Encom Discover PA™ 2013 User Guide

Pitney Bowes Software

pbEncom™

Encom Discover PA™ 2013 User Guide

Encom Discover PA 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

ENCOM DISCOVER PA RELEASE HISTORY

BHP Gemex Initial Release

v1.0 February 2000

Commercial Release

v2.0	February 2001	v7.0	August 2007
v3.0	November 2001	v8.0	April 2008
v3.1	July 2002	v9.0	March 2009
v3.2	December 2002	v10.0	February 2010
v4.0	July 2003	v11.0	May 2011
v5.0	May 2004	v12.0	February 2012
v5.5	August 2004	2013	February 2013
v6.0	October 2005		

Pitney Bowes Software

Table of Contents

1 Introduction

Encom Discover PA is an interactive interpreter's tool for analysis of data in profile, map and 3D display formats. The information that can be analysed by this software includes single or multi-channel data acquired from airborne or ground surveys, including electromagnetics (AEM), potential field (gravity or magnetic) data or a combination of data types. Encom Discover PA accesses data directly from advanced processing systems and industry-standard databases. This approach significantly increases the speed of data access and assists in data management. The use of database technology makes Encom Discover PA especially applicable to large datasets that are typical of modern airborne geophysical techniques.

Encom Discover PA provides a method of accumulating geophysical knowledge by using databases to store details of interpreted features. Access to the geophysical knowledge can be made directly in Geographic Information Systems (GIS) such as MapInfo™, Encom Discover and ArcView™.

Encom Discover PA also provides advanced visualisation using both 2 and 3 dimensional displays. 3-dimensional presentations of images, sections, graphs, flight paths and data objects provides interactive display manipulation with zooming, pan and fly-through techniques.

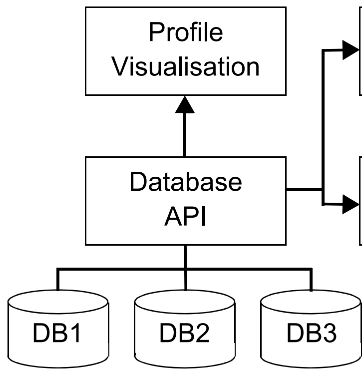
Templates allow fast creation of application-specific displays for magnetic, gravity, EM and spectrometer data. A number of templates are supplied with the software, along with the ability to modify or create your own. The template capability of Encom Discover PA enables combinations of various display types. For example, you can easily combine profiles, sections, maps and graphs together for high quality printed output. The routine production of maps, sections and profiles is simple when templates are used.

Overview

Encom Discover PA is designed for rapid and convenient visualisation of exploration data stored in industry standard databases. Currently supported databases include Geosoft Oasis montaj™, MapInfo Professional or DFA Intrepid™. Other database formats such as Microsoft Access are used for feature storage.

A range of display formats including profiles, image maps and graphs are supported and can be simply combined to form scaled or report quality output. Templates are used to save and retrieve display formats used routinely. Direct importation of ER Mapper grids (including ECW) and algorithms can produce quick image hardcopies.

Applicable databases contain input line data, ancillary metadata and any created features and interpretive information. A number of databases can be opened consecutively.

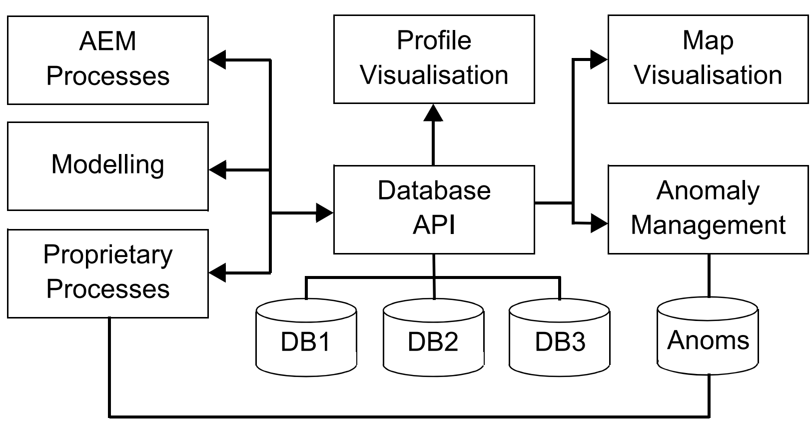


Schematic of Encom Discover PA architecture

Encom Discover PA can be used as a profile analysis tool with interaction and rapid navigation between lines with data linking to 2D maps. Real time analysis of data is also available through facilities such as the optional decay curve tool.

Encom Discover PA provides a means of communication with external processing or modelling modules. These programs form options that you may be interested in for specific exploration applications. Visualisation of both the observed and computed results can be undertaken in Encom Discover PA.

Pitney Bowes Software can supply airborne processing modules (such as EM Flow) or potential field modelling programs (such as ModelVision Pro) that can be connected to the Encom Discover PA software.



Batch processes and feature generation modules can interface to Encom Discover PA

External batch processing operations interface to the Database API. The processes can read data from the databases and return data that can be viewed in the map or profile modules. Importantly, information derived from modelling or picking of features can be transferred to a feature database that can be visualised with data or passed to external programs such as Microsoft Excel or Access.

Encom Discover PA operates under Microsoft Windows XP Professional and Windows 7.

Encom Discover PA Structure

Encom Discover PA consists of three packages designed to simplify the choice of available modules.

Encom Discover PA Viewer – A freely distributable version of Encom Discover PA, allowing all PA session files to be viewed and printed without a licence installation, with the limitation of not allowing changes to the properties of any views. This is available as a separate installation which can be downloaded from the Pitney Bowes Software - Natural Resources web site (*www.pbencom.com*)

Encom Discover PA Explorer – Contains all visualisation methods (in curve profiles, section profiles, 2D maps and 3D displays), data linking, import utilities and basic processing, conversion and display utilities.

Encom Discover PA Professional – Includes all modules available in PA Explorer plus a number of additional utilities for extending the interpretation functionality of Encom Discover PA, such as digitizing features in 1D, 2D and 3D.

When you run Encom Discover PA, the software begins a new session. A session is made up of various graphical windows that collectively can form:

- A **session**, which can open one or more Geosoft montaj™ or Intrepid databases. A session can also consist of one or more documents. Sessions can be saved to file (.EGS) and if transported between machines may require re-linking to local data.
- A **document**, which is contained within a window and holds a page definition as well as one or more graphical components. The position, layout and appearance of a document can be saved or recalled with a Template.
- A **template**, which is a description of all the graphical components in a document. A template can be saved to a file. It does not contain the data that the components display consequently a template is transportable across machines and can be linked to different data sources.

- A **component**, which is a graphical object that can be selected and manipulated by the user. A component can be saved to a file. As certain types of components may contain other components, inserting components into a document from a file can provide a flexible and powerful mechanism to construct complex documents built from a number of different components.

- A **frame**, is a container designed to contain graphical objects. It can also contain other types of objects such as:

  text

  pictures

  scale bars

  north arrows

  floating axes (colour bar)

Encom Discover PA produces a number of different types of plot objects. These objects are contained within a Frame, within a Document. Objects include:

- **Graph** (2D and 3D) – A graph presents data by joining the data value points with a line. A graph has no restrictions on the number of axes it can contain. There are few restrictions on the type of data objects a graph can have.

- **Scattergram** (2D and 3D) – A scattergram is used to display relationships between different data fields on 2 (2D) or 3 (3D) axes. Symbols are primarily used to locate the data points and their values. A scattergram is essentially a graph but with data points marked by a symbol rather than joined by a line.

- **Profile** (2D Only) – A profile is a subset of a graph but uses a horizontal axis derived from accumulated distance rather than a specific data channel. In addition, it can display layer data (e.g. CDI sections) since it is usually related to distance. It can only contain data from a single line, although the line data can be sourced from multiple databases.

- **Map** (2D and 3D) – A map contains X and Y axes (plus Z for 3D maps). There are few restrictions on the type of data objects a map document can contain.

- **Decay Curve** (2D Only) – A specialised graph designed to display EM decay curves. Display of decay curves is an optional module.

A data object requires a link to external data that may be stored in a database, raster grid file, vector file etc. Encom Discover PA can display the following types of data objects.

- **Curve or Lines** (2D and 3D) – A multi-banded curve (e.g. flight line data). Curves can be displayed as lines, filled lines and symbols. Curve and symbol attributes, including labels, can be modulated by data in a number of ways. Lines are simply multiple collections of Curves.

- **Section or Section Group** (2D and 3D) – A multi-banded layer (e.g. CDI data). Layers are derived from line data as either top, bottom and property or depth and property. They are displayed as filled colour sections. Section Groups are simply multiple collections of Sections.

- **Drillholes** (2D and 3D) – Specific drillhole databases contain depth and sample readings that can be displayed in profile sections, Vertical Graphs (log plots), maps or in 3D below surface. Drillhole traces can be modified by downhole reading, assay or lithology values.

- **Feature** (2D and 3D) – Displays feature data as symbols, lines or polygons. Symbol attributes can be modulated by data to assist in their interpretation.

- **Flight Path** (2D and 3D) – Displays the flight path or data location for a specified database.

- **Grid Image** (2D and 3D) – Displays raster data and in 3D, depth surfaces are also used. An image can contain multiple grid surfaces, each of which is derived from one or more raster grids. Surfaces can be Pseudocolour images, RGB images, ER Mapper Algorithms or Contours.

- **Vector File** (2D) – Imports a vector file in a variety of formats for display in a 2D or 3D map.

- **Vector File** (3D) – Supports import of 3D DXF files into a 3D graph or map.

- **Voxel Models** – Displays 3 dimensional voxel (volume element) models derived from modeling programs that output solid 3D volume models (eg UBC or CEMI).

- **TrackMaps** – These tracks are designed to be used with profiles and relate the path of a data line over an image such as a scanned map or raster image.

- **EMGui Database** (3D) – Displays models and system objects from an EMGui database.

Development History

Encom Discover PA commenced its development from the commercial software product known as GEMeX. GEMeX evolved over time to meet the exploration needs for airborne geophysical display and processing within BHP Pty Ltd. During 1999, BHP made the decision to commercialise their internally developed program called GEMeX to ensure ongoing support and development without ongoing internal costs. Through a consultative and tendering process, Pitney Bowes Software (formerly Encom Technology) was granted the rights to develop and commercialise the software. The term Profile Analyst was coined by Pitney Bowes Software to refer to the display and visualisation functions of GEMeX.

Within BHP, GEMeX was initially developed to operate on Silicon Graphics computers and make use of the powerful processing and visualisation capabilities of this hardware. The program was initially called TEMPER (Monks and Asten, 1993). More recently, the program was converted to operate on Windows 95 and NT/2000 environments plus Unix hardware with visualisation provided by AVS and AVS/Express.

Commercialisation of GEMeX by Pitney Bowes Software required a rewrite of the package with the initial emphasis on maintaining functionality and speed. Unix support was not required by BHP since the majority of their internal operations were now (during 1999) on PC Windows-based platforms. It was also considered that ongoing development could best be managed external to BHP without having to maintain internal support. The initial implementation of the GEMeX software by Pitney Bowes Software involved a major rewrite of the visualisation. This rewrite resulted in the GEMeX software which in turn became the commercial Profile Analyst software.

During 2006, it was decided to alter the commercial name of Profile Analyst to Encom Discover PA since the software functionality had grown to do much more than merely analyse profile data. With the introduction of a new name a restructuring of the available expert modules was implemented to provide clients with a more supportive package that allows access to greater features. The restructure provided the choice between three products.

- Encom Discover PA Viewer
- Encom Discover PA Explorer
- Encom Discover PA Professional

Supported Data Methods

Encom Discover PA supports a wide range of exploration data. These methods include:

Electromagnetic surveys

- Interactive feature analysis
- Presentation of processed results
- Manual feature tagging updated in database
- Conductivity depth image (CDI) display
- Composite display of data, CDI and features
- Decay Curve analysis

Magnetic surveys

- Variable area scaling of magnetic fields
- Display of filtered processing
- Vector data visualisation
- Interactive feature tagging updating in database

Gamma Ray Spectrometry

- Simultaneous KUT and Total Count displays
- Variable scaling for direct data comparison
- Analysis results
- Feature tagging
- Spectra displays

Gravity – Line Data

- Variable scaling of gravity profiles
- Visualisation of filter processes

- Display of vector gradiometer data
- Interactive feature tagging

Geochemistry – Line Data

- Combined simultaneous displays with other data types
- Statistical analysis display
- Interactive feature tagging
- Image display flipping for pattern convolution

Exploration and Engineering Applications

Numerous practical exploration and engineering applications can be undertaken with Encom Discover PA. Some of these include:

- Massive sulphide exploration with multi-channel EM, magnetics, geochemical and radiometric surveys.
- Regolith studies using multi-channel EM surveys, magnetics and spectrometry surveys.
- Geological mapping using multi-channel EM, radiometrics, geochemistry and magnetics.
- Combine geochemical and geophysical data for drill hole targeting.
- Diamond exploration using detailed magnetic analyses combined with EM and radiometric analysis.
- Uranium exploration using detailed analysis of gamma ray spectrometry data. Combine magnetics and EM data to visualise these methods in context across lines.
- Detailed image analysis applied to geological mapping using multi-band images, contour and vector data displays.
- Salinity studies of frequency and time domain EM data. Combine magnetics and radiometric data to complement these studies.
- Environmental studies merging specialised EM data with remote sensing methods such as Landsat TM imagery.

Template Applications

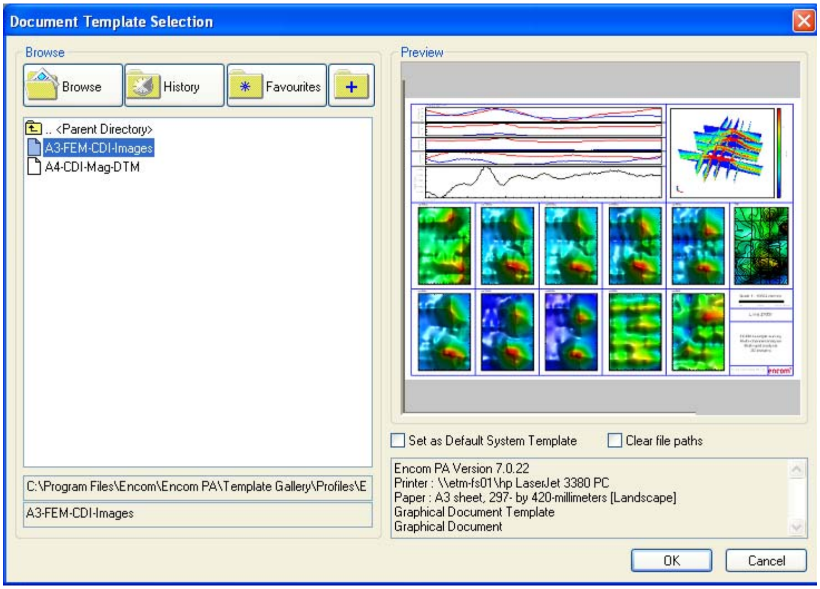
One of the most powerful and effective uses of Encom Discover PA comes from its template availability. Complex displays can be configured to present profiles, images and many combinations of display formats.

The configuring of the most effective displays for various exploration applications can be time consuming. It is not difficult, but it requires a good understanding of Encom Discover PA to achieve the desired result when complex presentations are required.

Consequently, to quickly provide powerful, application-oriented displays, a number of templates have been designed specifically for various applications. The templates are installed in the `Template Gallery` folder during installation. Some template applications include:

- Map Outlines
- Potential Field Templates
- Uranium Search
- Diamond Search
- Time Domain Method
- Frequency Domain Method
- Geochemistry Templates

To assist in the selection of templates, when the selection dialog is displayed, a small image of an example of the template is provided, plus a text description.



Template selection dialog with preview and text description

Using this Guide

Pitney Bowes Software has taken care to ensure that the information presented in this guide is accurate at the time the software was released. From time to time, both major and minor releases of Encom Discover PA are made. Changes in the program are not necessarily synchronised with the release of a new version of this manual.

This guide is available in printed and electronic formats. The electronic versions of this guide are supplied on the Encom Discover PA DVD in Microsoft Windows Help (CHM) and Adobe portable document file (PDF) formats. The PDF version is also available from the Pitney Bowes Software - Natural Resources web site (*www.pbencom.com*). Tutorials, available as PDF files, also provide a hands-on introduction to Encom Discover PA.

For information on the types of documentation available, see *Getting Help*.

2 Getting Help

Should difficulties or questions arise while operating Encom Discover PA, there are several sources of help available.

In this section:

- *User Documentation*
- *Technical Support and Updates*

User Documentation

User Guide

This comprehensive user guide provides all the required information for the operation of Encom Discover PA and associated modelling. The user guide is supplied as a PDF electronic document, which can be displayed from the **Help** menu.

You need Adobe Reader to view the PDF. Although supplied with the Encom Discover PA software, Adobe Reader is also available (at no cost) from the Adobe web site *www.adobe.com*.

Help

The user guide is also available in the form of interactive Help, which can be displayed at any time by:

- From the **Help** menu, selecting **Help Topics**.
- Selecting the **Help** button on most dialogs. This displays the relevant help topic.
- While a graphic or dialog is displayed, press the keyboard F1 key. This displays the relevant help topic.

Online Knowledge Base

The PBEncom Online Knowledge Base contains useful information that is regularly updated, including how-to articles, tips and tricks, error messages and their resolutions, video tutorials, and such. You can access the knowledge base by visiting *http://encomkb.encom.com.au*.

Encom Discover PA Tutorials

To assist with learning how to use Encom Discover PA, a set of tutorials is installed with the software. The tutorials provide step-by-step instructions on how to perform common taks in Encom Discover PA. The data files that are used in the tutorial exercises are also installed. All tutorials are based on real exploration situations in which Encom Discover PA can be used to display, enhance and visualise data. The tutorial is available as a PDF document, which can be displayed from the **Help** menu.

Technical Support and Updates

You can obtain product support for Encom Discover PA by e-mail or from your PBS reseller.

Contact details for Encom Discover PA support are:

Telephone	+61 2 9437 6255
Fax	+61 2 9439 1773
E-mail	*pbbi.support@pb.com*
Web	*www.pbencom.com*

3 Installing

This section describes the installation and operation of the software under the Microsoft Windows XP and Windows 7 operating systems.

- *Computer Hardware Requirements*
- *Installing Encom Discover PA*
- *Licensing Encom Discover PA*

Computer Hardware Requirements

The computer hardware and software requirements to install and operate Encom Discover PA are described below:

- *Windows Operating Systems*
- *CPU*
- *Graphics Monitor*
- *Disk Space*
- *Adobe Reader*

Windows Operating Systems

Encom Discover PA 2013 will run under the following operating systems:

32-bit operating systems

- Windows XP Professional (Service Pack 3)
- Windows 7

64-bit operating systems

- Windows 7

CPU

It is recommended that Encom Discover PA is operated on a Pentium 4 series 700 MHz (2 GHz recommended) PC. Extended memory also assists the operation of this and other Windows application software. A minimum of 512 megabytes (2 GB recommended) of RAM is required to operate Encom Discover PA.

Graphics Monitor

A high-quality graphics monitor of at least SVGA capability (1024 x 768 pixel screen display) is recommended for Encom Discover PA. Higher screen resolutions and a well configured, high memory graphics card will improve the appearance and ease of use of Encom Discover PA. A 64 MB graphics card is required as a minimum, but a 256 MB video card is strongly recommended.

Note If the computer is running an NVIDIA graphics card it is recommended that the NVIDIA Desktop manager is not enabled as it is known to cause conflict with Encom Discover PA.

Disk Space

Encom Discover PA requires approximately 650 MB of available disk space to store the executable, library files, and data files supplied with the software.

Adobe Reader

Adobe Reader version 6 or later is required to view PDF files supplied with the software.

Installing Encom Discover PA

The software can be supplied on either a DVD-ROM that includes installation, documentation, tutorials, slideshows etc. or you can download the software and selected items of documentation from the Pitney Bowes Software - Natural Resources web site *www.pbencom.com*.

Software installation is a two-stage process:

Stage 1: Install the software

For instructions, see either:

- *Installing from DVD-ROM*, or
- *Installing from the Web*

Stage 2: License the software

For instructions on how to perform different types of licensing procedures, see:

- *Licensing Procedures*

Installing from DVD-ROM

Place the supplied Encom Discover PA software DVD in the DVD-ROM drive and wait a few seconds. The DVD-ROM should register with your computer automatically and display an installation menu list. From this menu you can select the components you wish to install. When selected, the install program will run automatically.

Note If the DVD-ROM does not autoload, from Windows Explorer, double-click `AUTOPTN.EXE` in the `\AUTOPTN` folder or double-click `SETUP.EXE` in the `\Software` folder.

The install program guides you through the installation procedure.

When installed, Encom Discover PA can be started from the Encom Discover PA icon on the Desktop, or by starting the program from the Windows Start button. Initially, Encom Discover PA is not licensed and, when you first run the program, it requests you license the software.

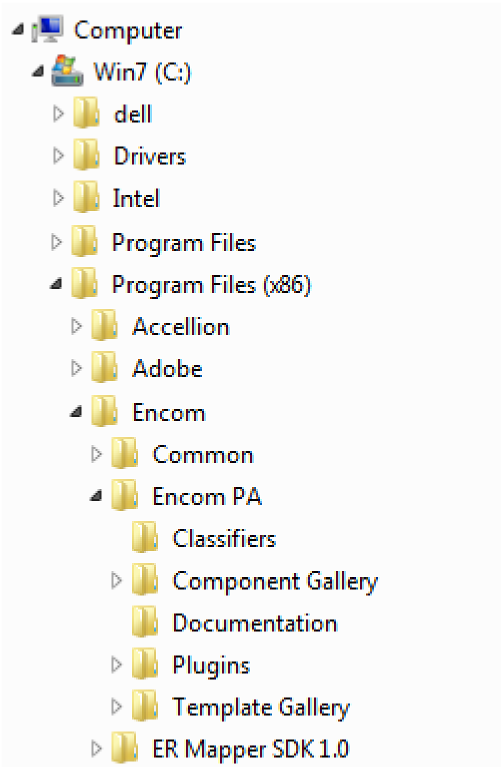
Installing from the Web

To download Encom Discover PA, visit *www.encom.com.au* and select the Download page. Encom Discover PA components are documented and their size indicates the time it will take to download. A separate installation for Encom Discover PA Viewer is available from the Pitney Bowes Software – Natural Resources web site and is a much smaller.

When you have completed downloading the installation file, from Windows Explorer, double-click the file name to initiate the installation procedure. Licensing procedures are identical for both a Web installation or a DVD-ROM installation.

Installation Folders

We recommend that you install this Encom Discover PA application in the default Encom installation folder under the Program Files folder on your local drive.



Example of the folder tree in which Encom Discover PA is installed.

The Encom folder may be placed under another folder by changing the default installation drive and folder name in the **Encom Discover PA Installation Folder** option when Encom Discover PA is installed.

Other folders may be created during the installation of Encom Discover PA.

Licensing Encom Discover PA

Encom Discover PA is protected under international copyright law. Pitney Bowes Software licensing systems are designed to protect against unlawful copying and use of the software. Encom Discover PA is supplied with a hard-disk licence system (see *Licensing Procedures*). Otherwise, if you are using a USB dongle or network licence, contact PBS Technical Support for assistance.

Licensing Procedures

When installing an Encom Discover PA licence, choose from the following licensing procedures:

Display information about Encom software installed

- *Displaying Licence Information*

Re-installing Encom Discover PA

If Encom Discover PA is installed on the computer with a valid hard-disk licence, the software does not need to be re-licensed when you re-install the software.

