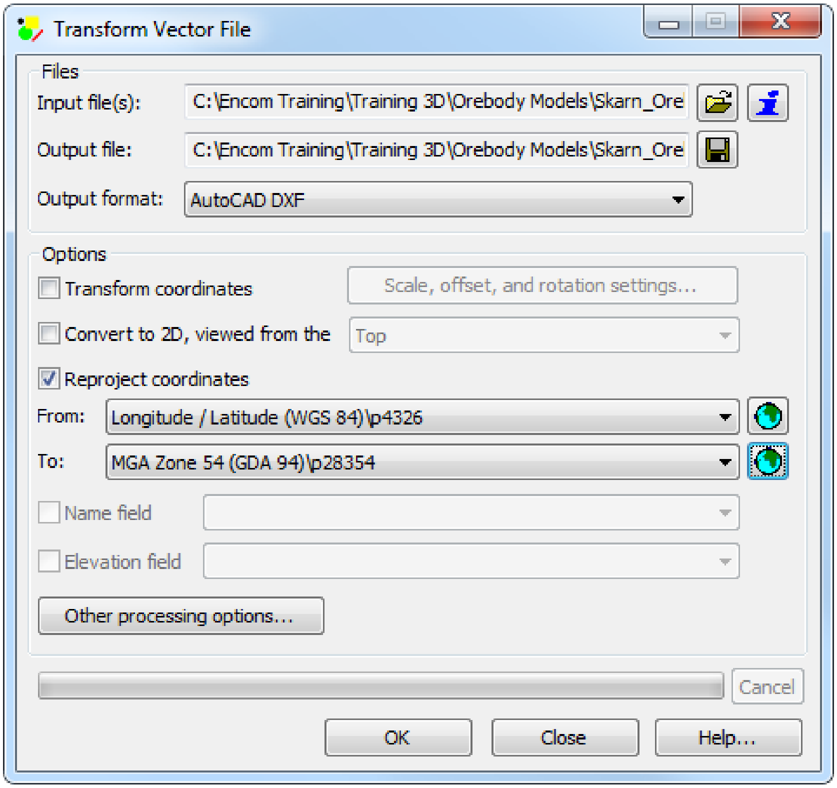
Google Keyhole Markup Language .KML

GPS exchange .GPX

LizardTech MrSID .SID
MapInfo Professional .MIF
MapInfo Professional .TAB
Surpac string .STR
Surpac wireframe .DTM , .STR
Vulcan triangulation .00T

When converting between file formats (with or without transformation and/or reprojection), the following output vector file formats can be created:

AutoCAD .DXF
CSV text file .CSV
Encom .TKM
ESRI shapefiles .SHP
GOCAD vector .TS
Google Keyhole Markup Language .KML
GPS exchange .GPX
MapInfo Professional .MIF
MapInfo Professional .TAB
Surpac string .STR



The Transform Vector File dialog

A range of Transformation options are available as independent X, Y and Z operations, allowing vector objects to be scaled, offset and rotated.

Transform options available are:

- **Transform coordinates** – enable the options button for Scale, offset and rotation settings. This is useful for converting feet units to metres or transforming from a local grid to UTM.
- **Convert to 2D, viewed from** – converts an appropriate 3D input Vector file into a 2D view. It can be viewed from each primary axis direction.
- **Reproject coordinates** – reprojection applied to coordinates. Note the from field is automatically populated with the first input file's projection if it found.
- **Name field** – certain formats only support a single attribute or label field for the vector objects. If enabled, select the desired field for labelling the points.

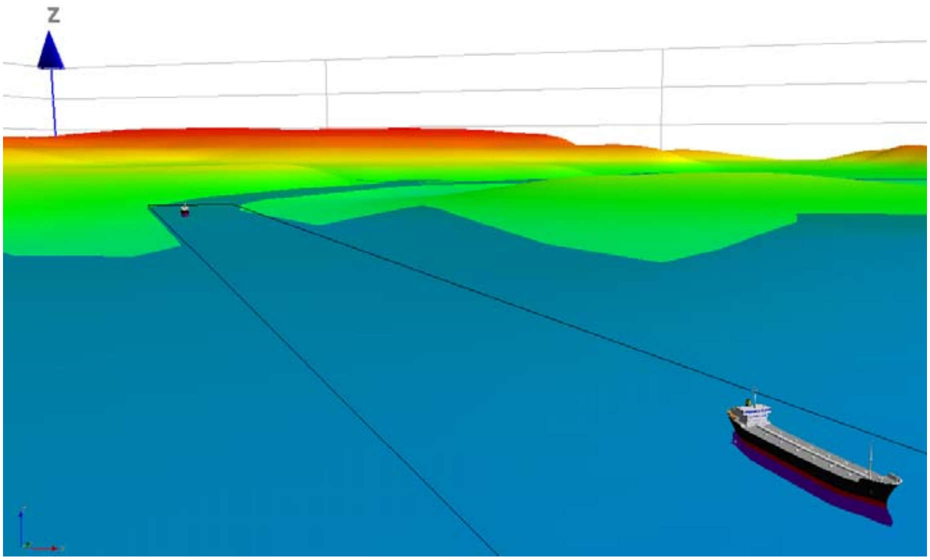
- **Elevation field** – when converting form a 2D (i.e. TAB file) to a 3D file format (i.e. DXF), this field is enabled to define the elevation of the object. For example this could be populated by using Surfaces>Assign Grid Values from a DEM.

Other processing options include:

- **Convert to point cloud** – converts all nodes/vertices in the input vector object to individual point objects in the output vector file.
- **Smooth polylines and polygons** – applies a line smoothing filter, with a width defined over the nearest nodes.
- **Correct polygon vertex order problems** – checks the node order in all polygon objects. Outer parts will be stored clockwise and inner holes will be stored counter-clockwise.
- **Compress polylines and polygons** – removes any nodes with a spacing less than the defined distance.
- **Convert closed polylines to polygons** – converts any closed polyline objects to a polygon object. This is particularly useful for formats such as DXF which do not have a polygon object type, but do have closed polyline objects.
- **Convert polylines with one point to point objects and polygons with two points to polylines** – removes any erroneous objects.

Vector files can also be **Reprojected** to accommodate absolute coordinate handling in Discover 3D. For instance, if all of your MapInfo data is in a UTM projection, but an ore body DXF model from a CAD program is in a different projection, this option will enable the DXF to be reprojected into the same projection as the rest of the 3D data. Enable the **Reproject Coordinates** option, and set the appropriate **Source** and **Target** projections using the **Choose** buttons.

Note The projection list available in the Discover Vector Import utility is stored in a separate file to the `MapInfow.prj` file. If you wish to reporejct into a custom coordinate system which has been added to the `MapInfow.prj` file then copy the custom projection line into the `Encom.prj` file located in the `...\Program Files\Encom\Common\Projections` folder.



3DS tanker ship models transformed (offset, rotation and scaling) into a 3D session to illustrate the scale over a shipping channel dredging project.

To use the Transform Vector File tool:

1. Under **Files**, select the input file and output file names and locations.

2. If a different output file format is required, select the format from the **Output format** box.

3. To explicitly position, scale and/or rotate the input file, select the **Transform coordinates** option, and specify the appropriate parameters.

4. If converting into a 2D vector file format (e.g. a TAB), enable the **Convert to 2D** option, and specify the desired view direction of the output file (default view from the Top).

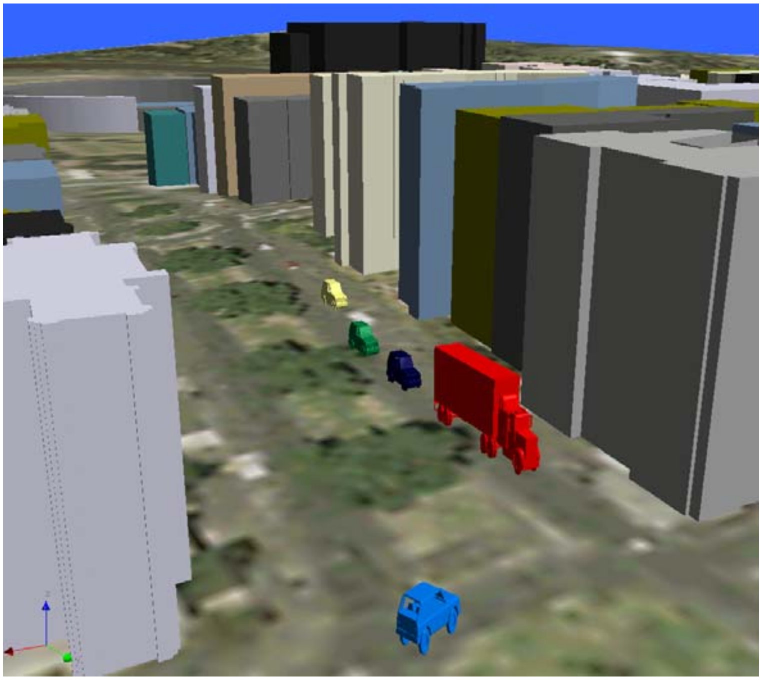
5. To reproject the input file, enable the **Reproject coordinates** option, and specify the input (From) and output (To) projections. See *Projections in 3D* for more information about reprojecting data.

6. A range of additional advanced transformation options (described above) are also available via the **Other processing options** button.

Batch DXF Transform

The **Batch DXF Transform** utility allows one or more vector models to be transformed into a large number of vector objects and placed into the 3D window. These vector objects are outputted into a single DXF file.

This is a useful way to rapidly populate a 3D environment with multiple individually positioned, scaled and rotated vector objects, based on a number of source DXF models. An example of this utility's use would be to create multiple vehicle objects from a single automobile DXF model. Similarly if a number of DXF models were available (e.g. car, truck and helicopter models), multiples of each model could be created.

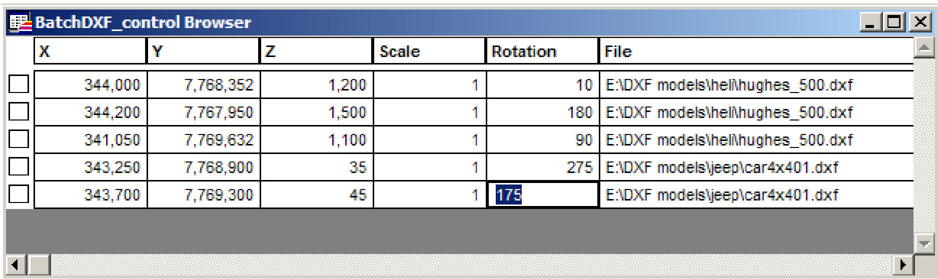


Multiple cars (behind the truck) in a cityscape produced using the Batch DXF Transform utility

This utility requires a Control table (TAB file) with the following attributes:

X	(compulsory)	The X coordinate of the output object
Y offset	(compulsory)	The Y coordinate of the output object
Z offset	(optional)	The Z coordinate of the output object

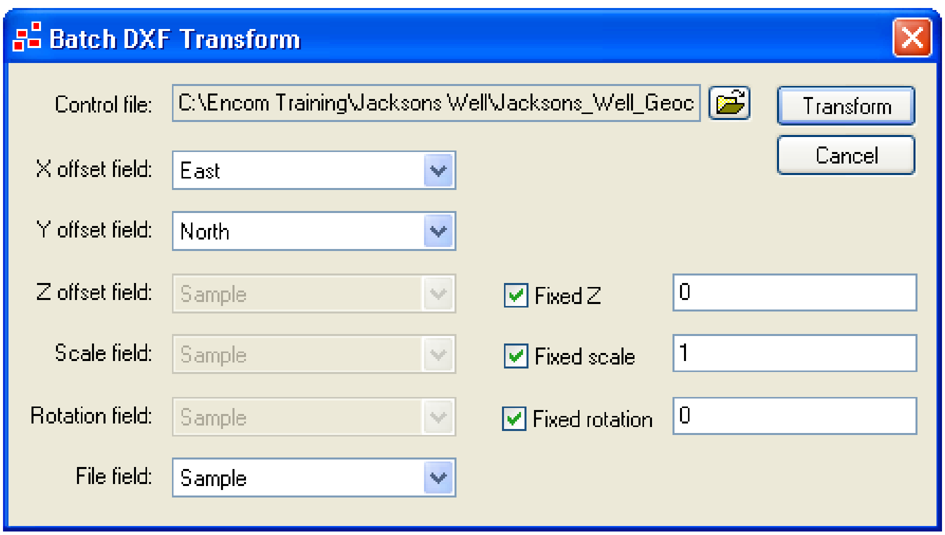
Scale	(optional)	The scale factor to be applied
Rotation	(optional)	The degree of rotation to be applied (positive value are anticlockwise)
File	(compulsory)	The file path and name of the source DXF model


BatchDXF_control Browser

X	Y	Z	Scale	Rotation	File
344,000	7,768,352	1,200	1	10	E:\DXF models\heli\hughes_500.dxf
344,200	7,767,950	1,500	1	180	E:\DXF models\heli\hughes_500.dxf
341,050	7,769,632	1,100	1	90	E:\DXF models\heli\hughes_500.dxf
343,250	7,768,900	35	1	275	E:\DXF models\jeep\car4x401.dxf
343,700	7,769,300	45	1	175	E:\DXF models\jeep\car4x401.dxf

Example control table

Use the **Browse** button in the **Batch DXF Transform** dialog to open the control table. This dialog will attempt to automatically populate the various parameters based on field names; otherwise use the drop-down menus to assign the various fields correctly. If the optional fields are not present in the control table, they will need to be assigned **Fixed** values by ticking the relevant option to the right, and assigning a value.



Batch DXF Transform dialog

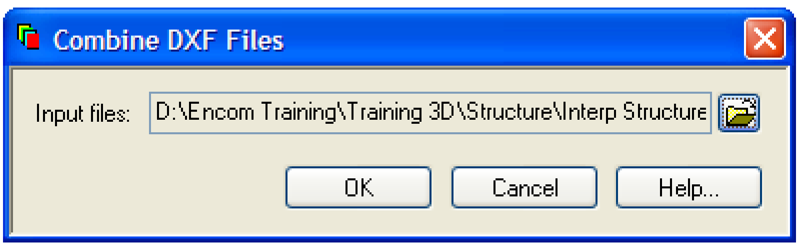
Clicking the **Transform** button will prompt for an output DXF file name and location. When this file has been created, it can be opened in the 3D window by dragging and dropping from Windows Explorer, or using the **Display>3D Vector** menu option.

Combine DXFs

The **Combine DXFs** utility enables multiple 3D DXF files to be combined into create a single 3D DXF file.

If there are a number of 3D DXF files that are generally opened in Discover 3D together, to save having to open each of them separately, a single DXF can be created. The DXF files must be in the same directory prior to combining.

On the **Combine DXF Files** dialog click on the button and browse to the directory containing the 3D DXF files to combine. Select all the DXF files to combine using the **CTRL** and **SHIFT** keys depending on whether they are consecutive in the file list.



Combine multiple DXF files.

Click **OK**. In the following dialog choose a new file name and location for the combined DXF file and click **Save**. The combined DXF file is not automatically displayed in the Discover 3D map window. Use the **Display>3D Vector** or **Display>2D Vector** menu option to display the combined DXF data.

Extrusion Wizard

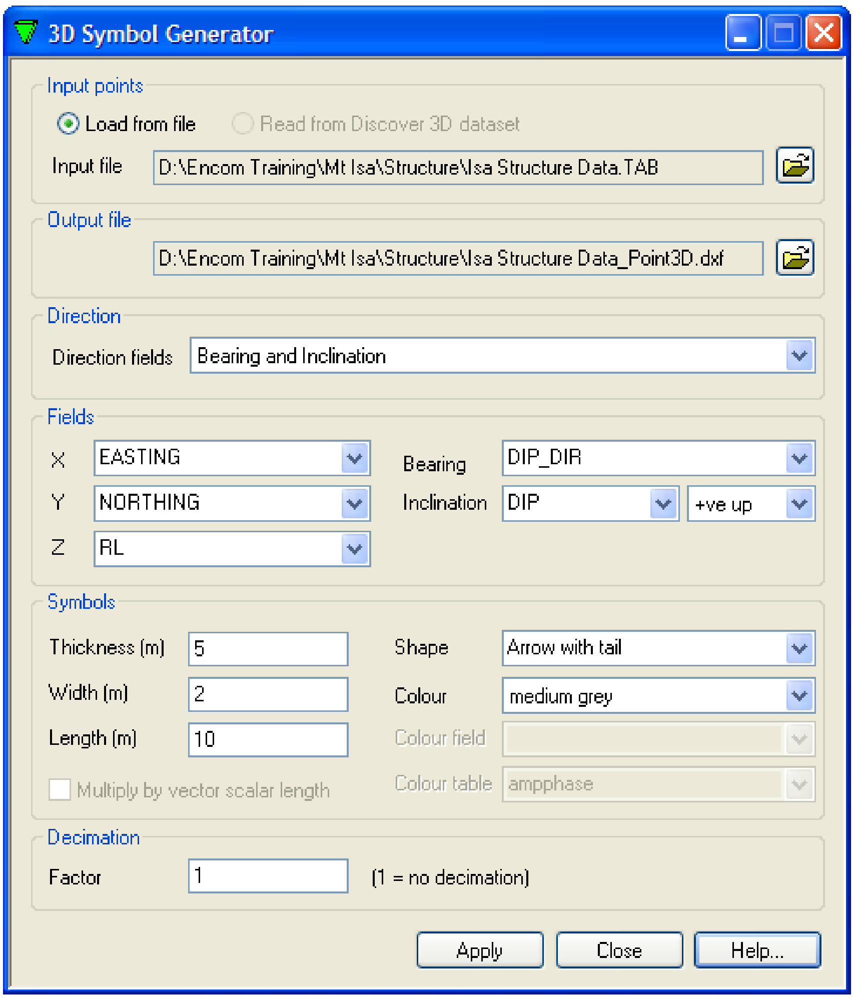
The Extrusion Wizard allows you to extend (extrude) the shape of a 2D or 3D object (e.g. a polygon or polyline) from a base surface to a second or upper/lower surface. This allows meaningful and useful visualisation of various bodies such as fault surfaces, mine shafts and workings, vein systems and buildings in three dimensions.

Refer to *Extruding Models from Points, Lines and Polygons* for further details.

3D Point Symbols

Use the **3D Symbol Generator** utility with point data to create orientated DXF symbols for display in Discover 3D.

The 3D Point Symbols utility can be used with any data table accessed by Discover 3D or with an external MapInfo Professional .TAB or .MIF file. The data table may be of line or point data type and should contain at a minimum Easting, Northing and Elevation or depth fields. For oriented data (eg. Downhole structure data), additional fields for Bearing and Inclination can be selected. Discover 3D uses these fields to create individually oriented symbols for each data entry.



3D Symbol Generator dialog with oriented data from TAB file

The following controls are provided:

Input Points

- **Load from file** - Location and attribute information can be converted to point symbols using MapInfo Professional .TAB or .MIF format files. Select this option and specify the input file location.

- **Read from Discover 3D dataset -** If dataset that has been opened directly into Discover 3D can be used for symbol creation at each data point selected by line.

- **Output File** - The created symbols are stored in an AutoCAD DXF format file. By default the output file is saved using the same name and location as the input file with a "_Point3D" extension. The name and location of the output file can be modified if required.

- **Direction** - 3D points can be orientated either via specification of Bearing and Inclination fields, or Vector component (u, v, w) fields.

- **Fields** - Data fields used for location (X and Y) and depth (Z) must be selected. The Direction option set (above) will control either Bearing and Inclination, or Vector (u, k, w) fields to be assigned.

- **Symbols** - The Width, Length and Thickness of the 3D symbols created can be specified (in scaled metres). The following 3D Shape symbols are available:

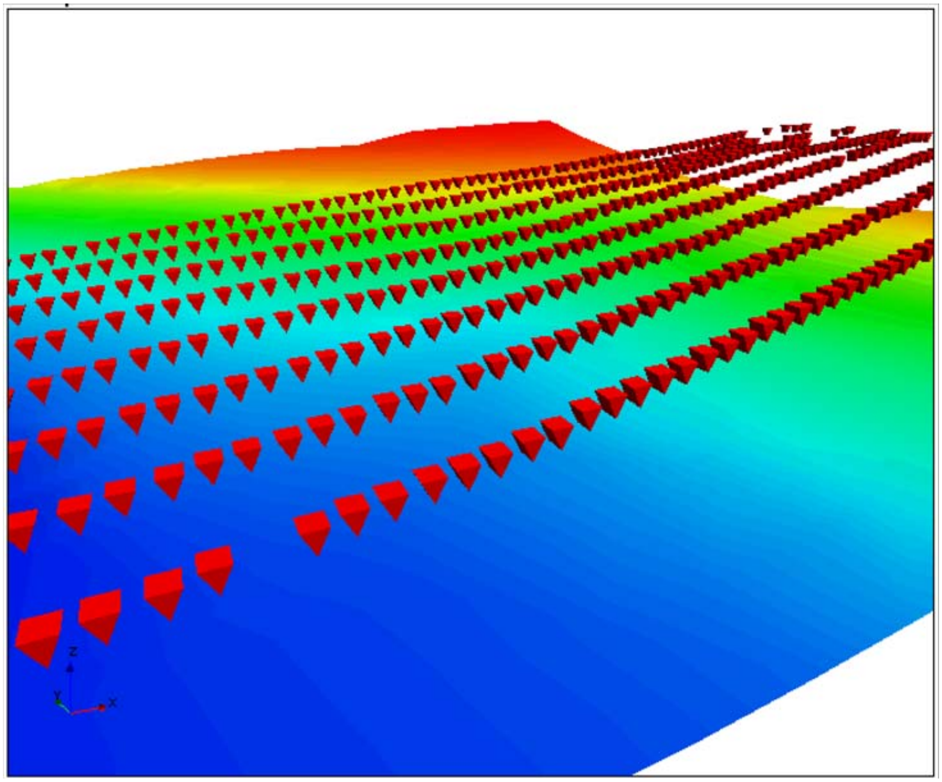
  Triangle
  Rectangle
  Tabular
  Arrow head
  Arrow with tail

- A **Colour** can also be set for the output point symbols. Setting this option to **Modulated from Field** allows each unique attribute in the specified **Colour field** to be coloured according to a selected **Colour table**.

- If Vector direction fields have been specified (u, v, w), an option to **Multiply by vector scalar length** is available.

- **Decimation** - Only available when a 3D dataset is being used. Displays only every Nth sample/point, where N is a user-specified value. For example, specifying 3 with display only every 3rd sample.

Click **Apply** to create the 3D symbols. When processing is complete the output DXF file is created and the symbols are automatically displayed in Discover 3D.Surfaces, other vector files, located bitmap images, drillholes, points, lines and voxel models can be added to this new 3D window.



Displaying soil geochemical points as .DXF objects using the 3D Symbol Generator

Drillhole Planner

Discover 3D provides the powerful capability to dynamically position and plan new drillholes directly within the 3D environment. This allows the user to target 3D objects such as voxel isosurface grade shells, extruded quartz veins, .DXF solids (such as alteration zone volumes generated from digitized section boundaries) or feature object interpretations.

Refer to *Drillhole Planner* for further details.

21 Creating Images and Movies

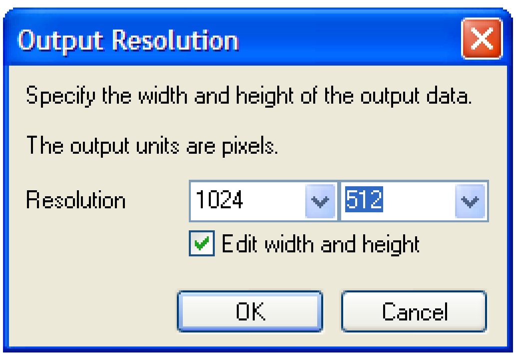
Discover 3D includes a range of output options for your 3D views:

- *Capturing the 3D View as an Image* (.BMP, .PNG, etc) for use externally (e.g. PowerPoint presentation).
- *Adding the 3D View to a 2D Map Window*, for use in a layout Frame Object.
- *Printing* using the Page Layout mode.
- *Creating a Geolocated Image*, either for use in 3D, or for adding to an existing drillhole project section.
- *Creating a Fly-Through Animation* of your 3D environment.

Capturing the 3D View as an Image

File>Save View As

This allows the current 3D view to be saved as an image at a user-specified resolution. A range of output image formats are available, including BMP, JPG, PNG, and TIF & EMF. This image is not georeferenced i.e. it is essentially a screenshot. These images could be used in a Word or PowerPoint document.



The Save View As dialog allowing output resolution specification

Consider the Orthographic View mode (see *3D Display Modes*) for output image for technical evaluations (i.e. without the distance distortion of the default perspective view).

Adding the 3D View to a 2D Map Window

View>Send to MapInfo

Add **3D View to MapInfo** button.

It is possible to transfer a 3D view back into a new MapInfo Professional mapper window. This is a useful function if you wish to have a snapshot of a particular display that can be used in a MapInfo Professional layout as a Frame Object (e.g. a cross-section layout with a small 3D snapshot to give the drillhole project a 3D perspective).

The transfer creates a bitmap of the Discover 3D screen display with an associated non-earth projection TAB file that is then displayed in a new mapper in MapInfo Professional.

To transfer a view from Discover 3D to MapInfo Professional:

1. Organise the layers and view position as required in Discover 3D.

2. Select the **Add 3D View to MapInfo** button or use the **View>Add 3D View** to MapInfo menu option.

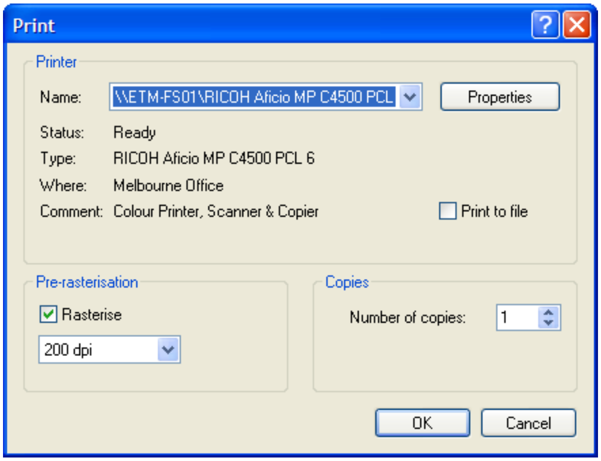
3. Return to MapInfo Professional and a new mapper display window should open containing the Discover 3D view.

Note The MapInfo Professional mapper view of the Discover 3D display is static and is not related in real-time to that of Discover 3D. To modify the view, return to Discover 3D, alter the view and repeat the above steps.

The TAB and bitmap files used for the Discover 3D view as displayed in MapInfo Professional are stored in the folder defined by the Discover Temp file type under the Discover 3D **Tools>Options>File Locations** tab. The files are named with a unique filename.