Encom Discover PA has not been previously licensed on the computer

If you installing the software for the first time, see:

- *Installing a New Hard-Disk Licence*

The licence is being transferred from another computer

You can transfer a licence from one computer to another without contacting Pitney Bowes Software. For detailed instructions, see:

- *Transferring a Hard-Disk Licence*

A later version of the software has been installed

If you are upgrading an installation with a valid licence to a later version of the software, see:

- *Upgrading an Existing Licence*

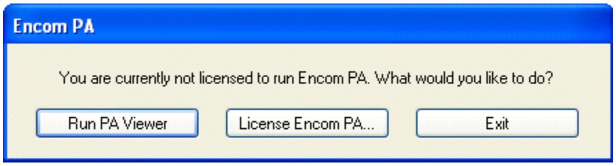
Installing a New Hard-Disk Licence

If your Encom Discover PA licence was delivered without a USB dongle, the following licensing procedure should be followed:

Start Encom Discover PA

1. Start Encom Discover PA from either the Windows Start button, or from the icon on the Desktop.

   A message is displayed indicating no licence was found.



Encom Discover PA Viewer can be run without a licence.

2. Click **License Encom Discover PA**.

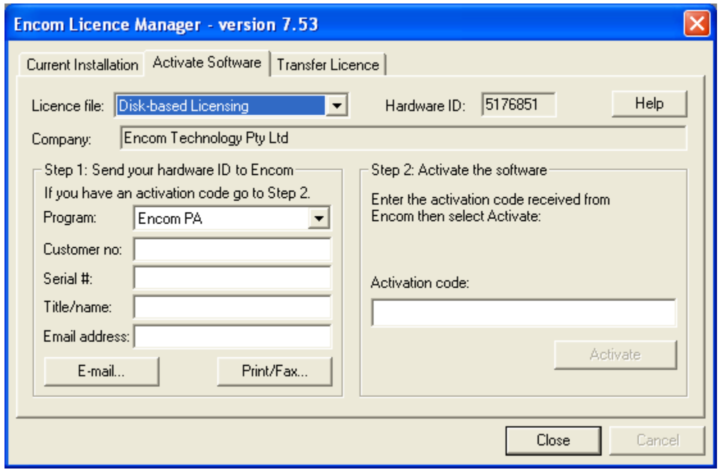
Create a new licence

3. The **Create Licence File** dialog is displayed. Type the company name and click **OK**.

   The **Encom Licence Manager** is displayed, which lists all Encom programs installed on this computer.

Request an activation code

4. Select the **Activate Software** tab.
5. In the **Program** box, select **Encom Discover PA**.
6. In the **Customer no.** box, type the 7-digit customer code supplied with the software.
7. In the **Serial #** box, type the 10-digit serial number supplied with the software.
8. Type your contact details in the corresponding boxes.



Encom Licence Manager

9. Click the **E-mail** button or, to generate a report that can be faxed to Pitney Bowes Software, click the **Print/Fax** button. The Pitney Bowes Software fax number is printed on the report.

Wait for Pitney Bowes Software to send the activation code

10. When your request and Hardware ID have been validated by Pitney Bowes Software, an Activation Code will be emailed or faxed to you.

Note If you are submitting your Activation Code request by e-mail, the automatic Pitney Bowes Software tracking system should respond within 30 minutes.

Activate the licence

11. If necessary, restart Encom Discover PA and display the Licence Manager as described in steps 1 through 3.

12. Select the **Activate Software** tab.

13. In the **Activation code** box, type the activation code supplied by Pitney Bowes Software. Upper or lower case characters can be used, but no blank spaces.

14. Click the **Activate** button. The displayed modules, expiry date and version number are updated.

Confirm details

15. Select the **Current Installation** tab and confirm the licence details.

16. Click the **Close** button.

Encom Discover PA is now ready to use.

Upgrading an Existing Licence

After installing Encom Discover PA, if a licence for an earlier version exists, you will be asked for an upgrade activation code when you run the software.

To upgrade the licence to the new version:

1. Start Encom Discover PA.

2. A message is displayed indicating that a valid licence was not found. Click **License Encom Discover PA**.

3. From the **Encom Licence Manager**, select the **Current Installation** tab.

4. In the **Licence File** box, select the type of licence.

5. In the **Encom Software Currently Installed** box, select **Encom Discover PA** and confirm that a licence for a previous version of the software exists, and then continue to step 6. If Encom Discover PA has not been previously licensed on this computer, select the appropriate licensing procedure for a new licence, as described in *Licensing Procedures*.

6. Select the **Activate Software** tab.

7. In the **Activation code** box, type the upgrade activation code supplied by Pitney Bowes Software. Upper or lower case characters can be used, but no blank spaces.

8. Click the **Activate** button. The displayed modules, expiry date and version number are updated.

9. Select the **Current Installation** tab and confirm the licence details.

10. Click the **Close** button.

Encom Discover PA is now ready to use.

Transferring a Hard-Disk Licence

If a hard disk licence is operating effectively on one computer (the source), you can transfer the licence to a second computer (the destination). The process of transferring the licence does not require contact with Pitney Bowes Software for an Activation Code. Follow the steps below to transfer a valid licence from one computer to another.

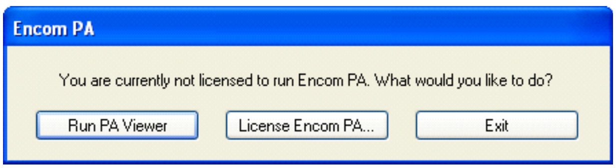
Install the software on the destination computer

1. Install Encom Discover PA on the second, destination computer. The version of the software on the source and destination computers must be identical.

Start Encom Discover PA on the destination computer

2. On the destination computer, start Encom Discover PA from either the Windows Start button, or from the icon on the Desktop.

   A message is displayed indicating no licence was found.



Encom Discover PA Viewer can be run without a licence.

3. Click **License Encom Discover PA**.

Create a new licence

4. The **Create Licence File** dialog is displayed. Type the company name and click **OK**.

   The **Encom Licence Manager** is displayed, which lists all Encom programs installed on this computer.

Obtain the hardware ID of the destination computer

5. Select the **Activate Software** tab.

6. Write down the 7-digit code in the **Hardware ID** box.

Display the Transfer Licence tab on the source computer

7. On the source computer, start Encom Discover PA.

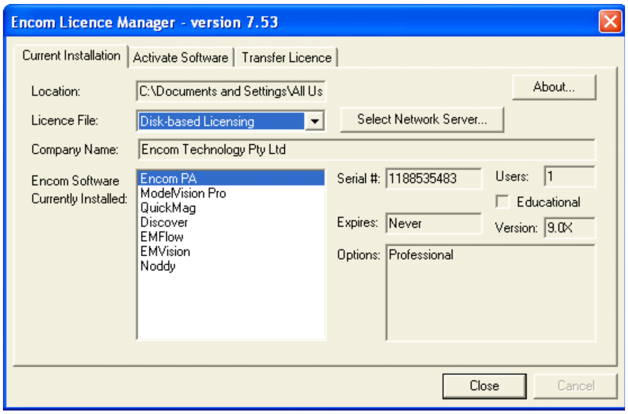
8. From the **File** menu, choose **Licensing**.

9. From the **Encom Licence Manager**, select the **Current Installation** tab.

10. In the **Licence File** box, select **Disk-based licensing**.

11. In the **Encom Software Currently Installed** box, select **Encom Discover PA**.

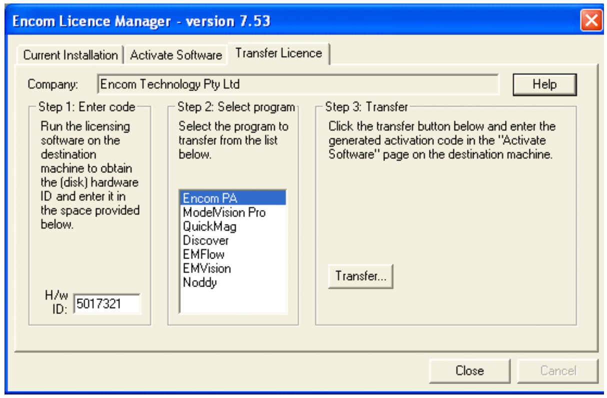
    The **Transfer Licence** tab is added to the dialog.



Encom Licence Manager on the source computer

Transfer the licence from the source computer

12. Select the **Transfer Licence** tab.



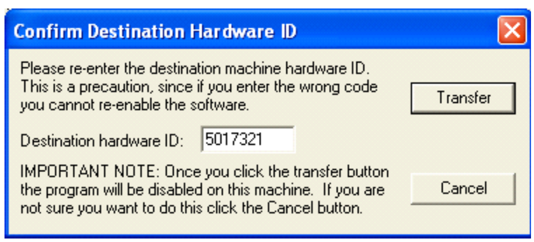
Transfer Licence tab on the source computer

13. Under **Step 1: Enter code**, in the **H/w ID** box, type the Hardware ID code from the destination computer.

14. Under **Step 2: Select program**, select **Encom Discover PA**.

15. Under **Step 3: Transfer**, click the **Transfer** button.

    A confirmation dialog is displayed.



Confirmation of the destination Hardware ID

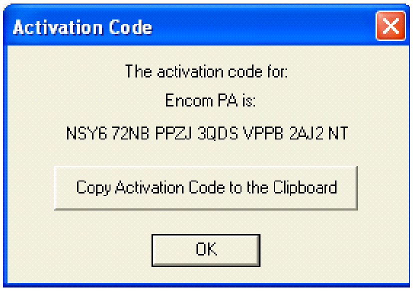
Important The licence transfer will disable the licence on the source machine. The source computer licence cannot be reactivated unless the licence is transferred back from the destination computer or a new Activation Code is obtained from Pitney Bowes Software.

16. In the **Destination hardware ID** box, type the Hardware ID that you obtained from the destination computer, and which you previously entered in step 13.

Important Take care that you enter the correct Hardware ID.

17. Click the **Transfer** button.

    The Activation Code for the destination computer is displayed.



Activation Code for the destination computer

Important The licence on the source computer is now disabled.

18. Click the **Copy Activation Code to the Clipboard** button and then paste the code into a Notepad or Word document, which you can either print or copy to the destination computer.

Activate the licence on the destination computer

19. If necessary, restart Encom Discover PA on the destination computer and display the Licence Manager as described is steps 2 through 4.

20. Select the **Activate Software** tab.

21. In the **Activation code** box, type or paste the activation code that you obtained from the source computer. Upper or lower case characters can be used, but no blank spaces.

22. Click the **Activate** button. The displayed modules, expiry date and version number are updated.

Confirm details

23. Select the **Current Installation** tab and confirm the licence details.

24. Click the **Close** button.

Encom Discover PA is now ready to use.

Displaying Licence Information

After you have installed and licensed the software, to display information about the Encom licences installed on the computer, from the **File** menu, choose **Licensing**. The Encom Licence Manager is displayed from which you can:

- Examine Encom applications installed on your computer
- Examine various modules, dates, and version settings that are licensed for your computer
- Install licensing for Encom Discover PA or other Encom applications
- Transfer an Encom application licence from one machine to another. When this is done, the operational licence on the source computer is disabled.

Troubleshooting

If you believe Encom Discover PA licensing and dongle (if used) are correctly installed but are unable to correctly operate Encom Discover PA, you should contact Pitney Bowes Software. Pitney Bowes Software will respond with a solution to your problem or with a request for further information.

4 Getting Started

In this section:

Introduction and Guidelines

Encom Discover PA is a simple but powerful program to operate and use for data visualisation and analysis. The following guidelines will assist you with the operation of the software.

Display the same data in multiple windows

Data can be displayed in a number of windows at any time. One of them (with the highlighted banner at the top) is current and active. To make a window current, move the cursor to it and click the left mouse button.

Use the Workspace Tree to select and manage objects

Selection and navigation of graphical objects and displays is best dealt with from the Workspace tree displayed by default on the left of the main window. Selecting the appropriate branch of the tree and clicking the right mouse button displays a pop-up menu with options relevant to the item selected. Additional information describing this can be found in *The Workspace Tree*.

Use the Properties dialogs to control the appearance of objects

The appearance of an object is controlled by a object property dialog, which can be displayed from the Workspace Tree or by double-clicking the object in any view. For information on using properties dialogs, see *Branch (Object) Properties*.

Choose between two graphical modes

Encom Discover PA operates graphical displays in one of two modes - on-screen (Normal) usage and print preview (Layout) mode. Either mode is interchangeable, but the Layout mode is usually used to design on-paper displays prior printing. The on-screen mode is used primarily for interactive interpretation and graphical access to your data. For more information, see *Controlling the Display*.

Use standard Windows keyboard shortcuts

Standard Microsoft Windows keyboard usage applies for dialog list controls. This means that the SHIFT key in combination with the left mouse button can be used to select multiple list items. The CTRL key can also be used in combination with the left mouse button when selecting non-consecutive items in a list.

Use the Data Manager to control which objects are displayed

A Data Manager tree provides easy visualisation of active data in the current session and is a good source for the drag-and-drop facility for addition to new and current window displays. The active data is grouped into seven Data Object folders with additional pop-up menus available from either the folders or the data files displayed as trees beneath these folders. Additional information describing this can be found in *Data Manager*.

Flexible data display

Encom Discover PA uses different display windows to present data, graphs and maps. These presentation windows may contain a number of graphical objects (such as profile traces, layers, legends, bitmaps, metafiles etc). You can mix different graphics objects with other objects in the same window for display purposes. Such objects can be made updateable in real time so a change in one object may affect another. For example, a line number change may be reflected in a linked profile). All displays have the same format which means that they can all be copied and pasted to any other. Each of the displays can be presented in either on-screen mode or layout mode for hard copy preview.

Use wizards to quickly perform complex tasks

The New Document Creation wizard simplifies the task of creating basic 2D and 3D displays of any data that you have established access to.

Starting and Closing Encom Discover PA

To start Encom Discover PA:

You can start Encom Discover PA in several ways:

1. Double-click the **Desktop Encom Discover PA** button. This icon is added to the Windows Desktop when the program is installed.

2. From the Windows **Start** button, choose **All Programs>Encom Programs>Encom Discover PA**.

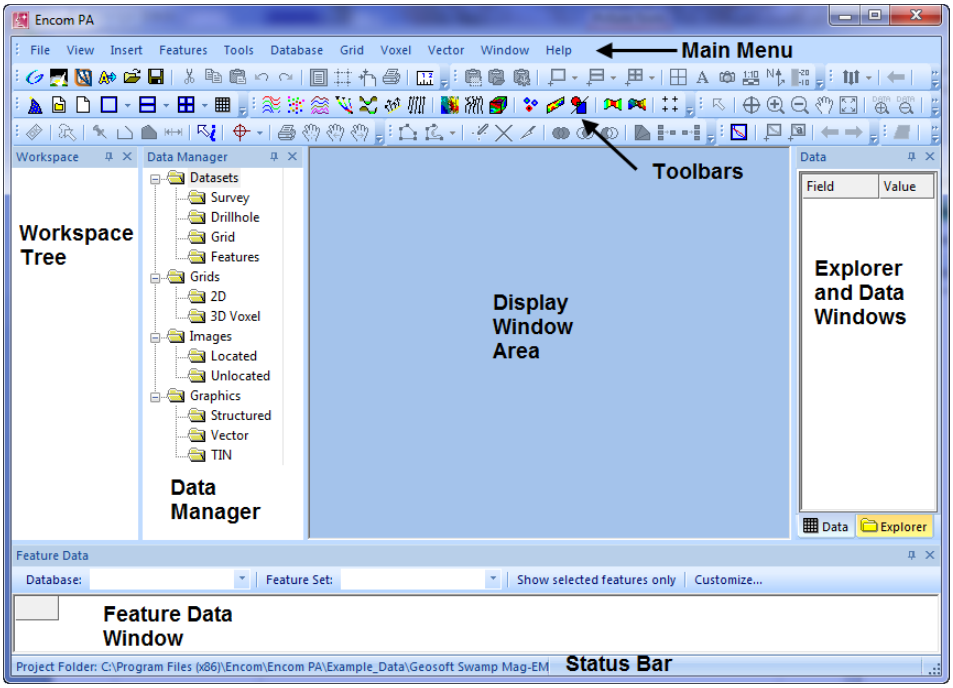
3. Other methods of creating shortcuts to executables are available in the Windows environment. Refer to Windows Help for additional information.

To exit Encom Discover PA:

- From the **File** menu, choose **Exit** (see *Main Menu*). Because projects and data contained in Encom Discover PA use database technology to store and retrieve information when you exit Encom Discover PA, any changes, processing or data alterations etc are automatically saved in the database.

Understanding the Interface

When Encom Discover PA starts, the main screen is displayed, which is divided into seven areas.



The main screen items of Encom Discover PA

- The *Main Menu* contains operational pull-down menu controls.
- *Toolbars* are available for controlling selected functions (e.g decay curves, features, etc.). These toolbars can be removed from view and restored using the Toolbars option in the View menu. Toolbars can be dragged into the window or docked at the perimeter of the main window. Tools on the toolbars may not always be available because their operation may depend on a specific task being performed.
- A *Display Window Area* that initially contains a grey background. This area is used to display windows and dialogs.
- A *Status Bar* at the base of the screen to indicate processing and display events, progress bar and other useful progress information.

- *The Workspace Tree* on the left of the display window. This toolbar provides a hierarchy view of all objects, data sources and controls of the various displays. Its contents alter when a dataset is accessed or a window is highlighted. The Workspace tree can be removed from view and restored using the option in the toolbars pop-up menu.

- The *Data Manager* window by default appears beneath the Workspace tree and is a tool to display and manage data objects open in a session file. The Data Manager can be removed from view and restored using the option in the toolbars pop-up menu.

- The *Explorer and Data Windows* may appear as a tabbed window or they can be placed or docked where convenient. The Data window displays the location, values, and field information when the cursor is located and moved in various display windows. The Explorer window provides a quick link to Windows Explorer for dragging and dropping data files. These windows can be removed from view and restored using the display tools on the toolbars.

Main Menu

The Main menu includes these general menus:

- *File Menu* – File handling for project sessions, display templates, and access to external databases and other file types are controlled using the options on this menu item. Packaging datasets, setting of licensing, printing and exiting are also available.

- *Edit Menu* – Standard Windows editing functions.

- *View Menu* – Options to display the various display types, toolbars and control dialogs, status bar and property dialogs.

- *Insert Menu* – Permits graphical objects and documents to be inserted into existing document displays.

- *Window Menu* – Control of the position and selection of windows.

- *Help Menu* – Online help and reference facility.

Plus a number of utility menus:

- **Features** – Controls the Feature module, options and Feature Spreadsheet. See *Features*.

- **Tools** – Provides options and access to editing default settings. Also contains some plug-in utilities that create various display objects from data opened inside Encom Discover PA. See *Display Utilities*.

- **Database**– Contains plug-in utilities that allow data to be processed (including line filtering) from within Encom Discover PA. See *Database Utilities*.

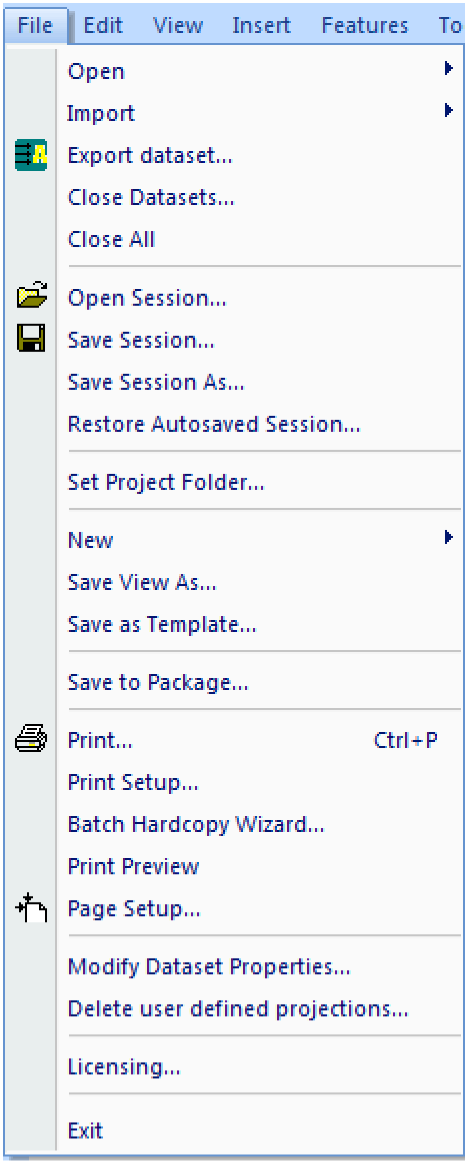
- **Grid** – Gridding line data and filtering gridded data plus other utilities for processing gridded data. See *Gridding Utilities*. The Geofilter module is also accessed from this menu through the*Grid Filter* option.

- **Voxel** – Options for 3D gridding and preparing data for 3D gridding. See *Voxel Utilities*.

- **Vector** – Options for converting and transforming 2D and 3D vector files. See *Vector Utilities*.

- **Display Utilities** – Provides access to plug-in utilities that create various display objects from data opened inside Encom Discover PA. See *Display Utilities*.

File Menu



File menu

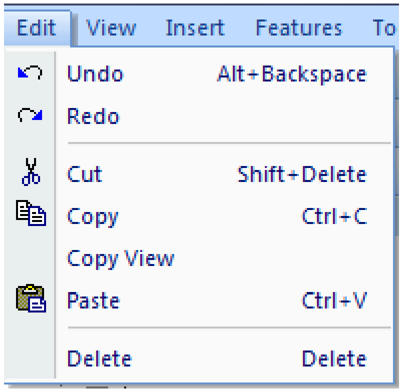
The options available from the File menu allow you to:

- Open and close point, line, and drillhole databases such as Intrepid, Geosoft montaj™, and MapInfo Discover (see *Working with Data*).

- Import ASCII data files, such as .TXT, .CSV, ASEG GDF, Crone, and SEG2 data (see *Importing ASCII Data Files*)
- Export Geosoft .GDB databases to ASCII files (see *Exporting Data*).
- Create, maintain and restore Encom Discover PA project session files (see *Managing Session Files*).
- Create new display documents of all formats or from templates (see *Page Layouts* and *Display Properties*).
- Create or restore templates to control the appearance of display windows (see *Templates*).
- Create a packaged dataset for archive or transmission to work associates (see *Packaging Datasets*).
- Save an image of a displayed window (see *Graphical Output*).
- Print, batch print, preview and access printer setup controls (see *Printing*).
- Restore the last few project session files (see *Session Files*).
- Modify the location data channels of a loaded dataset (*Working with Data*).
- Access the licensing management (see *Licensing Encom Discover PA*).
- Exit Encom Discover PA (see *Starting and Closing Encom Discover PA*).

These functions are described in the corresponding sections of this guide.

Edit Menu



Edit menu

The Edit menu is used to undo, copy, delete, paste and insert various graphical items in the displays. The Edit menu options include:

Undo

A limited number of undo steps are available in Encom Discover PA. The number of available steps is defined in the **Tools>Options** item. Note that for large, complex datasets and with a large number of undo steps, computer memory is used and performance may suffer.

Redo

Only enabled once an Undo operation has been performed, the redo option will reinstate the operation that was undone.

Cut

Copy a selected graphical item to the clipboard and remove it from display.

Copy

Copy a selected graphical item to the clipboard but do not remove it from display.

Paste

Insert a captured item from the clipboard and place it in the display at the location defined by the cursor.

Copy View

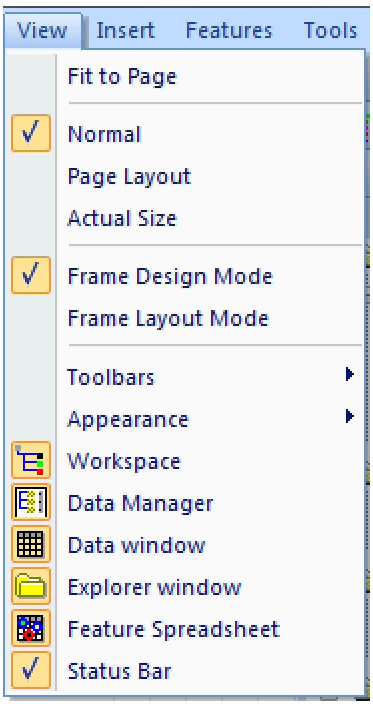
Selection of the **Edit>Copy View** option saves the contents of the active window to the Windows clipboard as a metafile. This capture can then be imported directly into other applications such as Microsoft Word or Excel etc.