Printing

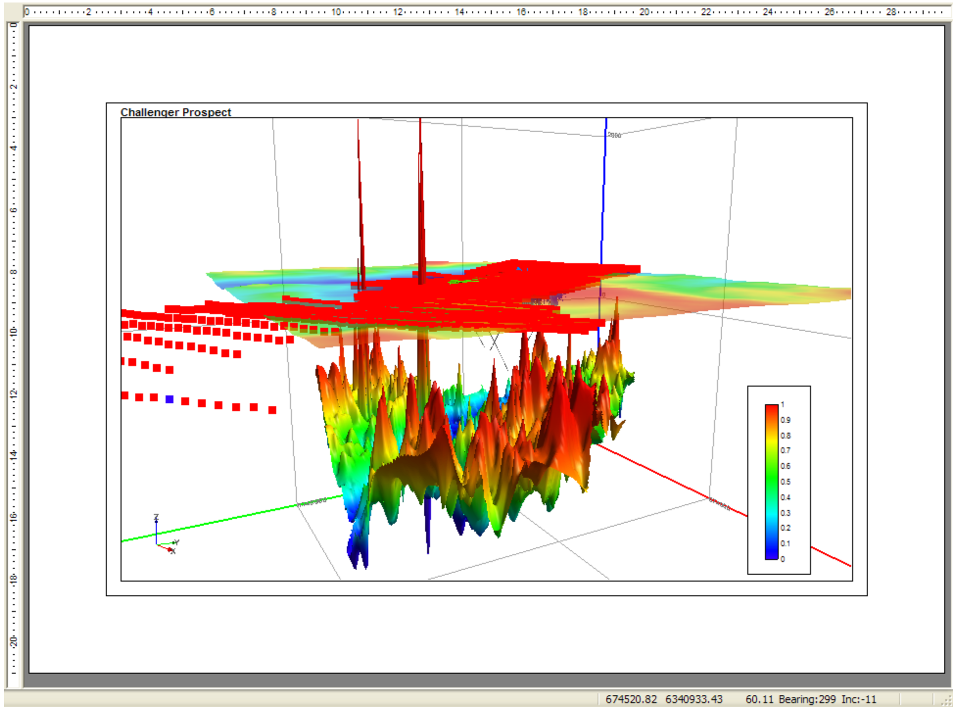
Every aspect of the current 3D view can be printed directly to a printer or plotter using the **File>Print** menu option, with printer and page size options configured via the standard **File>Page Setup** menu option.



Printer and Rasterize options

Note Discover 3D can use standard Windows drivers directly for printing **OR** it can use a rasterizing option to speed up and produce higher quality printed output. The rasterize option is included on the **Print** dialog and is recommended for routine use in printing. It is recommended that a rasterizing level of between 150 and 200 dpi is optimal for speed and quality.

However professional looking results are better achieved when Discover 3D is in Page Layout mode (see *3D Display Modes*): this allows the actual output 3D display for the currently set page size and orientation to be previewed and modified prior to printing.



The Page Layout view of the current 3D environment.

Within Page Layout mode, the full set of buttons for *Navigating in 3D* are available to precisely position the view within the page frame. Additionally, Discover 3D provides a number of controls on the *Zoom Controls Toolbar* specific to this view mode:

The **Zoom In** and **Zoom Out** controls allow adjustment of the overall zoom ratio of the page display.

The **Pan** button allows you to pan around the page view, useful when zoomed into larger page sizes (e.g. A1).

The **Fit to Page** button is only available in Page Layout view, and automatically resizes the page to fit to the extents of the view.

The *3D View Manager Tool* allows the user to save custom views within the 3D window.

The Orthographic View mode functions in Page Layout mode (see *3D Display Modes*), allowing hardcopy production for technical evaluations (i.e. without the distance distortion of the default perspective view).

Customising the Page Layout

The Page Layout view can be fully customised with titles, modified frame colours/ thickness, modified margins, etc via the *3D Map Properties* dialog in the Workspace Tree. Options include:

- **Title** - the Title tab allows the addition of a title string in the Coded Title String area with a selected Font. The Horiz and Vert options control the Title Position with respect to the 3D window display. The Anchor options locate the title locally (this can be fine tuned using the Add Offset controls).

- **Border** – the Appearance tab allows Inner and Outer borders to be displayed, with colour, thickness and line style options.

- **Background** - Data Area and Border (the area between the inner and outer borders) colours can be specified in the Appearance tab. The Data Area control is the same control as the Background Colour button on the *Main Toolbar*.

- **Minimum Plot Margins** – these can be set as millimeters recessed from the Outer Border position.

The position and size of the 3D view frame inside the page layout can be modified by enabling the Select button.

The position and size of the 3D view frame inside the page layout can be modified whilst in **Select** mode. Click on a border of the 3D view frame (the 4 corner points should display square nodes), and drag either a corner point (to resize) or an edge (to move).

Default and Offline Printer Configuration

The *Page Setup Tab* of the **Tools>Options** dialog in the 3D Window allows the default printer and page size/orientation to be configured.

It also allows a default offline page size to be set.

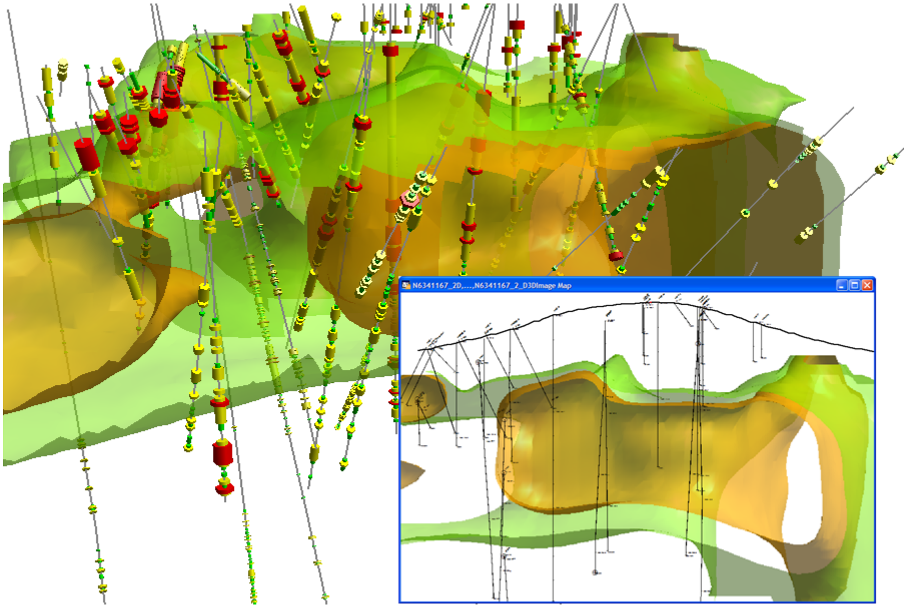
Creating a Geolocated Image

Tools>Georeferenced Image Export

Discover 3D's **Georeferenced Image Expor**t tool allows a geolocated image of the Cursor Plane to be created (i.e. a .PNG image with associated *Encom Georeferenced Bitmap (EGB) Format* header file). The image is created perpendicular to the cursor plane in Orthographic view mode (see *3D Display Modes*), and the width/extent of data captured can be controlled by the standard cursor plane Clipping options (see *Changing Cursor Plane Properties*).

This is a powerful way to capture one or more slices through your dataset (e.g. voxel models, .DXF solid models) as georeferenced images, for:

- Displaying in other applications (e.g. PowerPoint or Word).
- Efficient display in Discover 3D of more complex and memory-intensive datasets (e.g. displaying a voxel model as a series of image slices).
- Exporting to associated Discover drillhole cross-sections for detailed data comparison and interpretation (e.g. exporting to each section it's associated envelope through a voxel model).

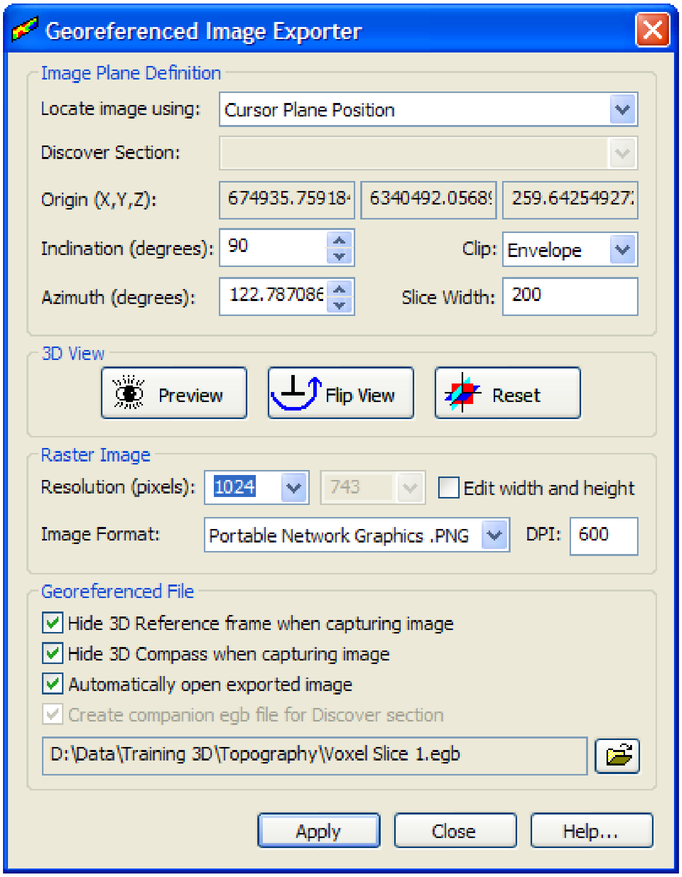


The Georeferenced Image Export tool can orientate the cursor plane via two methods:

- *Creating an Image Using the Cursor Plane*: use either the normal keyboard controls or individual specification of the necessary parameters (origin XYZ, inclination, azimuth).
- *Creating a Drillhole Cross-section Image*: use an open Cross-section in a Discover drillhole project to align the cursor plane.

Creating an Image Using the Cursor Plane

1. Open the Georeferenced Image Export tool (on the Tools menu in Discover 3D).
2. Set the ‘Locate image using’ to **Cursor Plane Position**.



Geolocated Image Export dialog configured to capture the manually positioned cursor plane with a 200m data envelope width

3. Orientate the cursor plane as required, using either the various parameter fields presented in the dialog, or the normal cursor plane keyboard controls (see *3D Cursor Keyboard Shortcuts*).

4. Set the desired data Clip type from the pulldown list (see *Changing Cursor Plane Properties*). The Envelope and Slice options (with appropriate Widths) are generally the most effective options.

5. Press the **Preview** button. The 3D view will change to orthographic mode (see *3D Display Modes*), and orientate perpendicular to the cursor plane. If the view is on the wrong side of the cursor plane, press the **Flip View** button to swap to the opposite side.

6. If the clipping options are not correct, alter these and repress **Preview** to visualise the result. The **Reset** button will take the view back prior to being perpendicular to the cursor plane, allowing the cursor plane be further manipulated.

7. The output image Format, Resolution and DPI can all be modified. The higher the resolution and DPI, the better the quality of the image, but the larger the image size (MB).

8. Ensure the **Automatically open exported image** option is enabled.

9. Specify an output directory and file name using the **Browse** button at the bottom right of the dialog, and press **Apply**.

The image will be displayed in the 3D view.

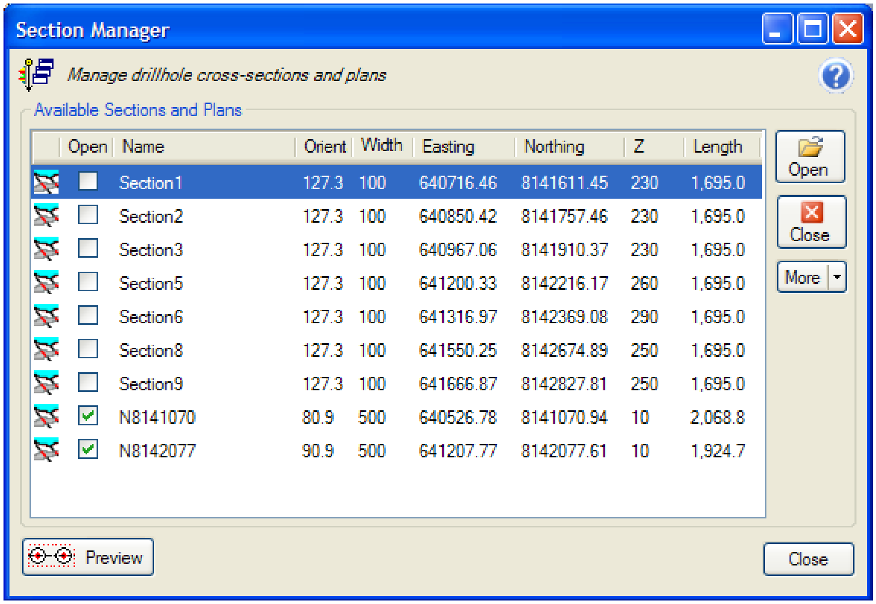
Note This EGB can be imported into Discover using **Discover>Import** or **Export>Encom PA located Image Import** only if the cursor plane is aligned exactly in the XY plane, by pressing the Z key. This is useful for generated surface map top down views of 3D objects.

Creating a Drillhole Cross-section Image

1. In MapInfo/Discover, if the drillhole project is not already open, open it with **Drillholes>Project Manager**.

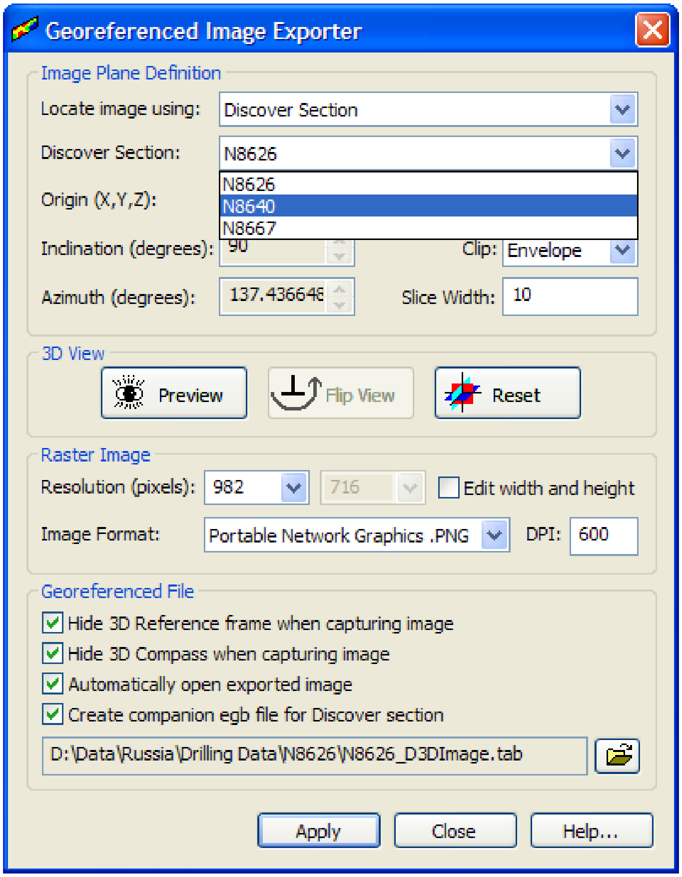
2. Still in MapInfo/Discover, use the **Section Manager** to open the destination cross-sections.

Note Horizontal plans are not supported. They can however be exported using the cursor plane method described above and align the cursor plane to the horizontal plane by pressing the Z key.



In Discover 3D, open the Georeferenced Image Export tool (on the Tools menu)

3. Set the ‘Locate image using’ to **Discover Section**.



Selecting a target cross-section from the Geolocated Image Export dialog

4. Choose the required destination **Discover Section** from the pull-down list.

5. The Clip type will be automatically set to Envelope (see *Changing Cursor Plane Properties*), with the Width set as per the cross-sections envelope width. These can be modified.

6. Press the **Preview** button. The cursor plane will reorientate to mirror the specified cross-section. Simultaneously the 3D view will change to orthographic mode (see *3D Display Modes*), and orientate to the cross-section's view direction (perpendicular to the cursor plane/section).

7. If the clipping options are not correct, alter these and repress **Preview** to visualise the result. The **Reset** button will take the view back prior to being perpendicular to the cursor plane.

8. The output image Format, Resolution and DPI can all be modified. The higher the resolution and DPI, the better the quality of the image, but the larger the image size (MB)

9. The image file will be automatically created in the cross-section or drillhole project folder (depending on your project settings) with a name `sectionname_D3DImage`. These can be changed using the **Browse** button at the bottom right of the dialog, and press **Apply**.

10. Press **Apply**. A successful creation message will be displayed. After pressing OK, the image will be automatically opened in the section mapper window in MapInfo/Discover if the **Automatically open exported image** option is enabled. It will also be displayed in the 3D view if the **Create companion egb file for Discover section** option is enabled.

Creating a Fly-Through Animation

Discover 3D allows 3D views to be captured for replay using a **Fly-Through Creation** Wizard. An automatic transition between view positions allows the playback to be smooth and provide the visual appearance of a ‘movie’. The ‘fly-through’ effect can follow a predetermined track or a series of joined view points with gradual transition between each.

Creation of a **3D Fly-Through Animation** is controlled by a wizard that generates a script. Within the script are a series of 3D location points that view the display with different angles and elevations. Smoothly moving from one position to the next creates the animation.

The **3D Fly-Through Animation** track can be specified by:

- MapInfo Professional® .TAB file.
- Flight path specified from data open in Discover 3D.
- Manually specifying various predefined views.
- Edit an existing Fly-Through (.FLY) file.

Other features of the **3D Fly-Through Animation** feature include:

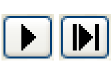
- Ability to repeat operation or reverse to replay backwards.
- Smooth view along predefined track using a Bezier curve approximation.
- Control frame rate, flight speed, view pause, view rotate and point examination.

- Create 3D DXF file of a created flight track including definable points to mark view points.
- Capture fly-through as an Video movie for later viewing. Selection of video compression (CODEC) provided.
- Audio identification at set points. These sound effects are heard while a script is being replayed through the computer sound system. Sound cues such as “Note the surface anomalism coincident with this drilling…” can be a useful adjunct to seeing the 3D display replayed. Note that when recorded to an Video movie file, the sound is not transferred.

To create a simple 3D animation

1. Ensure the required datasets are open and visible in the 3D window
2. Select the **Utilities>Fly Through Wizard** menu option
3. In the Fly Through Creation Wizard dialog that opens (Step 1) select the **Manually from the current Discover 3D view** option. Press Next
4. In the Step 3 dialog, ensure that the **Loop Style** is set to None, and **Bezier Smoothing** is enabled with a Low Tension. Press Next.

5. In Step 5, use the 3D nVideogation controls (see *Navigating in 3D*) to orientate the 3D view at the starting view for your animation. Press the **Camera Capture** button to capture this – a new row will be added to the dialog with this point’s details.
6. Use the 3D nVideogation controls to move to the next view point and again use the **Camera Capture** button to capture this location. In this view point’s row entry, specify the Duration in seconds the animation will take to travel between the previous (start) point and this location.
7. Repeat step 6 until all required view points have been captured and durations entered.



8. Play the animation in the 3D window by pressing the **Play** button. A selected number of view points can be previewed using the **Preview** button to the right of the Play button.
9. Edit the animation by adding, inserting, modifying or deleting the necessary view points. These controls and additional are discussed fully in *The Fly-Through Creation Wizard* section below.

10. If the overall speed of the animation is too fast, rather than respecifying each individual duration, use the Back button to go to the **Step 3 – Options** dialog, and alter the **Playback Speed** accordingly (e.g. reduce it from 1 to 0.5 to double the time taken between each view point).

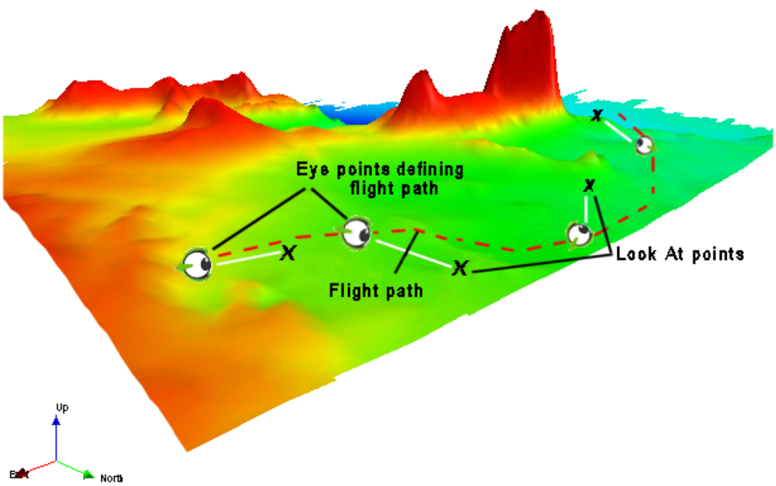
11. When satisfied with the flight path and durations, press Next to open the final Step 5 dialog.

12. Specify an Output FLY file using the adjacent Save button (this file will allow you to come back and modify your flight path, add new datasets, etc). Also specify the name of an output Video file.

13. Press the **Compressor** button, and select the compression CODEC to use for your output movie (see *Step 5 - Output Formats* below for further details). Once selected, press the **Create Video** button to generate an .Video movie file. The quality of the output movie can be altered by adjusting the **Frame Rate** (i.e. frames per second) in the **Step 3 – Options** dialog.

How Fly-Through Animation Works

A fly-through animation operates by positioning an **Eye Point,** (also called a **Capture Point** or **Position**) and directing the view to a **Look At** point. When played back, the view is defined from each **Eye Point** towards the **Look At** point and then a gradual change of view to the next successive **Eye Point** to the next **Look At** point. Both point types require an X, Y and Z specification as shown in the diagram below:



Flight path of 'joined' Capture Points as they direct their view to Look At points

Each **Eye Point** can be recorded from a manually positioned location in the 3D map display or at interpolated locations from a predefined flight path defined from a database or MapInfo Professional TAB file. The **Eye Points** look directly to their associated **Look At** point and then gradually move and update with the next **Eye** and **Look At** points during the animation. The movement from one **Eye** point to the next can be smoothed or direct (determined by the **Smoothing** and **Tension** settings). The speed at which the fly-through travels is dependent on the **Playback Speed** and **Flight Speed** (see *Step 3 - General Playback Options*).

Each **Eye Point** (**Capture Point**) can have a number of events and properties associated with it. An event may be a pause, sound, change of display property (eg transparency, application of a different LUT, etc) or a rotation of the view about an **Eye Point** or **Look At** point.

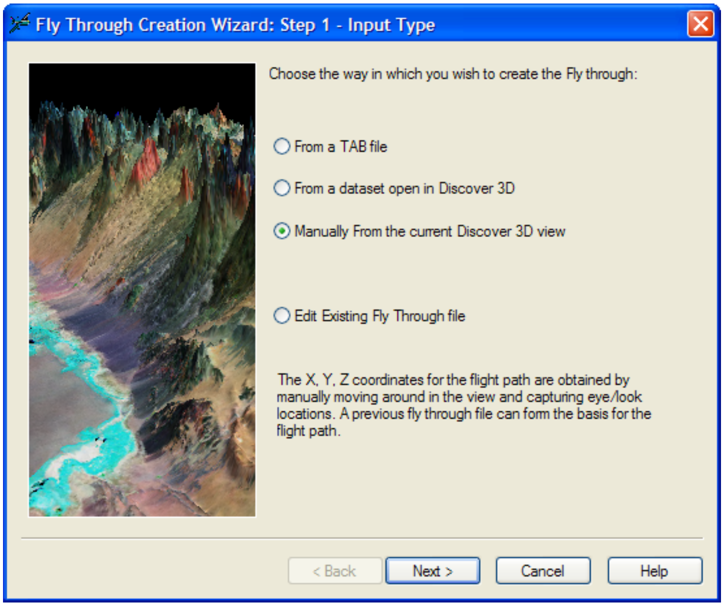
The individual **Eye Points** and their associated **Look At** point, including any associated events, are saved to a script file called a **FLY** file. These files, created for a given project area, provide the mechanism to replay the fly-through animation as desired.

The Fly-Through Creation Wizard

The following sections describe in detail the controls available within this tool.

Step 1 - Select Input Type

The **Fly-Through Wizard** can be accessed from the **Utilities** menu. The **Fly-through Creation Wizard: Step 1 – Input Type** dialog is displayed:



Fly-Through Wizard dialog and options to create a flight path script

A fly-through operates by different views being displayed along a predetermined path. The positions of the views are recorded in a script file (FLY file extension). The flight path may be specified by one of the four following methods:

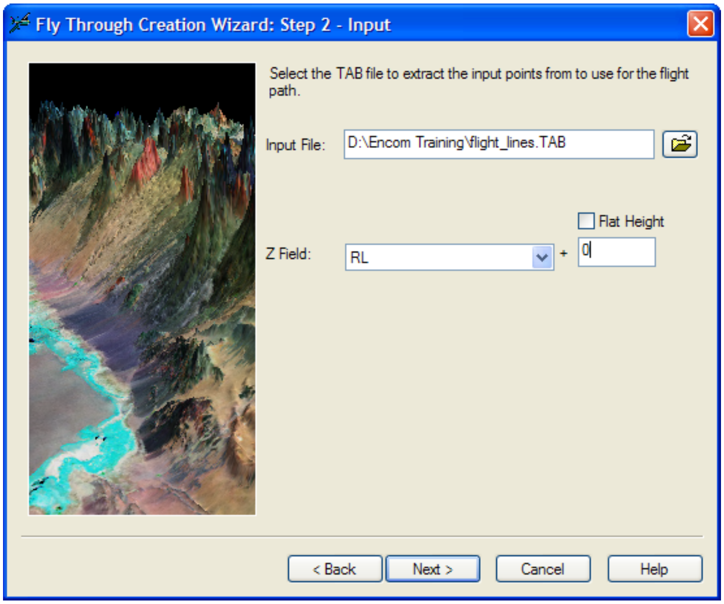
- From a TAB file – MapInfo table that contains XYZ coordinates representing the flight path course.
- From Discover 3D Dataset – XYZ coordinates for flight path based on values in fields in a Discover 3D dataset.
- Manually from current 3D view – Specify a number of 3D map window views to use as **Eye** and **Look At** points which are linked together to form the flight path.
- Edit Existing Fly-through file – Modify a previously created fly-through script file (FLY).

Depending on the flight path generation option selected, a different series of dialogs may be displayed.

Step 2 - Input Parameters

MapInfo TAB File

Select the **From a TAB file** option and click the **Next** button. In the **Fly-through Creation Wizard: Step 2** dialog use the **Browse** button to locate the appropriate MapInfo TAB file. This TAB file must be mappable and contain points, polyline or polygon map objects which represent the course the fly-through will follow. The X and Y coordinates do not need to be attributes in the browser but if Z values are to be used for individual locations there must be appropriate Z field in the flight path table.