Delete

Remove a graphical or selected item from the display or table entry. This is a permanent deletion and the item is then lost.

View Menu

The Encom Discover PA View option enables different data views and display control dialogs to be presented. The View menu also controls the placement and visibility of the floating Toolbars.



View menu

Fit to Page

Reshapes a display at the Frame level to fit the page format.

Normal and Page Layout

Applies only to individual and selected windows. Consequently, it is possible to have some windows in Normal display mode while others are in Page Layout mode. See *View Modes* for additional information.

Actual Size

The Actual Size option uses Page Layout Mode but zooms the display to a level on-screen that matches a hard copy print when scaling and range setting limits are specified.

Frame Design Mode

Two modes of display are available: Normal and Design. Design mode makes cell boundaries and linework visible. To view the final output, switch to Layout mode where final linework is not visible. To make this switch, use the **Grid Design** or **Page Layout** button.

Frame Layout Mode

This is the mode opposite to the Design mode (see above).

Appearance

Controls the appearance of the interface including windows, toolbars and menus based upon selected applications style.

Toolbars

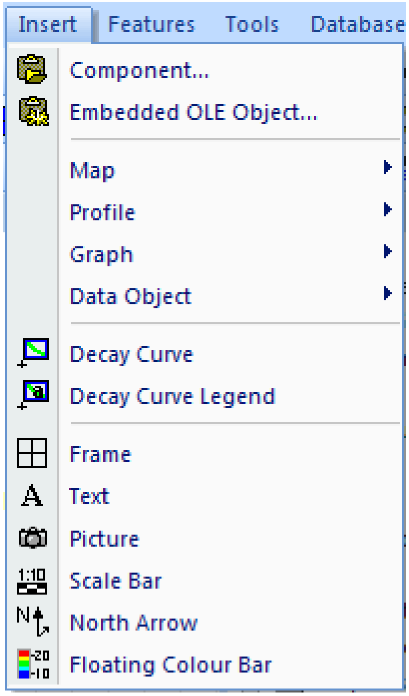
A variety of toolbars can be used in Encom Discover PA. For more information, see *Toolbars*.

Status Bar

Displays a range of status items relating to processing, data access, coordinates and data queries.

Insert Menu

The Insert menu provides you with options to add Map, Profile, and Graph display frames, OLE objects (for example, Microsoft Word documents), components (combinations of drawing objects), and data objects, such as text, bitmaps, and other graphics objects.



Insert menu

Features Menu

See *Features*.

Tools Menu

See *Display Utilities*.

Database Menu

See *Database Utilities*.

Grid Menu

See *Gridding Utilities*.

Voxel Menu

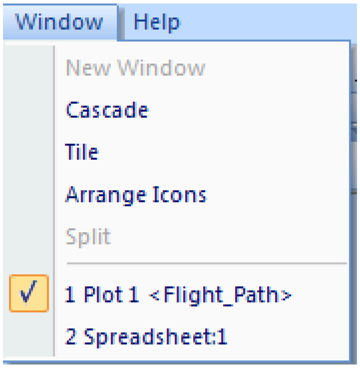
See *Voxel Utilities*.

Vector Menu

See *Vector Utilities*.

Window Menu

The Windows menu controls the various window displays that may be presented on the screen at any time. The menu appears as below:



Windows menu

Options contained in the Windows menu include:

New Window

Creates a new document window.

Cascade

Windows are arranged on the screen with uniform size and with overlap from the top left corner of the Encom Discover PA display area.

Tile

Windows are arranged with equal area and no overlap in the Encom Discover PA display area.

Arrange Icons

Arranges minimised window icons at the base of the Encom Discover PA display area.

Split

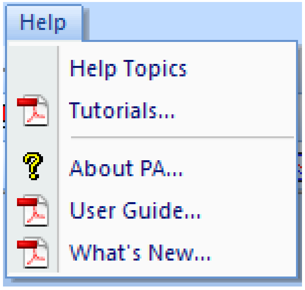
Enables a display such as a Spreadsheet to be split and resized.

Window list

Encom Discover PA displays a list of all the available windows, included those that have been minimised. You can use the cursor to point to one of these windows to activate it (bring to the front of overlapping displays). The name used in the Window list is that shown in the window's title bar.

Help Menu

The help option enables access to on-line help and information.



Help menu

Help Topics

Access on-line help. Refer to *Getting Help* for further detail.

About Encom Discover PA

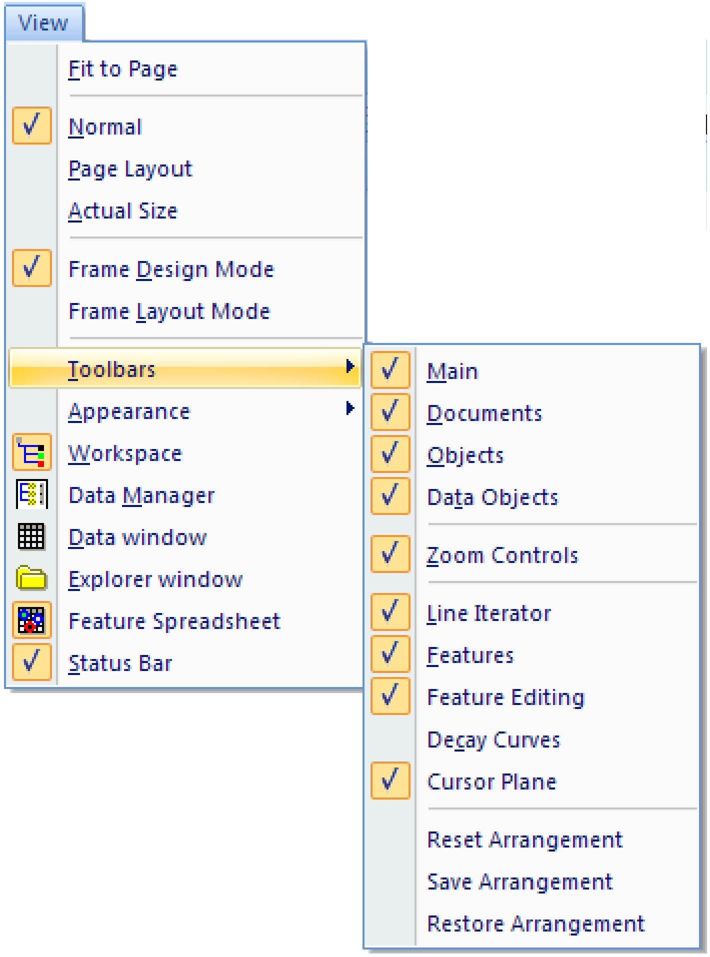
Displays developer, version and history information.

Other menu options providing access to electronic manuals and documentation are also provided.

Toolbars

Toolbars are a convenient way to utilise the functions available in Encom Discover PA. The buttons on a toolbar are grouped by similar or associated function. See *Available Tools* for descriptions of the function of the available toolbar buttons. For information on using toolbars, see *Using Toolbars*.

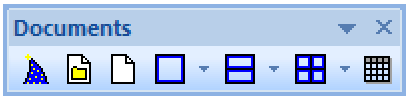
To hide and show toolbars, choose the toolbar from the **Toolbars** option on the **View** menu:



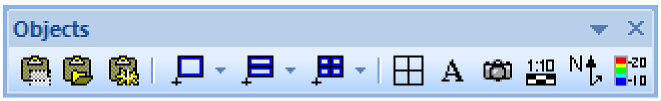
Toolbars on the View menu

Main toolbar – Used for data access, operational modes, printing, display, help and data selection.

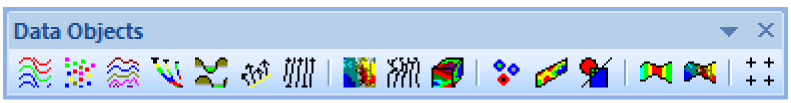
Documents toolbar is used for creation of initial displays including profiles, flight path maps, spreadsheets, graphs, decays and multiple grid presentations. Templates are also accessed from this toolbar. Buttons on this bar allow creation of 2D and 3D displays.



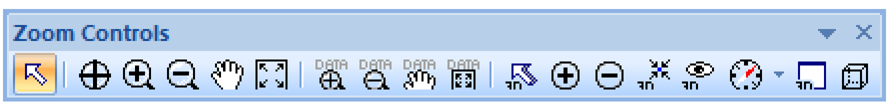
Objects toolbar enables the addition and control of graphical objects to a display window. This toolbar allows frames, maps, decay legends, bitmaps, OLE objects, text and components such as title blocks to be added.



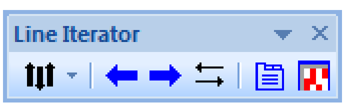
Data Objects toolbar provides single, push-button addition of the various display objects into the active window.



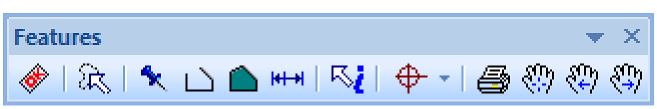
Zoom Controls toolbar provides access to window or real data zooming and panning control in both 2 and 3 dimensions.



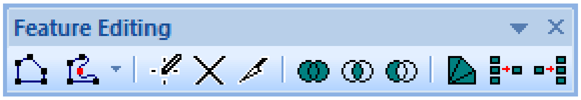
Line Iterator toolbar enables line navigation.



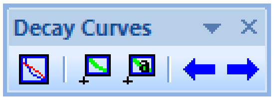
Features toolbar is used to select and assign properties to created features derived from interpretations.



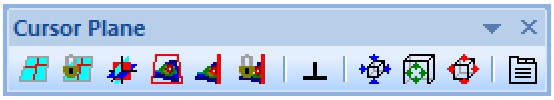
Feature Editing toolbar provides all the tools required for advanced editing for Feature Objects.



Decay Curves toolbar is used for displaying a display curve profile and reading the decay across time channels. It is therefore only fully active when a multi-banded, time-related EM dataset is displayed in a profile display and selected using the cursor in select mode.



Cursor Plane toolbar provides button controls for locating and creating 3D features in three dimensional displays. A 2D plane is used to constrain a cross-hair cursor that can accurately be located to determine a position in three dimensions. Additionally, the plane can be used to slice objects.



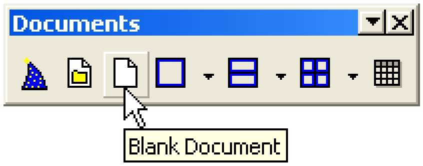
Toolbar Shortcut Menu

Other window components can be shown and hidden from the Toolbar shortcut menu, which is displayed when you right-click anywhere inside the main toolbar area of window:

- **Workspace** – This is the main object tree tool used to show the hierarchy of items being displayed.
- **Data Manager** –. Use this window to visualise what data files are open in the current session. Pop-up menu also allow additional operations for each file.
- **Data** – The data window will show data representing the display object that is selected by the cursor at the time.
- **Explorer** – This is a shortcut to Windows Explorer which can be used to drag and drop data files into the display windows.

Using Toolbars

To assist in understanding the function of each button, a message (called a Tool Tip) is displayed when the cursor is placed over a button for a short period of time. Descriptions of the available tools can also be found at *Available Tools*.



Toolbars can be moved so that they float in the screen or docked into the perimeter of the main window.

To move a toolbar:

- If the toolbar is docked, place the cursor over the three dots at the end (left or top, depending on the orientation) of the toolbar until the cursor changes to a four-way arrow. Then click and drag, holding down the left mouse button, until the toolbar is in the desired position, and then release the mouse button.

- If the toolbar is floating, position the cursor over the toolbar title bar. Then click and drag, holding down the left mouse button, until the toolbar is in the desired position or docked at the top of the window.

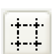
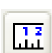
To save, restore, or reset your toolbar arrangement:

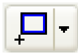
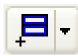
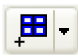
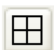
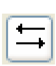
- From the **View** menu, under **Toolbars**, choose either **Reset Arrangement**, **Save Arrangement** or **Restore Arrangement**.

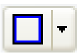
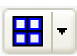
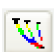
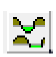
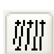
Available Tools

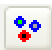
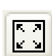
The tools available on the Encom Discover PA toolbars are described below:

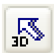
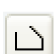
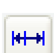
Tool	Description
	Open Geosoft database
	Open Intrepid database

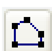
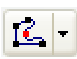
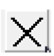
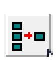
Tool	Description
	Open Discover drillhole database
	Import ASCII data – this will open the Import ASCII data dialog
	Open session
	Save session
	Cut the highlighted object
	Copy the highlighted object
	Paste a copied/cut object into the highlighted object
	Undo last action
	Redo last action
	Normal View/Page Layout View toggle
	Frame Design Mode/Layout Mode toggle
	Page Setup – allows specification of page size, orientation, etc.
	Print – Print a document
	Distance/Bearing Measurement tool
	Paste Into – this allows the insertion of a cut/copied object into a specified area of a document
	Insert Component into a document. This will open the Insert Component Preview dialog

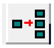
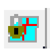
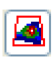
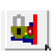
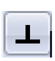
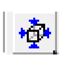
Tool	Description
	Insert an Object into a document, e.g. MS Word document, MS Excel spreadsheet.
	Wizard dialog to insert a Map Frame into a document. Use the black down arrow button to display menu options for quick insertion of a map display type.
	Wizard dialog to insert a Profile Frame into a document. Use the black down arrow button to display menu options for quick insertion of a profile display type.
	Wizard dialog to insert a Graph Frame into a document. Use the black down arrow button to display menu options for quick insertion of a Graph/Scattergram display type.
	Insert a Frame into a document
	Insert a Text Object into a document
	Insert a Picture into a document
	Insert a Scalebar into a document
	Insert a North Arrow into a document
	Insert a Floating Colour Bar into a document
	Line Iterator: Select an Active Line from a loaded database
	Line Iterator: Moves a Graph or Profile display backwards one line in a loaded database
	Line Iterator: Moves a Graph or Profile display forwards one line in a loaded database
	Line Iterator: Reverses the direction of the line shown in a Graph or Profile display
	Line Iterator: Displays the Line Iterator Properties dialog

Tool	Description
	Line Iterator: Displays the Line Iterator Line Report dialog
	New Document Wizard dialog
	Open from Template
	Open Blank Document
	Open new Map display wizard
	Open new Profile display wizard
	Open new Graph/Scattergram display wizard
	Open Spreadsheet display
	Add a curve to the current display (Profiles, Graphs and 3D displays only)
	Add a Points series to the current display (2D maps and 3D displays only)
	Add a Lines series to the current display (2D maps and 3D displays only)
	Add a Drillhole group to the current display
	Add Stacked Profile series to the current display
	Add a Flight Path series to the current display (2D maps and 3D displays only)
	Add a Directional Vector series to the current display (2D maps and 3D displays only)
	Add a Grid to the current display (2D maps and 3D displays only)

Tool	Description
	Add a Grid Contour to the current display (2D maps and 3D displays only)
	Add a Voxel Model to the current display (3D displays only)
	Add a Feature Set to the current display
	Add a Located Image to the current display
	Add a Vector file to the current display (2D maps and 3D displays only)
	Add a Section to the current display (Profile, Graphs and 3D displays only)
	Add a Section Grid to the current display (Profile displays only)
	Add an Axis (XY or XYZ) series to the current display
	Select an object
	Zoom Page in Place
	Zoom In to Page
	Zoom Out from Page
	Pan around the Page
	Fit window to page extents
	Data Zoom In
	Data Zoom Out

Tool	Description
	Pan around in the Data area (Profiles, Graphs and 2D maps only)
	Fit data to frame extents (Profiles, Graphs and 2D maps only)
	Navigate in 3D (3D displays only)
	3D Zoom In (3D displays only)
	3D Zoom Out (3D displays only)
	Reset 3D View (3D displays only)
	3D View Manager (3D displays only)
	Activate Fast View (3D displays only)
	Select the viewing direction for a 3D display from pre-set directions.
	Toggle between Perspective and Orthographic view (3D displays only)
	Confirm Feature Creation prompt
	Select feature(s) with polygonal frame
	Create a Point Feature
	Create a Polyline Feature
	Create a Polygonal Feature
	Create a 3-Point-Range Feature (Profile and Graph displays only)

Tool	Description
	Feature Information spreadsheet
	Activate SnapTo when digitising feature objects
	Batch print feature objects
	Centre view on selected feature object
	Select previous feature object and centre view on it
	Select next feature object and centre view on it
	Toggle Reshape feature mode
	Activate Elasticity for when editing feature objects
	Add mode
	Delete mode
	Break mode
	Combine selected feature objects
	Intersect the selected features
	Cut selected features from the target
	Triangulate - converts a selected polygon to a surface or ensures a selected surface contains only triangular faces
	Aggregate - group multiple selected features into a single feature and preserve all

Tool	Description
	Disaggregate - ungroup aggregate features into individual features
	Open a new Decay Curve display
	Insert a Decay Curve display into a document
	Insert a Decay Curve Legend into a document
	Move Decay Curve backwards to previous line
	Move Decay Curve forwards to next line
	Display Cursor Plane (3D displays only)
	Lock the cursor plane at current orientation while editing feature (3D displays only)
	Change the Orientation of the Cursor Plane (3D displays only)
	Bond cursor plane to data or image (3D displays only)
	Clip a 3D display with the Cursor Plane (3D displays only)
	Clip the display with current cursor plane orientation and move the cursor plane (3D displays only)
	Orient the view to be perpendicular to the Cursor plane
	Shrink the size of the 3D Focus box (3D displays only)
	Enlarge the size of the 3D Focus box (3D displays only)
	Fit the Focus box to the 3D display extents

Tool	Description
	Display the Cursor Plane Properties dialog

Display Window Area

The display area is the main screen for graphic presentations. The area is also used for dialogs, spreadsheets and layouts containing print preview displays. Descriptions of each of the display formats are provided in the Graph and Map Display sections.

Status Bar

The Status Bar is located at the base of the Encom Discover PA window. A range of items are displayed in the status bar including:

- Messages to inform you of processing or data access
- Progress bars to advise of data processing status
- Information relating to the readiness state of Encom Discover PA

On the right end of the bar are specific information areas that display the setting of particular functions (for example, modelling enabled etc) or location information.

The Workspace Tree

The Workspace tree is the main means of control used in Encom Discover PA. It operates by listing all entities used in a display. These entities may be graphical objects, profiles, axes, annotations etc. Each object displayed is represented by a branch on the Workspace tree. The relationships between the branched objects are shown in the Workspace tree as a hierarchy.

For example, in the simplest case, a document may contain a frame of one or more profiles. Each profile uses an X axis for its distance plotting (using the ProjectedDistance channel). The vertical scaling of data is controlled by a Y axis plus a Y axis control for annotation. The profile may also have an associated vertical colour bar (called a Colour-Axis). The data source is controlled by a Curve (of which there may be several in a banded dataset). Each data field can also have its own colour, symbol and rotation controls presented under the Curve branch in the hierarchy shown.

From this simple illustration you can see there can be a complex interaction of the components of a simple profile. Encom Discover PA permits these and many other complex interactions. The Workspace tree displays these interactions and provides controls for each of them.

The structure of the example is logical and provides immediate access to every branch of the plot for property control.

The hierarchical structure of the tree has distinct advantages over more conventional graphing techniques:

- Each item can be selected for property control with no ambiguity.
- Operations such as Cut, Paste, Delete and Clone are made available by selection.
- Even sophisticated profiling relationships are no more complex than those shown above.
- Data object branches of the tree can be dragged to another position in the tree.

Other displays such as maps, images, 3D plots etc. can all be positioned as a hierarchy in a display. In each case the relationship is presented in the workspace.

For more information about using the Workspace tree, see:

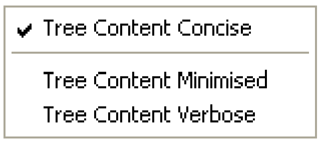
- *Controlling the Tree Content*
- *Hiding and Showing Branches*
- *Renaming Branches*
- *Expanding and Collapsing Branches*
- *Branch Controls*
- *Working with the Workspace Tree*
- *Branch (Object) Properties*

Controlling the Tree Content

The Workspace tree grows incrementally as various objects and their branches are added or changed. Because the size of the tree is limited by screen space, and complex designs can entirely fill the available area, you may need to adjust the window display to find and view the details you need:

- Use the scrollbars on the right and bottom margins that appear when the entries become too long to fit horizontally or there are too many entries to fit vertically.

- Change the Tree Content setting to expand or collapse the various elements of the tree. To change the Tree Content setting, position the cursor in a vacant area of the window and right-click to display the Tree Content shortcut menu.



Each option reveals a different level of detail in the tree. The options are:

- **Concise** – Only a minimum number of branches are shown. For most operations this is adequate.

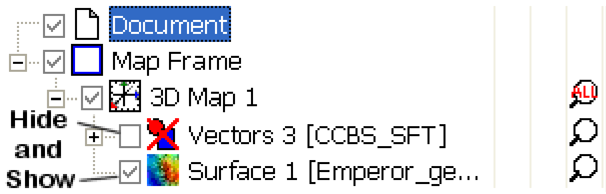
- **Minimised** – An increased number of branches are shown but not individual channels or rarely accessed items.

- **Verbose** – All branched items used in the display are shown.

However, the lowest branch of the tree (the individual data fields or bands) is not affected by this setting and is controlled only by the expand ( ) or collapse ( ) button on the branch.

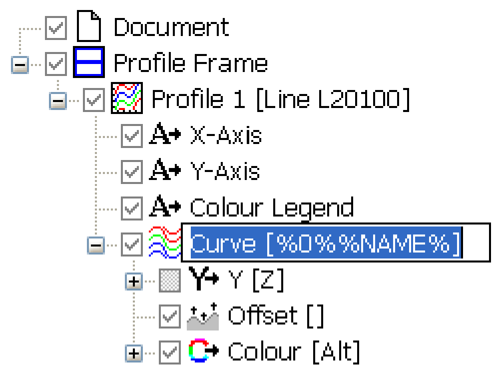
Hiding and Showing Branches

Beside each displayed object branch is a check box. Click this box to hide (no tick) and show (ticked) this object branch. Hiding a branch will also hide all dependent sub-branches.



Renaming Branches

The branches of the Workspace tree can be renamed. This can be helpful in complex tree designs. To rename a branch, position the cursor over the branch label, click once and the text appears selected. You can then type in a new label or edit the current label. Click outside the label box to save your changes.

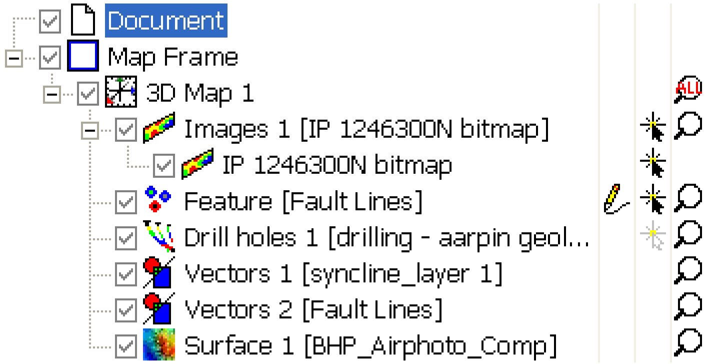


Expanding and Collapsing Branches

To the left of each subordinate branch is a small button, which appears as either ⊞ or ⊟. These boxes indicate that lower levels exist in the branch. By clicking on these boxes you can expand or collapse the sub-branches.