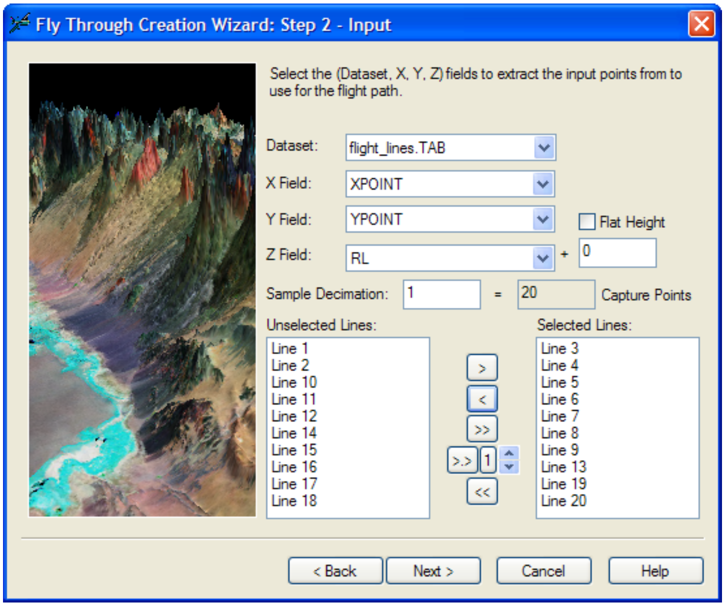
Select MapInfo table and Z values

Select the **Z Field** in the table that contains the height or elevation to use in the fly-through path. The Z field must be of numeric datatype. A constant value can be added to the existing Z values i.e. to convert local RLs to AHD. If there is no Z field in the table or all the **Eye** and **Look At** points are to viewed from the same elevation, check the **Flat Height** box and manually enter a Z value.

Discover 3D Dataset

Fly-through animations can be created from point or line data open in Discover 3D. Select the **From Discover 3D Dataset** option and click the **Next** button. In the **Fly-through Creation Wizard: Step 2** dialog use the **Dataset** pull-down list to locate the appropriate 3D dataset and assign the X and Y fields.

The Z values may be selected from a **Z Field** in the dataset or by checking the **Flat Height** box and entering in a constant Z value for all lines. A constant value can be added to the existing Z values i.e. to convert local RLs to AHD.



Select 3D dataset and assign XYZ fields

Manually from Current 3D View

This **Fly-Through Wizard** option is the simplest to use as the flight-path is compiled from a list of **Capture Points** (**Eye** and **Look At** points) which are specified by rotating and zooming to different views in the current Discover 3D map window. The **Capture Points** are then displayed in the capture order to simulate the flight-path. There are no additional setup parameters for this option.

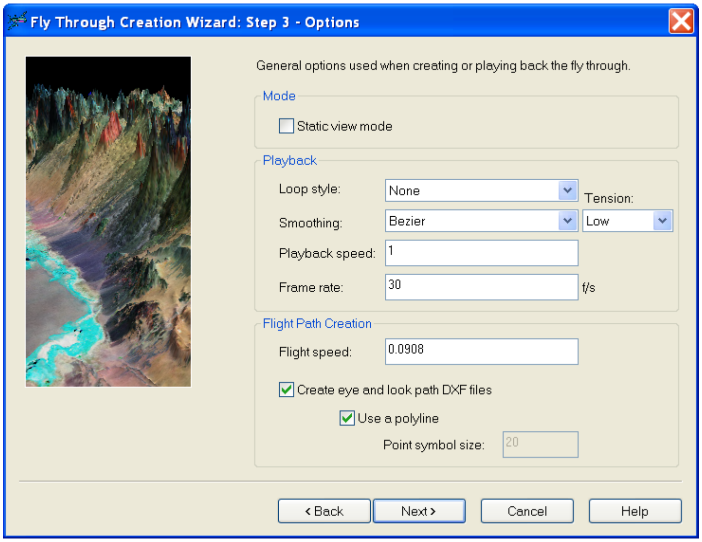
Edit Existing Fly-Through File

To load an existing fly-through file (FLY) into the **Fly-through Wizard** select the **Edit Existing Fly-Through File** option. In the **Fly-Through Creation Wizard: Step 2** dialog browse to an existing (FLY) script file.

Note If you use a script file with flight path capture points which are for a region that is not in the same area as the displayed in the 3D map window, when you play back the script, no image animation will be seen.

The (FLY) script file specifies the flight path and the general playback and recording options including capture point information.

Step 3 - General Playback Options



General playback and flight path options dialog

These options relate to the recording **Mode**, **Playback** controls and **Flight Path Creation**.

Mode

The default capture behVideoour of this tool is dynamic mode, allowing movement within 3D space.

Static view mode is designed solely for capturing the movement of the cursor clipping plane through your 3D dataset (e.g. clipping into a complex dataset over a period of time to highlight core area). In this mode, movement of either the eye or look-at positions in 3D is not possible. Enabling this option will disable many functions of the wizard.

Playback

The following **Playback** controls are available:

- The **Loop Style** options control the way a script is re-played. Use the pull-down list to select the desired option. A script can be set to:
- **Repeat** - the fly-through script is played and then repeated from the start continuously,
- **Repeat Reverse** - replay the script continuously but in reverse, or

- **Repeat Reverse Look -** replay the script continuously and in reverse but from the view looking back to the capture point.

- **Smoothing** – As each capture point is a location in three dimensional space, the flight path joining these points can be a smoothed line (Bezier) or straight line (None) between each point. If smoothing is selected, a Bezier smoothing algorithm with Tension is applied. The higher the degree of tension, the more closely the replayed flight path will be to the direct path between capture points.

- **Playback Speed** – Actual speed the playback is performed. For example, a playback speed of 1 is normal, a value of 0.5 would take twice as long as normal, a value of 2 would play twice as quickly as normal, etc. If a script takes 10 seconds playback then changing the Playback Speed to 0.5 would change the playback to 20 seconds. Similarly, if a value of 2 were assigned, the playback time would be 5 seconds.

- **Frame Rate** – Number of display captures in one second. The higher the Frame Rate, the smoother the animation. Note the smoother the animation, the greater the amount of storage required, especially if outputting to an Video movie.

Flight Path Creation

- **Flight Speed** – Time taken to move between two capture points. Speed is expressed in units per second (normally in metres per second). If you find that the time being calculated for movements is too large then increasing the flight speed shortens the movement times. All replay times get recalculated with new values when this property is altered.

  The speed of the replay is set in **Playback Speed**.

- **Create Eye and Look Path DXF Files** – The flight path and capture points of a script can be saved to 3D DXF files. Two DXF files get created, one for the **Eye** points and one for the **Look At** points. The DXF filenames are dependent on the name given to the fly-through file. The DXF files are created in the same directory as the fly-through file. For example, if the output fly-though file path is: `C:\DATA\FLY1.FLY`, then the DXF files created would be called:

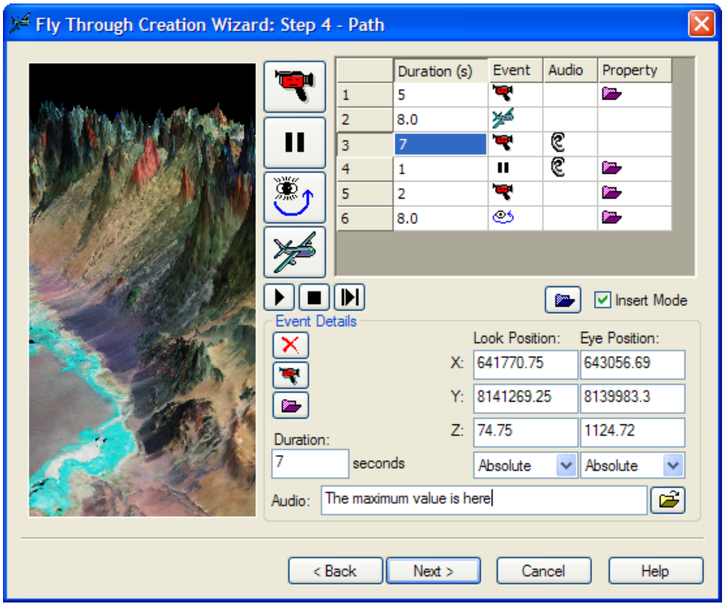
  `C:\DATA\FLY1_EYE.DXF` and `C:\DATA\FLY1_LOOK.DXF`

- **Point Symbol Size** – This parameter defines the size of a symbol to represent the **Eye** and **Look At** points in the created DXF files.

Step 4 - Capture Points and Event Details

The **Fly-Through Creation Wizard: Step 4 – Path** dialog is made up of three components:

- **Capture Point List** – The details and properties of the various capture points are listed with replay duration and icons to indicate events. If you select any of the capture point records by clicking on the corresponding row number, the Discover 3D map updates to that view.
- **Capture Point Events** – Controls to capture a point view, pause playback, rotate view, preview fly-through and stop a playback.
- **Event Details** – Control parameters for position and time specification, plus any audio accompaniment to a **3D Fly-Through Animation**.



Specifying and assigning events to the capture points of a fly-through script

Capture Point List

If a flight path is specified from a MapInfo TAB or MIF file, Discover 3D dataset or existing fly-through file (FLY), the capture point details are automatically added to the **Capture Point List**. New points can be added or existing points modified in this list. If the **Manually from 3D view** input option was selected in **Fly-through Creation Wizard: Step 1 – Input** dialog, new capture points must be created. See *Capturing Point Events* for more information.

The **Capture Points** are listed incrementally with **Duration**, **Event**, **Audio** and **Property** field for each view.

- **Duration** – The time in seconds that it takes to move from one Capture Point record to the next.

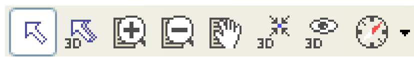
- **Event –** The type of capture point is indicated by one of four Events: **Capture Point, Pause, Rotate Eye** and **Rotate Look At.** See *Capture Point Events* for more information.

- **Audio –** If an audio entry is to be used at a **Capture Point**, an audio symbol is displayed. Enter some text in the Audio entry line or specify a pre-recorded sound file in WAV format. If you type an entry, such as “The value at this location is high”, when the animation script is replayed, the computer will repeat the text through voice recognition processing and the text is spoken from the computer. Alternatively, if a WAV file is selected the content of this file is played.

Capturing Point Events

The **Camera** button adds a new **Capture Point** to the Capture Point List. Each entry in the list represents a point along the flight path that is saved in the (FLY) script file. The script replays the capture points in sequential order (forward or reverse depending on the Loop Style).

To create a new capture point use the 3D nVideogation tools (see *Navigating in 3D*) in the 3D view to rotate and zoom to the next view point:



To insert a new capture point view between two existing views, check the **Insert Mode** box and click the **Camera** button to insert the current 3D view immediately after the highlighted record in the Capture Point List.

This button causes the replay script to **Pause** for a specified duration of time. By default, the time is 1 second, but using the **Duration** entry field, you can modify this time. Note that a record of this type in the list uses the last **Capture Point** and then pauses for the required time with no movement.

The **Rotate View** button causes the replay script to halt at the last used **Eye** position (determined by the previous capture point record) and then rotate the display by an amount determined by an **Azimuth** and **Inclination** (looking from the top down). The time to complete the rotation is specified by the **Duration**.

The **Rotate Around** button causes the replay to rotate the 3D view around the last **Look At** position (determined by the previous capture point record).The time taken to complete a rotation (as defined by the **Azimuth** and **Inclination** settings) is defined by the **Duration.**

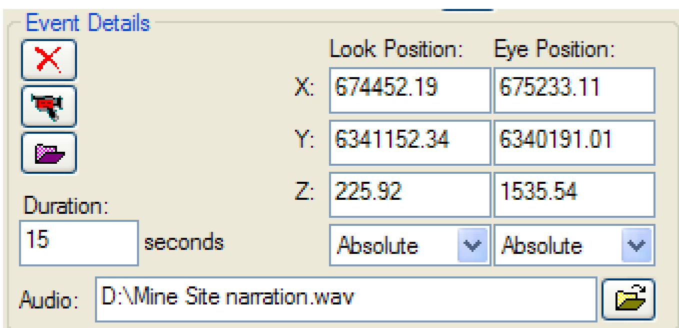
The **Properties** for the current event can be captured using this button. This will save any changes made to any data objects in the Workspace Tree since the previous event (e.g. visibility/transparency/legends/object thickness or size/etc). To refresh this, select the point event, make the necessary changes in the Workspace Tree, and press the Properties button.

This allows, for example, different datasets to turned on and off throughout your animation, e.g. a large scale satellite image could be displayed at the start (while you are zoomed out), then turned off, and a small scale image made visible when zoomed into the prospect scale.

Note Properties applied to a Capture Point Event are also applied to all successive Events, unless a following Properties entry alters this.

Event Details

The **Event Details** controls are concerned with the **Capture Point** attributes. Depending on the **Capture Point** record selected, the content of the **Event Details** may differ.



The three buttons included under **Event Details** are:

- **Delete** the selected **Capture Point** from the flight path.

- **Update** the **Eye** and **Look** Positions for the current event with the current 3D view.

- Displays the **Properties** of all 3D objects for the selected event, including display, transparency, lighting, line and fill colours, etc. To modify this, make the appropriate changes in the necessary items of the Workspace tree, then press the **Properties Capture** button (see *Capturing Point Events* above).

The attributes of each **Capture Point Event** are displayed here and can be manually edited:

- **Look Position** – The X, Y and Z location of the **Look At** point in Relative or Absolute coordinates.
- **Eye Position** – The X, Y and Z location of the **Eye** point in Relative or Absolute coordinates.
- **Duration** – The time in seconds that it takes to move from one Capture Point record to the next.
- **Audio** – Either a text entry or specification of a pre-recorded WAV file.

Each entry in the **Capture Point List** contains two pairs of position points (the **Eye** Position and the **Look At** point which determines the direction the fly-through path will follow). The points can be in either **Relative** or **Absolute** coordinates.

Absolute values are used to display a capture point view in real world coordinates. **Relative** values are used to display a capture point view *relative* to the previous capture view coordinates. For example, if a point capture view has a relative coordinate of (10, 10, 10) then the view will be moved 10 units in the X, 10 units in the Y and 10 units in Z direction from the previous capture position. Units are map distance units e.g. metres. If the previous capture position is in **Absolute** values the new view will move in world coordinates by 10 m in the X, Y and Z directions.

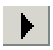
Note The flight path created using one of the various options should optimally be above or close to the 3D objects in the Discover 3D map window. For example, a flight path created over a topographic surface would not be appropriate to use in a fly-through of a non-topographic gridded surface representing assays for the same area as the Z values will be quite different.

To get around this problem:

1. Add an appropriate offset to the flight path to re-locate the flight path height to the new surface values.

2. Display the surface in the Discover 3D map window and in the **Enhanced Layer Control** highlight the **Surface** branch. Choose **Properties** from the right-mouse shortcut menu. In the **Vertical Scaling and Offset** controls select either the **Offset to grid maximum** or **Offset to grid average**. This operation relocates the grid surface automatically to a level that should be approximately zero.

Previewing your 3D Fly-Through



When more than one **Capture Point** record is present, you can **Replay** or preview the fly-through script. Click the **Play** button to activate the fly-through. The replay uses the **Capture Points** according to the **Loop Style** setting and the various speed and event controls (see *Step 3 - General Playback Options*).

To halt the replay, use the **Stop** button.

To display selected capture points in the fly-through replay use the button shown left. Only the capture points highlighted in the list will be included in the replay. This is useful for editing a long fly-through without hVideong to view the entire flight path every time.

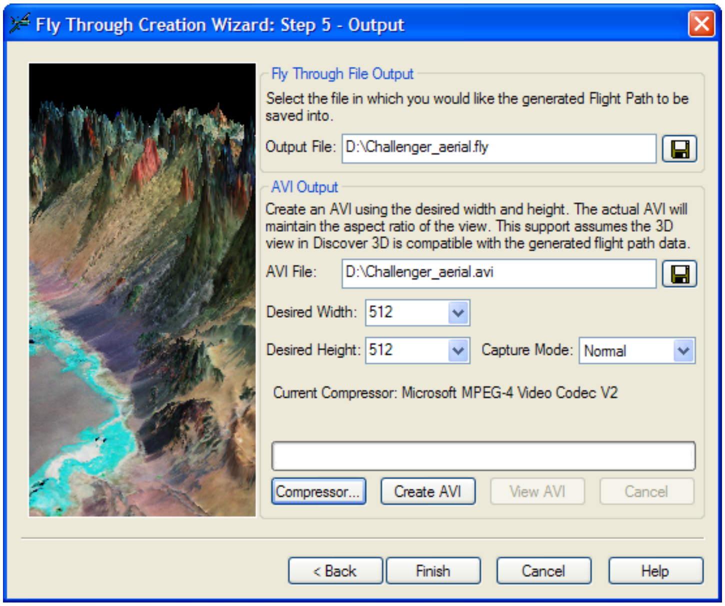
Once a Fly-Through file has been saved, it can be played back via a number of options within Discover 3D: see *Playing an Existing Fly-Through File* below.

Step 5 - Output Formats

Two output formats are available in the **Fly-through Creation Wizard: Step 5 – Output** dialog:

- Save the **Capture Points** and settings of the fly-through script by assigning a file name and location in an **Output File.** The settings are saved as a **FLY file** and can be restored for later use with the same project dataset. A **FLY file** can be activated via the **Display Fly-through** option on the Discover 3D map window right-mouse click shortcut menu.

- Save the fly-through as an **Video Output movie file**. Movie files allow replay of a fly-through animation completely separated from Discover 3D and are therefore useful for management and overview of a project. For more information, see *Codecs and Movie File Format*.



Retain the fly-through script and create a movie of the replay

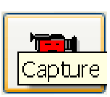
Static View Mode

The Fly Through Creation Wizard can also capture the movement of the cursor clipping plane over a period of time, for instance gradually clipping into a voxel model to incrementally expose slices of the interior cells values.

1. To activate this, enable the **Static view mode** option in the Step 3 dialog.

2. In 3D, align the view to the desired position.

3. Enable the cursor plane and position it at the desired start location. Setup the required clipping options (Envelope or Slice width, Near, plane Appearance etc) in the Cursor Plane Properties dialog. It is also recommended to set the **Move by user defined step** value to a useful value (for instance, if the cursor clip plane will be moving 500 m, try a value of 20-50 m).



4. In the Step 4 Path dialog, capture the initial point location (recommend unclipped, with or without the cursor plane enabled).

5. Then enable the cursor plane, and move to the final clipping position (i.e. in the depths of the data). Capture this point. Set an appropriate duration for the cursor clip plane to move from the start point to this location (e.g. 4 seconds)

6. Preview the result.

Notes

- Once the initial view point is set (step 4 above), the view orientation cannot be moved in any successive view capture
- The orientation and position of the cursor clipping plane can be modified throughout the process
- Datasets can be modified throughout the process (visibility, transparency, etc)
- If a static clipping move is required as part of a normal dynamic movie, it will need to be merged with the separate dynamic movie using a movie editing package such Camtasia Studio or similar. It is recommended to save the initial view position using the 3D View Manager, to ensure view continuity in the final product.

Codecs and Movie File Format

When creating a movie file, Discover 3D will display a list of any Codecs installed on your operating system, whether installed by the operating system or downloaded from a third-party vendor.

Important Discover 3D does not install any codecs.

Note Most default codecs included with Windows will return an error message, because they are Decodes only, and do not support encoding or creating your own movies. It is recommended to download and use a codec pack which includes the open source FFDShow standard encoders, such as the CCCP pack (*http://www.cccp-project.net/*).

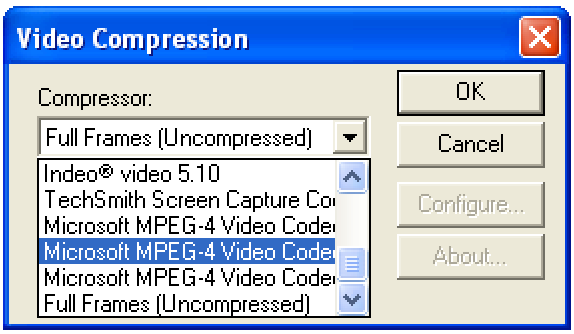
To create an video movie:

1. Specify an Video path and file name.

2. Specify the desired **width** and **height** for the movie. By default the pixel size is 512 x 512; increasing this will increase the quality of the output movie, but also increase both the output file size and processing time required to create the movie.

3. Leave the **Capture Mode** at normal. If you experience problems such as corrupted output or it is taking along time, you can adjust this to **Advanced** and lastly, **Expert** levels to troubleshoot. This reduce the amount of memory required and the complexity of the recording process. Note that you will need to leave the Discover3D window front most ontop of other applications when using these modes.

4. Select the **Compressor** button. Choose the **Codec** to use in the **Video Compression** dialog from the **Compressor** pull-down list. (see *Codecs and Movie File Format f*or more information*).* A slider bar controlling the **Compression Quality** is also displayed. While an Video is being created, it can be halted by clicking the **Cancel** button.



Selection of a codec for Video movie creation

It is recommended that an ffdshow codec is selected (if available) as this codec generally offers the best video compression.

5. Adjust the quality, file size, and speed of the output file under the Codec configuration by clicking **Configure**. The most important parameter is the bit-rate, which determines how much information is stored for each second of playback.

6. Select the **Create Video** button. A progress bar should commence being written as the frames of the movie are created. The rotation and display of the Discover 3D map window will also update as the movie scenes are captured. When the movie is completed, you can see the movie by clicking the **View Video** button.

Note If you receive an ERROR message after selecting a codec and clicking the **OK** and **Create Video** buttons it is likely that the Codec you have selected is for decoding rather than encoding (i.e. for the playback of an Video movie rather than the creation/writing of an Video file). Select another codec and try again.

7. When the Video file is created, you can display this movie file using a wide range of free movie players. Such players include the GOM Player (*http://www.gomlab.com/eng/GMP_download.html*):

Playing an Existing Fly-Through File

There are three ways of replaying a fly-through (*.FLY) file created using the Fly-Through Creation Wizard:

- Open the file using the Fly-Through Creation Wizard, by selecting the **Edit Existing Fly-Through File** option. This allows the file to be edited as well as played.
- Using the **File>Open** menu option, and setting the Files of Type to “All supported Fly-Through Files”. Browse for the required .fly file and click **Open**.
- Right-clicking on the 3D Map branch in the Workspace Tree, and selecting **Display Fly-through**.

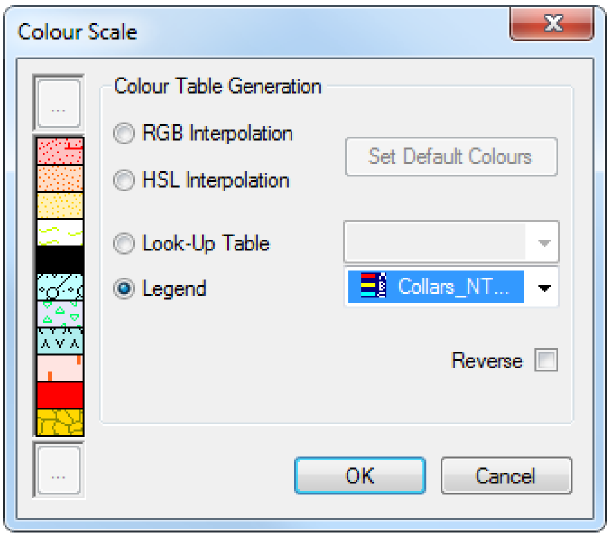
The second and third options will display a **Flight Control** bar upon opening the Fly-Through file, allowing playback control.

22 Working with Colour Tables and Legends

When displaying data such as an elevation, geochemical or geophysical gridded surface, it is useful to colour the grid using a Colour Look-Up Table (LUT) in order to show the variation in data values. Instead of creating colours for individual data values a colour look-up table is used which is made up of a number of colour shades to form a continuous colour spectrum. When a colour look-up table is applied to the gridded data the colours are applied evenly between the minimum and maximum values.

For data which is to be displayed visually according to individual data categories or data ranges, a Legend table can be created. For example, Downhole drillhole data can be displayed showing patterned lithological units or discrete coloured assay ranges. Geochemical data can be displayed according to mesh size or sample type. Legends can be created manually or automatically from a selected dataset field.

To apply LUT and Legends to datasets use the **Colour Scale** or **Colour Lookup Selection** dialogs (below). Access to these dialogs is via the relevant **Properties** dialog. In addition these dialogs provide standard HSL and RGB Interpolation options with end member colour control.



The Colour Scale dialog

In this section:

- *Using the Colour Look-Up Table Editor*
- *Using the Legend Editor*
- *Advanced Colour Mapping*

Using the Colour Look-Up Table Editor

The Colour **Look-Up Table** (**LUT**) **Editor** is accessed from the **Tools** menu in the 3D interface. It can also be accessed from the **Colour Lookup Selection** dialogs (via the **Point** and **Line Properties** dialogs).

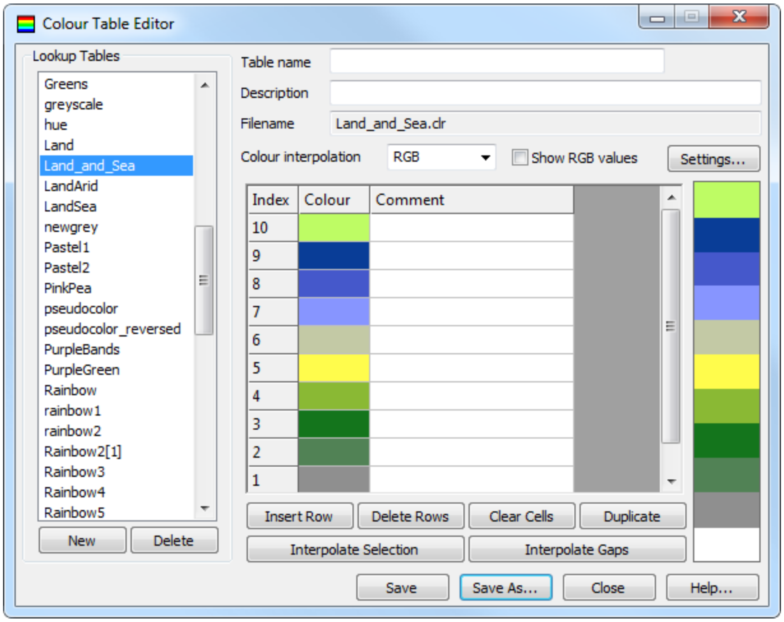
LUT files are stored by default in the `C:\Program Files\Encom\Common\LUT` (Windows XP) or `C:\ProgramData\Encom\ Common\LUT` (Windows Vista or Windows 7) directory. The format of LUT files varies depending on the saved format type. Supported types include:

- ER Mapper (files with extension .LUT)
- MapInfo Professional (extension .CLR)
- Geosoft Oasis (extension .TBL)

The **LUT Editor** dialog is divided into functional areas. On the left is a scrollable list with the available **Lookup Tables**. Beneath this list are **New** and **Delete** buttons for creating a new LUT and removing lookup tables from this list. In the centre of the dialog are the various colour settings for the number of rows specified in the selected LUT with descriptive comments if available.

The lookup **Table name**, **Description** and the corresponding **Filename** are displayed at the top of the dialog. Underneath the **Colour interpolation** can be selected from RGB or HSL with an option to show the individual RGB values for each row. The **Settings** button shows the file path of the stored LUT and can be changed to another location such as a server directory if required. A preview of the selected LUT is displayed on the right of the dialog showing the entire LUT colour spectrum.