Branch Controls

To the right of many branches are three controls.



Editability – enables objects to be added or edited in the selected *feature database*. Only one Feature database can be editable at a time. Making a feature database editable will automatically turn on the *Cursor Plane*, ready for feature object digitization.

Selectability – enables objects within the selected dataset to be selected. For instance, when multiple image branches are open, this is a useful way of limiting which branches (or images) the Cursor Plane can be bonded to. Selectability can also be used to control which *feature databases* can be interrogated when multiple complex databases are open.

Browsable – attribute information for a Selectable dataset can be dynamically viewed in the Data tab of the Data tab. This only applies to datasets such as those used for 3D Points, 3D Lines, and Drillholes.

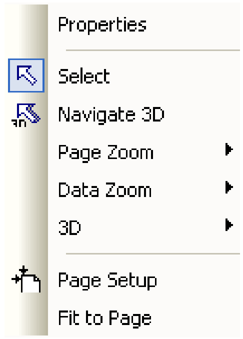
Zoom Extents – zoom the view to the extents of the selected data object, or

Zoom All – zoom the data view to the extents of all data within a frame.

Working with the Workspace Tree

A wide range of functions and tasks are performed from the Workspace tree. The functions available from shortcut menus, which are displayed by right-clicking different types of branches.

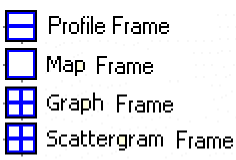
Right-click on | **To display the shortcut menu** | **For this branch type**

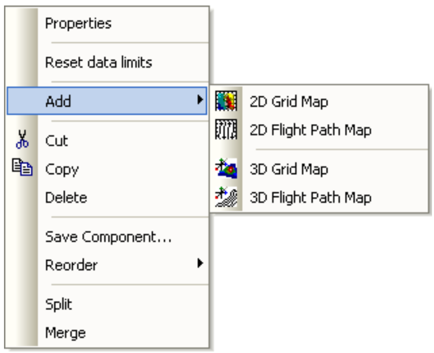


Document Branch

Apply some of the commonly used cursor controls such as Pointer (2 and 3D), Page and Data Zoom options (see *Toolbars*). Access to the Properties of the display is also provided.

Specify the Page Setup for a specific printer or plotter. The Default Layout option reverts a display to the current Page Setup setting.

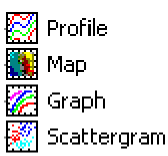


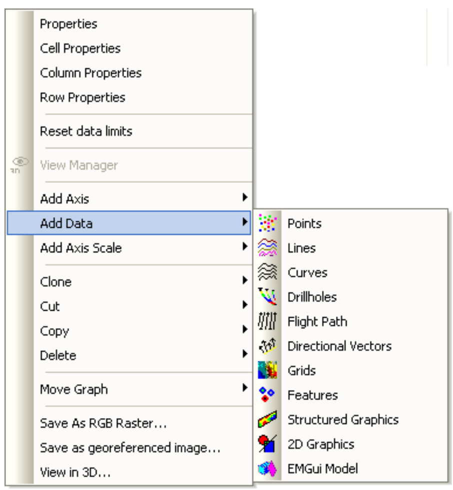


Frame Branch

Add a new display to the document frame (relevant to that Frame type). You can Cut, Copy and Delete or Save the objects as a Component.

If cell edge selection is used, the cell can be Split or the adjacent cells can be Merged.





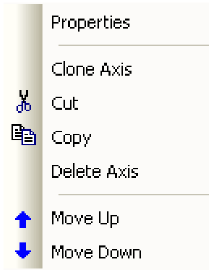
Data Object Branch

Properties, adding axes and/or additional data objects (profiles, sections, images, models, vectors and features) or axes. If features are to be displayed with these objects, they must be added as an additional object type.

Cut, Copy and Delete selected items or make a Clone copy of a selected object. An object can be positioned Up, Down, Left or Right and Forward/Back if overlapping. Displays contained in a window can be saved as a raster.

Right-click on	To display the shortcut menu	For this branch type

X-Axis
All axes relating to X, Y, Colour and Size axes (see Axes)

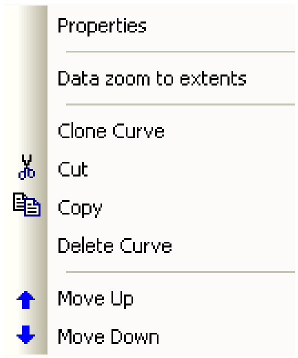


Axis Branches

This menu deals with the labels, positions and other properties related to axes and their annotations. Display the Property dialog of the axis. Make a Clone copy, Cut, Copy or Delete annotation.

You can position the annotation axis up or down if required and hide or show it.

Curve
Section
Relates to all Data Objects except the Map Document.



Data Source Branch

This menu relates to the data being presented in the data object. You can display the Property dialog and Copy, Cut, Delete or Clone the data object.

You can position the annotation axis up or down if required and hide or show it.

X
Y
Colour
Size
Angle
Label
Reference
Upper
Lower
Colour

Field/Band/Layer Branch

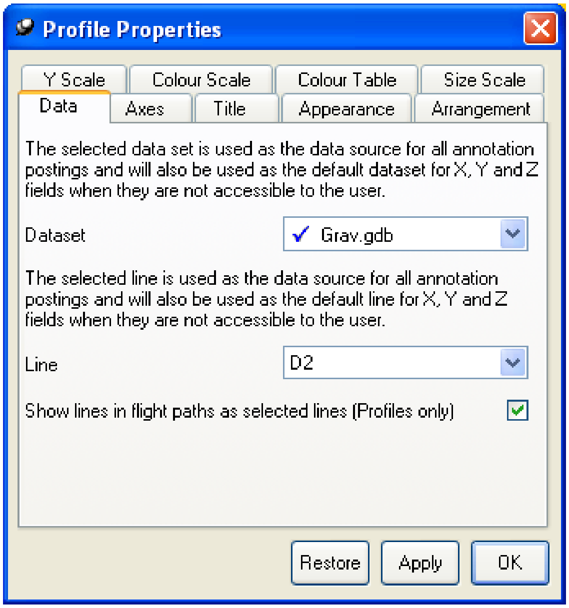
Display the Property dialog. Select All bands for displaying or scaling. Delete All bands from display.

Branch (Object) Properties

The properties of every object are controlled from the Workspace tree. To display the properties of a branch either:

- Right-click the branch and choose **Properties** from the shortcut menu, or
- Double-click the branch.

In most cases, the Properties dialog contains several tabs, and each tab controls a group of related properties.



The Profile Properties dialog and assorted tab controls

You can keep the Properties dialog open by clicking the pin symbol on the title bar. The dialog will update automatically with each object selection.

Changes to properties are not made until you click **Apply** or **OK**. If you make a mistake when entering changes, select the **Restore** button to return the control settings to their previous state. Clicking the **OK** button also dismisses the dialog.

For information on how to use the properties dialogs, refer to the relevant topic in this guide.

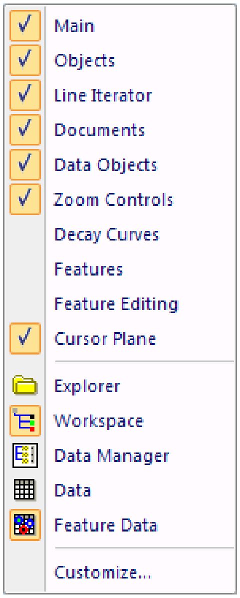
Data Manager

The Data Manager is an optional window, similar to the Workspace Tree, that can can be toggled on and off, moved, and docked. The purpose of the Data Manager is to control the visualization of data files loaded into a session of Encom Discover PA.

- *Hiding and Showing the Data Manager*
- *Information Displayed by the Data Manager*
- *Adding Objects to Displays*

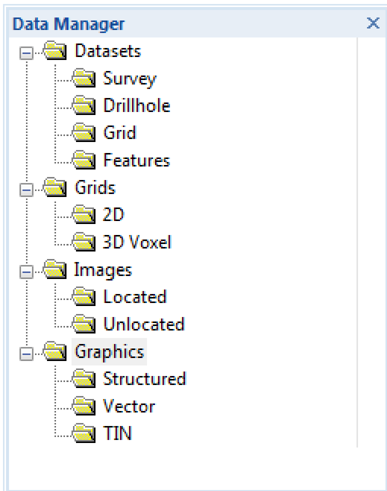
Hiding and Showing the Data Manager

If the Data Manager window is not visible when Encom Discover PA starts, right-click in the toolbar area below the Main menu and select it from the Toolbar shortcut menu.



Information Displayed by the Data Manager

When a new session is first opened in Encom Discover PA, the Data Manager will display seven empty folders, one for each of the object types that can be loaded in Encom Discover PA:

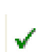


An empty Data Manager window

When a file for one of these object types is loaded, the file name is listed below the object folder for that category. A Feature Database will be displayed under the Features folder, located under the Datasets folder, and any associated Feature Sets for this database will be displayed as branches beneath the feature database entry.

The Data Manager also has shortcut menus options for each folder, from where you can open and close data files (instead of using the File menu), and display information about the current files:

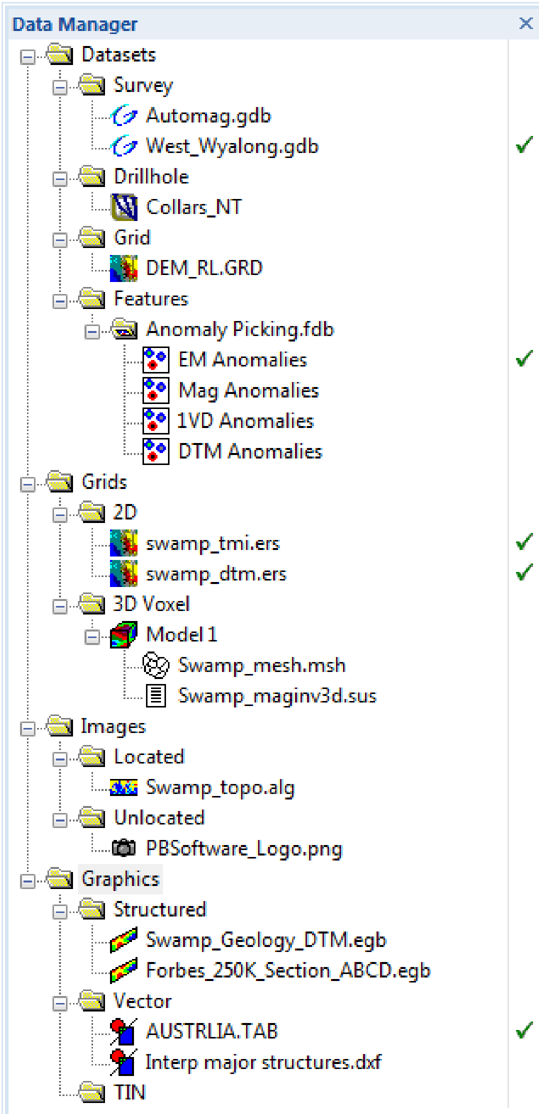
View Reference Count – If this option is enabled then a number is displayed in a column located to the right of a loaded file. This indicates the number of times this file is being used in all display windows of the current session.



View In-use Flag - If this option is enabled then a green tick located to the right of a loaded file indicates that the file is in use in a display window.

View Long Names - This option will show the file location as well as the file name in the Data Manager window.

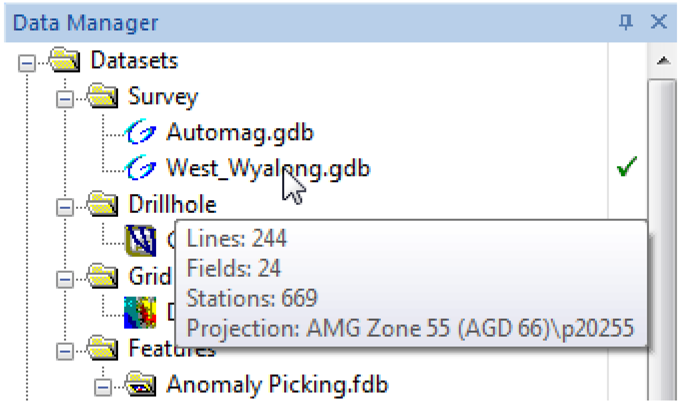
View Short Names and By default this option is enabled. This will show only the object file name in the Data Manager window.



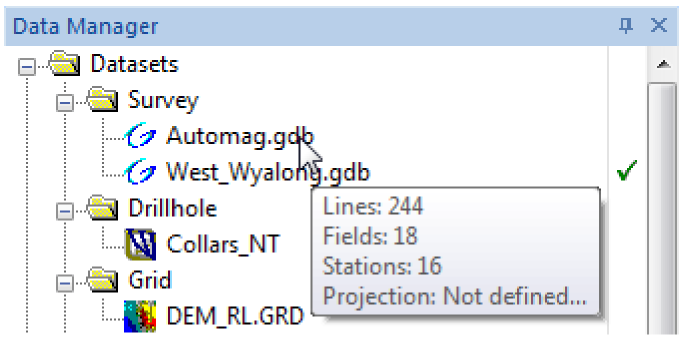
Example of a populated Data Manager window

Coordinate System Information

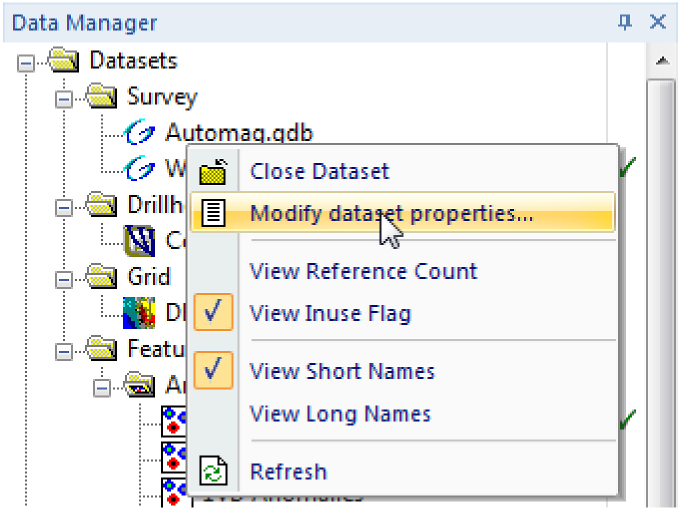
Once a data file is open in Encom Discover PA and is visible in the Data Manager window, it is possible to view the coordinate system associated with the data by hovering the mouse over the title in the Data Manager window and a temporary window will appear showing this information."



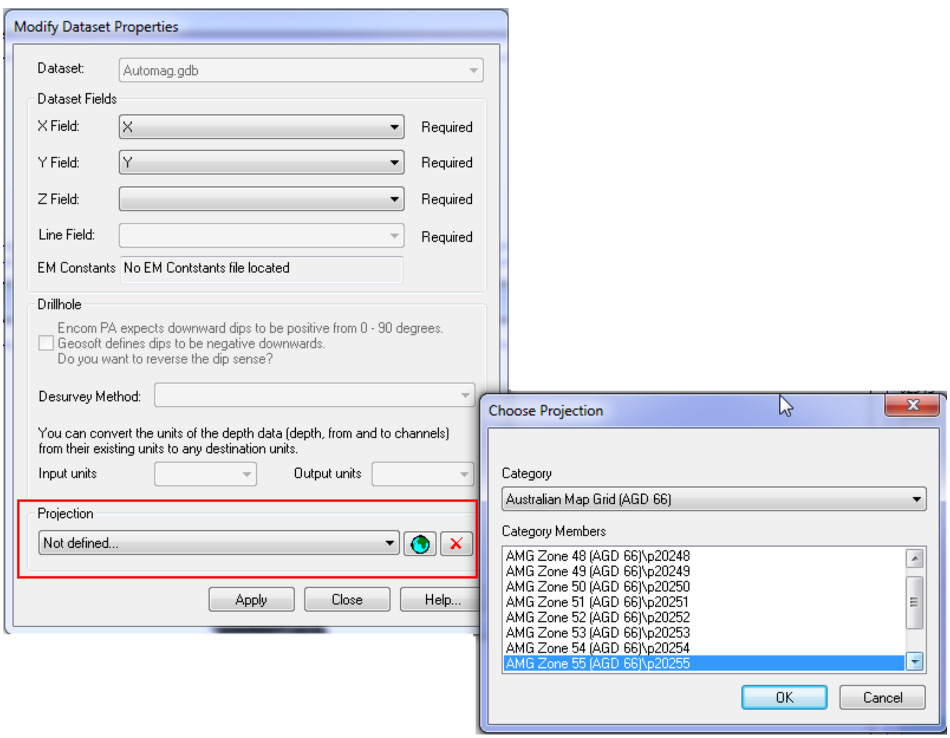
If no coordinate system is associated with the data file, "Projection: Not defined..." will be displayed.



To associate a coordinate system with the data file, right-click on the file name in the Data Manager and select **Modify dataset properties** from the pop-up menu that appears.



In the **Modify Dataset Properties** dialog press the **Choose Projection** button in the **Projection** section and nominate a **Category** and **Category Member** for the Coordinate System. Press **OK** and then **Apply**.



Alternatively if the coordinate system has already been used within Encom Discover PA, select this from the drop-down list in the Modify Dataset Properties dialog.

Check the Data Manager again to ensure that the coordinate system definition has been updated.

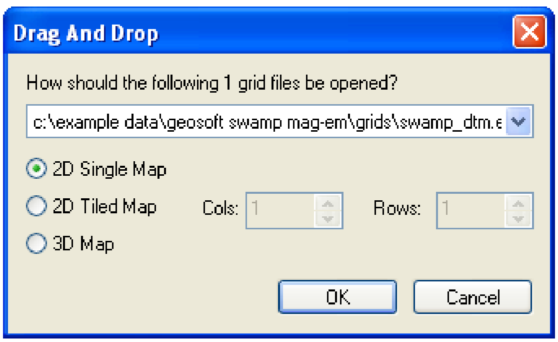
Adding Objects to Displays

You can drag-and-drop objects from the Data Manager into a new or existing display window.

To display objects in a new frame:

1. In the Data Manager, select the files you want to display, and drag it into the grey display window area. Multiple files can be drag-and-dropped in one action using the Ctrl key.

2. From the **Drag and Drop** dialog, select a display window type.



Drag and Drop prompt for loading a data object into a new display window

To add objects to an existing frame:

- In the Data Manager, select the files you want to add to a display, and drag into the display frame.

Explorer and Data Windows

The Explorer and Data windows can be positioned and docked where convenient. By default, these windows are docked to the right margin.

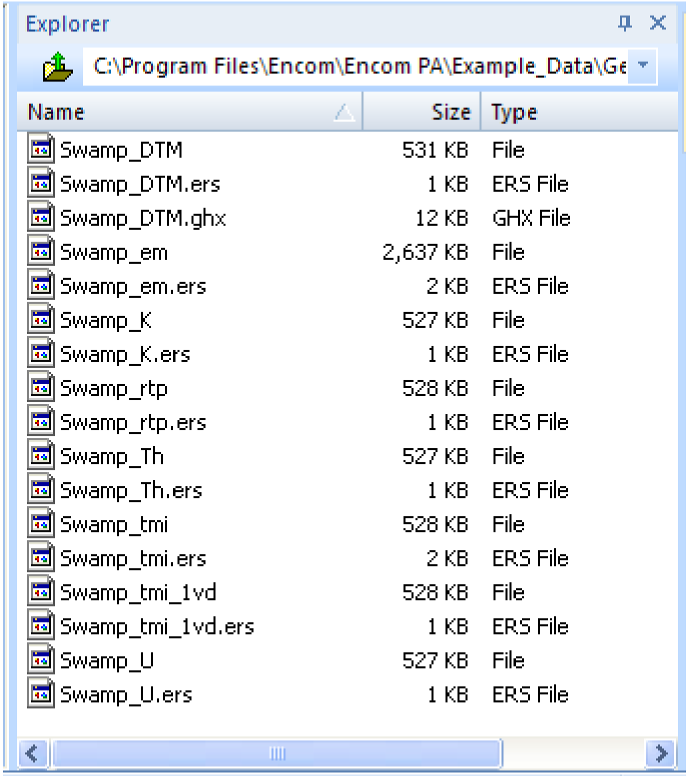
- *Explorer Window* – provides a quick link to Windows Explorer for the drag-and-drop operation of data files.

- *Data Window* – displays data location, values and field information when a cursor is located and moved in various display windows

Each of the window tabs can be removed from view and restored using the Data or Explorer options in the Toolbars shortcut menu.

Explorer Window

The Explorer window operates similarly to Microsoft Explorer and allows you to access, list and report on files within any of the drives available to the computer. You can use the Explorer in the same manner as Microsoft Explorer such that the pop-up menu (from a right mouse click) can be used to alter the view, the folder, paste/copy and examine file properties.



Explorer tab used for navigating and accessing files

The Explorer view also saves its last 10 user selected directories and reloads these into the address bar on the next run of the application. The Explorer view initialises to the last used directory from this saved list.

Warning When the Explorer tab is used, it interrogates all available drives to the computer. This will include mapped network drives, if connected, and as a consequence, it may slow down Encom Discover PA operation.

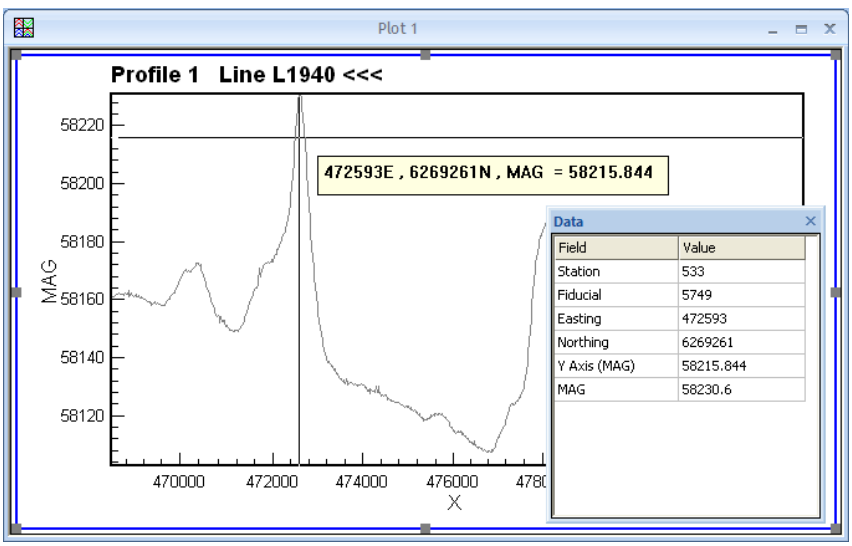
To prevent opening of the Explorer tab at startup:

1. From the **Tools** menu, choose **Options**.
2. Select the **View** tab.
3. Clear the **Create Explorer View at startup** option.

Data Window

The Information window Data tab can be used to interrogate data values displayed in profiles, graphs and maps. With one of these displays present, by placing the cursor in the window, the relevant data values for the objects are shown in the Data tab. As the cursor is moved within the display window, the data values update. This feature can be used in multi-banded data such as EM or multi-tracked data such as elevation, altimeter and topographic profiles.

An example of using the Data tab is shown below:



Data displayed in the Data tab from the cursor located in a multi-band profile

Managing Windows

In the top right corner of the Workspace Tree, Data Manager, Explorer, and Data windows there are options for controlling the visibility.

The **Pin** button allows the user to auto hide the window when it is not in use. When auto-hidden the window can be accessed from the vertical bar to the right of the PA interface.

The **Close** button will remove the window from view. To display again the option needs to be accessed from the pop-up menu by right mouse clicking on the main toolbar area.