At the base of the dialog is a series of buttons used to control the distribution of colours in a new or existing lookup table. Once a LUT has been specified it can be stored using the **Save** button. To save an existing LUT under a different name, location or file format use the **Save As** button. The above LUT formats are provided as options when the **Save As** dialog is displayed. The LUT Editor dialog appears as below:

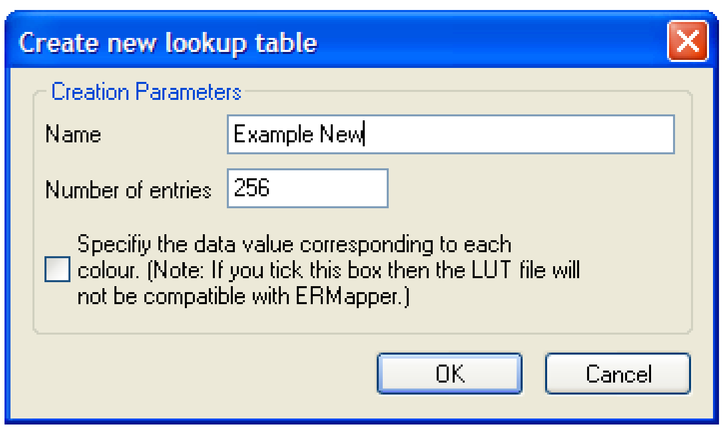


The dialog of the LUT Editor

A colour lookup table can contain any number of rows. Usually, this number is 16, 32, 64, 128, 256 etc. to enable an even colour spread. Each row can have a colour individually assigned or a number of rows can be selected and the colours graduated over the entire row range.

To create a new colour look-up table:

1. Click the **New** button.



Dialog for creating a new LUT

2. The **Create new lookup table** dialog requires a **Name** plus the **Number of entries** of rows (colours) to be assigned.

3. Check the **Specify the data value corresponding to each colour** box to precisely match data values with a colour from the table. For example, if the data has a range from 51,000 to 51,500, you could evenly space 500 definable colours for each of the 500 data values. Note that if this utility is used, you cannot use the created LUT within ER Mapper.

When the new table is created it is added to the available LUT list but all the colours are blank. Double click on any of the **Colour** cells of a row and a list of available, standard colours is displayed. If you select the **Custom** option, any of the Windows-supported RGB or HSL colour specifications can be created.

By selecting two end-member cells (indicated by highlighting the cells in two non-adjacent rows); you can use the **Interp Selection** button to fill any intermediate blank cells. In a similar way, you can make blank any highlighted cells by using the **Clear Cells** button or interpolate colours between specified colours using the **Interp Gaps** button. Selected cells can be all set to match the FIRST cell using the **Duplicate** button. This always operates from the top-most selected cell down, irrespective of the order in which the cells were selected. Extra rows can be added or deleted using the **Insert** or **Delete Rows** buttons. To store the current LUT settings click the **Save** button.

Using the Legend Editor

The **Legend Editor** is accessed from the **Tools** menu in the Discover 3D interface.

The **Legend Editor** can be used for the following:

- Create colour legends for numeric or text fields.
- Create thickness legends for numeric fields (e.g. for custom thickness modulation of drillholes by an assay field).

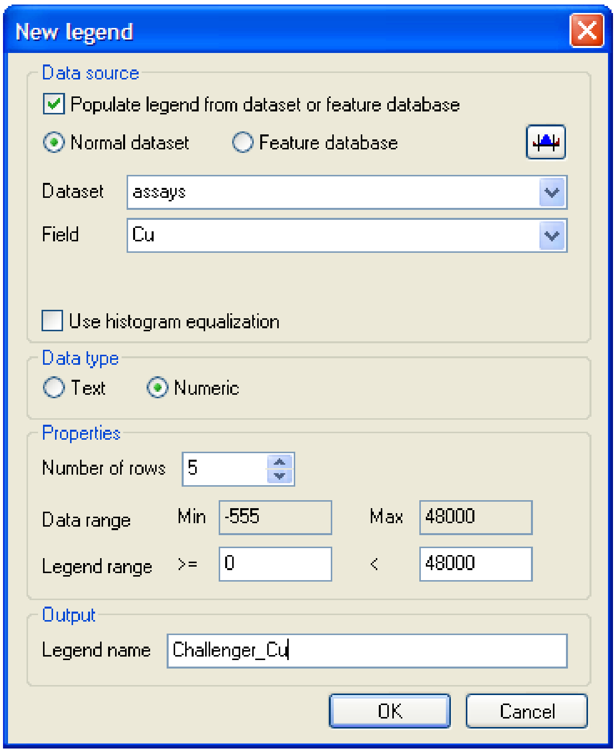
- Modify and maintain existing legends.

To create a new legend:

1. Ensure an appropriate dataset is open in Discover 3D (e.g. a drillhole project or 3D points/lines dataset).

2. Select the **Tools>Legend Editor** menu option.

3. Press the **New** button at the bottom left of the dialog. This will open the New Legend dialog.



4. Enable the **Populate legend from dataset…** option, and select the **Normal dataset** option.

5. Select the Dataset and target Field from the pull-down lists.

6. The field Data Type can be either a numeric or text field- the dialog will automatically recognise this.

7. If numeric, set the Number of rows to be the required number of ranges. Do not alter this value if a text field - the Legend Editor will have determined the number of unique attributes on the selected field.

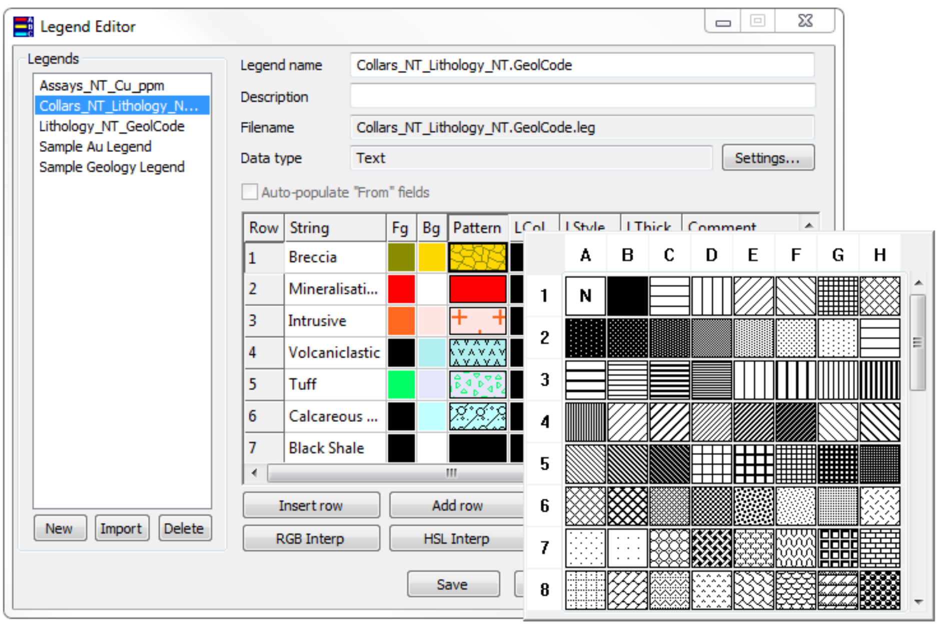
8. Specify an output **Legend Name** (or leave as the default) and press OK. The legend will be displayed in the Legend Editor.

To edit a legend:

1. In the Legend Editor dialog, select the required legend from the list on the left. It's attributes will be displayed on the right.

2. Numeric legends will display a From (>=) and To (<) field. Click in the appropriate cell and enter a value to alter the ranges as required. Enabling the **Auto-populate From fields** option at the top of the dialog will fill the next range's From value from the previous ranges To value after it has been entered. Extra ranges can be added/deleted using the Row controls at the bottom right of the dialog (**Add/Insert/Delete Rows**).

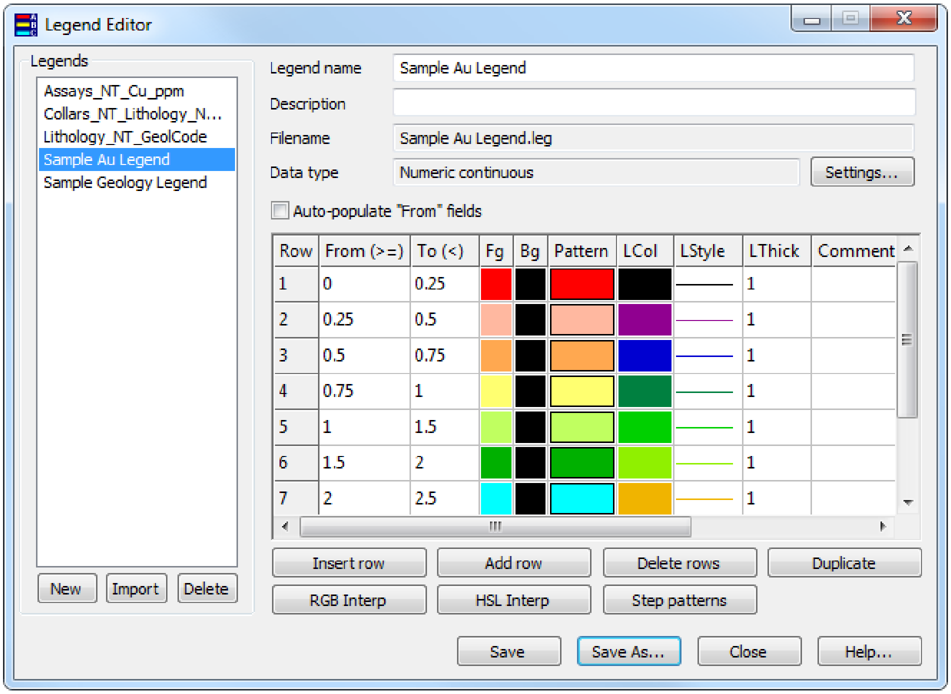
3. Text legends will have a String field. If the legend has been auto-populated in the New legend dialog, it is not recommended to alter the string entries, as altered entires may not match dataset attributes.



4. If the legend is to be used for **colour modulation** (e.g. drillholes), the primary foreground colour can be set for each row using the colour palette under the **Fg** field. If patterns are to be applied (e.g. geology), these can be set using the palette available under the **Pattern** field, in tandem with a background (**Bg**) colour (if applicable). Colour schemes can also be automatically applied using the **RGB/HSL Interp**olation buttons.

5. If a numeric legend is to be used for thickness modulation (e.g. drillholes), alter the LThick column entries to the required thicknesses.

6. Press the Save button when complete, and close the dialog.



Legend Editor Controls

The **Legend Editor** dialog lists the available legends down the left side and the details of the currently highlighted Legend including colours, patterns and associated string entries on the right side.

At the top of the **Legend Editor** dialog the selected **Legend name** is displayed with a **Description**, the **Filename** and the legend **Data Type** which may be text or numeric. Next to these entries are buttons to save and/or **Close** the dialog. By default, the **Save** button overwrites and saves the legend file into the folder as set by the **Tools>Options** default path. The **Save As** button allows you to save a legend in a new folder and with a different filename if required.

Column headings indicate the display attribute of a row. These are:

- **Row** – incremental row number.

- **String** – a text string to describe the item of the legend row. Double-click to edit.

- **From/To** – the range end values for numeric-based legends. Double-click to edit the value.

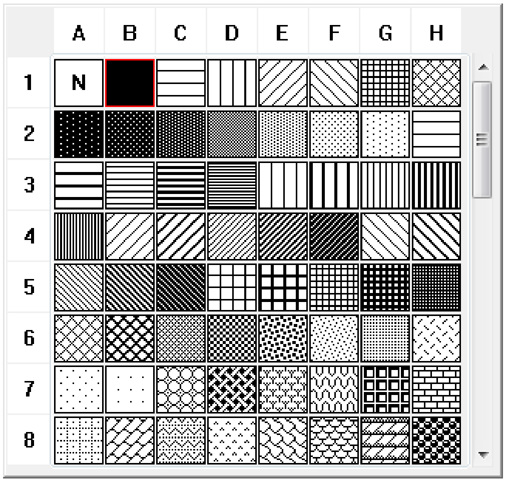
- **Fg (Foreground colour) –** placing the cursor over the Fg cell of this row and clicking the left mouse button displays the colour pick options.

  Standard colours are available but if you wish to create or chose custom colours, click the **Custom** item. The legend item can also be set to **Transparent** which renders items with this legend entry see-through.

- **Bg** (**Background colour)** – setting a background colour highlights patterns when assigned. Selection of background colours is identical to selection of foreground colours (see above).

- **Pattern** – specify the pattern to be displayed for a legend row. Place the cursor in the Pattern row location and click the left mouse button. A scrollable list of available patterns is displayed and allows individual pattern selection.

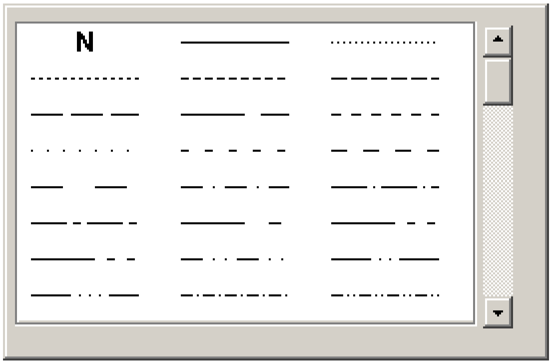


The pattern selected is drawn with the colour of the nominated **Background Colour** (see above).

Pattern styles are stored in bitmap files located as specified in **Tools>Options>File Locations** tab.

- **LCol (Line Colour)** – used to form a drawn line boundary around the specified legend row entry. Selection of Line Colours is identical to selection of foreground colours (see above).

- **LStyle** – Double-click to select an appropriate boundary Line Style.



- **LThick** – Input value defines the boundary Line Thickness, used in *Thickness Modulation* of drillholes.

- **Comment** – a descriptive text entry for each legend row can be stored with the legend if required.

Operational buttons along the base of the dialog enable you to:

- Create **New** legend files.
- **Import** a .TAB file used by Encom Discover 3D to populate a new legend.
- **Delete** an existing legend file.

Various operations can be performed over a selected range of legend rows within a column. Select the first cell then, holding the SHIFT or CTRL keys, click on the last cell for the desired range. Selected cells are highlighted in the editor. The following operations relate to selected cells:

- **RGB Interpolation** automatically interpolates Red:Green:Blue data values across rows of selected cells. To use this feature, select a vertical column of cells.
- **HSL Interpolation** automatically interpolates Hue:Saturation:Lightness colour data across selected legend rows. Its operation is identical to the RGB Interpolation option.
- **Step Patterns** assigns patterns from the pattern list incrementally from the first selected pattern cell to the last selected cell.
- **Duplicate Patterns** copies the pattern of the first selected cell to the last selected cell.
- **Insert row** places an empty row before a selected row or cell.
- **Delete rows** removes single or multiply selected rows. Any cell selection that is highlighted is removed with this operation. A confirmation message is displayed before the rows are deleted.

Warning When rows are deleted, and the modified legend has been saved, the deleted rows cannot be restored.

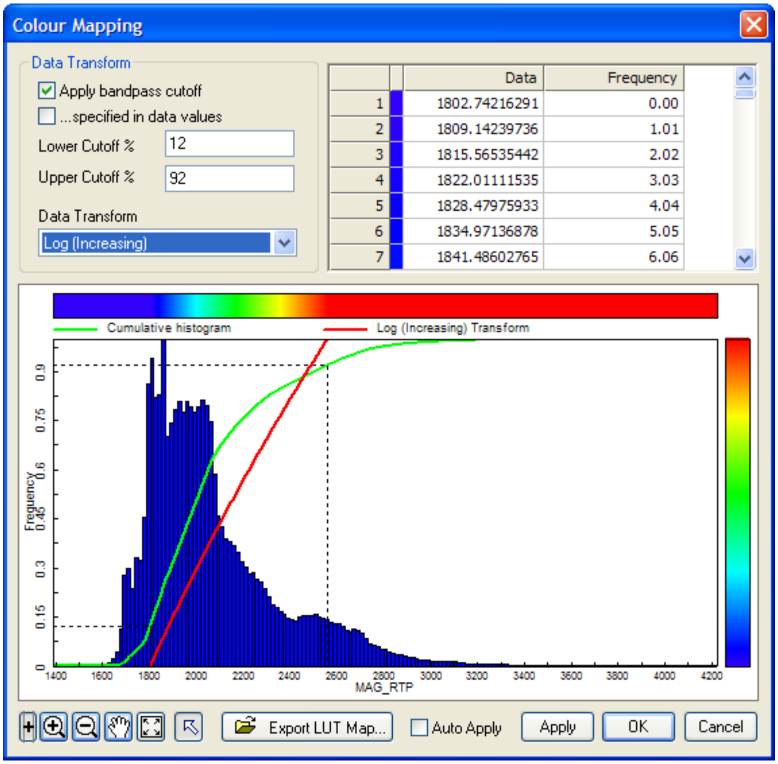
Legend Location

Legend information is stored in a file with a .LEG extension. Legend files are stored in the `C:\Program Files\Encom\Common\LUT (Windows XP/2000)` or `C:\ProgramData\Encom\ Common\LUT (Windows Vista or Windows 7)` directory or the directory as specified in the **Tools>Options>File Locations** tab.

In addition to the legend files, associated patterns are stored in the `C:\Program Files\Encom\Common\Patterns (Windows XP/2000)` or `C:\ProgramData\Encom\ Common\Patterns(Windows Vista or Windows 7)` directory.

Advanced Colour Mapping

In many properties dialogs, wherever a Colour or other property such as Thickness or Transparency is selected to be modulated, an Advanced data mapping dialog is provided.



The Colour Mapping dialog configured for an increasing logarithmic data transform, with bandpass cut-off limits set in data values

This dialog presents a preview screen of the data distribution, with the Cumulative histogram displayed as a green line, and the applied data transform a red line (and are also replicated in the main Surface Properties dialog preview screen), as well as the bandpass cut-off limits (dashed lines). The colour distribution for the specified data transform is also displayed around the preview edges as colour bars. A spreadsheet displays transform inflexion points as a series of coordinates, where X is the data value and Y is frequency. The preview screen can be controlled using the buttons at the bottom left (**Zoom**, **Pan**, **Fit view to data**).

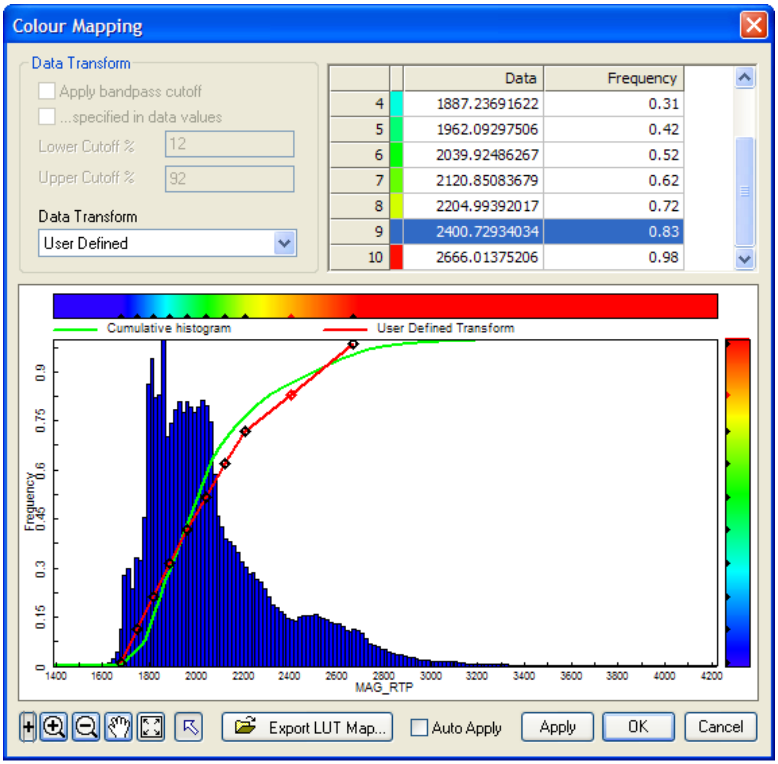
The following transform options are available in this dialog from the pull-down list:

- **Linear (increasing or decreasing)** - the transform linearly distributes colour between the minimum and the maximum data values.
- **Log (increasing or decreasing)** - colour is distributed logarithmically between the minimum and the maximum data values.
- **Histogram** - this transform produces an image with an equal amount of colour area across an image.
- **User-defined** – having set one of the above transforms (with or without a bandpass cut-off), choosing this option allows you to customise the transform curve. A series of inflexion points are displayed as black icons along the curve; additional control points can be added by placing the cursor over the transform and clicking when it changes to the Add Point cursor. Control points can be edited via the following methods:
  - Clicking on a control point in the graph and moving it moves all control points in an elastic motion. This move is reversible; moving a control point and then returning it to its original position will return all other control points to their original positions. The editing mode compresses control points in front of the selected point and expands them behind the selected point. This mode allows quick modification of the shape of the transform curve whilst retaining the flavour of the original curve.
  - Clicking on a control point with the CTRL key held down and moving it moves the selected control point only. Points cannot be moved horizontally beyond the two adjacent control points. Directly editing a control point in the spreadsheet applies the same rules. This mode allows control over the exact coordinates of individual control points.
  - Clicking on a control point with the SHIFT key held down and moving it. This editing mode compresses control points in front of the selected point but does not modify control points behind the selected point. The same effect can be achieved by moving a control point in one of the colour bars.

    Moving a control point horizontally is equivalent to moving a selected colour to a new data value. Moving a control point vertically is equivalent to moving a selected data value to a new colour.

- Clicking between two control points in a colour bar and dragging moves the control points to the left and right of the cursor. The distance between these two points will not be modified but control points in front of the leading point will be compressed. Control points behind the trailing point will not be modified.

  Moving a segment horizontally is equivalent to moving the selected colour range to a new data range. Moving a segment vertically is equivalent to moving the selected data range to a new colour range.



Creation of a User-defined transform

For each of these options (except User-defined), a **Bandpass cut-off** can be applied either as percentage or data values. All data values outside the specified range will be assigned the minimum or maximum colour value and the colour stretch will then be restricted to the data within the bandpass.

23 Customising Discover 3D

In this section:

- *Customising 2D Interface Settings*
- *Customising 3D Interface Settings*
- *Customizing Toolbars*
- *Assigning Custom Keyboard Shortcuts*

Customising 2D Interface Settings

The **Discover 3D>Options** menu item (accessed from the 2D MapInfo menu system) opens the **3D Options** dialog, which allows control of various system and display settings.

- *System Tab*
- *Display Tab*

System Tab

Options available from the **Systems** tab include:

File Locations

- **Temporary Directory** - During a work session various files are created for display and management of the 3D displays. By default, this is the same directory as the Discover Temp directory.
- **Remove Temporary Files on Exit** - When the Discover 3D window is closed, any files present in the nominated Temporary Directory are removed if this option is enabled.

XYZ Compass

A small principal compass indicator is displayed in Discover 3D to assist in orientation of the three dimensional view. The XYZ axis directions can be displayed all the time, only when rotating or never. In addition, the axis labels can be renamed (e.g. Up, East, and North).

3D Surface Display Options

A number of methods are provided to vary the default compression factor applied to grid surfaces. For speed considerations, surface grids are decimated in their display i.e. not all rows and columns are displayed depending on the compression selected. The higher the degree of Compression, the faster the redraw speed with an associated decline in the level of surface detail displayed.

- **Loading Max Columns** - Maximum number of columns in a grid which will be used to load the grid into system memory. If the grid exceeds the column number Discover 3D will use large grid handlers to load the grid into Discover 3D. It is recommended that only small grids e.g. <50 MB are loaded into system memory and large grids e.g. >50 MB employ the large grid handlers.

- **Loading Max Rows** - Maximum number of rows in a grid which will be used to load the grid into system memory. If the grid exceeds the row number Discover 3D will use large grid handlers to load the grid into Discover 3D. It is recommended that only small grids e.g. <50 MB are loaded into system memory and large grids e.g. >50 MB employ the large grid handlers.

- **Rendering Max Columns** - Maximum number of columns in a grid before the grid will be decimated for display in Discover 3D Display Window. Decimation is employed to reduce the system overhead and speed up the refresh rate of the 3D Display Window. The decimation will only be used when surface compression mode is set to Constant and the level of decimation is set to System size, refer to Gridded Surfaces in 3D chapter for more details on compression.

- **Rendering Max Rows** - Maximum number of rows in a grid before the grid will be decimated for display in Discover 3D Display Window. Decimation is employed to reduce the system overhead and speed up the refresh rate of the 3D Display Window. The decimation will only be used when surface compression mode is set to Constant and the level of decimation is set to System size, refer to Gridded Surfaces in 3D chapter for more details on compression.

- **Normal texture size** - Pixel resolution of rendered surface in Discover 3D. When a surface file is imported into Discover 3D the display is actually composed of a bitmap image, the resolution of the bitmap image is determined by the normal texture size setting, a higher resolution setting the greater the detail, however, the slower the refresh rate of the 3D Display Window.

- **Maximum texture size** - Pixel threshold set for a georeferenced image imported into Discover 3D, before the image will be tiled. If a georeferenced image does not reach the maximum texture size a single image drape is displayed. If a georeferenced image imported into Discover 3D exceeds the maximum texture size a series of seamless image tiles will be created. Tiling an image is generally conducted due to limitations in graphics cards and allows the display of very high resolution images in the 3D Display Window. Generally, set the maximum texture size to 1024 as most graphics card support this texture size.

Display Tab

The **Display** tab allows control of the following options:

3D Window Display Options

Raster images are used to show draped pictures over surfaces or at defined elevations etc. The amount of detail used in these images impacts on the drawing speed of Discover 3D. If the detail of an image is increased the redraw speed is decreased but the information provided in the images is increased. If you select a value of 2x, Discover 3D creates an off-screen bitmap that is 2x the size (in width and height) and 4x the size in pixels of the selected Map window.

The **3D Point Symbol Size** option sets a default size for 3D Point symbols in Discover 3D displays.

Customising 3D Interface Settings

The **Tools** menu in the Discover 3D window provides access to the 3D Interface **Options** dialog, which enables the control of numerous 3D window settings.

Options that can be specified include:

- Paths for data and templates.
- Default page size (in the absence of an installed printer).
- Preferred view options.
- Specific file locations.
- Default printer page size and orientation.
- Auto-reloading of modified files in use.