Controlling the Display

- *View Modes*
- *Page and Data Zoom Modes*
- *Data Zooming, Panning and Frame Data Linking*
- *Map and Profile Zoom Relationships*

View Modes

Encom Discover PA display windows (whether profiles, maps, spreadsheets or decay plots) vary in appearance depending on the View mode. Two view modes make efficient use of available screen space and account for the fact that the requirements of on-screen interpretational work are frequently different to those of printing. The two modes are:

Normal View Mode – This is the default viewing mode. In normal mode, Encom Discover PA displays profiles, maps and other objects using all available area of the display window and provides the most efficient view of data.

Layout Mode – In Layout mode you can see how objects are positioned on the printed page. This mode partially replicates a paper view although a more accurate preview before printing is achieved from the **File>Print Preview** option. This view is also useful for editing borders, margins and the placement of objects prior to printing. The size and orientation of the page is defined by printer settings specified in the **File>Print Setup** option (see *Printing*).

You may switch between Normal and Layout views quickly by using the **Page Layout** button or from the Window menu option.

Note To see a plot as it will be printed use **File >Print Preview**.

Page and Data Zoom Modes

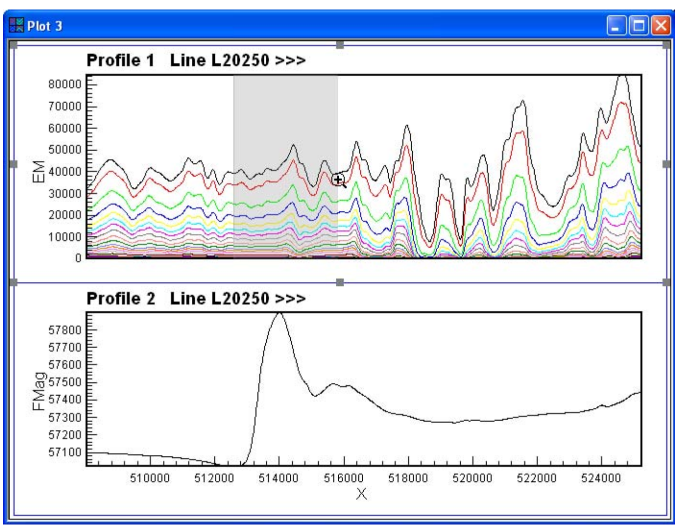
As well as two viewing modes, two data display modes are also available. These modes dictate the data content displayed in the various document windows. The two modes are:

Page Mode – This mode uses the **button group** and controls zooming out/in, panning and fitting data in normal or default operation. For a description of each button's operation, see *Available Tools*.

Page mode zooming increases the zoomed area as if magnifying the paper on which it is presented. This means that text, data, line thickness etc are all increased in size.

Data Mode – This mode controls the data content displayed in a window, rather than the view. The data content can be zoomed, roamed and fitted similarly to Page Mode, but the content is displayed with axes etc for relative assessment. The Data Mode uses the set of buttons.

When zooming in Data mode, a greyed rectangle over the zoomed area is drawn to provide an indication of the area to be zoomed. Resultant zooming does not affect the size of annotation or line thicknesses but acts only to increase the level of data seen in a display. Smooth scrolling of data in profiles is available to allow panning and roaming of data, especially in profiles (see below).



Smooth scrolling of profiles and maps uses the third button or depressed scroll wheel of recent mouse device types. To initiate the smooth scrolling of a map or profile, zoom to a portion of the profile or map window using the **Data Zoom button**.

With the profile in Data Pan mode (click button), press the third button (or **scroll wheel**) on the mouse and move the cursor horizontally in the window. The map or profile then moves smoothly left or right.

You may notice that as a profile scrolls, it automatically adjusts the vertical scaling to accommodate the changing data range. To have fixed scaling, display the properties of the Profile dialog, select the Y Axis Properties Tab and set the **Y Axis Range and Scaling** to be Fixed. The velocity of panning is proportional to the distance from the initial mouse placement.

A third group of buttons available from the Zoom Controls toolbar are used exclusively for 3D window navigation. The buttons are described in *3D Toolbar Controls*.

Data Zooming, Panning and Frame Data Linking

Encom Discover PA distinguishes between two types of Zoom and Pan operations.

- **Page Zoom** – This modifies your view of a displayed document. Page Zoom enlarges annotation text, axes and the paper view of a window.
- **Data Zoom** – This form of zoom allows you to zoom and pan the data range within a plot object. A defined Data Zoom area is used to fill the available display window. Annotation text etc adjusts its data coverage values but not its font size or style.

When you Data Zoom in a plot object, it is likely that other plot objects in the frame will also respond and modify their data range. This happens because the axes of frames can be linked using a global setting. In a map, the X and Y ranges are usually global. The frame linking facility can be disabled by setting the data ranges to be not global if required.

The mechanism to keep data ranges linked from one frame to another can operate in the same, or different, documents. A typical operation is to have map frames data linked or to have a profile frame linked to a map data range. In the latter case, when you Data Zoom or pan in a map, it broadcasts its data range to the profile which then zooms to show only that portion of the profile that is within the map data range. If a profile data range is completely outside the data range of the map then the profile may displays no content.

To enable this option, ensure that the frame you wish to use as the data range source is a Broadcaster and that the frame you wish to have respond to these data range changes is set to Listener. Furthermore, you need to specify whether the listening frame should respond to broadcasting frames in the same document only, or in other documents, or both. To initiate the first link you need to modify the data range in the broadcasting frame.

These settings are made from the Options tab of the frame properties. Additional information on this topic is available in *Map and Profile Zoom Relationships*.

Map and Profile Zoom Relationships

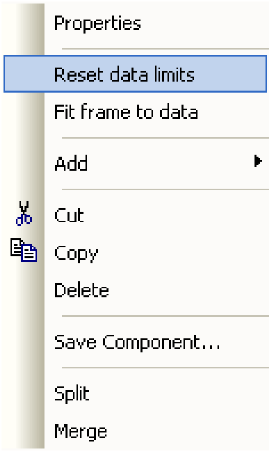
Frequently data is zoomed and panned in maps and profiles. In particular, the flight path map is a powerful means of controlling which data lines are selected and displayed in accompanying profiles. In these circumstances, if a map is zoomed in so that only a small portion of the overall line length is viewable, it is desirable that only the same length of (zoomed) profile is displayed.

This relationship of having the zoom level in a map controlling what is displayed in a profile is a powerful feature. Encom Discover PA also updates a profile display in such a way that if the controlling map is panned or re-zoomed, the profile coverage is automatically updated.

To achieve the linked zooming relationship between a map and a profile, select the Map Frame branch of the Workspace tree and display the **Options** tab. From this tab, the map must Broadcast its globally zoomed limits to other responding windows. Consequently, from the zoom control window, enable the **Broadcast** option. From the Frame level of the responding window’s property dialog, again select the **Options** tab and enable **Respond to Global** in both X and Y.

When you zoom or pan using the Data mode (only), the portion of the line displayed in the map matches that displayed in the profile.

Reset Data Limits



From the Frame and Object (Profile, Map, Graph and Scattergram) branches of the Workspace tree, a right mouse click is used to display the controlling pop-up menu. This menu contains an option called **Reset Data Limits**. When selected, this action re-assigns the data range in all subordinate graphs or axes and redraws. The result is a data zoom to extents that resets all axis restrictions.

Data Zoom to Extents

At certain branches of the Workspace tree, a variation of the above option is provided in the context menu. This item **Data Zoom to Extents**, causes any zoomed view of a map to be displayed as a Zoom Fit. This option operates identically to the **Data Zoom Fit** button.

Accessing Data

Encom Discover PA is designed to access data directly from industry standard databases. Unlike other geophysical packages, Encom Discover PA can use existing import and raw data reading facilities of external packages to provide processing and access to the data. Encom Discover PA also has the capability of reading data from external, non-database files (such as located data and ASCII files of various formats). It does this by external processing and import modules supplied with the software which read the specified data and then place it into a database format.

The use of databases and the ability to use the import capabilities of other software has a number of advantages:

- A wide variety of data formats, data sources and file formats are catered for by other packages and are not required to be supported by Encom Discover PA.

- The manipulation and editing of data obtained from a database provides greater power and increased flexibility.

- Reading of data from a binary database is much faster than from raw (text) data files and does not have the overhead of checking record or data validity.

- Navigation of data from one point (or traverse line) to another can take advantage of search and indexing techniques to increase speed and simplicity.

Encom Discover PA is designed to act as an interface between external data and databases (containing data), to display and provide interpreter interaction (through accessory processing or interpretation modules). The data links are transparent to the user as Encom Discover PA detects the type of database present in a folder and builds the required links automatically.

When Encom Discover PA opens a database, editing or changing the attributes of the data entries is not currently possible. However, external processes such as batch procedures, can create and add new data to the database. Additional utility modules allow you to rename, delete or merge data into array fields.

After starting Encom Discover PA, to access a valid database, you initially need to know where and what type of database is to be accessed. The location is the path, in a root folder, and the type is either:

- Geosoft Oasis montaj™

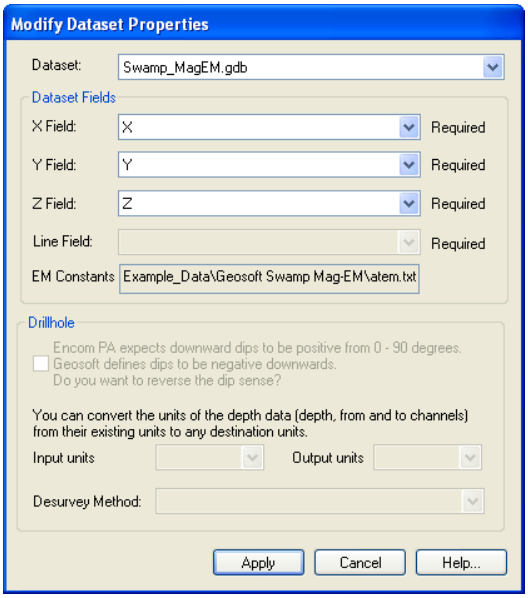
- Intrepid

- MapInfo Professional

Note ASCII data can be loaded directly into Encom Discover PA. This operates by transferring the ASCII data into a Geosoft database and then directly loading the database (see *Importing ASCII Data Files*).

Defining the Z-Channel

Since Encom Discover PA can display data in three dimensions, it uses a specific data channel (termed Z) for the third dimension. If you load a database that does not have a Z-data channel, the program asks if you wish to define a Z-channel. If you are unlikely to use 3-dimensional displays, you may opt not to select a Z-channel. The dialog displayed below makes this clear.



Dialog to nominate an X, Y or Z channel

Note If you select the **OK** button without defining a Z data field, Encom Discover PA can open the dataset, but no three dimensional display is available.

Project Directory Specification

Encom Discover PA can use one of two concepts to access data files. These are:

- Specify a project directory to assist with navigation and access to data. Typically, when working on a project, a specific folder is created and various data components created or used for that project are placed in sub-folders beneath the main, project directory.

The menu command **File>Set Project Directory** displays a dialog allowing you to specify a folder location. If using this method of data navigation it is recommended that you specify the project root directory location rather than a sub-folder of the project. Note that when a session (of a project) is restored, the path of the session is assumed to be the Project Directory. The Project Directory as specified is displayed in the Status Bar.

The Project Directory is also used when project specific files are created. For example, if you create a colour look-up table or legends which relate only to that project, they can be stored into the project folder and be available only while the Project Directory setting is set to that path.

- A second useful facility for file navigation is the concept of a Global Directory. This setting (set from the **Tools>Options>File Locations** tab) stores the last accessed path when any data type is saved or loaded. This means that if you for example, navigate to a project grid file, when you next again search for a grid, the software path is initially returned to the same location as the previously accessed grid. This approach is data type specific so a path for a grid will be different for a database etc. The **Use Global Directory for all file location dialogs** option enables this location for all data object that are opened and is activated by default.

Specifying which of the two data navigation methods is used is determined by a setting on the **Tools>Options>File Location** tab. Set the desired method by enabling the option as required.

Working with Graphical Objects

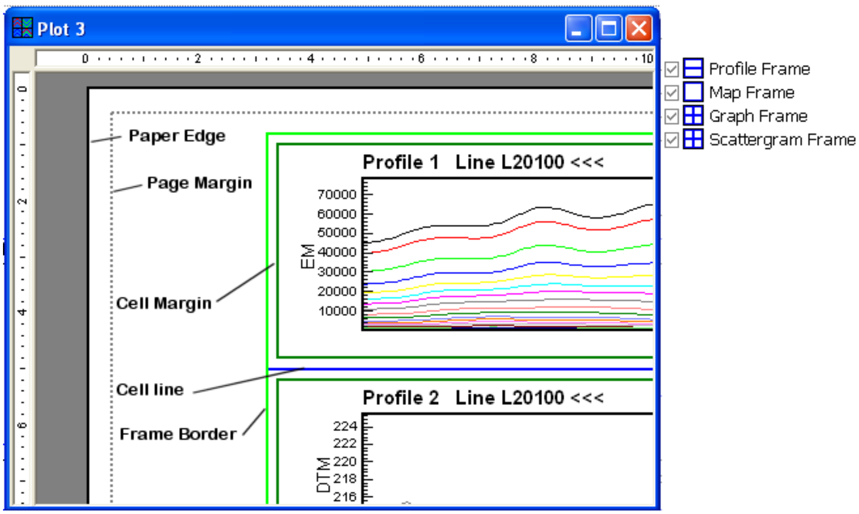
Encom Discover PA uses containers (called Frames) to hold graphical objects in position on a page. Different frame types exist for each of the object types that can be displayed. All displays have frames in the Workspace tree directly beneath the document type. Although they are referred to as Profile frame or Map frame, their manipulation and operation for each frame type is identical.

- *Frames*
- *Components*
- *Frame and Plot Axis Control*
- *Snap Operation*
- *Size Units*
- *File Drag-and-Drop*
- *Line Navigation*

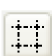
Frames

Frames or containers are used to hold graphical objects in position on a page. These frames can have any number of rows and columns to form cells, similar to a table in word processing software. Each cell can contain one or more graphical objects. Using cells enables objects to have alignment as well as easily controlling their edit operations such as cutting/pasting etc. As an alternative to positioning items in frames, refer also to *Snap Operation*.

Each frame has its own properties to control the appearance of the cells irrespective of the display objects it contains. The property dialog for frames in all cases has an **Appearance** tab. Controls on this tab position the Frame within a Border and each contained cell has Margin controls. Boundary lines surround each of a frame's cells.

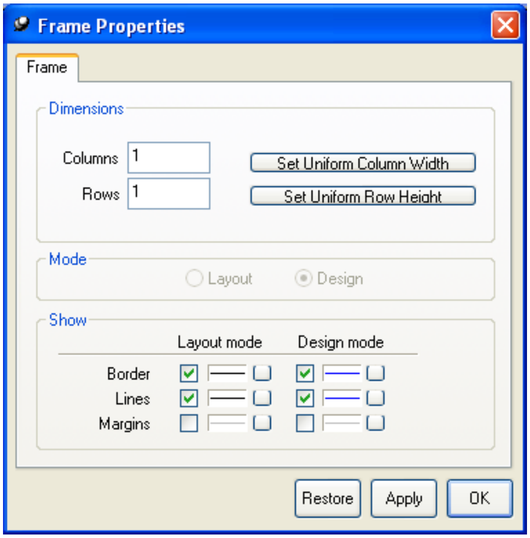


The components of a document display



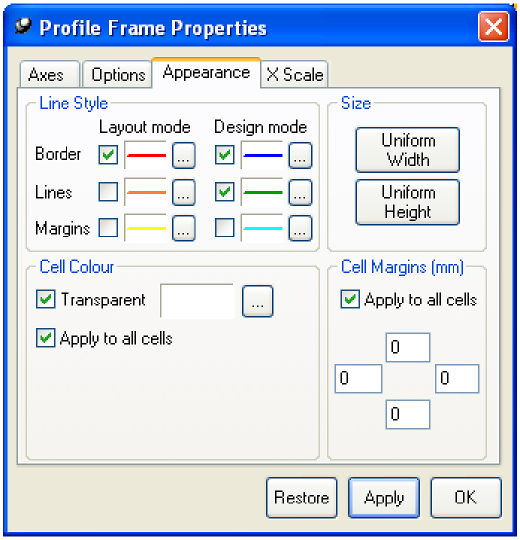
Frame borders, cell lines and margins can have individual line styles and these can be made visible or invisible. Two modes of display are available to assist with frame and cell design and positioning – Layout and Design. The purpose of these modes is to provide one setting that makes the cell boundaries and linework visible (in Design mode), but to quickly preview the final output by switching to Layout mode where the non-drawing frame and cell linework is not visible. To make this switch, use the **Frame Design/Layout Mode** button.

When initially creating a frame, the Frame properties dialog allows control over the number of Columns and Rows.



The Frame property dialog allows the dimensions (Columns and Rows) of a frame to be specified

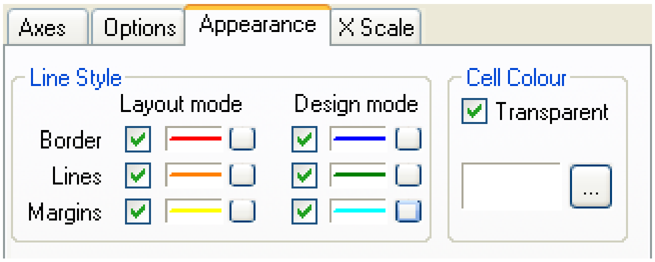
The offset of a frame's Margins, Lines and Borders is defined by properties specified in the frame's **Appearance** tab. These are discussed for each frame type in the relevant sections. An example is shown below:



Appearance controls to view the Line Style and Cell properties

A background colour (other than the white default) can also be used in profiles and graphs by setting a **Cell Colour** and disabling the **Transparent** option.

The offset of Margins and Lines from each other and their Border is defined by properties specified in a document's **Appearance** property. These are discussed for each document type in the relevant sections. An example of the Property dialog is shown below:



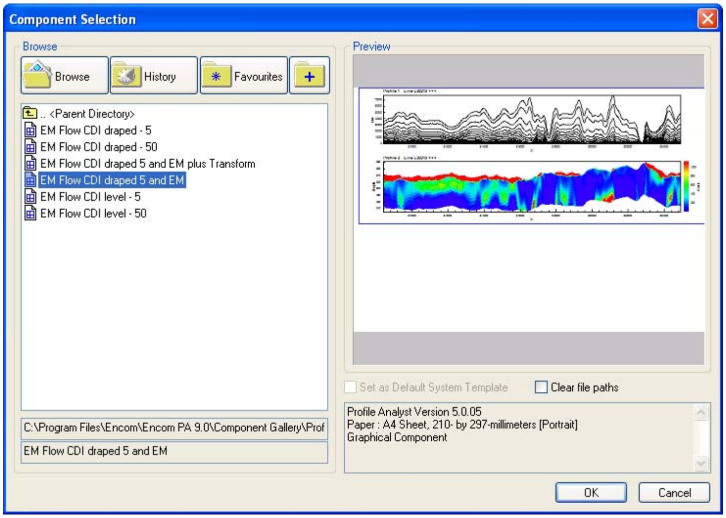
Appearance controls to view the Grid Cell properties

Components

Encom Discover PA can create larger applications used for interpretation or final interpretative products. Usually such applications consist of individual components that have various available displays (eg profiles, maps, images etc). Components can be saved and re-used in Encom Discover PA. Components may consist of a single item (such as a graph, profile, logo or scalebar etc), or may be composed of combined objects such as title blocks with text, logos, grids, maps etc. By combining various components it is easy to create larger, more complex presentations that would, by themselves, take a much longer time to create.

Components are saved from the **Save Component** item found on most pop-up menus and the saved Component file has the file extension of .EGC.

Components are restored from the **Insert>Component** menu item or the **Insert Component** button. When you want to insert a component, a selection dialog presents a preview of the available items.



Component selection dialog for restoring a component

Browse, History, and Favourites

Buttons allow fast search and retrieval of frequently used components. These options operate similarly to the Windows Internet Explorer utility.

Once you select a component, position the drawing cursor on the active page to contain the component and drag a rectangle with the left mouse button pressed.

Frame and Plot Axis Control

Plot objects are contained in the cell of a frame. Plot objects cannot be floated out of the frame in which they are contained although objects can be copied and pasted into other frames. Frames assume a critical role in the control of the plot objects they contain.

The Axes property page of the frame enables you to modify settings that are applied to the plot objects in the frame.

1. You can specify that the range of axes is global to the frame. In this case the data range of the axes automatically fits all the data in all of the plot objects in the frame.

2. You can specify that the data range be derived from the current clipped data range or from the entire unclipped data range. This is the **Zoom Global** option and is primarily used with profiles. As you pan or zoom data along a line, the axes fit to the range of the data displayed unless Zoom Global is set in which case the range is always derived from the entire line.

3. You can specify that axis annotations be global (when the axis range is global). This allows you to set annotation properties for all axes (of a particular type) in all plot objects in the frame in one place.

4. You can specify that the X and/or Y axes are scaled and specify the scaling factor.

5. You can make all plot objects in the frame isotropic. When set, Encom Discover PA automatically adjusts the data ranges on the axes to ensure that the plot objects are isotropic. For example, a map has this option set by default.

6. You can specify that Encom Discover PA should modify the data ranges to ensure that the plot objects all fully occupy the frame cells in which they are held (option Data zoom to fit grid cell). This option is usually combined with the isotropic setting. It can affect a scaled plot object and if not set, the plot object may be smaller or larger than the frame cell. When enabled, the plot object always fits the cell.

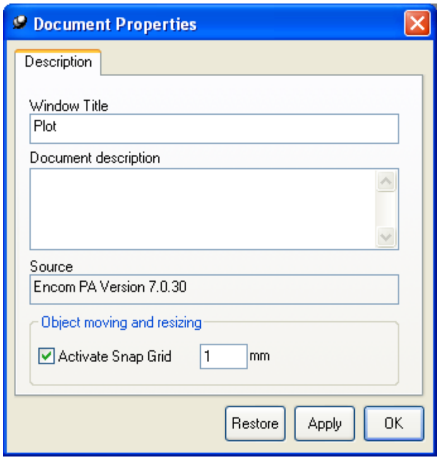
Every plot object contains an **Axis** property page that is exactly the same as the frame property page described above. If a setting is defined at the frame level then it is set and not editable at the plot object level. In other words, if a property is not controlled by the frame, it can be controlled on an individual basis by the plot objects in the frame. The properties set in the plot object are then applied to every axis in the plot object. For example, the Y range for all Y axes in a plot object can be controlled by setting the **Range Global** at the plot object level of the Workspace tree. This structured concept continues down the controlling Workspace Tree to individual axes where properties can be modified if they are not controlled by the plot object or by the frame.

Axes in a plot object have an ordered structure starting at X then Y, Z, Colour, Size. This means for example, that you can add a Y axis to an X axis, but you cannot add an X axis to a Y axis. This structure defines data domains in the plot object within which you can display data objects. When combined with the plot object and frame global options, Encom Discover PA provides a flexible data graphing capability that can display your data almost any way you want.

Snap Operation

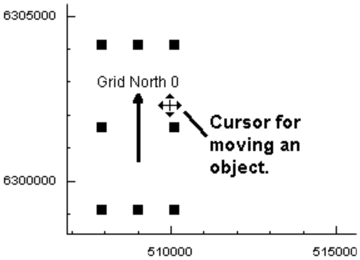
Encom Discover PA provides the ability of graphics objects to be snapped. This means that an invisible grid (of a definable size in millimetres) is present in a display and this can be used to move or resize objects (including Frames) so they align or overlay each other precisely.

The Snap control is turned on or off from the **Document Properties** dialog. The size of the snap movement is also specified.



Document properties and Snap Operations

Operation of Snap is best used when dragging or resizing a graphics object. Select the item graphically or from the Workspace tree and the object's handles should appear. As you move the cursor to the edge of the surrounding border highlighted by the handles, the cursor changes to a move-style cursor. This can then be used to shift the object and snap can be used to align it accurately with other objects.



Moving an object and snapping

Size Units

Throughout Encom Discover PA the properties of displayed objects are accessible. In many cases the position of these objects is made relative to a defined point (such as an anchor, a boundary or a margin). Where distances from that point are controllable, the units of distance may be:

Millimetres

Millimetres as used in a final, paper layout plot (not on-screen)

Page Points

A point is a printing measurement whereby 32 points equals one inch on printed paper. This unit applies to printed output.

Screen Points

One pixel of screen resolution equals one screen point of distance. This distance format is used for on-screen presentation and retains plot size across computer monitors of different resolutions.

Percentage

If a measurement is made in percentages, it refers to the available distance being divided into units of 100 and the object is placed at the relative percentage location. For example a 50% distance positions the object at half the available distance.

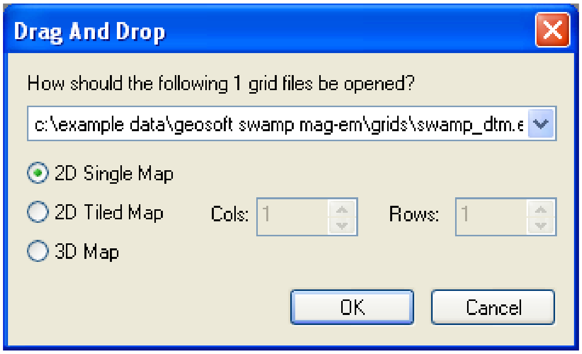
File Drag-and-Drop

Files (of supported types) can be dragged and dropped from either the Explorer tab or Microsoft Explorer and displayed without having to create a display window for them. The process of dragging and dropping is to select individual or multiple suitable files, and with the left mouse button depressed, move the cursor to the display area of Encom Discover PA and release the left mouse button.

The following actions can be performed:

- Drag a suitable file and drop it into the open display area of Encom Discover PA. An appropriate window opens and displays the file contents.
- Drag a suitable file and drop it onto an existing, open window. If the window type can accommodate the dropped file, it displays it. For example, if an image grid were dropped into a 2D map already displaying say a raster image, the additional image would also be displayed.

- If a dropped file can be displayed in either 2D or 3D, a message and choice is displayed.
- If multiple files suitable for display in a map are selected and dropped, a message is displayed providing a choice of the number and orientation of frames to display in the window.



Choice of 2D or 3D single or tiled map for multiple files dropped for display

- If a database is dragged and dropped, it is opened and available for access rather than displayed.

Example file types that can be dragged and dropped are as follows:

Type	**Action**
.SHP	Opened as a vector file object
.TAB	Opened as a vector file object
.MIF	Opened as a vector file object
.DXF	Opened as a vector file object
.GSF	Opened as a vector file object
.FDB	Opened as a feature database
.EGA	Opened as a feature database
.GDB	Opened as a dataset
.EGB	Opened as an Image object
.ERS	Opened as a Surface object

Type	Action
.GRD	Opened as a Surface object
.TAR	Opened as a Surface object
.DEM	Opened as a Surface object
.BMP	Opened as a Picture object
.JPG	Opened as a Picture object
.GIF	Opened as a Picture object
.PNG	Opened as a Picture object
.TIF	Opened as a Picture object
.FLY	Opened as a Fly-Through script file

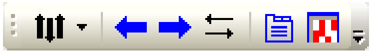
Line Navigation

Encom Discover PA is a data processing and display system that uses traverse lines as the basis for analysis and presentation. Data lines contained in an open database can be selected for display or processing in a variety of ways.

- Individual navigation tools, such as drop-down line lists are available from properties dialogs and the Line Iterator toolbar.
- The Line Iterator toolbar can be used to select lines specifically or sequentially.
- The Flight Path Map displays traverse lines that can be selected graphically. Profiling displays such as Profile Document windows or Decay Tools can use this selection procedure.

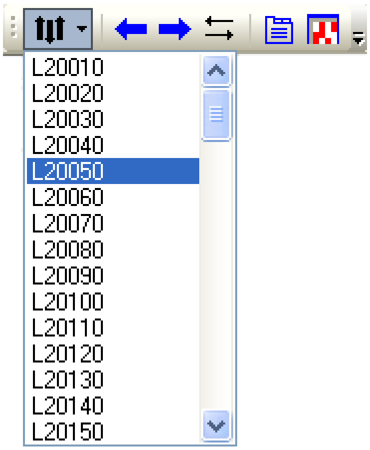
Line Iterator Tool

The Line Iterator is a floating toolbar that can be moved around the Encom Discover PA screen or docked with the main Encom Discover PA screen. If this toolbar is not visible select the Line Iterator option from the pop up menu that appears when the main toolbar area is clicked on with the right mouse button.

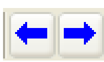


The Line Iterator toolbar

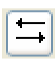
The pull-down line list on the toolbar shows all lines compatible with the datasets in the current display. This may not be all lines in the database. In certain circumstances (such as for tie lines and traverse lines with approximately perpendicular direction azimuths), it is inappropriate to display these in the one graph, with the same base axis.



Line Selection list on the Line Iterator toolbar

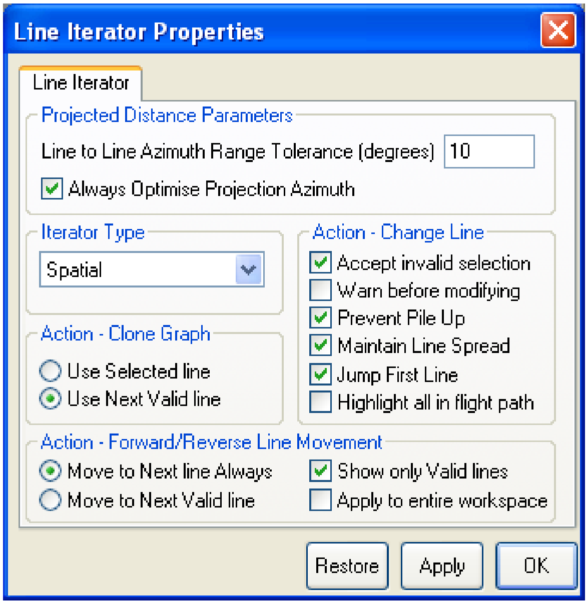


Selecting a line moves all graphs to the selected line or with their appropriate line offset. Moving lines by clicking the buttons of the Line Iterator toolbar, moves all lines in the plot forward or back in the iterator line list.



The sense of a display for a profile of data can be reversed (flipped), by using the **Reverse Line Direction** button. The display reversal applies to all profiles contained within a document (even if contained in different frames).

The operation of the Line Iterator is controlled by the properties available from the **Line Iterator Properties** button which will display the Property dialog. This same dialog can also be accessed from the **Tools>Options** menu item using the **View** tab.



Line Iterator Properties dialog

Properties that can be adjusted from the Line Iterator Properties dialog are:

Line to Line Azimuth Range Tolerance (degrees)

Encom Discover PA displays profile data by projecting lines onto a common baseline. Encom Discover PA only displays lines that have an azimuth within +/- the azimuth tolerance of the baseline azimuth. For more information see *Axis Properties*. This variable determines whether two lines are azimuth compatible - defined as when the difference between the line bearings is less than twice the specified tolerance. The default tolerance is 5 degrees.

Always Optimise Projection Azimuth

Encom Discover PA adjusts the baseline azimuth for any group of lines selected so that it lies within the centre of the range of line azimuths. When enabled (the default), the azimuth of the lines are examined each time the data content in a frame changes and an optimal azimuth for the projected distance computation is calculated.

Iterator Type

The Line Iterator toolbar uses one of three modes – Standard, Spatial or Sequential for the ordering of survey lines.

- **Standard** - lists lines in the order they are stored in the database.

- **Spatial** - re-orders the line list spatially such that moving from one line to the next is spatially meaningful and according to line azimuth criteria described in Profile Horizontal Axis. Only valid lines (that is lines that meet the line azimuth criteria are displayed in the Spatial line list.

- **Sequential** – this mode iterates through lines similarly to the Standard line iterator except that it forces a movement to the next/previous line even when the line azimuth rules would normally prevent it. It can be used in more complicated survey line situations or in cases where multiple survey databases have overlapping flight path tracks.

Action – Change Line

- **Accept invalid selection** - When moving forward or back through the Line Iterator, the next lines may not meet the specified display criteria (e.g. a line exceeding the azimuth tolerance). This option allows the user to click on a flight path (or set the line through property pages) and change the line even when such action invalidates the azimuth compatibility rule. One or more graphs will usually be invalidated.

- **Warn before modifying** - If an invalid line combination is about to be displayed, a warning is displayed. It warns the user that a selection is invalid and allows the user to cancel the operation.

- **Prevent pile up** - If you iterate to the end of the available line list, no further iteration is allowed if this option is enabled. If not enabled, the Iterator moves the selection to the last available line.

- **Maintain Line Spread** - If a specific combination of multiple lines is chosen for a profile display, this combination is maintained, even if a flight map is used for line selection.

- **Jump First Line** - If this option is OFF, if the selected line is prior to the lines in the frame then the rear line in the frame is moved back to the new line. If the selected line is in advance of the lines in the frame, then the leading line in the frame is moved forward to the new line. When the selected line is within the frame line group then nothing happens. If this option is ON the rear line in the frame is always moved to the selected line

- **Highlight all in flight path**

Action – Clone Graph

- **Use Selected Line** – When a new profile is created as a copy of an existing one, the same line is used if this option is enabled.

- **Use Next Valid Line** - When a new profile is created as a copy of an existing graph, the next line in the Line Iterator tool is used. When you clone a graph it can use either the line of the graph you are cloning or use the next line along. As you usually want the next line, it is the default. Note that it uses the next VALID line. This is the next line in the spatial order that is azimuth-compatible with the lines currently being displayed. It may not be the nearest line in the spatial order.

Action – Forward/Reverse Line Movement

- **Move to Next Line Always** - If enabled (the default), this option forces a move to the next line in the Iterator order. This option may invalidate the line selection. This option relaxes the Use Next Valid Line restriction. The option permits a move to the next valid combination of lines.

- **Move to next valid line** - The Line Iterator list only displays lines that are valid for the current baseline azimuth. This is the default.

- **Show only Valid lines** - Displays the list of only azimuth-valid lines. This setting is the usual default.

- **Apply to entire workspace** - Applies the Line Iterator selection to all available windows. This is not the default

If you can always move to the next line in the spatial order then you can iterate through an entire dataset. You can move between incompatible lines (eg flight and tie lines) at will. The spatial order preserved and rules followed, it will just allow you to move to a line that is incompatible with the current lines.

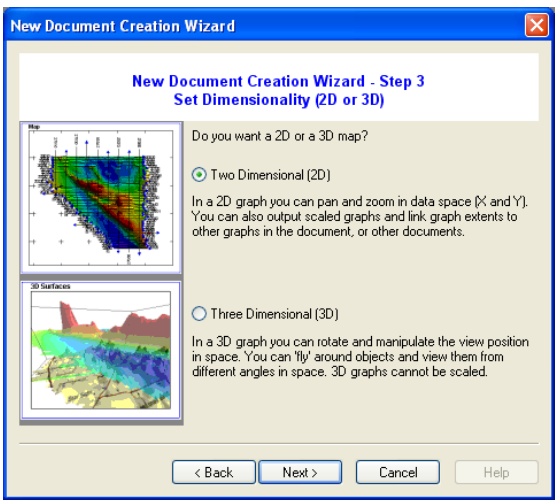
If you can only move to the next valid line, you can never shift between tie and flight lines as they are incompatible. The only way to display a tie line is to manually change the line in the graph property page or interactively select the line from a flight map. When you iterate forward/back with **Next Line Always** the program moves one line at a time. With **Next Valid Line** it may jump many lines each move and searches for the next azimuth-compatible line. If you enable **Show Only Valid Lines**, the drop down list on the Iterator toolbar only displays the lines that are azimuth-compatible with the current line selection. **Apply to Entire Workspace** applies a Line Iterator movement to all views in the workspace.

Using Wizards

Various operations within Encom Discover PA use wizards to simplify tasks that otherwise are complex to a user (for instance association of features to line databases). Other wizards are used for performing simple operations but for new users, such procedures may seem difficult (for example, displaying a 2 dimensional map). Wizards can be initiated from the Wizard button or in other cases, within the menu items.

Wizards consist of a series of logical steps to arrive at the desired outcome. The various dialogs describe each step and contain explanatory text to make your choices simple. In many cases, you may need to make decisions about options but if you make a mistake, you can back up or advance by using the Back and Next buttons. If you wish to skip intermediate steps of a wizard, often you can proceed directly to the Finish button and thereby accept only the defaults specified by the wizard. Generally the defaults would be as you would have chosen anyway. If you decide not to use the wizard at all, select the Cancel button.

In the example below, some choices are provided, with examples of the option output.



Wizard with a choice of 2 or 3D map and example illustrations

New Document Creation Wizard

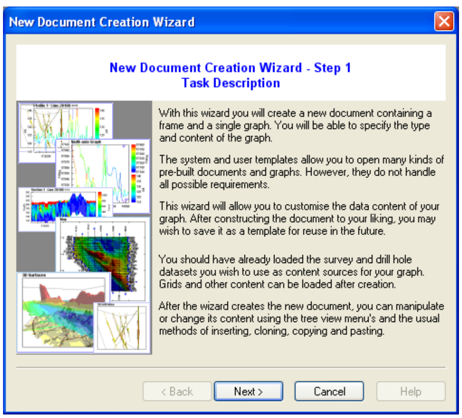
The New Document Creation wizard is a fast method of producing a range of different displays by responding to questions asked in a number of wizard screens. Before using the wizard, ensure that the databases and files you wish to use are available (see *Accessing Data*).

Initiate the wizard from the button on the Document toolbar. A series of screens are displayed, which will guide you through the options for displaying data. You can navigate backwards and forwards between screens using the **Back** and **Next** buttons.

When you have completed your selections, click the **Finish** button, and the program will then create the view.

Step 1: Task Description

The first screen is descriptive only and describes the task you are about to perform.

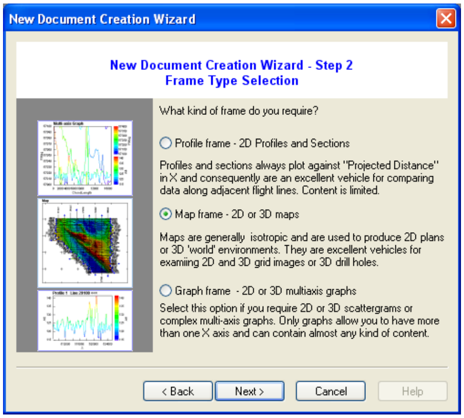


Step 2: Frame Type Selection

Choose which of three basic display types you want:

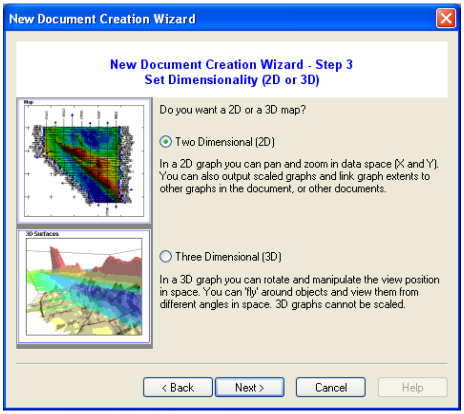
- Profile frame
- Map frame
- Graph frame

Note The term “frame” is used to describe the display container and it provides flexibility in being able to specify scale, resizing, editing and display control (see *Profile Frame Properties*).



Step 3: Set Dimensionality

The next screen asks whether you want a 2- or 3-dimensional display. This will create either a map (2 dimensions, see *Two-Dimensional Maps*) or a display volume (3 dimensions, see *Three-Dimensional Displays*).



Step 4: Select Data Objects

The final wizard screen asks you to choose which data objects you want to display. Only available objects are presented.

When you have selected the display objects, click the **Finish** button, and the wizard will create the display with the available data. In some cases (for example, requests for external files such as grid, vector or bitmaps), the wizard may ask you to select additional files.

To modify a display created by the wizard, you will need to use the normal editing and control functions available in Encom Discover PA.

5 Display Formats

In this section:

About Display Formats

Encom Discover PA is an interactive software package for one (profiles), two (maps) and three dimensional display of geoscience data. Geophysical information that can be analysed includes single or multi-channel data acquired from airborne or ground surveys such as electro-magnetics (AEM), geochemistry, potential field (gravity or magnetic) data or a combination. Encom Discover PA accesses data from industry-standard databases which allows high speeds of data transfer and assists in the management of data. The speed and usage of database technologies make Encom Discover PA especially applicable for large datasets as are typically derived from airborne geophysical techniques.

Encom Discover PA serves many roles through its design. Its function is not purely to display geophysical data, but to provide interpretation links to other software packages (such as MapInfo Professional and ER Mapper) as well as maintaining a feature database for recording and documenting interpretation.

Encom Discover PA acts as an interface to numerous processing algorithms. Depending on the geophysical technique being used, processing tools that are usually operated as batch processes are available. Encom Discover PA has also been designed to accommodate growth in processing algorithms and data types.

The main design principles used in Encom Discover PA are to:

- Interface to other commercial software (for example, Geosoft’s Oasis montaj and DFA’s Intrepid) wherever possible. This interface provides direct data communication to and from supported databases.
- Ensure Encom Discover PA is operational on the most commonly available computing platforms. These environments include Windows XP, WIndows Vista, and WIndows 7 systems.

- Divorce the data from the processing procedures such that Encom Discover PA can present all supported data types and interface to tailored processing algorithms.

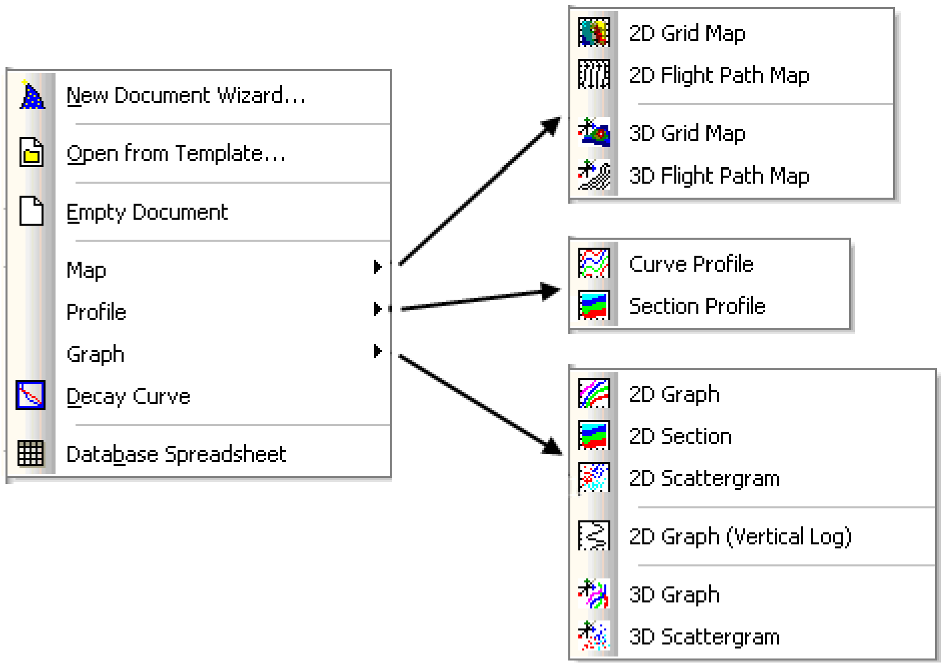
What Can Encom Discover PA Display?

Available display formats provided within Encom Discover PA include:

- 2D and 3D curve profiles of line-based data
- Layered sections within profile tracks
- 2D and 3D graphs with any data channels used to specify the graph axes
- Flight and traverse path maps
- 2D and 3D images and raster presentations
- Scattergrams
- Spreadsheets of data
- 3D representations of objects such as drillholes and volume models.

Creating a New Display

Use the **File>New** menu or the corresponding toolbar tool to create a new display.



New display documents for all formats can be created by selecting the appropriate item, or documents can be created using a predefined format and layout via a template (see *Templates*) .

The New menu item can also be used to display a Decay Curve or Database Spreadsheet.

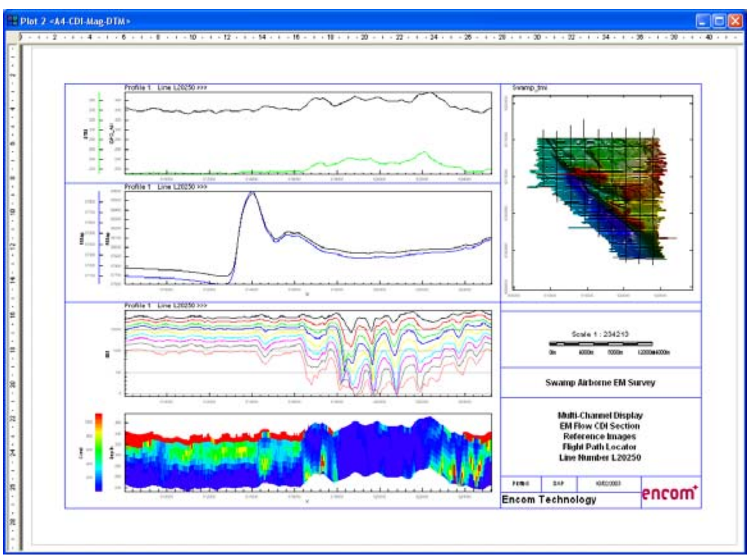
Display Types

- *Profiles*
- *Graphs*
- *Spreadsheets*
- *2-Dimensional Maps*
- *Map Mosaics*
- *Points*

- *3-Dimensional Displays*
- *Drillholes*
- *Decay Curves*
- *Graphs and Scattergrams*
- *Vertical Log Graphs*
- *Stacked Profiles*
- *Voxels*
- *TrackMaps*
- *Combination Displays*

Profiles

A primary tool used to examine data in Encom Discover PA is the profile. A profile is defined as the display of one or a number of data fields with a common horizontal distance axis. A document or window page displaying profiles may also include other display types such as maps, graphs and other objects to provide a direct comparison of data or for preparation to printing. An example of a combined document is shown below.



Two data lines with magnetics, elevation, EM data, inversion results, titling and flight map within a layout mode Profile Document

All documents are created in a plot window:

- Multiple plot windows can be defined containing multiple display objects
- Each graphed area in a Profile document may contain multiple profiles
- The horizontal axis of a profile is scaled according to a distance channel called ProjectedDistance. This computed channel (based on the chord or straight line length and position of the relevant line(s)) allows direct comparison from one line to another.
- Although the ProjectedDistance channel is used for plotting and scaling the horizontal axis of profiles, other channels may be used to annotate the X-axis (such as easting, fiducial etc).
- Each profile may contain many Y-axes, where each Y-axis is used to control data fields with their own units or physical properties. For example, XChannel and ZChannel could be on one Y-axis, Diurnal and Residual Magnetics in a second Y-axis etc.
- Each Y-axis may contain many channels or data bands. A band is the lowest hierarchy component of a profile. For example, one band may display Magnetics, another may display multiple channels such as in EM data. Each band may have its own independent attributes of colour and style.
- The data fields used to display profiles in a document may not be derived from the same database but must be common to the one line if separate fields for one line are to be combined.

A new profile is added below the bottom profile in an active plot window but it can be moved, cut or pasted. All profiles automatically re-size when you add a new profile, or when you change the aspect ratio of the profile window. The relative size of a profile in a document window can be controlled but its horizontal scaling. X-axis plotting is controlled by a computed distance channel called ProjectedDistance.

For more information, see *Profiles*.