For more information, see:

- *File Tab*
- *Size Tab*
- *View Tab*
- *File Locations Tab*
- *Page Setup Tab*
- *Fullscreen Tab*

File Tab

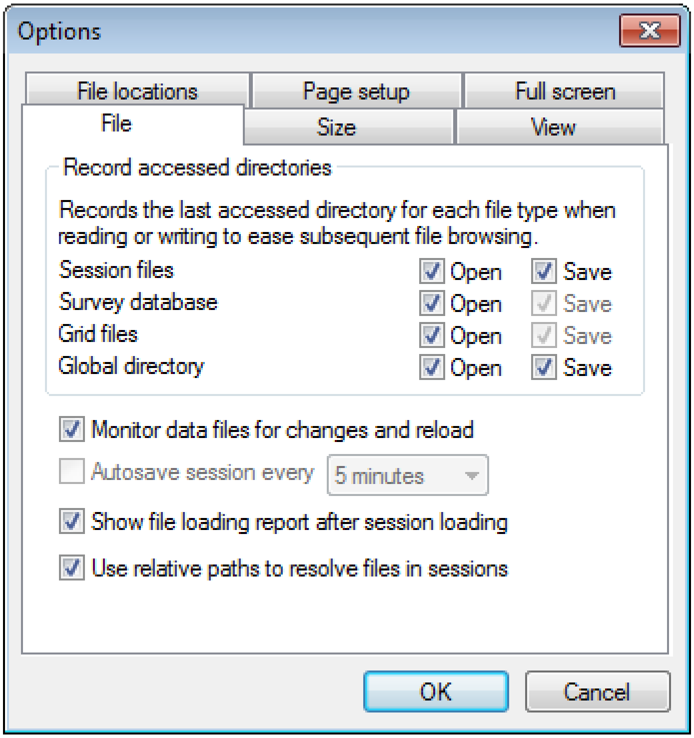
Record accessed directories enables or disables the last used file paths when using open or save dialogs. These include the general Global directory and session open/save, as well as survey database (drillholes) and grid files.

Monitor data files for changes and reload adds support for automatic update/ refresh when data files are edited.

The **Autosave session** setting defines how frequently the recoverable autosaved session file (**File>Restore Autosave Session**) is updated.

Show file loading report after session loading enables a list of files automatically found or skipped after opening a session.

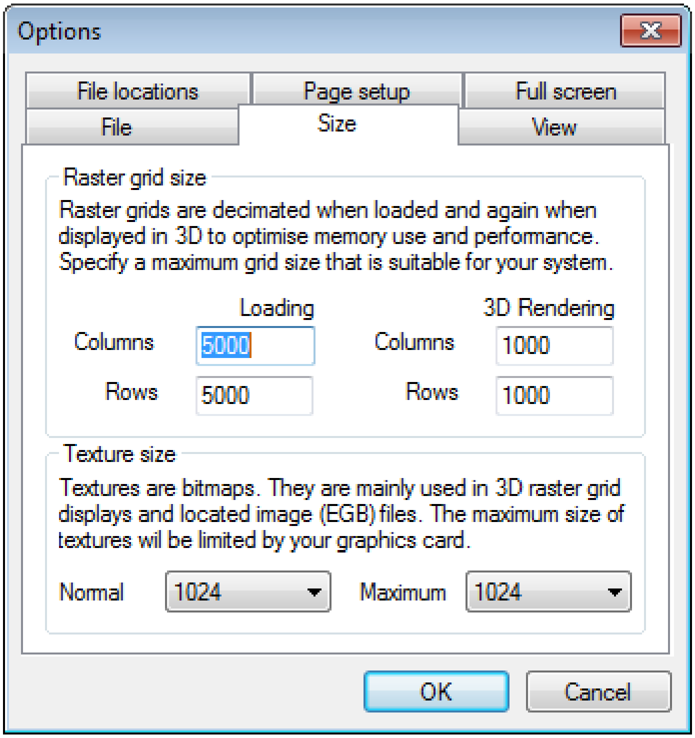
Use relative paths to resolve files in sessions improves the automatic resolving of moved or missing files when opening session files.



Size Tab

Raster grid size limits the largest surface grid that will be loaded by default in a surface or image branch. In Discover3D, as there is only a 3D map the Loading option is not applicable. Only the 3D Rendering limit applies.

Texture size defines the resolution or raster images such as JPG, bitmaps or ECW files which are usually loaded in an Image branch. Generally only up to 4096x4096 is supported by graphics cards. Image files larger than 4096x4096 pixel resolution will be decimated to this size.



View Tab

Support for the 3DConnexion SpaceNavigator™ device is enabled in this dialog: see *Using the 3DConnexion SpaceNavigator™* for use of this device in navigating Discover 3D.

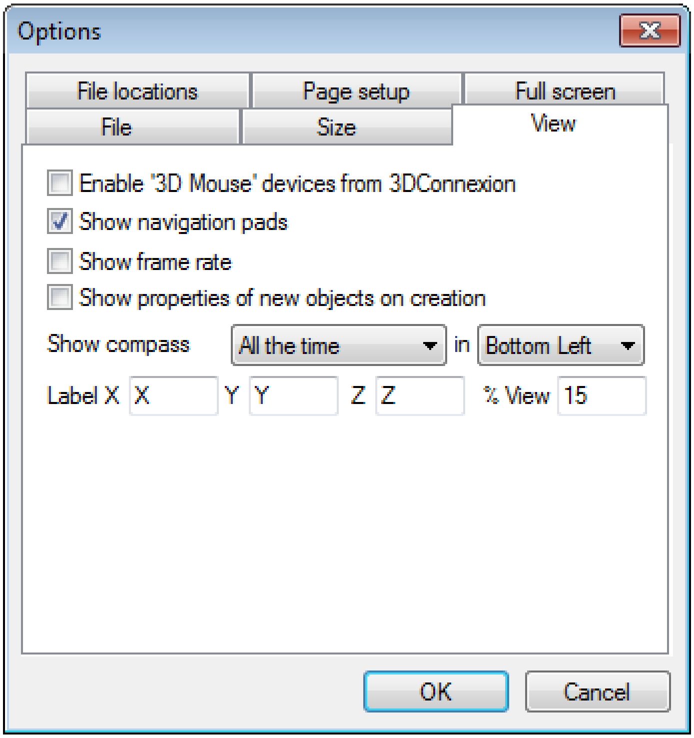
Show navigation pads enables the navigation buttons that are embedded in the 3D window when 3D navigation cursor mode is selected.

Show frame rate displays frames per second at the top left of the 3D window.

Show properties of new objects on creation automatically opens the properties of a branch when a new branch is added/opened in the 3D Map.

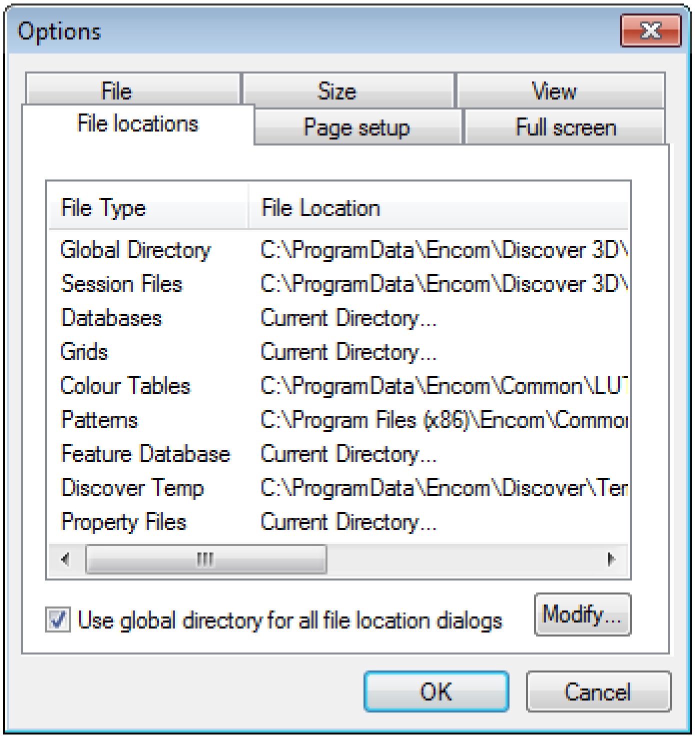
The position of the XYZ indicator is also controlled on this tab.

Note These XYZ compass options are replicated in the **Discover 3D>Options** dialog accessed in Discover under the **System** tab. Changes applied in either dialog are automatically updated in both dialogs.



File Locations Tab

Preferred datasets, sessions, surface grids, colour tables and patterns can be saved in specified default locations. These locations usually are defaulted but these values can be overridden by selecting the item to be specified from the displayed list and then using the **Modify** button.



Specify the default locations of various files

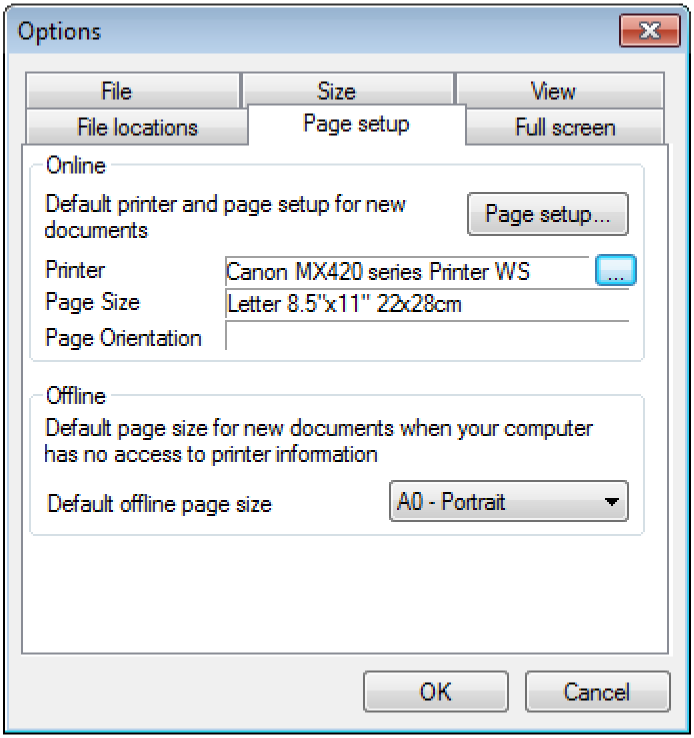
The option to **Use Global Directory** is enabled by default. This setting is used for accessing all file types from a single source folder. The folder is defined by selecting the Global Directory item and clicking the **Modify** button to set the Global Directory location.

Note The Discover Temp directory cannot be modified from this dialog. The Temp directory can only be changed in the **Discover>Configuration>Settings** dialog within Discover.

Page Setup Tab

The **Page Setup** tab of the **Options** dialog provides support for default printers when Discover 3D is connected to or remote from printers or plotters. This default sets the automatically displayed page when a **Page Layout** display is presented. The layout may alter depending on whether Discover 3D can find the specified drivers for the defined printer/plotter. If these Windows drivers cannot be found on the machine, the Windows default printer settings are used.

To modify the printer default setting for Online printers, select the **Page Setup** button.



Page Setup tab of the Options dialog

Fullscreen Tab

This dialog allows you to specify **Show Main Menu**, which will include the main menu toolbar when in fullscreen view.

Customizing Toolbars

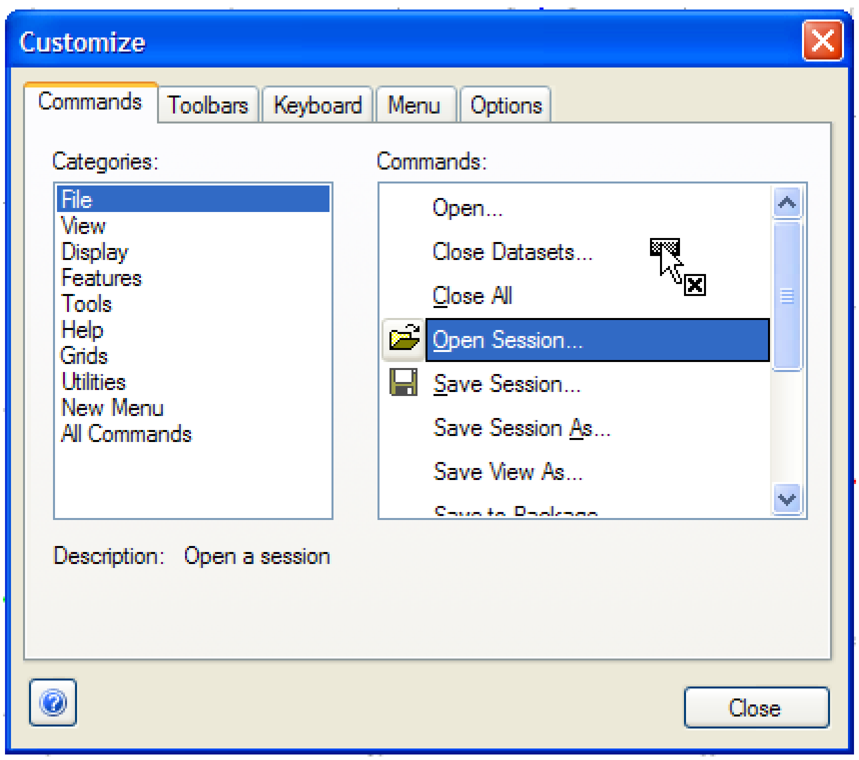
- *Adding and Removing Tools*
- *Restoring Toolbar Settings*
- *Toolbar Button Size*
- *Show and Hide Tool Labels*
- *Creating New Toolbars*
- *Other Customizations*

Adding and Removing Tools

Existing tools on the toolbars can be quickly displayed or hidden by clicking on the dropdown arrow on the selected toolbar. Select **Add or Remove Buttons** and the appropriate toolbar to customise. And then select or deselect the tool to display or hide.

The tools available on ioolbars can also be customized to suit the user requirements. To open the **Customize** dialog either right mouse click on the **Main** menu bar and select the **Customize** option or from a toolbar navigate to dropdown arrow on the selected toolbar and select **Add or Remove Buttons** followed by the **Customize** option.

From the **Customize** dialog navigate to the **Commands** tab, select a tool, from the **Commands** list drop and drag a select command into an existing toolbar. Next time Discover 3D is started the customized toolbar will appear.



Drop and drag tool into toolbar.

Restoring Toolbar Settings

To restore a toolbar to factory settings, click on the dropdown arrow on the selected toolbar. Select **Add or Remove Buttons** and the appropriate toolbar to customise. And then select or **Reset Toolbar**.

You can also reset one or all toolbars by selecting the **Customize** option on any toolbar dropdown menu, and selecting the **Toolbars** tab.

Toolbar Button Size

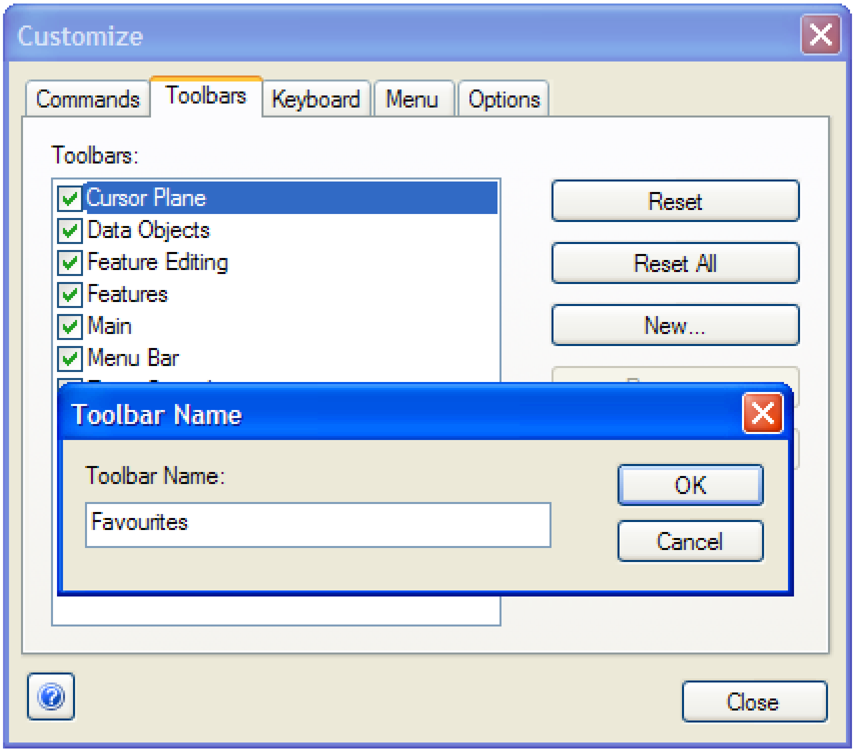
To display toolbar icons in a larger style, navigate to the **Customize** dialog by right mouse clicking on the **Main** menu bar. On the **Options** tab check the **Large Icons** option.

Show and Hide Tool Labels

To hide and show tool labels, navigate to the **Customize** dialog by right mouse clicking on the **Main** menu bar. Check the **Show text labels** option on the **Toolbars** tab of the **Customize** dialog, the selected toolbar will display the tool label as well as the icon.

Creating New Toolbars

As well as customising existing toolbars up to ten user-defined toolbars can be created. To create a toolbar navigate to the **Toolbars** tab on the **Customize** dialog and press the **New** button, type in an appropriate name for the toolbar. Using the procedure for customizing individual toolbars navigate to the **Commands** tab, select a tool, from the **Commands** list drop and drag a select command into the new toolbar.



Create new toolbar with custom tools.

Other Customizations

Located on the **Menu** tab of the **Customize** dialog are options for menu list effects and shadows.

Located on the **Options** tab of the **Customize** dialog are options for displaying screen tips, shortcut keys and enlarging the toolbar icons.

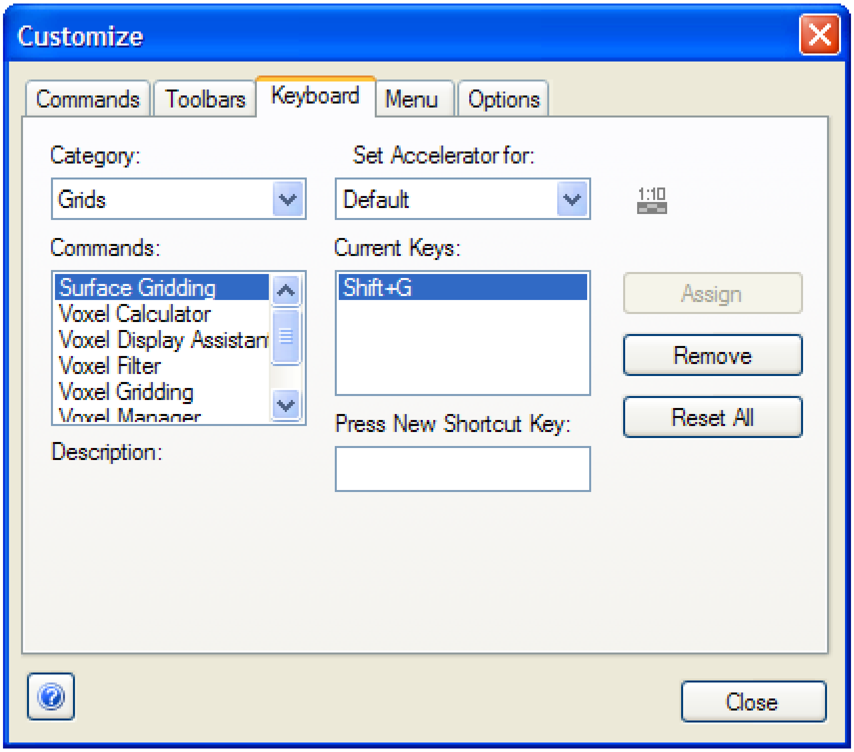
Assigning Custom Keyboard Shortcuts

The **Customize** dialog also allows you to link existing Discover 3D menu commands with a keyboard shortcut. Some commands already contain built-in shortcuts, such as Print (CTRL+P). Additional shortcuts can also be set up for these menu commands using the **Keyboard** tab in the **Customize** dialog.

To add a keyboard shortcut to Discover 3D open the **Customize** dialog and select the **Keyboard** tab. Select a **Category** from the pull-down list and select a **Command** from the list displayed. Click the mouse cursor in the **Press New Shortcut Key** box and select the keyboard shortcut (use **CTRL** and **SHIFT** keys if desired). Click the **Assign** button to complete the shortcut creation.

To remove a keyboard shortcut, select a command click the **Remove** button. To remove all custom shortcuts click **Reset All**.

To display the custom shortcut in the menu system navigate to the **Customize** dialog and select the **Options** tab, check **the Show shortcut keys in Screen Tips** option.



Create keyboard shortcut for a tool.

Appendices

A *3D Cursor Keyboard Shortcuts*

B *Statistics Explorer Tool*

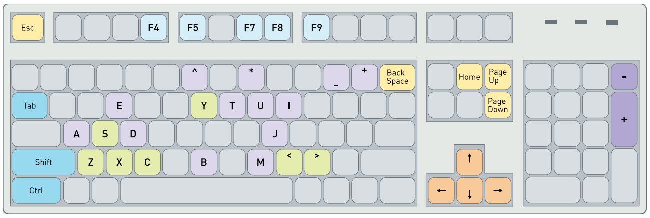
C *Field Data Conditioning Tool*

D *Full Colour 3D Stereo Projection Configuration*

E *Voxel Calculator Syntax*

A 3D Cursor Keyboard Shortcuts

Encom Discover 3D Keyboard Shortcuts



Using this Card

This card can be used as a quick reference for the Encom Discover 3D keyboard and cursor shortcut keys. As an aid to productivity, these can be used in conjunction with the standard menu items and toolbar icons.

Key

Shortcut keys apply once functions have been activated.

- Navigation
- Snapping
- Cursor Plane Display
- Cursor Plane Position and Orientation
- Bonding Cursor Plane to Images
- Clip with Cursor Plane
- Feature Editing

Mouse Symbology

- Left button click
- Right button click
- Dragging mouse with the left button pressed down
- Dragging mouse with the right button pressed down
- Dragging mouse with the left and right buttons pressed down

Navigation

Mouse	Rotates the view
Mouse	Zooms in and out
Mouse	Moves the viewing centroid horizontally in the XY plane
Shift + Mouse	Moves the viewing centroid vertically and parallel to the screen plane
Ctrl + Mouse	Pans the view from a fixed "look from" position

Snapping

S	Toggles snapping on and off

Cursor Plane Display

F4	Toggles cursor plane display
F5	Centres the 3D view at the current cursor position
F7 or Home	Extends the focus box to encompass the current 3D cursor location (does not move the cursor plane)
< >	(hold the Shift key) Decreases and increases the cursor plane size
Shift + C	Toggles the cross-hair style between none, 2D and 3D

Cursor Plane Position and Orientation

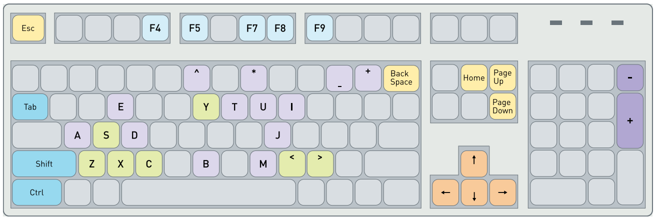
Tab	Switches the cursor plane orientation between three standard view planes: XY, YZ and XZ
Shift + X	Orientates the cursor plane parallel to YZ plane
Shift + Y	Orientates the cursor plane parallel to XZ plane
Shift + Z	Orientates the cursor plane parallel to XY plane
Page Up Page Down	Moves the cursor plane towards or away from the view point
↓ ↑	Changes the cursor plane inclination/dip
← →	Changes the cursor plane azimuth/bearing
Shift + F7	Centres the cursor plane without changing its orientation

Note: The position/orientation of the cursor plane changes at a slower/faster rate if the Shift/Ctrl keys are pressed.

Bonding Cursor Plane to Images

F9	Bonds or unbonds the cursor plane to the selected image
Shift + + or -	(numeric keypad) Iterates through all selected images
Shift + Ctrl + + or -	Switches bonding between the first and last images in a series

Encom Discover 3D Keyboard Shortcuts



Creating Features

(left click)	Creates a point feature or new points of polygon/ polyline (depending on the current default)
(left click and drag)	Creates a polygon/ polyline feature (if it is the current default type)
Esc	Cancels creating a feature
Back Space	Removes the last created point of a multi-point feature

Selecting Features

(left click)	Selects a feature
Ctrl + (left click)	Adds a feature to the current selection

Editing Features

Applies to a reshapable feature

(left click)	Depends on the current edit mode (Add, Delete, Break, None) and the type of element (node, edge, face)
(left click and drag)	**Node:** moves node on the cursor plane **Edge:** inserts a node
Ctrl + (left click and drag)	**Node:** moves node perpendicular to the cursor plane **Edge:** inserts a new node and moves it perpendicular to the cursor plane
Shift + (left click and drag)	Moves feature on the cursor plane
Shift + Ctrl + (left click and drag)	Moves feature perpendicular to the cursor plane
Ctrl + B	Toggles edit mode between Add, Delete, Break and None

Feature Operations

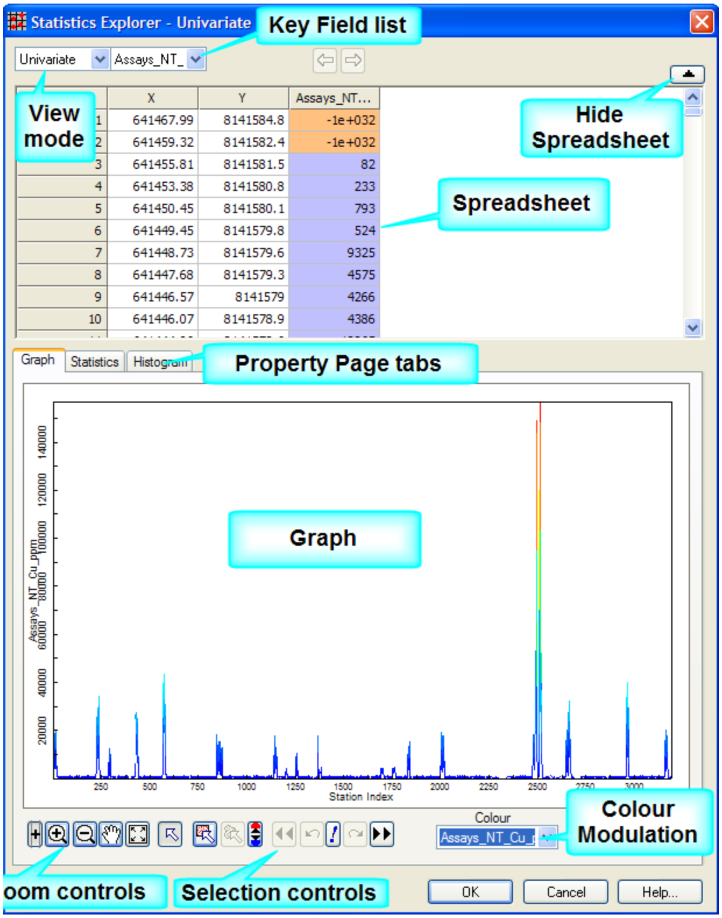
With two or more features selected

Ctrl + I Ctrl + *	Creates the intersection of the selected features
Ctrl + U Ctrl + +	Creates the union of the selected features
Ctrl + E Ctrl + ^	Creates the exclusive union of the selected features (overlapping areas removed)
Ctrl + _	(minus key) Cuts the first selected feature with all other selected features
Ctrl + A	Aggregates the selected features, preserving the original geometries
Ctrl + D	Disaggregates or ungroups aggregated features into individual features
Ctrl + M	Consolidates the selected features by removing identical elements (nodes, edges, faces)
Ctrl + T	Triangulates the selected features (breaks each polygonal face into a set of triangular faces) When at least one of the selected features is a point or a line, creates a surface by triangulating all nodes from all selected features

B Statistics Explorer Tool

The **Statistics Explorer** is a series of dialogs that enables you to examine any dataset both statistically and spatially.