Graphs

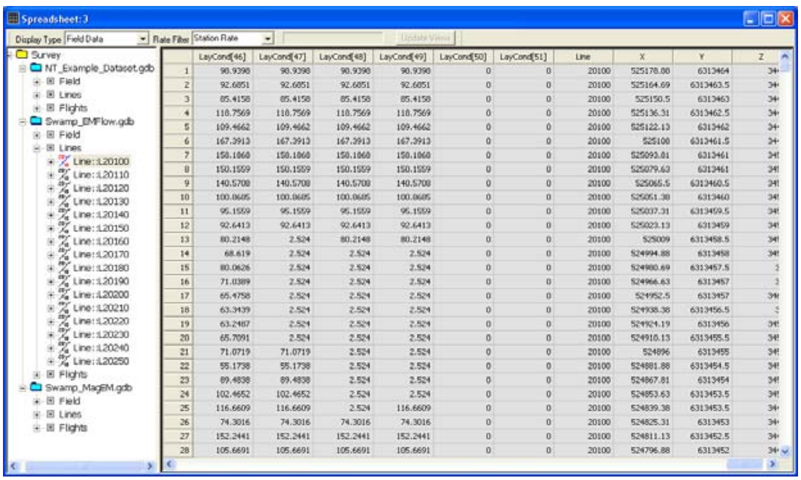
The Graph object is essentially identical to a Profile except the horizontal, scaling axis is defined. A Graph uses horizontal scaling you define according to a data field and does not use the ProjectedDistance or chord length channel as used in a Profile horizontal axis. All other channel or line combinations, as described in Profiles apply to Graphs.

For more information, see *Graphs*.

Spreadsheets

To provide a numerical display of the content of a database, Encom Discover PA provides a data spreadsheet facility. The spreadsheet can present the numeric contents of a database, statistics or data ranges. Other functionality includes the ability to export the database content or subset.

Databases accessed by Encom Discover PA typically contain flight information, lines and various data channels of sensor information. This display format is useful for quickly examining statistics of a database or contained data fields.

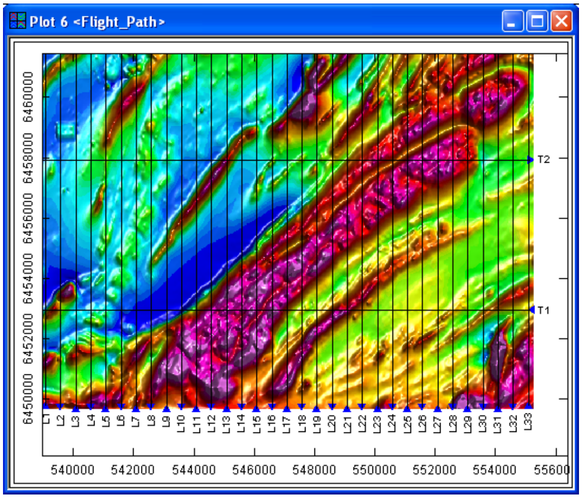


A spreadsheet with flight, line and channel information available

For more information, see *Spreadsheets*..

2-Dimensional Maps

A 2D Map is used to contain flight path maps, images, contours and other two dimensional displays. Encom Discover PA can use most industry-standard grid formats (including but not limted to ER Mapper, Geosoft, Geopak, Vertical Mapper, Surfer, ASEG GXF, Landmark and USGS).



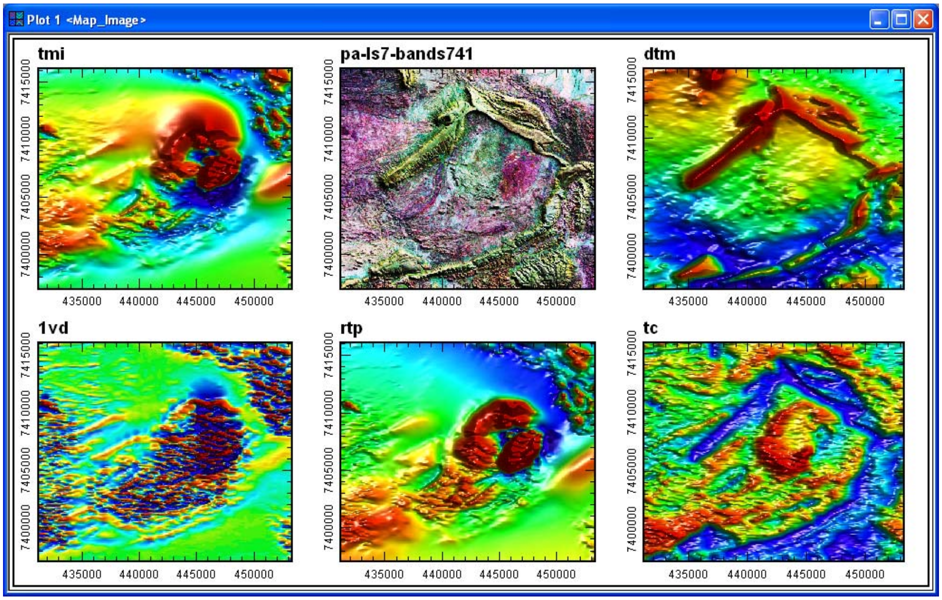
A Map with annotated flight lines, image, contours and annotations

For more information, see *Two-Dimensional Maps*.

Map Mosaics

It is possible to create a series of map displays whereby each map is geographically linked to the others when contained within a single Map Frame.

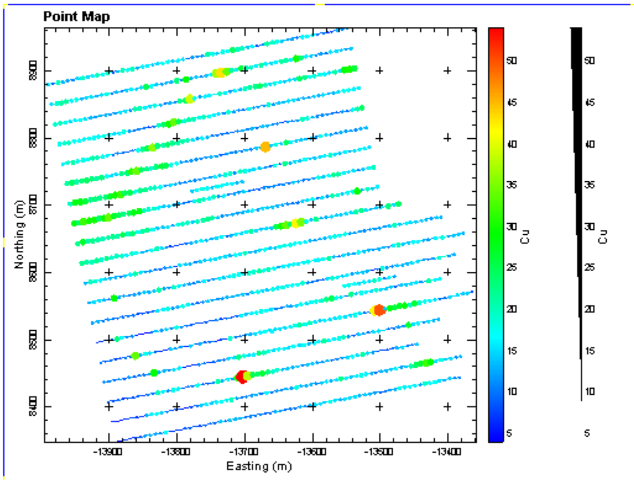
The map mosaic can be created from a single 2D Map display and choosing the **Clone>Graph** option from the pop-up menu displayed when the Map branch of the Workspace Tree is selected with the right mouse button. Alternatively the map mosaic can be created by selecting multiple surface files in Windows Explorer and using the Drag-and-Drop facility available in Encom Discover PA.



A map mosaic representing multiple data files loaded into Encom Discover PA using the "drag and drop" facility

Points

Encom Discover PA can display data points that have easting and northing locations specified. The data can be accessed from a Geosoft montajTM or a MapInfo Professional .TAB data file. Often the individual data points may be clustered into lines but this is not mandatory. Encom Discover PA can use its line support capability (such as the Line Iterator) to navigate point datasets, but if each data record represents a single point, this can be displayed quite satisfactorily.

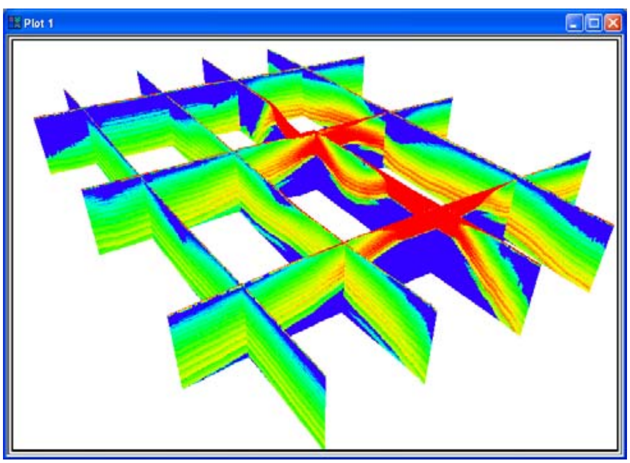


Example point display with size modulated points

For more inforamtion, see *Displaying Points*.

3-Dimensional Displays

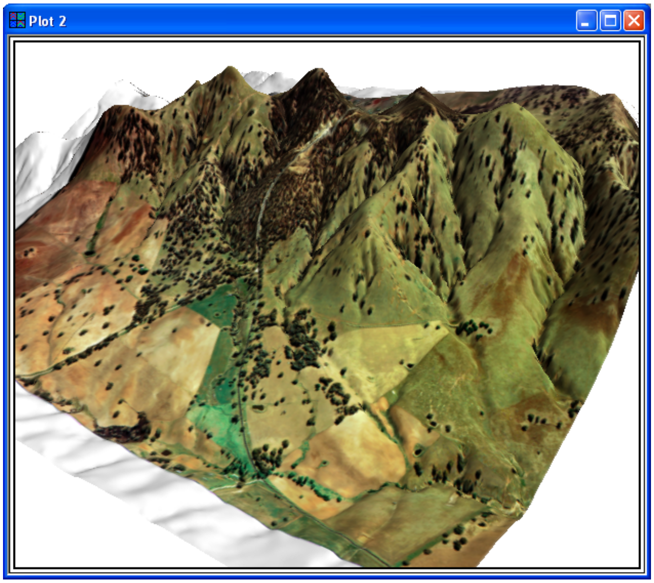
Maps, graphs, flight paths and other display objects can be presented both in two and three dimensions. Complex displays and data with elevation or depth information can be viewed more clearly in three dimensions. Views can be zoomed, panned and even fly-through is available for impressive visual display of data.



Layered sections of Conductivity Images derived from EM Flow software

Three dimensional displays are effective in the display of surface grids as well as display objects requiring 3D volumes (such as drillholes, voxel and 3D .DXF vector files). Many combinations of surfaces can be displayed such that:

- Single or multiple surfaces can be displayed one above the other
- Where one surface overlaps another (or other data), a transparency can be applied to allow the lower surface to be viewed through the obscuring surface
- The colour rendering of a surface can be supplied by one grid onto a second shown as the viewed elevation. For example, a radiometric grid could be displayed but coloured by magnetics to note any detection correlation.
- One surface (such as topography) can have another (such as an air photograph or satellite image) overlaid to provide a topographic relief presentation.

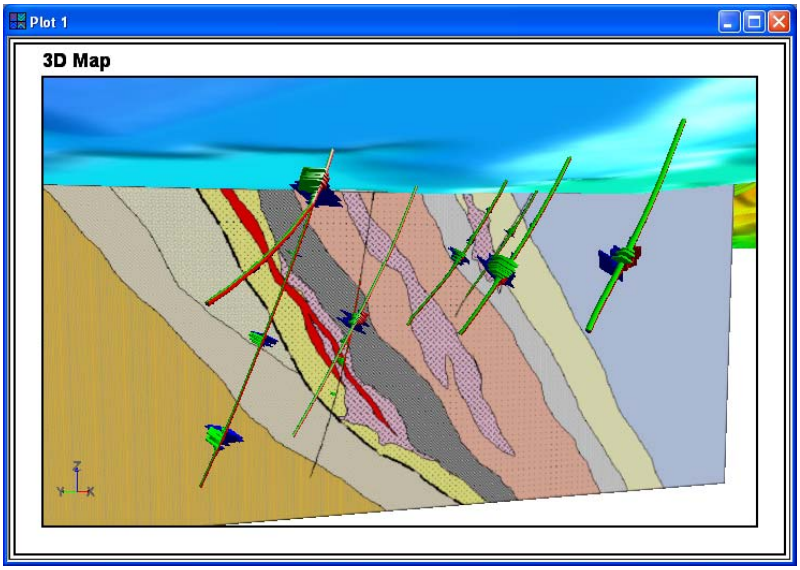


Aerial photograph draped over a topographic grid surface

For more information, see *Three-Dimensional Displays*.

Drillholes

Encom Discover PA can import and display drillhole locations and downhole data in a variety of formats. The displays can be coupled with three dimensional bitmaps to indicate geological sections or derived sections from depth-conductivity analysis. Three dimensional navigation with interactive zooming, roaming and fly-through capability make volume and 3D visualisation simple.

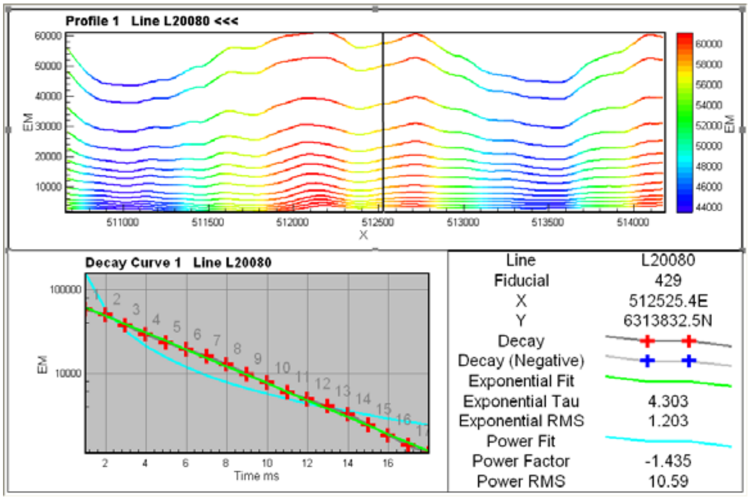


Display of multiple drillholes in a 3D display with assay and geological section information

For more information, see *Drillholes*.

Decay Curves

Encom Discover PA contains an interactive display tool for the presentation and analysis of multi-banded data. Typical use of the Decay Tool is with time domain EM geophysical data. The Decay Tool is enabled from the **File>New>Decay Curve** or the **Decay Curve** button on the Decay Curve toolbar.

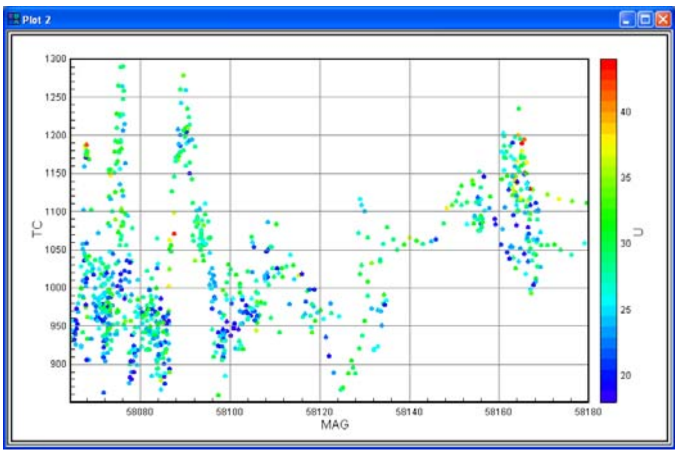


Decay tool with an interactive profile for updating the decay plot

For additional information, see *EM Decay Curves*.

Graphs and Scattergrams

For displaying the relationship of one data field with another, Scattergrams and Graphs can be used. This display format is useful in comparisons of data such as radiometric analysis (Total Count to say Potassium) or for elevation versus raw magnetic data. The plot can use symbols to indicate the population distribution, lines or both. The Scattergram and Graph displays are similar in that they provide individual data field control of both the vertical (Y) and horizontal (X) axes. The Scattergram by default presents data as symbols while the Graph display uses lines to join the data values at their required location.



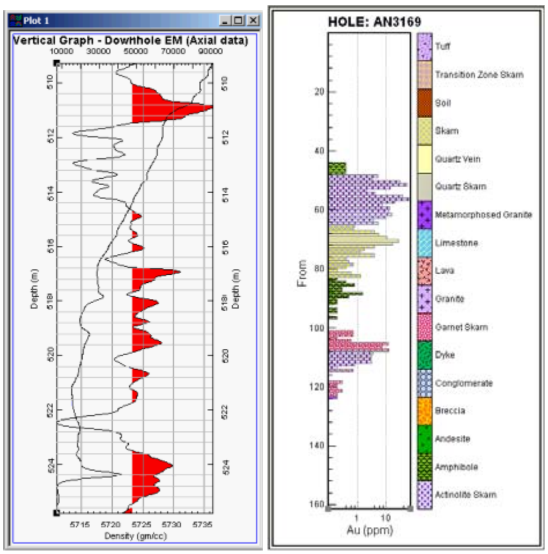
An example of a Scattergram

For more information, see *Graphs* and *Scattergrams*.

Vertical Log Graphs

In some instances (such as downhole logs), it is advantageous to rotate the standard, horizontally-oriented graph such that axes and cursor movement are oriented vertically. The Vertical Graph operates exactly like its horizontal equivalent where individual data field control of both the vertical (Y) and horizontal (X) axes is available.

When created, these objects have the standard zoom, pan and cursor movement vertically oriented to allow data analysis up and down. An example of this format is shown with drillhole EM data below:

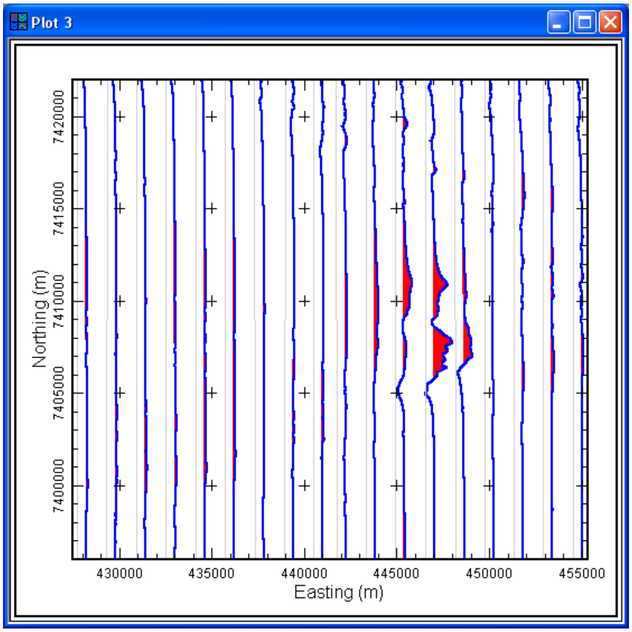


An example of a Vertical Graph document

For more information, see *Vertical Graphs*.

Stacked Profiles

A Stacked Profile display produces a 2D vector file (.DXF or .GSF file format) representing a plan view of survey lines from a database with amplitude representing a field within the database.



Example stacked profile map

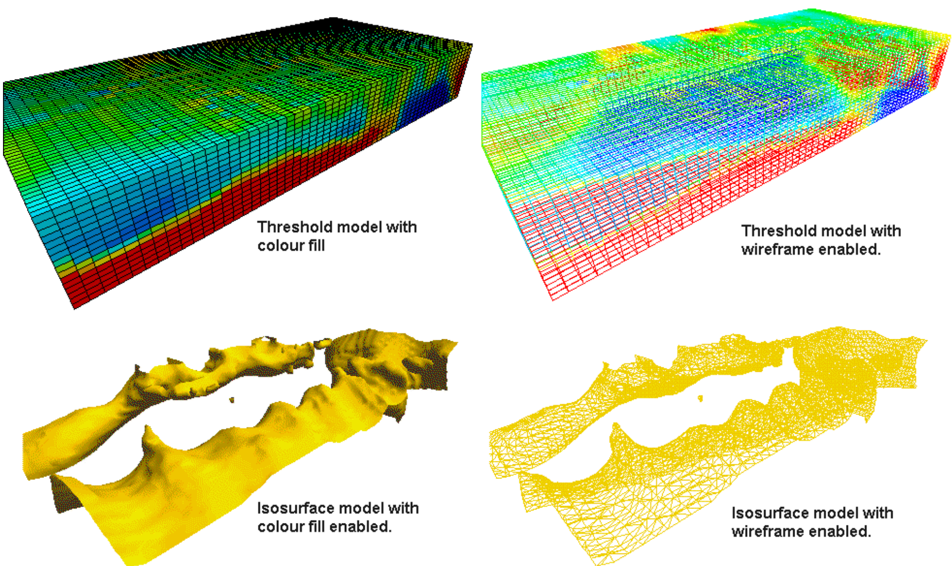
For more information, see *Stacked Profiles*.

Voxels

The following functions are provided for displaying Voxel Models:

- Import and interpret three-dimensional model structures as created by research projects managed by UBC (University of British Columbia) and CEMI (Consortium of Electromagnetic Modelling and Inversion – University of Utah). Other 3D objects such as Gemcom .BT2, DataMine and Surpac are also supported.
- Display three-dimensional model volumes as voxel (cell-based) entities that can be visualised interactively and combined with other 3D object types (surfaces, drillholes, vector files and other models)
- Allow real-time interactive voxel display control through look-up table manipulation, visual navigation and slice or block presentation

- A range of display formats including row, column or cell slicing, multiple slicing (to create a chair view), attribute thresholding and wireframing
- Creation of isosurfaces whereby model attributes of similar value are three-dimensionally displayed as a surface
- Specific look-up tables can be created to simulate lithological analogies and so present voxels of a colour that matches a particular property range.
- Multiple voxel models can be added for display with independent attribute controls to compare property distributions. For example, magnetics models could be displayed at the same time as gravity inversion model results.

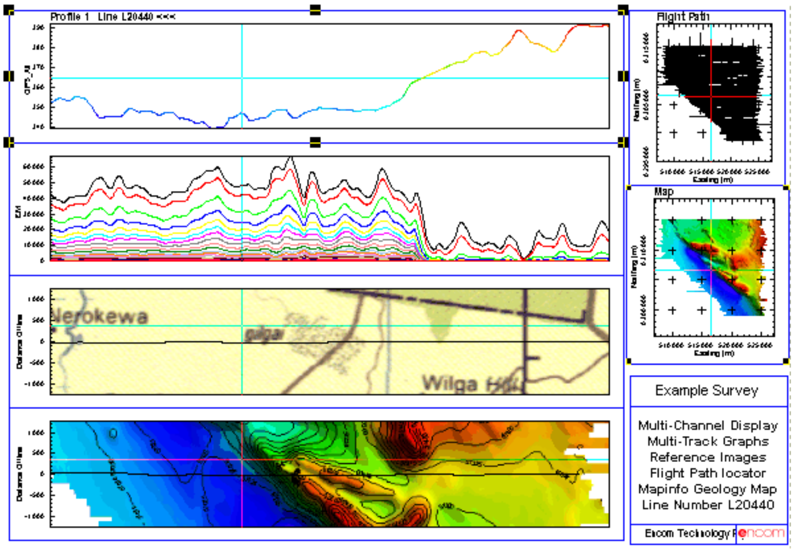


Various display formats of the Voxel Modeller

For more information, see *Voxel Models* and *Voxel Utilities*.

TrackMaps

TrackMap displays can be used as shown in the example below:



TrackMap example showing the flight path location over a scanned image and over a map of the magnetic response

A TrackMap display has the following functionality:

- The traverse of a selected profile (shown in the top two tracks) can be configured to show in the bottom two tracks
- Tracks can have different background images
- The track can be combined with other profiles or sections. The TrackMap track has all the display attributes of a normal profile. It can be sized vertically and scaled horizontally for hard copy purposes
- The background displayed in a TrackMap uses an RGB raster image file (such as ER Mapper)
- TrackMap track display is isotropically displayed. That is, the vertical and horizontal scaling is identical such that no distortion to the image is introduced.