The Statistics Explorer can be accessed via the statistics button in the various Field Data Conditioning dialogs accessed from many Properties dialogs, as well as the 3D gridding tool.



The components of the Statistics Explorer window

The Statistics Explorer window is divided into two main sections; the upper *Spreadsheet* window and the lower Property Page window. The contents of these depend on the view selected; the Statistics Explorer has four views, selectable from drop list at the top left of the window. These present graphical and statistical information using different Property Pages, which can be controlled using the Property Page tabs in the middle of the window.

These views are:

- The *Univariate* view examines a single field in the input data and presents basic summary statistics and histogram analysis.
- The *Bivariate* view examines any two fields in the input data and presents scattergram plots and basic summary statistics.
- The *Spatial* view plots the data using its spatial X, Y and Z coordinates and presents basic summary statistics.
- The *Variogram* view computes a variogram grid and displays directional semi-variogram data. This allows the creation of sample variograms, and creation and editing of model variograms.

The Statistics Explorer allows data to be selected in the spreadsheet or any of the univariate, bivariate or spatial views using the *Selection Tools* at the base of the window. The explorer can then ‘collapse’ the dataset to the current selection, and display the statistics for this new data subset.

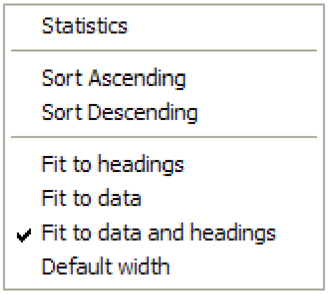
Spreadsheet

The **Spreadsheet** comprises the top half of the Statistics Explorer dialog, and displays the records for the currently viewed selection.

It uses the following colour codes to highlight records:

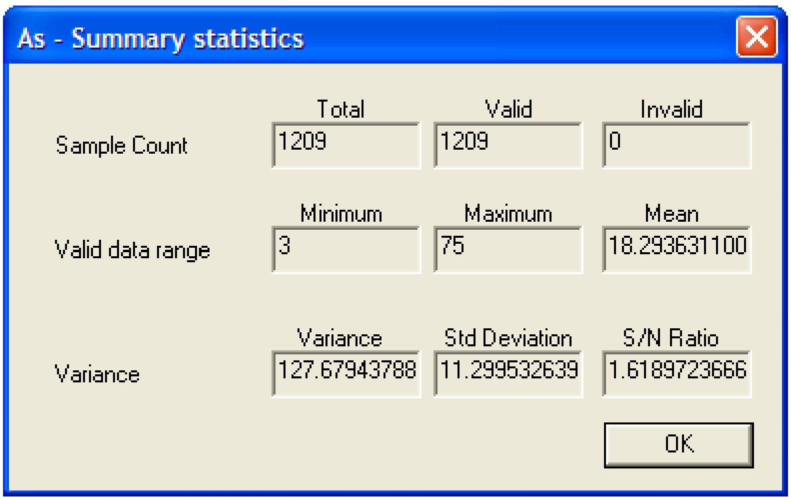
- The currently selected data points are highlighted red.
- The currently selected data point is highlighted purple.
- Invalid data is highlighted orange (e.g. coincident data).
- The current key and ancillary fields are highlighted purple.

The spreadsheet shortcut menu can be accessed by right clicking on any column header. This provides a number of column resizing options (including the various ‘Fit to’ options). Alternatively, columns and rows can be resized by clicking on their boundaries and dragging.



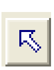
The Spreadsheet shortcut menu

The spreadsheet can also be sorted by any column using the Sort menu options. The Statistics option returns a basic statistical summary of any field.



The statistics summary for an arsenic field

Data points can be selected from the spreadsheet (if valid for the current view) by clicking on the row header; use the CTRL and SHIFT keys to unselect stations and make multiple selections. A range of cells can be also be selected by clicking and dragging the mouse. These selection operations are treated exactly the same way as graphical selections and the undo/redo operations can be applied.



When a single data point is selected in any graph (using the **Pointer** tool, see *Selection Tools*) the spreadsheet will scroll to the appropriate record.

The key field (whilst in univariate mode) or the ancillary field (whilst in bivariate mode) can be changed by double clicking on the field header in the spreadsheet.

Zoom, Selection and Display Controls

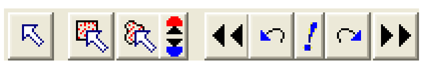
Each of the view types incorporates a number of property pages, discussed further under the relevant view sections. Depending on the view type, some or all of the following controls may be available within a property page window:

Zoom controls

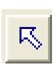
The Zoom controls toolbar

- Use the three buttons in the middle to **Zoom In**, **Zoom Out** and **Pan**.
- The **Restrict Zoom** button on the left restricts zooming and panning to the horizontal dimension only.
- The **Fit to Data** button on the right cancels any zoom and returns to the default view showing all data.

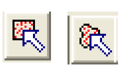
Selection Tools



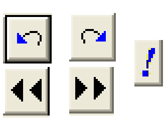
The Selection controls toolbar



The **Selection** tools provide a powerful data selection capability. The **Pointer** button displays a cursor which can dragged through the graph. It may be displayed as either a vertical bar or a cross hair depending on graph type. In all cases, the pointer 'snaps' to the closest station and this station is then highlighted purple in the spreadsheet.



Data can also be selected using **Rectangle** or **Polygon** selection, although the polygon selection is not available in all graphs. Selected stations are coloured red unless colour modulation is enabled in which case they are coloured black. Each selection operation can either select the enclosed stations (include option) or unselect the enclosed stations (exclude); this is toggled using the **Include/ Exclude** button.

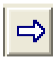


Each selection operation is placed onto a stack which allows you to **Undo** and **Redo** operations. Buttons are also provides to **Unselect All** and **Select All**. The selection can also be **Inverted**.

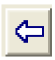
Display Modulation

The graph can be also be **Colour** and/or **Size Modulated** by any field in the dataset. Select the required fields from the drop lists at the base of the graph. To cancel colour or size modulation, select **<None>.** A simple pseudocolour look-up table is used and a linear colour stretch is employed.

Subsetting



Making a selection within a graph (which supports selections) or the spreadsheet view will activate the **Collapse** button at the top of the Statistics Explorer window.



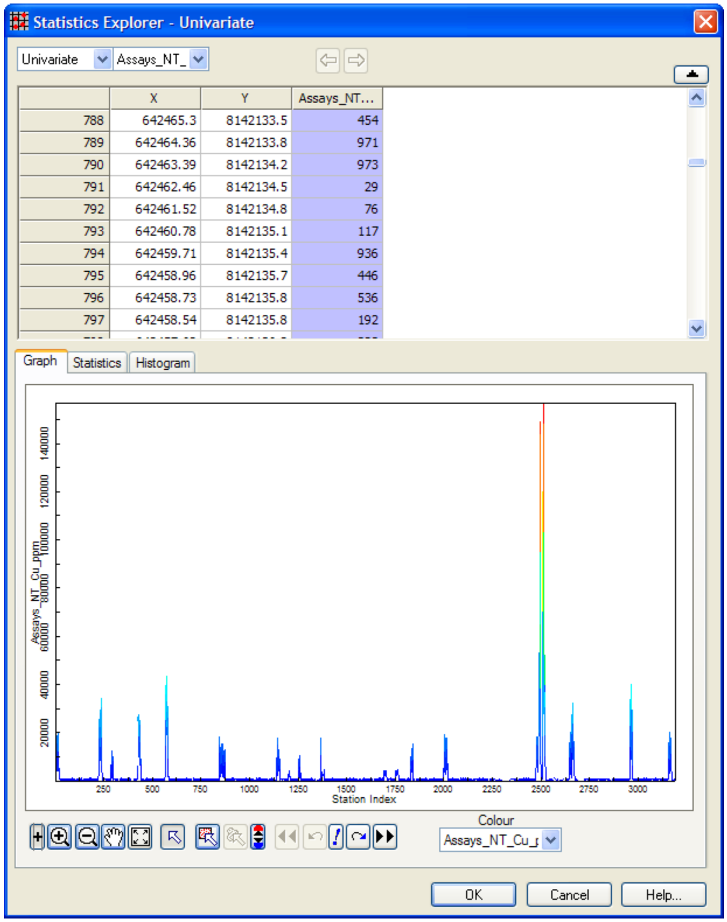
This button extracts the selected data points from the dataset to create a new subset. The spreadsheet and graph views will be redrawn to focus on this subset. Only the subset will be considered in any statistical computations or graphs. Further selections can be made to examine the data in greater detail. The **Expand** button allows a previous level/subset to be redisplayed. If this button is disabled then the view has returned to the original dataset.

Univariate

The univariate view examines a single field in the input data and presents basic summary statistics and histogram analysis. It incorporates three property pages:

- A **Scattergram** graph of the data point index verses key field.
- **Statistical information** for the key field.
- A **Histogram** of the key field. When this is displayed, the spreadsheet will show a detailed breakdown of the histogram data.

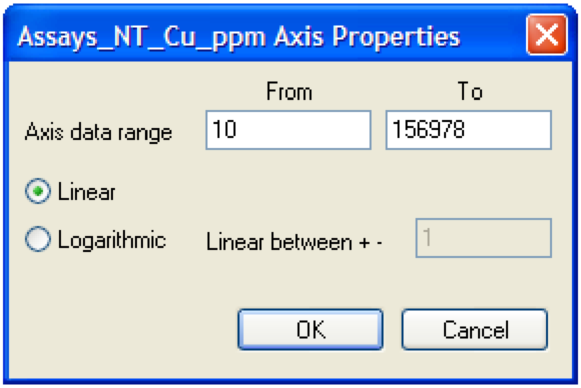
The univariate view requires a *key field* to be defined. This field is selected from the second drop list at the top of the dialog. The key field can be changed at any time to examine any field in the dataset.



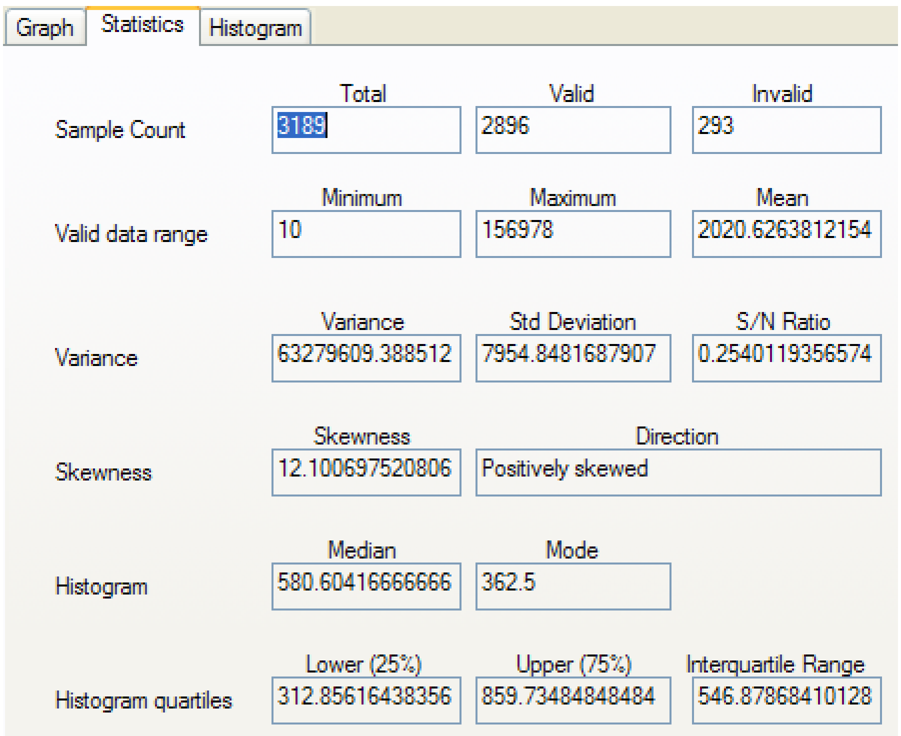
Univariate graph display

The **Graph** page plots the key field versus station index. Graph controls are detailed in *Zoom, Selection and Display Controls* above.

The properties of the graph can be obtained by double clicking on an appropriate area of the graph. For example, click in the bottom and left margins to obtain properties of the X and Y axes. This property page enables you to change the axis extents and change the axis mapping between linear and logarithmic. When using logarithmic axes, you must elect to leave some data space surrounding zero as linear.



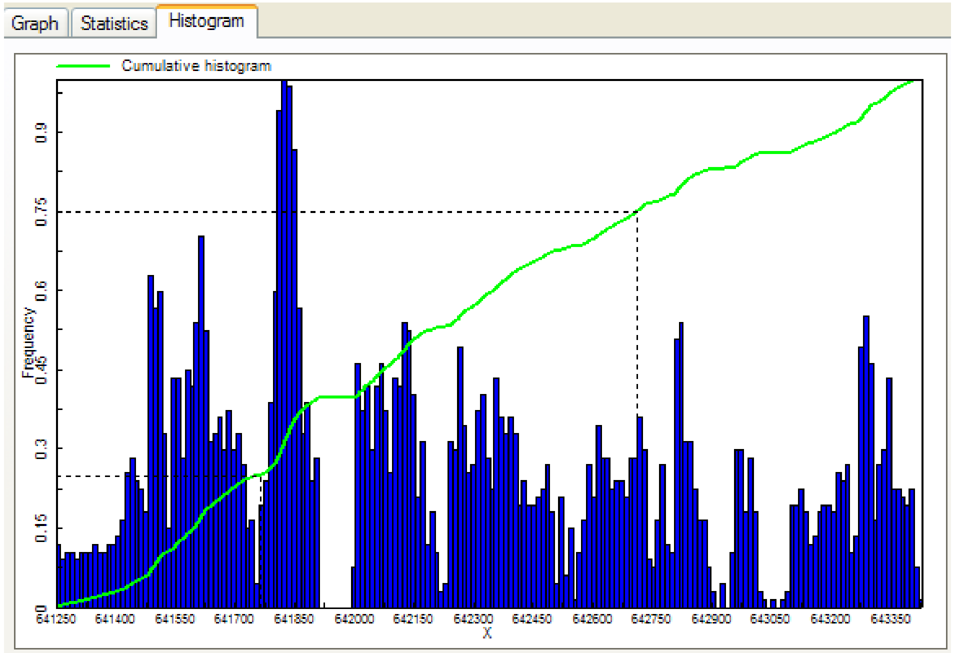
Specifying an axis range



Statistics of the chosen field

The Statistics page presents basic summary statistics for the key field, as detailed below:

Sample Count	The total number of samples in the dataset.
Valid Sample Count	The total number of valid (non-null/selected) samples in the dataset.
Invalid Sample Count	The total number of invalid (null/unselected) samples in the dataset.
Minimum	The minimum value of all valid samples.
Maximum	The maximum value of all valid samples
Mean	The mean (average) value of all valid samples.
Variance	The variance of all valid samples.
Standard Deviation	The standard deviation of all valid samples.
S/N Ratio	The signal to noise ratio of all valid samples.
Coefficient of skewness	The skewness of all valid samples.
Skew direction	The skew direction (positive/negative) of all valid samples.
Median	The value of the centre value in the sorted dataset.
Mode	The most frequently occurring valid value in the dataset.
Lower quartile	The value of the dataset at the 25 percent quartile.
Upper quartile	The value of the dataset at the 75 percent quartile.
Interquartile range	The range of the data between the lower and upper quartiles.



Histogram distribution of the selected field

The **Histogram** page presents an ‘equal width’ histogram of the key field. An equal width histogram is one built by dividing the valid data range into a large number of equal width bins and then computing the frequency of occurrence of key field values in each bin.

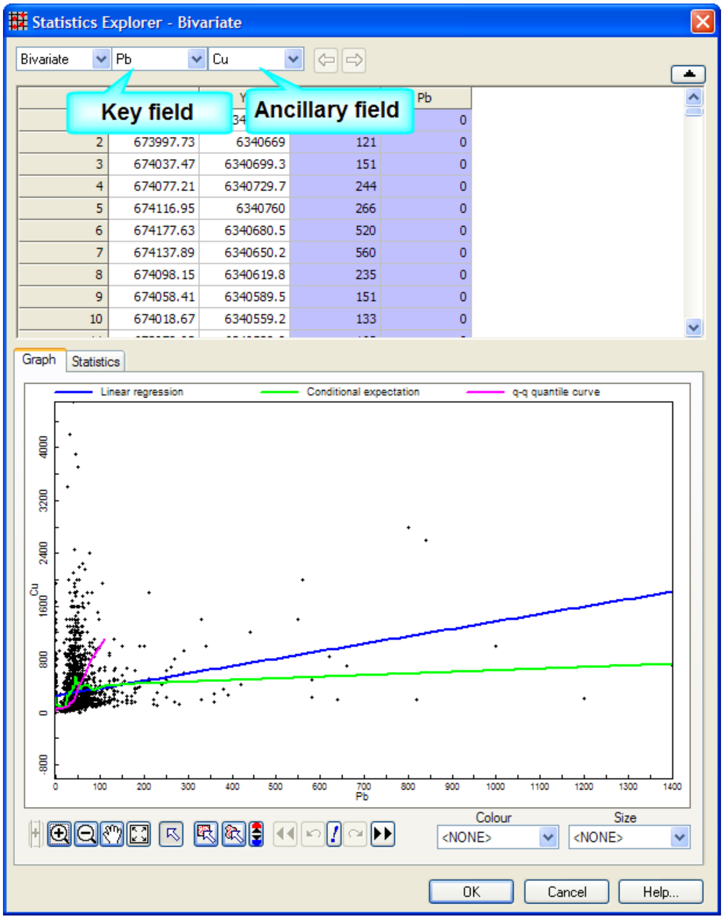
To be effective a large number of bins are used and these are grouped together depending on the scale at which the histogram is viewed. When multiple bins are being grouped together each consolidated bin is drawn with a back border. When zoomed in sufficiently to see individual bins, no black border is drawn. Also, the thickness of the bin is no longer equal to the assigned min/max values of the bin; it now relates to the actual min/max values of the data assigned to the bin.

The cumulative histogram is also displayed as a heavy green line. This indicates the percentage of data that is below the current value at any point. The upper and lower quartiles are displayed against the X and Y axes as dotted lines.

The spreadsheet displays the bin information including the bin data range, number of samples in each bin and the range of the actual data assigned to the bin. It also displays the cumulative histogram values.

The *Selection Tools* work normally in this graph. The selections are indicated as a percentage of stations selected in each bin. This is displayed as a red base. Note that no selections can be made from the spreadsheet in this mode.

Bivariate



Display of Bivariate data distribution with Key and Ancillary selection fields indicated

The **Bivariate** view requires the user to define two input fields – the **key** (horizontal) field and the **ancillary** (vertical) field. Make these selections from the drop lists at the top of the explorer (indicated in screenshot above).

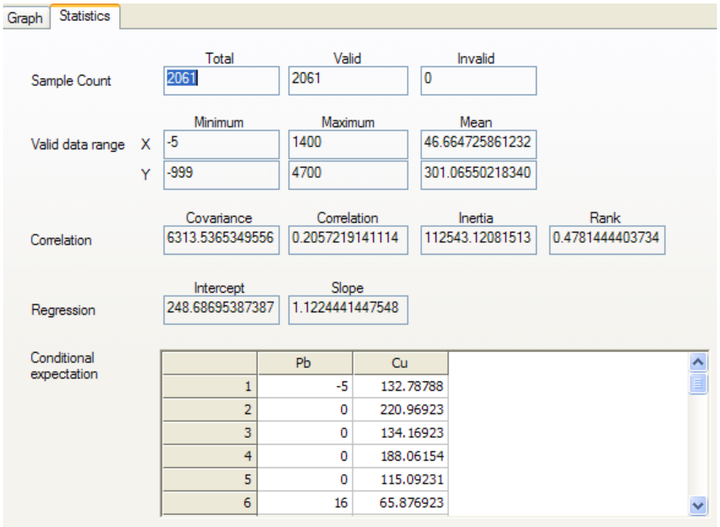
Two property pages are presented. The **Graph** page shows a scattergram of the ancillary field plotted against the key field. The **Statistics** page presents summary statistics for the bivariate distribution as well as a spreadsheet of the conditional expectation. You can copy and paste from this spreadsheet into Microsoft Excel.

The Graph page displays a linear regression as a blue line. A conditional expectation curve is also displayed as a green line. It also displays a quantile vs. quantile (q-q) curve as a purple line from 5% to 95% at steps of 5%.

For both pages, the Spreadsheet displays the complete dataset. A full range of selection tools are available both graphically and in the spreadsheet.

The following statistical definitions are used:

Sample Count	The minimum number of samples in both fields.
Valid Sample Count	The total number of valid (non-null/selected) samples in one or both fields.
Invalid Sample Count	The total number of invalid (null/unselected) samples in one or both fields.
Valid data range	The range of valid common samples in the key and ancillary fields (X,Y).
Minimum	The minimum value of all valid samples.
Maximum	The maximum value of all valid samples.
Mean	The mean (average) value of all valid samples.
Covariance	Sum of squares of product of the difference between the field mean and each sample.
Correlation coefficient	Covariance normalised by the product of the fields standard deviations.
Inertia	Half the sum of squares of difference between the two fields.
Rank coefficient	Spearman rank coefficient.
Regression	Linear regression fit to all valid samples.
Intercept	Value of the ancillary field at the origin.
Slope	Slope of the ancillary field versus the key field.
Conditional Expectation	Expected value of V at any U.

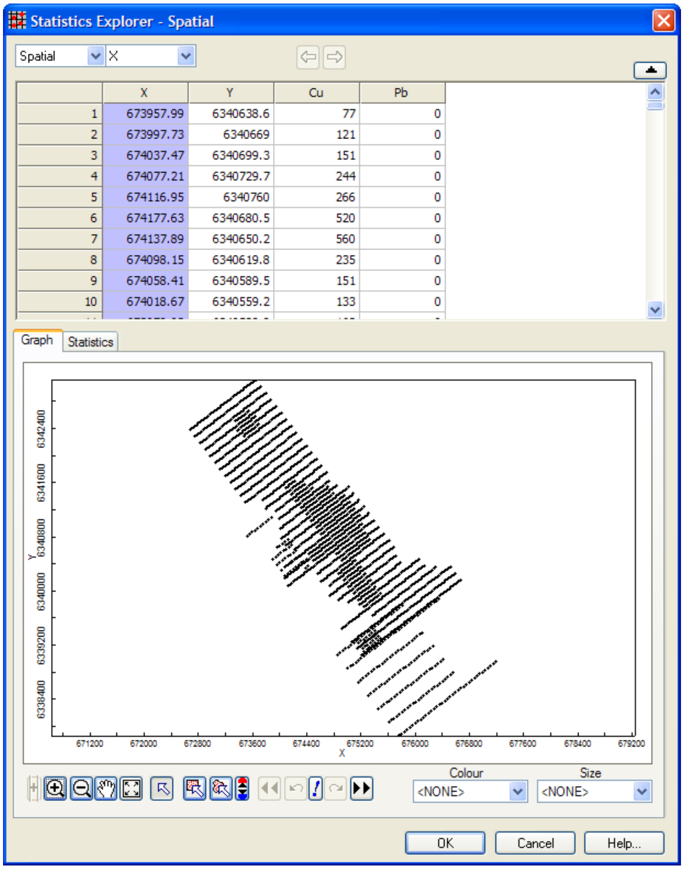


Bivariate statistical data ranges and distribution

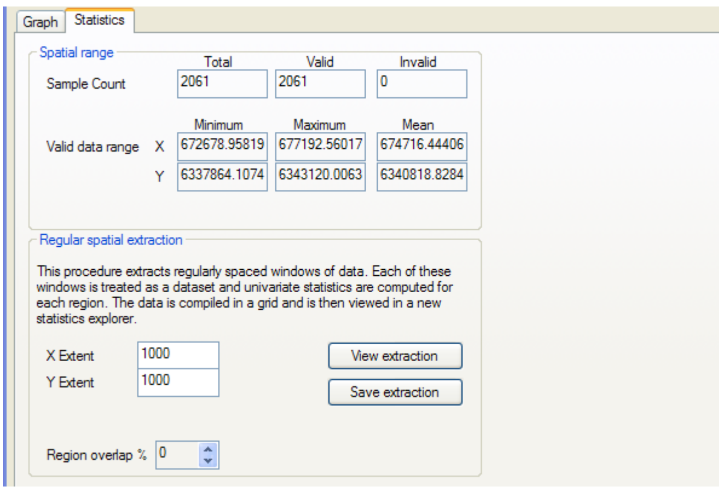
Spatial

The **Spatial** view does not require the user to define any fields (unless the **View Extraction** option on the **Statistics** page is used; see below). The data will be automatically plotted using its spatial coordinates in an isotropic view. If the data has less than two spatial dimensions no spatial view will be available.

A **Graph** page showing the spatial plot is presented. All *Zoom, Selection and Display Controls* are available. Data points can be colour or size modulated.



Displaying the Spatial distribution of the data



The Spatial Statistics pages

The **Statistics** page displays basic summary statistics for the spatial fields. It also has a **Regular spatial extraction** export option. This creates a new dataset using multiple regular sized cells covering the entire spatial extents of the existing dataset. The size of these cells are defined using the **X** and **Y Extent** windows. The **Region Overlap** control allows cells to overlap adjacent cells by up to 50%.

Regular spatial extraction requires the prior specification of a **Key field** in order to calculate summary statistics for each new output cell. The output dataset can be viewed using the **View extraction** button, which opens it in a new instance of the Statistics Explorer. The **Save Extraction** button allows this output dataset to be saved as a multi-banded ERMapper grid file.

Variogram

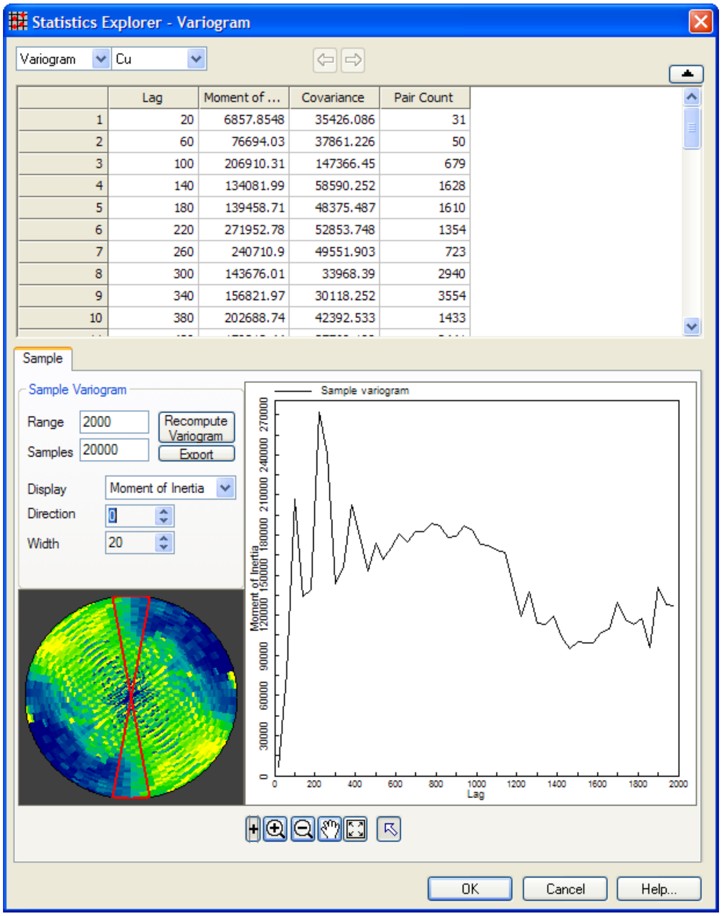
The **Variogram** view requires the user to define a key field which will be used in tandem with the spatial coordinates to compute a sample variogram. If the data has less than two spatial dimensions no variogram view will be available.

A variogram shows the degree of correlation between data in a spatial dataset in different directions and at different distances.

It can take a long time to compute a variogram and in most cases it will not be possible to compute the complete variogram for the entire data set unless it is relatively small (< 10,000 samples). Variogram computations are restricted by:

- Capping the maximum range (distance between points) that will be considered.

- Capping the number of input samples that will be considered.



Display of the calculated variogram

A default spatial variogram is computed using parameters that should ensure that the computation time is of the order of a few seconds. Thereafter the range and maximum sample number can be modified and the variogram can be recomputed by hitting the **Recompute Variogram** button. Note that if the maximum number of samples is set to minus one, then all samples will be computed. It is not advised to include all samples in the data set if it exceeds 10,000 samples as the computation time required to build the variogram could be extremely long. If the number of samples is restricted the algorithm will look at a sub-set of samples that are evenly distributed spatially within the dataset. In many cases it is desirable to increase the range and increase the maximum number of samples to improve the statistical reliability of the variogram.

This procedure generates a radial variogram grid (lower left). The directional variogram can then be quickly extracted from this grid. The directional variogram is plotted on the right and the source data is shown in the spreadsheet. Only the zoom tools are available in this graph (no data selection is possible).

The plot on the left shows the variogram grid. The red sectors represent the area of the grid that was used to extract the directional variogram. This area is controlled via **Direction** (0 to 360 degree clockwise from North) and **Width** (degrees of arc) controls. If the width is 180 degrees then you have obtained the omni-directional variogram and direction is irrelevant. Otherwise, you will obtain a direction dependent variogram. You can change the direction of the variogram from the **Direction** spin buttons or simply click and drag you mouse across the variogram grid plot. The width can only be changed via the **Width** spin control.

The variogram records several parameters including the moment of inertia (semi-variogram), covariance and pair count. Both the variogram grid plot and the directional variogram plot display the parameter selected in the **Display** drop list.

Model Variograms

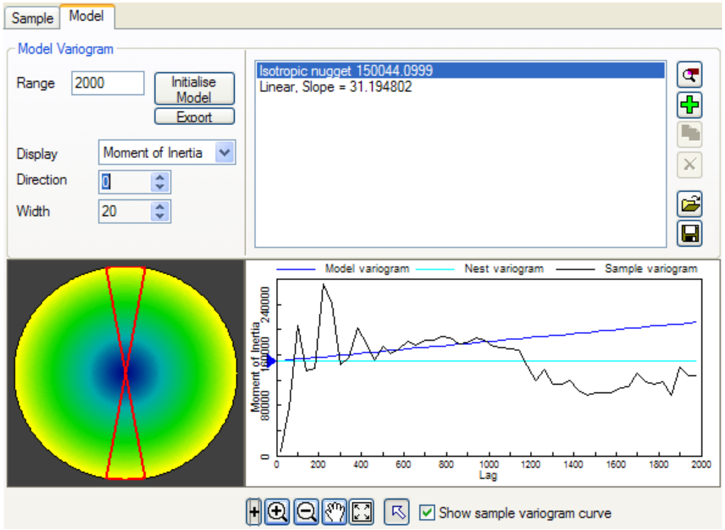
If the Statistics Explorer has been opened via the **Variogram** button within the *Kriging* wizard of the Gridding Tool, a **Model Variogram** also can be created, edited and displayed. In this case an option is added to the **Sample** page to show the model variogram curve in addition to the sample variogram curve.

A Model property page is also available to manipulate the model variogram. The model variogram will be automatically initialised to a reasonable model fitting the data. You can press the **Initialise Model** button at any time to reset the model to this default.

The model is plotted to a range controlled via the range edit parameter. By default this is equal to the sample variogram range.

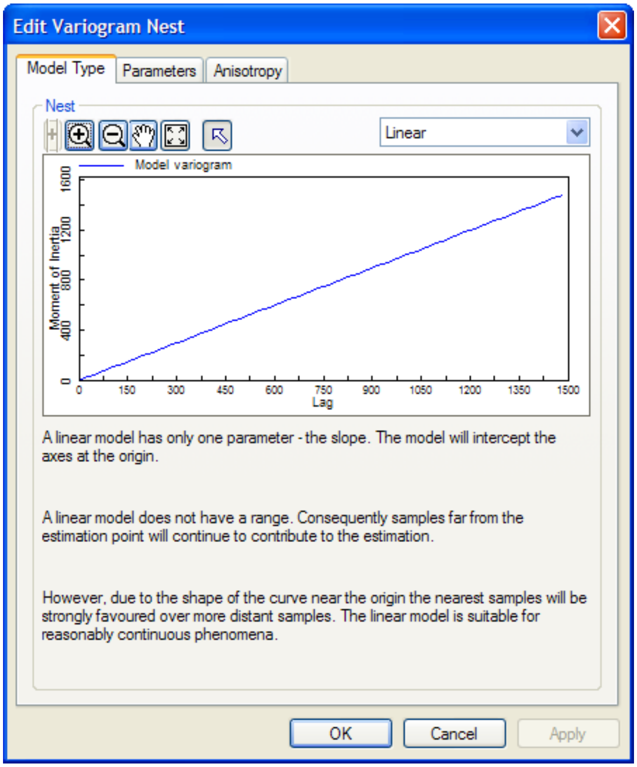
The model variogram is displayed as a grid and also as a directional extraction – just like the sample variogram. The sample variogram curve can be plotted for comparison. As before, the extraction direction can be controlled via the edit buttons or by dragging the cursor across the variogram grid display.

In the upper right the model nests are displayed. Each nest corresponds to semivariogram model. Individual nests can be edited, added, cloned and deleted via the buttons on the right. Alternatively double clicking on a nest in the list allows editing.



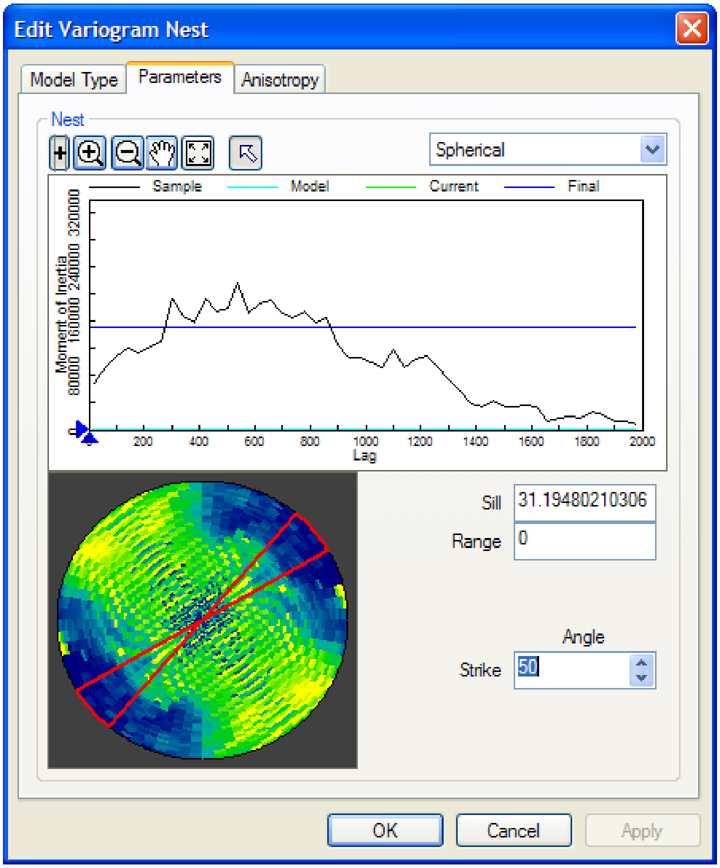
Displaying the sample and model variograms

Editing or adding a nest presents the following dialog; if adding a new nest the dialog is presented as a wizard.



Select the Model Type from the range available

The **Model Type** page allows you to change or select an appropriate model for the nest. A description of the model is presented. The coordinates displayed for the model are not representative of the actual model coordinates.

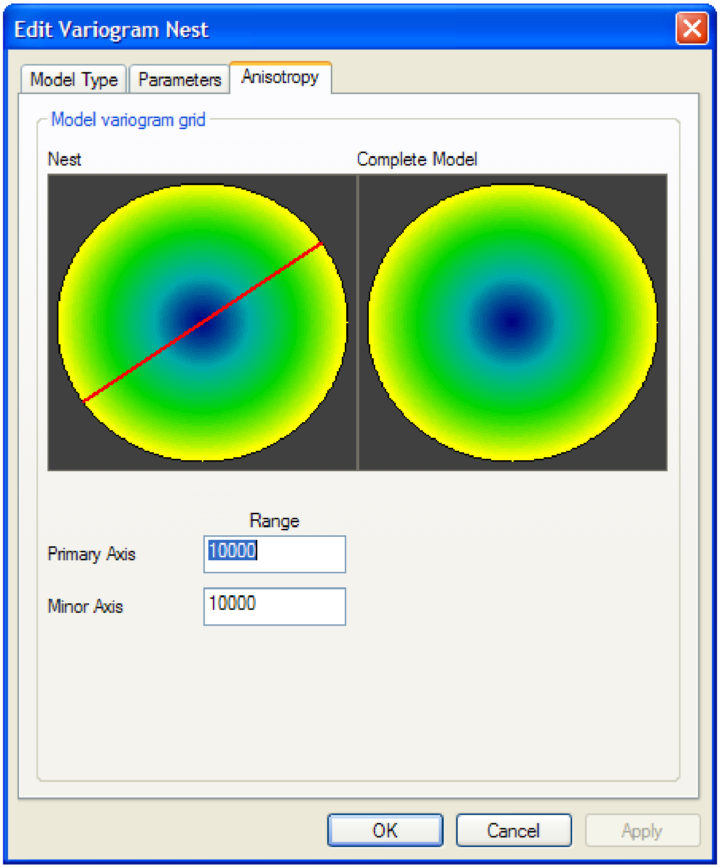


Controlling the parameters of the variogram

The **Parameters** page allows the editing of the model parameters. At the top right is a drop-list of model types.

The graph presents the sample variogram, the model variogram *excluding* the current nest you are editing or adding, the current nest and the final model variogram *including* the current nest. The model parameters are presented below the graph and can be directly edited. Also, most model parameters can be edited graphically by dragging the blue tags that are presented on the graph axes. For example, with the spherical model the left vertical axes allows the editing of the sill and the bottom horizontal axis allows range to be edited.

The sample variogram grid is displayed in the bottom left. Drag the cursor in this graph to edit the direction of the model variogram. This will also change the direction for the sample variogram extraction. Note that the width of the extraction is equal to the width of the extraction on the main ‘Sample’ page and cannot be modified here.



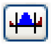
Editing the variogram nest

The **Anisotropy tab** controls the range of the model along two axes – the major and minor axes. The direction of the major axis is shown on the nest grid preview on the left. The final model preview is shown on the right.

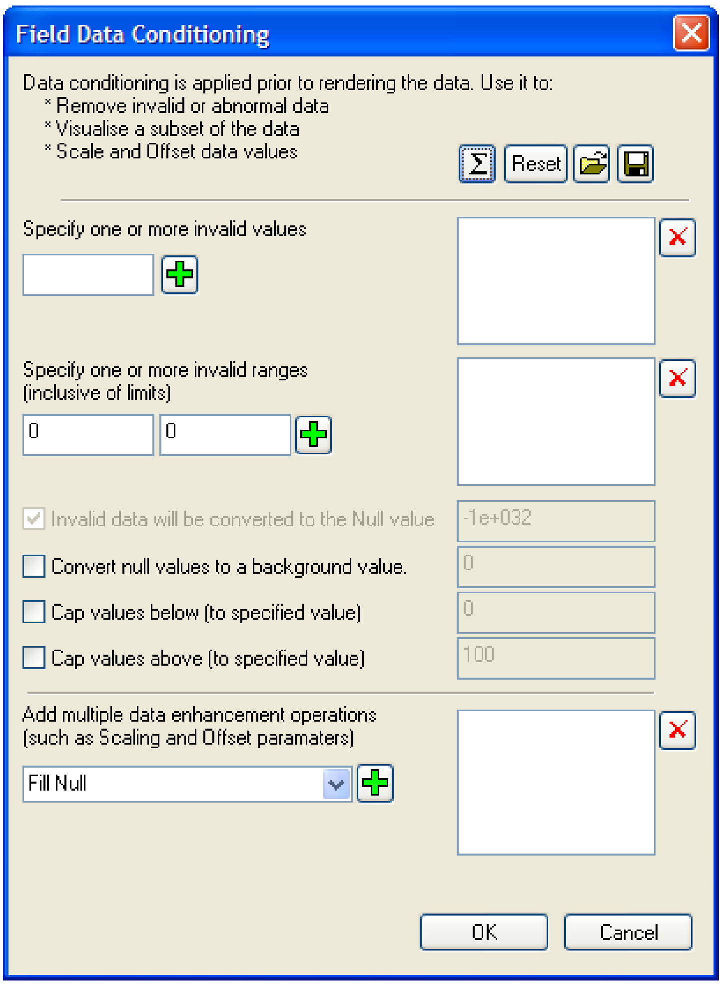
The major axis range will be equal to the range established on the ‘Parameters’ page. The minor axis range can be used to introduce anisotropy to the model. If the minor axis range is equal to the major axis range then the model is isotropic. If it is larger than the major axis range then the model will have a smaller contribution in the minor axis direction.

The principal parameters of each nest can be altered graphically via the ‘model’ page. Select the nest from the model list and manipulate the principal parameters via the blue edit tags in the variogram plot.

C Field Data Conditioning Tool

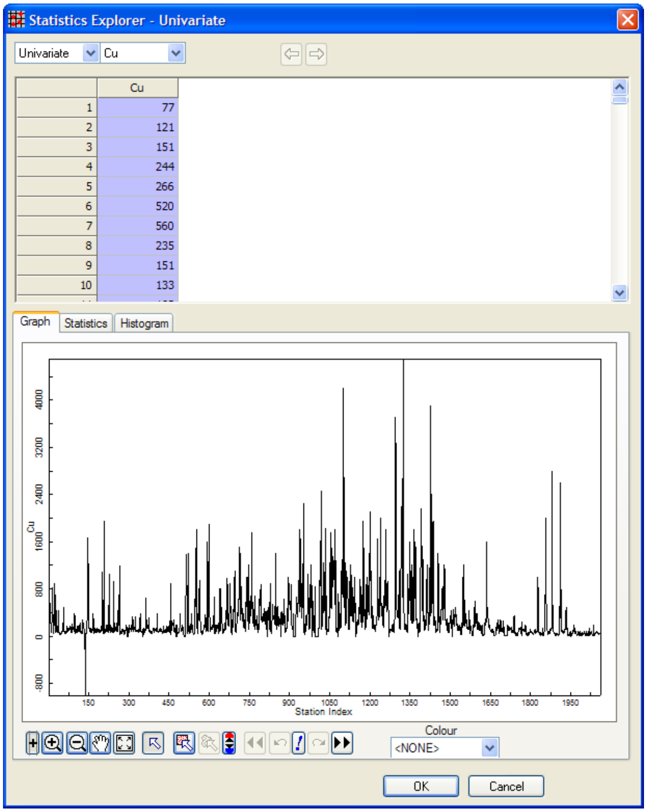


The **Field Data Conditioning** dialog can be accessed from a number of Discover 3D tools using the **Advanced Settings** button (shown left). It allows you to remove abnormal or invalid data, such as negative values representing missing samples and spurious outliers, as well as providing Null value assignment and handling. It can also be utilised to query out and display specific portions of a dataset, for example all downhole gold assays between 2g/t and 5g/t, or only intervals with a QBX or QV lithological code.



The Field Data Conditioning dialog

The **Statistics** button at the top of the dialog opens the *Statistics Explorer Tool*. This tool provides a powerful means of analysing the dataset for geochemical outliers, invalid data ranges, data distribution, etc via Graphical, Statistical and Histogram views. This dialog is dynamic: it will reflect any invalid data ranges or cap values set in the **Field Data Conditioning** dialog, allowing the effect of any conditioning applied to the data range to be checked.



The Graph view of the Statistics Explorer

Within the Graph and Histogram views of the *Statistics Explorer Tool*, data subsets can be selected using the **Rectangle Selection** tool.

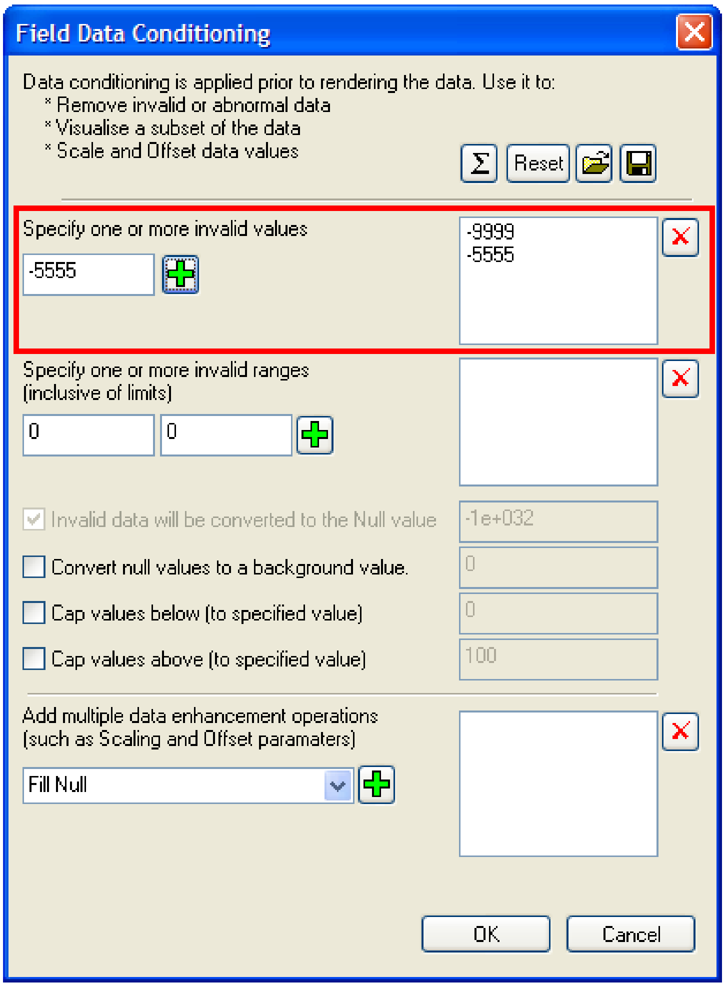
Then viewed using the **View Current Selection** button.

Return to the previous selection (or entire dataset) using the **View Previous Selection** button.

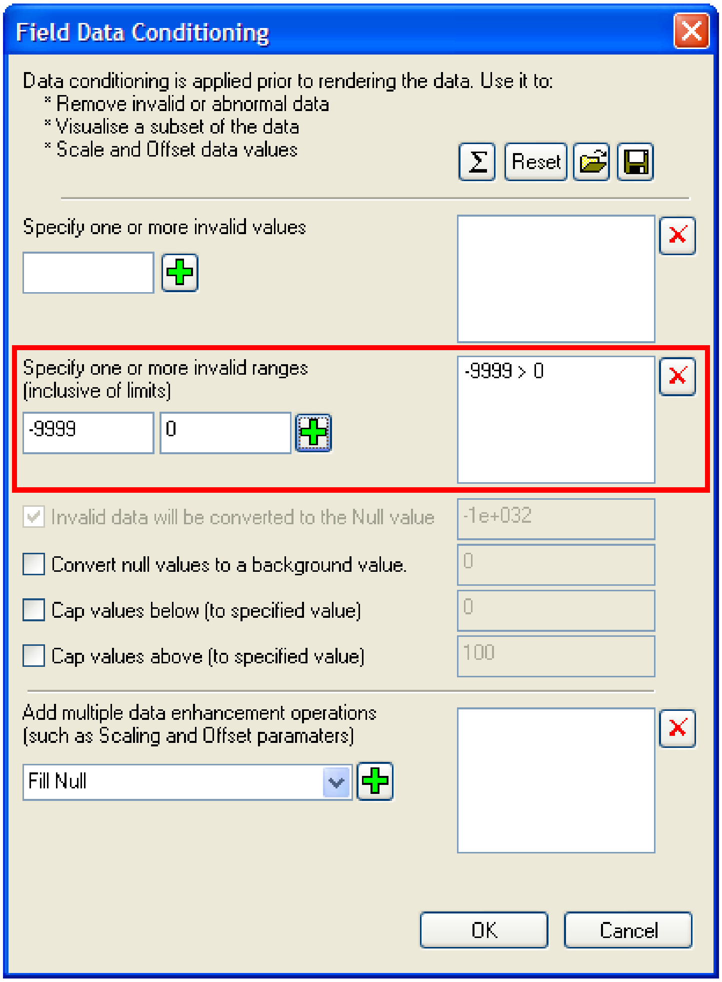
The central part of the **Field Data Conditioning** dialog allows the specification of invalid data and/or data ranges. The first option auto-populates with the dataset's default null value: generally -1.0×10^{32}. Invalid data (either user-defined or pre-existing null input data) will be handled as a null value internally and not displayed.

The following options allow specification of a list of user-defined invalid data values and/or ranges specific to the data field. Enter the invalid value or range at the left and then click the green cross to add it to the list at the right.

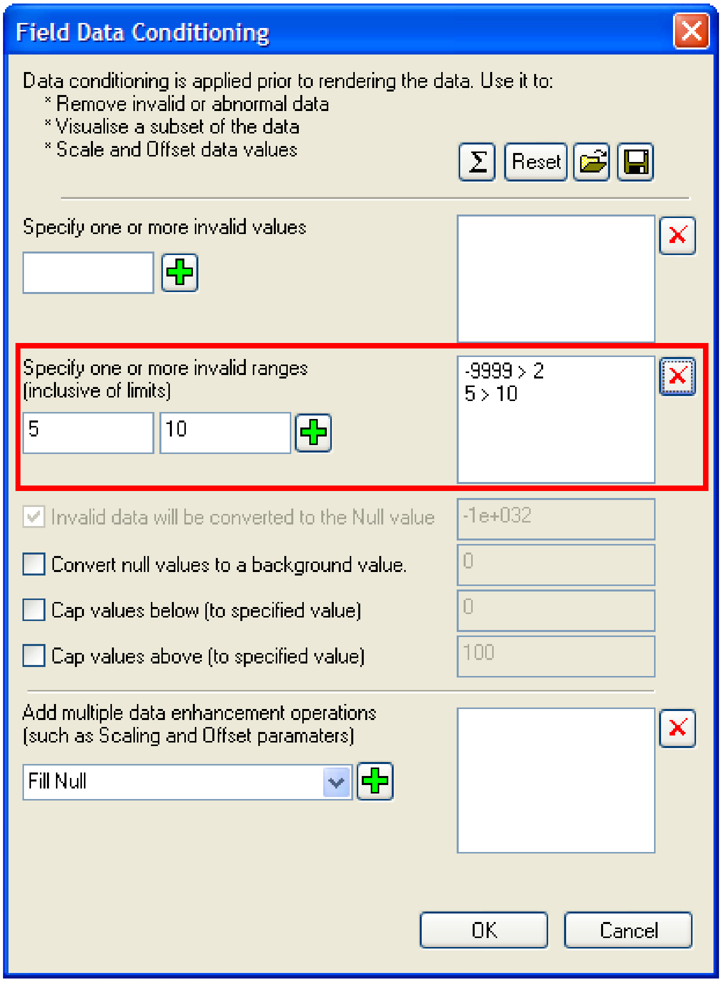
List entries can be removed using the **Delete** button. Some examples of use are presented below:



Specifying individual invalid data values. This is appropriate for removing negative values representing BDL (Below Detection Limit), SNR (Sample Not Received), etc.



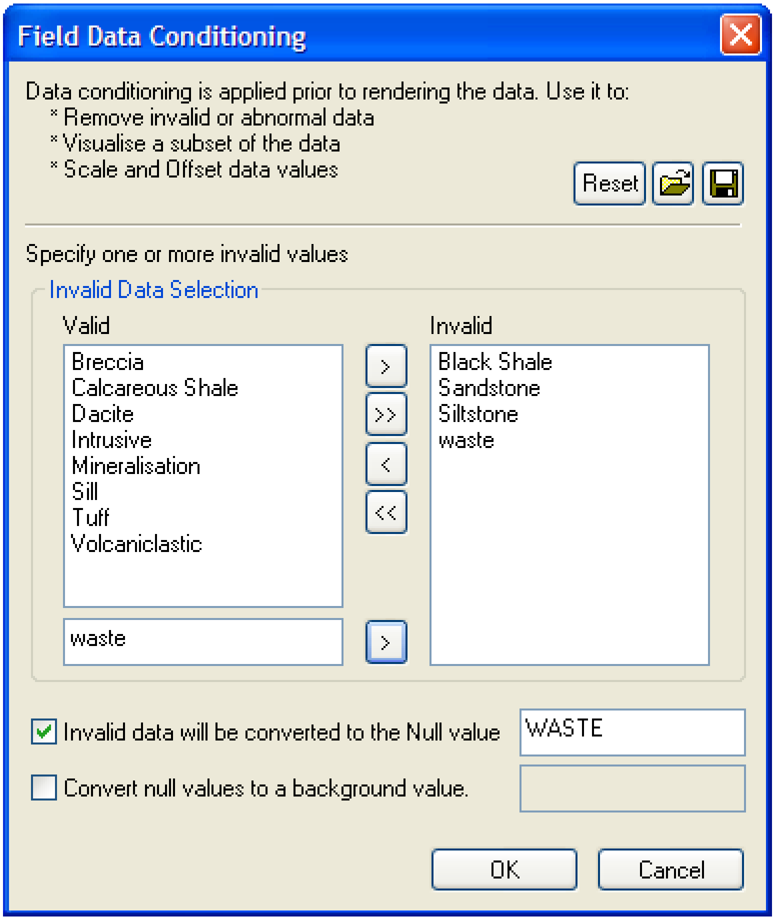
Specifying an invalid data range to remove a range of negative values in the dataset from the 3D gridding process. This is a useful way to remove multiple negative values, rather than entering each one individually.



Using the Field Data Conditioning dialog to display only a data range of interest, in this case only gold assays between 2 and 5g/t. This is done by specifying all data outside this range as a series of invalid ranges.

Setting a **Cap values below** or **values above** will cap source data outside the set limit to the limiting value. For example, with **Cap values above** set to 500, a gold assay on 725ppm will be handled as a 500ppm value. It is also possible to **Convert null values to a background** value specified by the user, in order to constrain the invalid data. For example, if plotting geochemical assays, much of the data may not have been sampled for and in these areas the assay result may be assumed to be equal to the background value. This helps prevent anomalies ‘ballooning’ into areas with no source data coverage.

Various other numerical functions are available for most datasets from the drop down at the bottom; including scaling, translate (offset) and averaging functions.



The Field Data Conditioning dialog when the source data field is a text field

If the data field is a **text** field (e.g. lithology codes for use in drillhole colour modulation, or voxel gridding), the **Field Data Conditioning** dialog display will be altered accordingly.

This allows individual text strings to be specified. The null value for invalid data can be set, and a background value to set nulls to.

Data conditioning parameters (invalid data ranges, cap values, etc) can be saved and reused by using the **Save** and **Load** buttons at the top of the dialog. These create and handle Data conditioning files (.edc) that store the user-defined parameters.

D Voxel Calculator Syntax

A series of advanced Boolean and algebraic operators are available for use in the *Voxel Calculator* tool to enable data merging, decision logic and complex arithmetic operations. The operators are implemented by applying a specified syntax that describes the input, output and required operation.

- *Logical Operators*
- *Arithmetic Operators*
- *Anti-Log Functions*
- *Comparison Operators*
- *Boolean Operators*
- *Trigonometric Operators*

Logical Operators

- *AND Operator*
- *NOT Operator*
- *OR Operator*
- *XOR Operator*
- *IF Operator*

AND Operator

Returns TRUE if both its arguments are TRUE; returns FALSE if one or both arguments are FALSE.

Syntax

```
AND(logical1,logical2)
```

Logical1 and logical2 are conditions you want to test that can be either TRUE or FALSE. The arguments must evaluate to logical values such as TRUE or FALSE. An alternative to the AND function is the operator ‘&’ (‘&&’ is interpreted as ‘&’).

Example 1

```
Out = IF(AND(A>5, B<=10), 1, 2)
```

The output Out is set to 1 if both A>5 and B<=10 otherwise it is set to 2

Example 2

```
Out = IF(A>5 & B<=10), 1, 2)
Out = IF(A>5 && B<=10), 1, 2)
```

This is an alternative syntax. These examples produce the same output as example 1.

NOT Operator

Reverses the value of the argument. Use NOT when you want to make sure a value is not equal to one particular value.

Syntax

```
NOT(logical)
```

Logical is a value or expression that can be evaluated to TRUE or FALSE. If logical is FALSE, NOT returns TRUE; if logical is TRUE, NOT returns FALSE.

Example 1

```
Out = IF( NOT(A<100), A, 100)
```

The output Out is set to A if A > 100 and is set to 100 otherwise. This is a trivial example which could be replaced with the >= operator

OR Operator

Returns TRUE if either argument is TRUE; returns FALSE if both arguments are FALSE.

Syntax

```
OR(logical1,logical2)
```

`Logical1` and `logical2` are conditions you want to test that can be either TRUE or FALSE.

The arguments must evaluate to logical values such as TRUE or FALSE. The '|' operator is an alternate to the OR function. '||' is an alternative to '|'

Example 1

```
Out = IF( OR(A<100, B<100), C, D)
```

This example sets the output Out to C if either A or B is less than 100. The output is set to D otherwise.

Example 2

```
Out = IF( A<100 | B<100), C, D)
Out = IF( A<100 || B<100), C, D)
```

This is an alternative syntax where these examples produce the same output as example 1.

XOR Operator

Returns TRUE if only 1 argument is TRUE; returns FALSE if both arguments are FALSE or both are TRUE.

Syntax

```
XOR(logical1,logical2)
```

Logical1 and logical2 are conditions you want to test that can be either TRUE or FALSE. The arguments must evaluate to logical values such as TRUE or FALSE

Example

```
Out = IF( XOR(A<100, B<100), C, D)
```

This examples sets the output 'Out' to C if one and only one of A and B is less than 100. The output is set to D otherwise.

IF Operator

Returns one value if a condition you specify evaluates to TRUE and another value if it evaluates to FALSE.

Use IF to conduct conditional tests on values and formulas.

Syntax

```
IF(logical_test,value_if_true,value_if_false)
```

Logical_test is any value or expression that can be evaluated to TRUE or FALSE. For example, A=100 is a logical expression; if the value in A is equal to 100, the expression evaluates to TRUE. Otherwise, the expression evaluates to FALSE. This argument can use any comparison calculator operator.

Value_if_true is the value that is returned if logical_test is TRUE. For example, if this argument B and the logical_test argument evaluates to TRUE, then the IF function returns the value stored in B.

Value_if_false is the value that is returned if logical_test is FALSE. For example, if this argument is C and the logical_test argument evaluates to FALSE, then the IF function returns the value stored in B.

Up to seven IF functions can be nested as value_if_true and value_if_false arguments to construct more elaborate tests. See the last of the following examples. When the value_if_true and value_if_false arguments are evaluated, IF returns the value returned by those statements.

Example 1

```
Out = IF( A<100, A, 100 )
```

This example will set the output to A when A is less than 100 and set it to 100 otherwise.

Example 2

```
Out = IF( A<100, IF( B>A, B+100, A+100), IF(C>0, C, A) )
```

This example shows how the two output expressions of the first IF can themselves be IF functions.

Operators specify the type of calculation that you want to perform on the elements of a formula. Two types of calculation operators are available: arithmetic, comparison.

Arithmetic Operators

To perform basic mathematical operations such as addition, subtraction, or multiplication; and produce numeric results, use the following arithmetic operators.

Arithmetic operator	**Meaning (Example)**
+ (plus sign)	Addition (3+3)
– (minus sign)	Subtraction (3–1) Negation (–1)
* (asterisk)	Multiplication (3*3)
/ (forward slash)	Division (3/3)
^ (caret)	Exponentiation (3^2)

Anti-Log Functions

To perform anti-log operations (opposite in operation to logarithmic function equivalents) the following are available:

Syntax

`aln(x)`

Anti-logarithm (base e) of x, that is e^x

`alog(x)`

Anti-logarithm (base 10) of x, that is 10^x

Comparison Operators

You can compare two values with the following operators. When two values are compared by using these operators, the result is a logical value either TRUE or FALSE.

Comparison operator	Meaning (Example)
`=` (equal sign)	Equal to (A=B)
`>` (greater than sign)	Greater than (A>B)
`<` (less than sign)	Less than (A<B)
`>=` (greater than or equal to sign)	Greater than or equal to (A>=B)
`<=` (less than or equal to sign)	Less than or equal to (A<=B)
`<>` (not equal to sign)	Not equal to (A<>B)
`MIN(value1, value2)`	Smaller of two values
`MAX(value1, value2)`	Larger of two values

Null support is also provided in the calculators.

Boolean Operators

Note that the following Boolean logic operators apply only to the Grid Calculator. Some examples:

```
OUT = IF(A>0, A, B)

OUT = IF(A>0 & B<(C/2), D, E+F)

OUT = IF(AND(A>0, B<(C/2)), D, E+F)

OUT = IF(A<100, Null, A)

OUT = IF(A=Null, B, C)
```

Note that the second and third examples above are identical, the first using the & operator and the second using the AND function.

Trigonometric Operators

The following trigonometric operators are available for both Line and Grid Calculators.

```
DEGTORAD(degree_value)
```

Convert degree values to radians.

```
RADTODEG(radian_value)
```

Convert radian values to degrees

```
SINH(a)
```

Hyperbolic sine of an argument in radians (Y = sinh(x))

```
COSH(a)
```

Hyperbolic cosine of an argument in radians (Y = cosh(x))

```
TANH(a)
```

Hyperbolic tangent of an argument in radians (Y = tanh(x))

Note that standard geometric operations such as SIN, COS and TAN are available from the Line and Grid Calculator standard interfaces.

Index

Symbols

.00T 153, 166, 356
.3DS 152, 166
.ADF 152, 153, 166, 216, 356
.ALG 152
.ASC 152, 216, 356
.BIL 152, 216, 356
.BMP 153
.BT2 153, 166, 356
.CSV 364
.DEM 153, 216, 356
.DM 152, 166
.DTM 166, 356
.DXF 152, 244, 356, 364
.ECW 152
.EGB 152
.ERS 153, 216, 356
.FLY 571
.FLY animation script 575
.GIF 152
.GRC 153
.GRD 152, 153, 216, 356
.GXF 152, 216, 356
.HDR 216, 356
.JPG 152
.MIF 170, 244, 356, 364
.MIG 152, 216, 356
.PL 153, 166, 356
.PNG 152
.SHP 166, 244, 356, 364
.STR 166, 356
.TAB 152, 170, 244, 356, 364
.TAR 153, 216, 356
.TIF 152, 153
.TIFF 152
.TIN 356
.TKM 152, 356
.TS 153, 166, 356, 364
.USG 153, 216, 356
.VS 153, 166, 356
.WAV format 581
.XYZ 152, 216, 356

Numerics

3D animation 571
3D cursor keyboard shortcuts 619
3D DXF models 165
3D Extrusion Wizard 27, 66, 508
3D Grid Toolkit 443
3D Gridding
  block kriging 473
  capping data 461
  cell size 463
  classification 453
  Continuously variable method 449
  Convert null values to background 255, 461, 645
  creating 3D or 2D grids 446
  Default numeric null 481
  Default text null 481
  elliptical search 467
  geology 453
  grid extents 463
  grid origin 463
  grid passes 466
  Inverse Distance Weighting 450
  lithology 453
  Load entire hole length 481
  nugget effects 461
  Null value handling 460
  ordinary kriging 473
  point kriging 473
  search expansion 466
  Segment break length 481
  simple kriging 473
  simple mode 467
  weighting models 469
3D Grids 443
3D Map 440

3D Stereo Visualization 128
3D Studio 166
  vector 152, 356
3D Tube
  of drillholes 289
3DConnexion SpaceNavigator™ 125

A

Absolute
  for animation 583
Absolute position
  in Extrusion Wizard 513
Add 3D View to MapInfo button 562
Add a lines display object 85, 97
Add a located bitmap layer 85, 98
Add a points display object 84, 97
Add a surface layer 85, 98
Add a vector layer 85, 98
Add a voxel model layer 85, 98
Add button 429
Add Image button 144
Add layers to an image 220, 221, 222
Add Points tool 173, 189, 203
Add Vector button 166
Advanced option 38
Advanced Settings
  3D Gridding 461
Akima 242
Akima Spline
  drillhole desurvey method 283
Algorithm Options
  3D Solid Generator 528
animation 571
anisotropy
  kriging model variogram 480, 639
Appearance Tab
  of voxels 412
Appearance tab 413
  of drillholes 289
Apply button 114
Arc
  grid 152
ASCII 364
ASCII files
  import 400
ASCII XYZ voxel 405
ASCII XYZ voxel model 398
ASEG
  grid 356
  GXF 216
  GXF grid 152
attribute ranges 427
Audio
  for animation 583
audio accompaniment 580
Audio entry line 581
audio identification
  in animation 572
Auto-Apply
  option 407
Auto-Apply option 217, 416, 427
Available window 51
AVI movie 572
Azimuth
  for animation 581

B

Back buttons 32
Back calculation
  drillhole desurvey method 281
Background colour 596
Band Pass Mapping
  of images 234
Band-pass operations
  for colouring 157
base of extruded object 511, 513
Bezier smoothing
  for animation 579
Bezier Spline
  drillhole desurvey method 283
BIL 152, 216, 356
bitmap 153
bitmap display and manipulation 309
Bitmap tab 308
bitmaps formats 143
Bivariate view
  gridding Statistics Explorer 630
block kriging
  3D gridding 473
block models
  creating 443

breaklines
    Gridding Tool 253

C

calculator for 3D models 494
Camera button 581
Cancel button 79
    for animation 587
capping data
    3D Gridding 461
capture a point 580
Capture Point List 580
cell size
    3D Gridding 463
CEMI 398, 401
Centre the bitmap 309
centred 99
Chair Clipping 412
Chair View option 418
Clear Cells button 592
Clip model option 416
Clipping
    of voxels 412
clipping operation 417
Clipping tab 415
closed polyhedral object 520
collar positions 278
COLLARS.TAB 279
Colour
    option of images 233
Colour and Intensity layers 219
Colour cells of a row 592
colour gradients 232
Colour layer 232, 233
Colour Look-Up Table (LUT) Editor 590, 592
Colour mapping 157
colour mapping 420
Colour Tab
    of voxels 412
column
    slicing 415
Compression
    3D Solid Generator 529
compression 229
Compuserve GIF 152
Continuously variable
    3D gridding 449
Contour 222
Contour Dropout 242
contour labels 242
Contour surface 219
contour surfaces 222
contour vector layers 239
Contours 222
Convert null values to background
    3D Gridding 255, 461, 645
Convolution filtering
    3D Gridding 481
core mesh 400
Cosinus kernel function
    density grid 269
Create a movie 584
Create a new LUT 592
Create AVI button 587
Create Eye and Look Path DXF Files 579
create objects See Feature Database
cumulative value of the histogram 234
Custom option 592

D

Data Conditioning
    3D Gridding 460
    Gridding Tool 255
data field
    in Extrusion Wizard 511
Data honouring
    Convolution filtering 482
Data Source
    of images 233
Data tab 406
DATA.TAB 279
Datamine
    vector 152
    voxel 399
    wireframe 166
Datemine
    wireframe 356
Decimation entry 243
default locations 609
Default numeric null
    3D Gridding 481

default page size 605
default printer page size 605
Default text null
  3D Gridding 481
defining the flight path 575
Delete an image layer 220
Delete rows
  legends 598
density correction
  IDW gridding 472
Density method
  3D Gridding 455
Depth Axis 480
descriptive text entry 597
digitization See Feature Database
dimensions
  3D Gridding 464
direction selected 99
Discover3D>Options 603
Discover3D>Toggle Auto 3D Selection menu 42
Display
  option described 77
Display Assistant
  3D Gridding 497
Display list 288
display the Properties Dialog 96
Display Window area 76
Display>Drillhole menu 278
Display>Image 144, 146
Display>Image item 147
Distance
  3D Gridding 466
distance grid 482
dockable toolbars 76
downhole log 305
downhole survey data 278
draw objects See Feature Database
drillhole
  selection 52
drillhole appearances 290
drillhole data 276
drillhole display object 85, 97
drillhole log 26, 59
Drillhole Project 279, 306
drillhole project 276
Drillhole Properties 308
drillhole trace 306
drillholes
  appearance 289
Drillholes button 278
Drillholes entry 278
dropout contours 243
Dropout Frequency 243
Dropout Threshold 243
DTM tab 158
Duplicate an existing surface 220
Duplicate button
  of LUT Editor 592
Duplicate Patterns 598
Duration
  for animation 581, 583
DXF 152, 166, 170, 244, 356, 364
DXF file of a created flight track 572

E

Edit button 157
Edit Existing Fly-through File option 577
elevation of a topography surface 229
Elliptical search
  3D Gridding 467
elliptical weighting
  IDW gridding 472
Encom
  3D grid 399
  georeferenced image 152
  grid 152, 216, 356
  GSF 170
  vector 356
Encom 3D Grid format 499
Encom Georeferenced Bitmap file 143
Enhance
  option 236
Enhance displays
  of drillholes 304
Epanechnikov kernel function
  density grid 269
ER Mapper algorithm 222
ER Mapper Algorithm button 238, 240
ER Mapper software 238
ER Mapper support 238
ERMapper

algorithm 152
ECW 152
ERV 170
grid 153, 216, 356
ERMapper surface 219
ESRI
3D shapefiles 166
grid 216, 356
SHP 170
TIN 153, 166
vector 356
ESRI .SHP files 244
Event Details 582
for animation 580
Event type 581
exact hit distance
IDW gridding 472
examples of point data 173
experimental variogram
Kriging gridding 451
Exponential weighting
IDW gridding 471
Export DH Logs to 3D 59
extrude regular shapes 515
Eye points 579
Eye Position 583
of animation 573

F

Feature Database
cosmetic layer 360
Create 353
digitization 370
editing 375
export 364
Import 355
interrogating 391
labelling 365
object attributes 364
querying 368
selecting objects 368
feature databases 350
File Tab
of voxels 412
File types
.FLY 575
.WAV 581
Fill 156
fill colours
of lines 189
filtering
3D Gridding 481
Finish button 39
in Extrusion Wizard 517, 518
fixed value of height
Extrusion Wizard 512
flight path specified
for animation 571
FLY file type 575
fly-through wizard 571
Follow baseline 242
Foreground colour 596
formats 398
Frame Rate 579
From input
in Extrusion Wizard 515
From-to segment option 481
Fullscreen mode
3D view 127

G

Gaussian kernel function
density grid 269
Gaussian weighting
IDW gridding 471
GDB file 280
Gemcom 166
vector 153, 356
voxel 399
Geopak
grid 153, 216, 356
Georeferenced Image 153
Georeferencing Wizard 144
Geosoft
grid 152, 216, 356
voxel 399
Geosoft drillhole database
modify dip 280, 281
Geosoft Wholeplot Databases 280
GeoTIFF 152
GoCAD 364
vector 153, 166, 356

voxel 399
Grid Calculator Syntax 647
Grid Compression 604
grid creation See 3D Gridding
Grid Flipper 217
grid formats 217
Grid Information button 224
Grid Management 500
Grid Passes
3D Gridding 466
Grid Selection button 238
gridding methods 256
gridding See 3D Gridding
Gridding Tool
Input Tab 253
Preview Display 252
grids
gridding methods 256

H

Histogram
transform 157
histogram 234
Histogram Colour Mapping
of images 234
histogram equalisation
GriddingTool 252
hole name 288
hole tops 292
Holes tab 287
horizontal Scale 308
HSL Interpolation 184, 200, 421, 598
Hue:Saturation:Lightness colour 598

I

Image layer 147
Image Properties dialog 153
Images 3D
Object Manager 28, 70
Import a .TAB file 598
Inclination
for animation 581
individual drillholes 51
Input Data
3D Gridding 458
Input Tab
Gridding Tool 253
Insert
option described 77
Insert row
legends 598
integration 441
Intensity layer 232, 234
Interactive mode
of thresholding 424
interactive mode of thresholding 424
Interp Gaps button 592
Interp Selection button 592
Interpolation Mode of an image 161
interpretational displays 440
inversion 399
inversion packages 397
isosurface
creation 429
using interactive mode 429
using lithologies 429
Isosurface slider bar 430
Isosurface Tab
of voxels 412
Isosurface tab 429

J

joining shapes
tie lines 536

K

Kernel density estimator
density gridding 456
keyboard shortcut 619
Kriging
experimental variogram 451
model variogram 451, 452
nugget 452
range 452
sample variogram 451
semivariance 451
sill 452
kriging
equation 452

L

labelling of data 181
Landmark
  grid 152, 216, 356
layer property 234
Layout Mode 127
LEG extension 598
Legend information 598
Lighting
  of Located Images 157
Line Colour 597
Line Data 36
line data 189
Line descriptor 189
Line display 231
line identifier field 189
line parameters 189
Linear (Decreasing)
  transform 157
Linear (Increasing)
  transform 157
Linear Colour Mapping
  of images 234
linear traverses 189
Linear weighting
  IDW gridding 470
lithological boundaries 26
Lithology mode
  of thresholding 424, 426
Lithology pull-down
  list 429
Lithology Tab
  of voxels 412
Load entire hole length
  3D Gridding 481
log
  of drillholes 59
Log (Decreasing)
  transform 157
Log (Increasing)
  transform 157
Log Display 305
Log profiles 59
logs displayed 308
Look At point 573, 583
Look At points 579
Look Position 583
look-up tables 515
Loop Style 581
  for animation 578
lost file 79
low gradient areas 231
LUT Editor 590

M

Manual
  transform 158
manually pick anomalies 97
manually specified views
  for animation 571
MapInfo 364
  grid 152, 216, 356
  MIF 170
  TAB 170
  vector 356
MapInfo Professional
  views 562
Mask check box 244
Match Grid option 464
measured depths of the hole 278
mesh
  model 400
  UBC model 400
Micromine
  voxel 399
mine simulation 397
Minex
  grid 152, 216, 356
Minimum Curvature
  drillhole desurvey method 282
minor contours 242
model variogram
  kriging 476, 636
  Kriging gridding 451, 452
ModelVision 152
Modify Database Properties
  Geosoft drillhole database 280, 281
modulate the colours
  in Extrusion Wizard 515
Modulated rotation 186
Moment of Inertia See Sample Variogram
Move the selected surface 221

MPEG-4 Video codec 587
multiple image objects 28, 70
multiple isosurfaces 429
multiple voxel displays 440

N

nests
  variogram 476
New button
  for LUTs 591
New legend files 598
Next buttons 32
Noddy 399, 403
Normal View Mode 127
nugget
  inverse distance weighting 470
  Kriging 452
nugget effects
  3D Gridding 461
Null value handling
  3D Gridding 460
nulls 401
number of cells
  UBC model 401
number of colours 184, 199, 420
Number of entries 592

O

Oasis montajTM
  Geosoft drillhole database 280
Object Manager 28, 70
offset
  of Located Images 158
Offset Tab
  of voxels 412
Offset to grid maximum
  for animation 584
Online printers 610
Open button
  of grids 217
operator
  AND 647
  arithmetic 651
  boolean 652
  comparison 652
  IF 650
  NOT 648
  OR 648
  trigonometric 653
  XOR 649
Options tab 608
ordinary kriging
  3D gridding 473
ore body outlines 26
Orthographic view 99

P

padding zone 400
Page Layout display 610
Page Setup tab 610
Pan button 564
pattern 596
pause
  for animation 580
Pause for a specified duration 581
Perspective view 99
Play button 584
playback
  animation 580
Point Data 36
point data 173
point data tables 173
point kriging
  3D gridding 473
Point Size 605
Point Symbol Size
  for animation 579
Points display 174
pop-up menu items
  described 111
Power weighting
  IDW gridding 471
preferred view options 605
preview
  for animation 580
preview display
  Gridding Tool 252
Primary Axis 480
primary coordinate directions 418
properties

examples 397
properties of a model
UBC 401
property attribute 397
Property colour controls
of isosurfaces 430
Pseudocolour 219
Pseudocolour image 222, 223, 232
pseudosection 143

Q

quality of the bitmap 309
quality texturing 309
Quartic kernel function
density grid 269
query
display data subset 447, 641
query data
Field Data Conditioning 645

R

Radial density estimator
density gridding 456
range
Kriging 452
raster 152
Raster Detail 605
raster imagery 145
rasterise option 563
recorded lithologies
of drillholes 276
Red, Green and Blue branches 237
Red:Blue:Green (RGB) imagery 236
Red:Green:Blue
images 222
Remove Temporary Files on Exit 603
removing lines 189
Render thresholded voxel model option 429
Repeat Reverse Look 579
Repeat Reverse replays 578
replay
of animation 571
Reset button
in 3D displays 230
Restore button 115
RGB colour mode 236
RGB Interpolation 184, 199, 420, 598
RGB surfaces 223
Rotate Around button
for animation 582
Rotate View button
for animation 581
rotation
for animation 580
row
slicing 415

S

Sample Variogram
Kriging 473
sample variogram
Kriging gridding 451
Scale button
in 3D displays 230
Scale factor
in Extrusion Wizard 512, 513
scaling
in Extrusion Wizard 512, 513
scanned product 162
Scope
of Located Images 156
script file
for animation 575
search expansion
3D Gridding 466
Search Parameters
3D Gridding 465
search sectors
3D Gridding 469
seed model
for inversion 440
Segment break length
3D Gridding 481
Segments
drillhole desurvey method 282
Select points within a 3D wireframe 540
Selected window 51
semivariance
Kriging method 451
session file restoration 79
set transparent 162
shade ball

intensity 235
shading control 235
shading to the display 234
Shape Matching
3D Solid Generator 531
Sharpen
drillholes 304
sides of the extruded shape 66, 508
sill
Kriging 452
simple kriging
3D gridding 473
Simple mode
3D Gridding 467
Simple option 37
single model
only 407
slice
model 415
Slices Tab
of voxels 412
slider bars
for chair view 418
of Clipping 417
Smoothing 242
for animation 579
of 3D vectors 168
smoothing
in Grid Wizard 450
SpaceNavigator
controls 125
device 125
Spatial view
gridding Statistics Explorer 632
specific file locations 605
Specify the data value
of colours 592
spreadsheet
Gridding Tool 622
Stack button 230
standard colours
in Extrusion Wizard 515
Statistics Explorer 254, 621
Gridding Tool 621
status bar 77
Step Patterns 598
Structure Analysis
3D Solid Generator 530
Style
of Located Images 156
Subsetting 625
sun illumination
GriddingTool 252
Surface Compress option 231
surface display 229
surface grid
in Extrusion Wizard 511
Surfer
grid 152, 216, 356
Surpac
DTM 166
string 166
vector 356
voxel 399
survey data
of drillholes 278
survey points
of drillholes 279
SURVEY.TAB 279
Symbol Generator utility 186

T

tab controls 114
Tapering
IDW gridding 472
Temporary Directory 603
Tension
of smoothing for animation 579
texture mapping 223
Texture Quality 224
texturing
images 28, 70
threshold
lines 189
Threshold by checkbox 425
Threshold Tab
of voxels 412
Thresholding 424
Tie Lines
3D Solid Generator 533
Tools
option described 77
Tools option 605

topography 401
Total Colours item 184, 199, 420
transfer a view from Discover 3D 562
Transparency 222
transparency
  of drillholes 58, 60
transparency value 223
transparent 162
tree branch 111
trench display object 85
Triangle kernel function
  density grid 269
Triweight kernel function
  density grid 269

U

UBC 398, 400
UBC model 400
Uniform kernel function
  density grid 269
Univariate view
  gridding Statistics Explorer 625
University of Utah 401
Use Global Directory 610
USGS
  grid 153, 216, 356
Utilities menu 77

V

Variable cell size option 464
Variogram view
  gridding Statistics Explorer 634
Vector Contour branch 240
vector files 165
Vector item 166
vector layers 240
Vectors layer 166
vertical bars using point data 509
Vertical Mapper
  grid 153, 216, 356
Video codec 587
View AVI button 587
View Drillhole 306
View Logs menu 60, 307
view modes
  Full screen 127
  Normal 127
  Orthographic 99
  Page Layout 127
  Perspective 99
View Section Boundary 63
View>Add 3D View to MapInfo 562
voxel
  defined 397
Voxel Calculator 494
Voxel Model properties 406
Voxel models
  creating 443
voxels 411
Vulcan
  triangulation 166
  vector 153, 356
  voxel 399

W

WAV file type 581
Weighting Models 470
weighting schemes
  IDW gridding 469
WHOLEPLOT 280
wireframes 433
wizard button
  for models 406
Workspace toolbar
  described 109
Workspace tree growth 109

Z

Z Scale
  of 3D displays 225
Zoom Toolbar controls 98