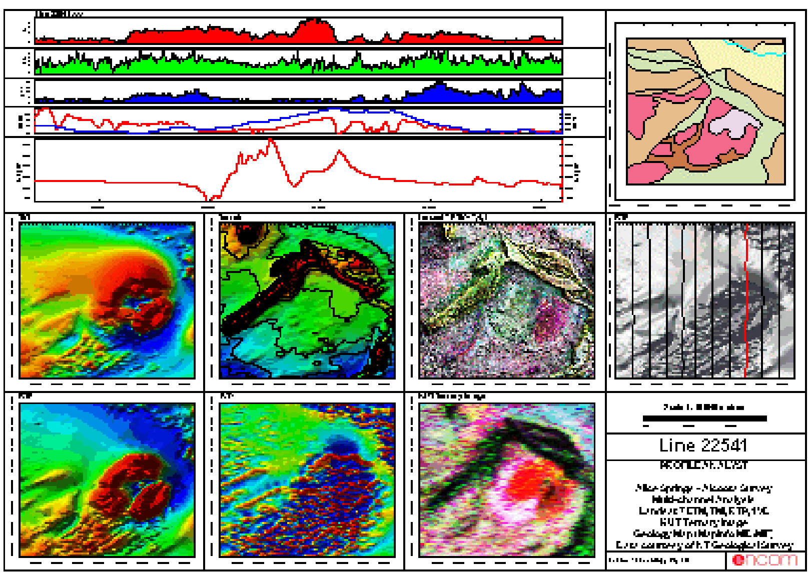
- The orientation of the traverses of a TrackMap are irrelevant. The TrackMap module automatically re-samples the background image as required to position the traverse and display the image. If the image or traverse are rotated, the program extracts and rotates only the required swath of image necessary to display the selected track.
- Positioning a cursor in a track indicates the location of the point in adjacent tracks (including the TrackMap tracks) plus any linked maps
- Data zooming of any of the profiles causes all tracks to zoom and pan or smooth scroll

For more information, see *TrackMaps*.

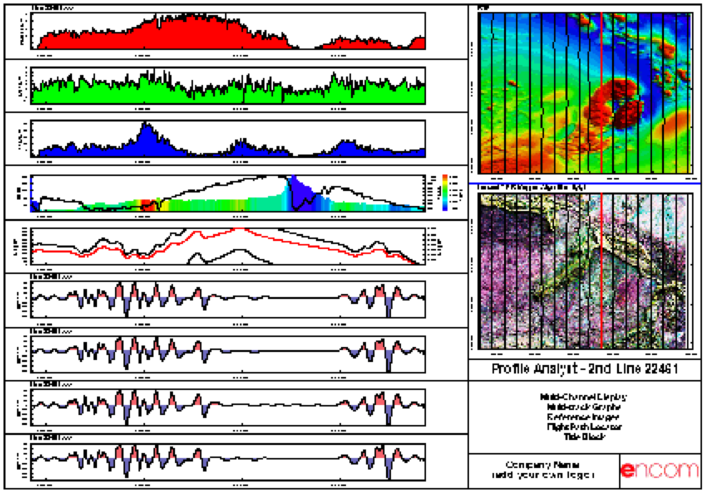
Combination Displays

Display formats that combine the various presentation objects of Encom Discover PA are powerful and useful for data interpretation applications. Some examples of these are shown below where combinations of maps, profiles, axes etc are placed into a document. The use of templates and components make these types of complex presentations simple to create (refer to *Templates and Components* supplied as a separate document with Encom Discover PA).

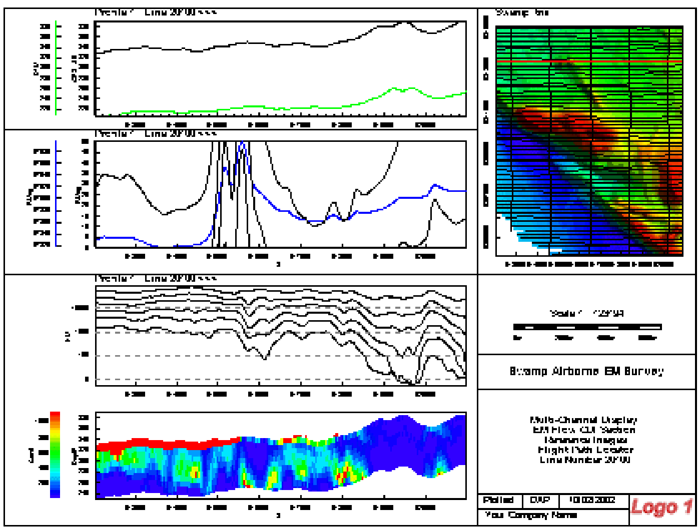
Examples of complex displays are included below. In each case the required templates or components to create displays of this type are supplied with your installation of Encom Discover PA.



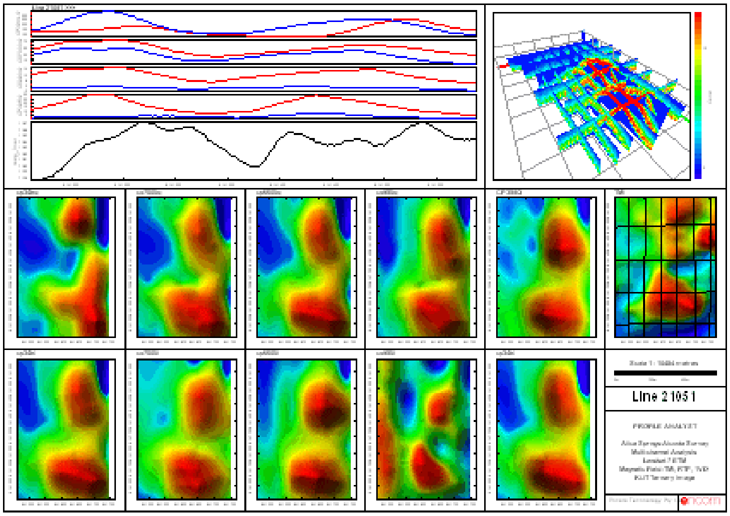
Combined image maps (including satellite imagery) with profile data and complete *annotation*



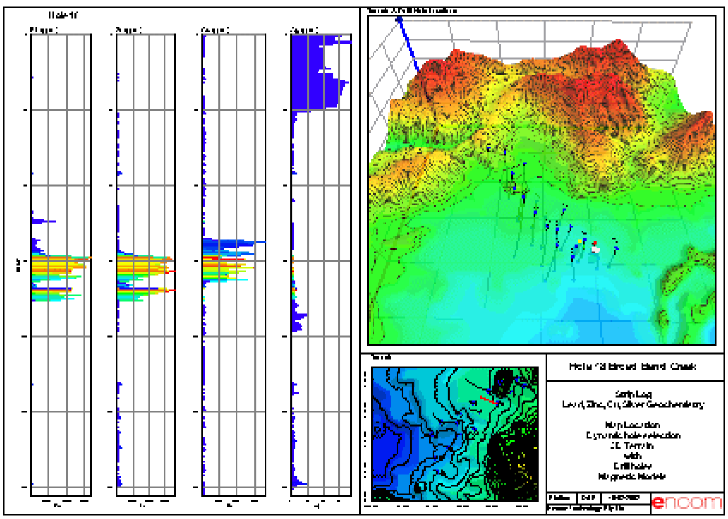
Variation of the display above with different components added



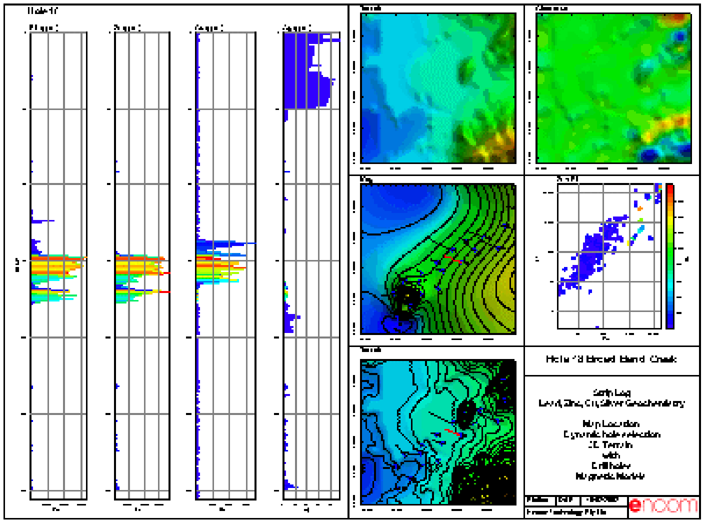
Profiles with EM data, section and maps for location of the data line



Multiple images of different resistivity slice images plus a profile and 3D section display



Drillhole log plots with 3 dimensional surface location view and map



Drillhole log plot woth locations, terrain, magnetics and scattergram of assay results

For more information, see *Page Layouts*.

Examples of Displays

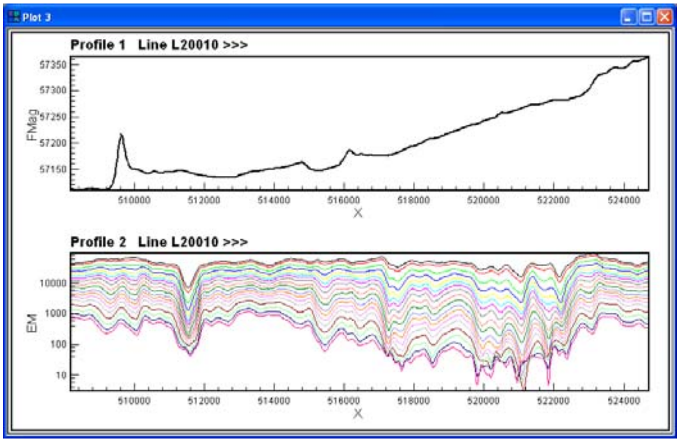
Encom Discover PA is capable of displaying complex data presentations that are tailored to your requirements. In this section are examples of many of the display format types PLUS descriptions of how the display was created. Combination plots (for example, profiles plus layers etc) are not described in this section. Such combined plots are merely copied and pasted from other documents. This flexibility to combine different displays together is a powerful asset of Encom Discover PA. For all examples, the associated Workspace tree is provided to assist in reproducing the results shown.

- *Example: Two Separate Profile Tracks*
- *Example: Two Profiles from the Same Track*
- *Example: Filling Profile Areas*
- *Example: Highlighting Major Channels*
- *Example: Modulated Data Tracks*
- *Example: Multiple Fixed Scaling Profiles*
- *Example: Adding a Depth Section to a Profile*

- *Example: Flight and Image Maps*
- *Example: Graph and Scattergram Presentations*
- *Example: 3D Grid Surface with Flight Path*
- *Example: 3D Grid Surface with Image Drape*
- *Example: 3D Grid Surface with Contours Overlain*
- *Example: Flight Path over a Grid Surface*
- *Example: Adding 3D Axes to a Display*
- *Example: Displaying 3D Located Images*
- *Example: Use of Transparency*
- *Example: Colouring a Surface with Another Surface*
- *Example: Texture Mapping*
- *Example: Multiple Surfaces*
- *Example: Merged Texture Surfaces*

Example: Two Separate Profile Tracks

This example has two tracks of different data fields created from a time domain EM database. The top track contains magnetics data while the bottom track is derived from EM multi-band data. Each EM data band has been presented with different colours.



A profile display consisting of two individual curve profiles, FMag and EM.

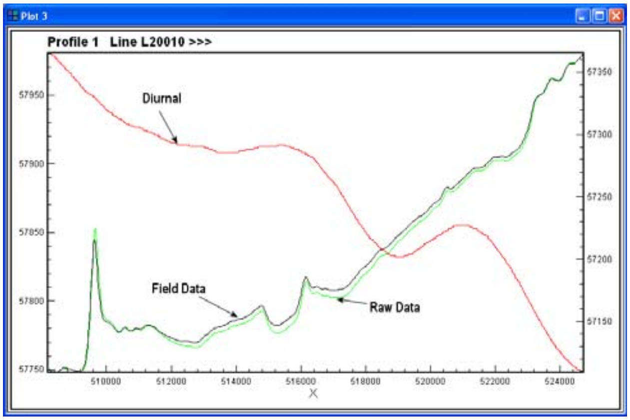
Using a new Curve Profile display (accessed from the **File>New>Profile>Curve Profile** menu item) view the Profile Properties dialog by double left mouse clicking on the Profile 1 branch of the Workspace Tree. View the Data tab dialog and choose the first line of the Swamp_MagEM database. Click the Apply button to update the display. Then View the Fields tab dialog of the same Profile Properties and choose the FMag field from the drop-down list available. Display the properties dialog from the Curve branch and in the Line Style tab dialog turn off Modulate Line Colour and edit the Default Line Style. The upper track should now be completed.

The next step is to highlight the Profile 1 branch, and Copy it using the pop-up menu from a right mouse click. Select the Profile Frame branch and again use a right click to Paste the copied track. A second track now displays beneath the first.

View the Profile Properties dialog for the 2nd profile and select the Fields tab dialog field to nominate the EM field. To select individual bands of this multi-channel field select the Y-Bands button to the right of this field drop-down list and select/deselect the appropriate bands.To make the EM data channels have different colours, select the Line Style tab from the Curve option and enable Cycle unique colours. Symbols can also be enabled from the Symbols tab if desired. Different scaling (Logarithmic) can be controlled from the Profile branch properties dialog in the Y Scale tab.

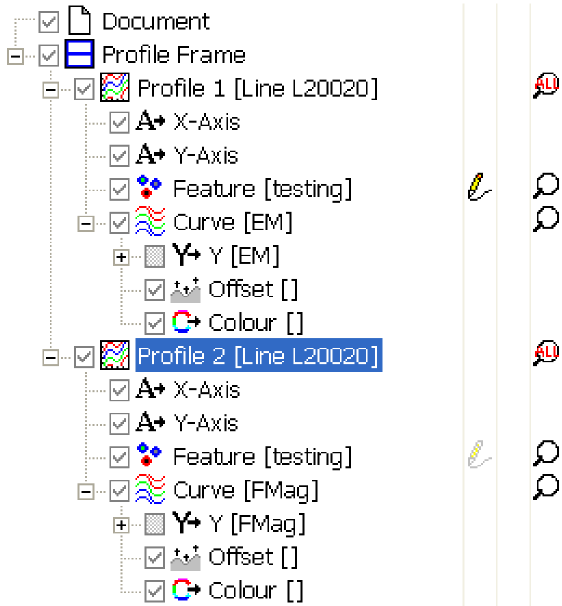
Example: Two Profiles from the Same Track

Comparison of two or more data fields can be done within one data track. Usually the data ranges and sampling interval of the various data fields are not compatible. Following is an example of data fields with different data ranges. In this case, diurnal data is approximately 400 nT greater than the final and raw magnetic profiles. In addition, the altimeter data ranges from 110 to 130 metres.



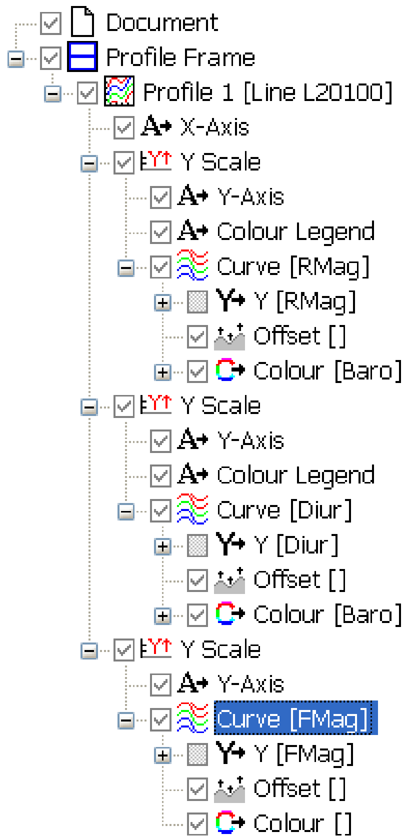
Magnetics comparative data (final, raw and diurnal)

Display a new Curve Profile and in the Profile Properties dialog ensure that the Swamp_MagEM database is chosen in the Data tab and then choose the Fields tab to nominate the FMag field from the drop-down list available. In the Curve branch Properties dialog deselect the Modulate Line Colour option and choose any colour from the Default Line Style button. Repeat this operation for the Colour branch but deselect the Band 1 entry. The upper track should now be completed and can be displayed in whatever colour or symbol pattern is required.



Example 2 - Multiple Profiles in a Single Track

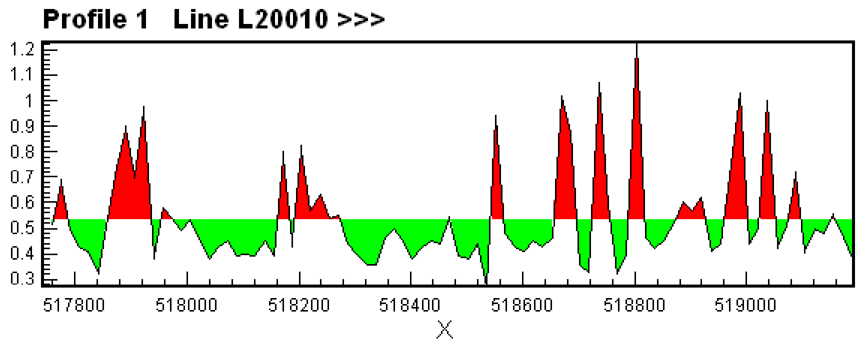
Since the second profile is to be added to the same graph (but with a new Y Axis), select the Profile 1 branch of the workspace tree and display the pop-up menu. Choose the **Add Axis Scale>Y Scale** option. This adds a new Y Scale branch to the Profile. The next step is to add a Curve branch to the Y Scale. This can be done with a cut and paste operation from the existing Curve branch. Alternatively, select the Y Scale from the pop-up menu and choose **Add>Curve** or press the **Add Curve** button on the Data Objects toolbar. The new curve can then be re-configured from the available data fields. This time add the Diurnal field. Some adjustment to the Y Scale properties may be required to position the axes. Axis scales can be changed manually from the Y Scale tab of the Y Scale branch using the Fix Y Range option.



Since the second profile is to be added to the same graph (but with a new Y Axis), select the Profile 1 branch of the workspace tree and display the pop-up menu. Choose the **Add Axis Scale>Y Scale** option. This adds a new Y Scale branch to the Profile. The next step is to add a Curve branch to the Y Scale. This can be done with a cut and paste operation from the existing Curve branch. Alternatively, select the Y Scale from the pop-up menu and choose **Add>Curve** or press the **Add Curve** button on the Data Objects toolbar. The new curve can then be re-configured from the available data fields. This time add the Diurnal field. Some adjustment to the Y Scale properties may be required to position the axes. Axis scales can be changed manually from the Y Scale tab of the Y Scale branch using the Fix Y Range option.

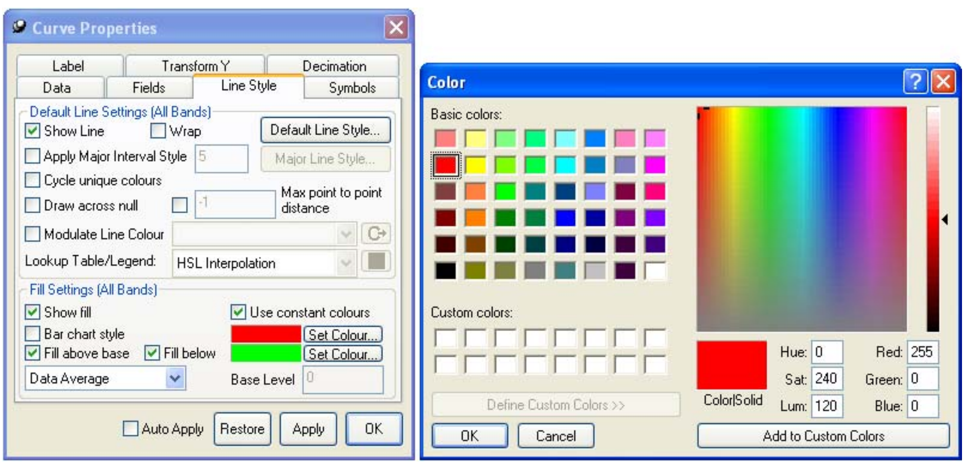
Example: Filling Profile Areas

Particular line styles can be used to enhance the appearance of profile presentations. To fill above or below in a profile can highlight certain trends or anomalies in data. Below is an example of a colour-filled profile.



Example of colour-filled profile

The fill properties are controlled by the Line Style tab of the Curve branch properties dialog. This dialog enables filling from a specified level determined by a variety of means (average, mode, median, minima etc). The dialog permits colour filling both above and below the profile trace with different colours.



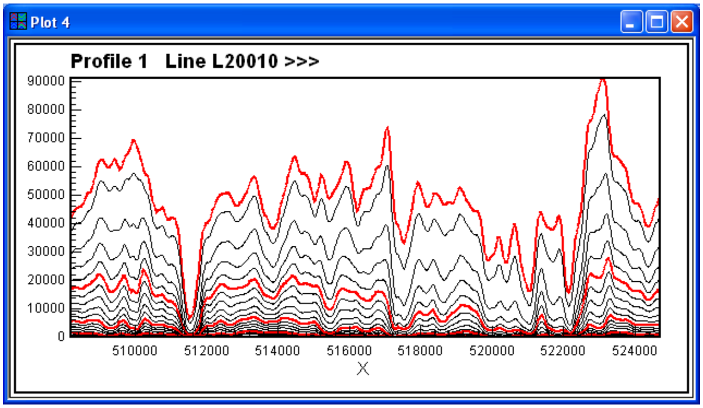
Curve properties dialog to fill above and below a defined level

In one profile, the fill above is set to a colour of red, while the second has filled below with a green fill colour.

Example: Highlighting Major Channels

Particular line styles can be used to enhance the appearance of profile presentations.

A second example presents multi-banded data where major and minor data traces have different styles. The Line Style tab of the Curve branch properties dialog is used to control this style.



Use of the Line Style tab to control major and minor data traces

The Apply Major Interval Style option controls the cycle interval. Minor traces assume the default line style, while the Major Line Style control is the major trace style.

This form of control is often used in time domain EM geophysical survey data displays.

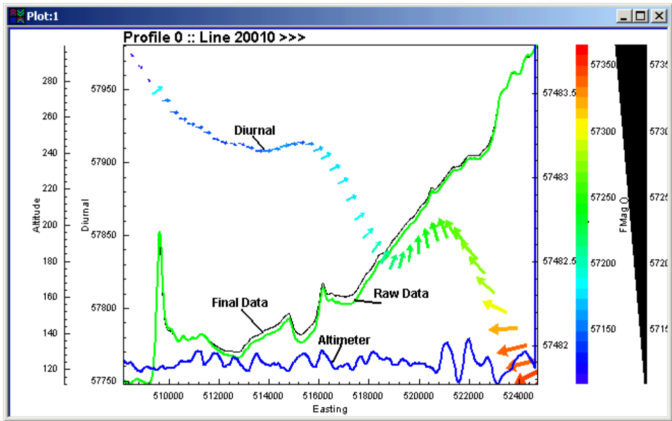
Example: Modulated Data Tracks

Individual data tracks of a profile can be altered in their appearance by colour, symbol size, symbol rotation. This process is called modulation.

An example of symbol modulation of the Diurnal data is shown in the following example. In this example, the line trace has been turned off and the arrow symbol is modulated by rotation, length and colour.

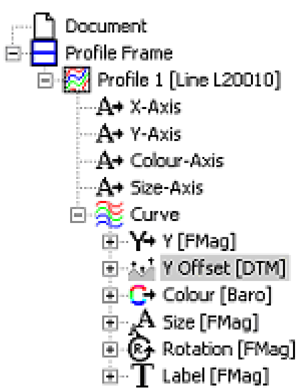
Symbol attributes such as colour and size are controlled by data fields. The symbol controls are activated by selecting the Show Symbols option in the Symbols dialog tab of the Curve branch properties dialog.

Many applications can make use of modulation. Examples include geochemical data presentations, amplitudes of geophysical data, relative rotational information such as EM components or drillhole information.



Modulation of the diurnal data for colour, size and rotation

Scaling of symbol sizes, colour ranges, units and increments of rotation etc can all be controlled from the properties dialogs. The Workspace tree for the modulated trace is shown below:



The modulation controls (that is, data fields that are used to change the appearance of another data set) are specified as data fields (in this case, all modulation is by the Final Magnetics data (MAG)).

Modulation of the appearance of symbols is achieved by assigning data fields to the Colour, Size, Angle or Label branch of the Workspace tree. Initially these branches (except Colour) are not visible. They become visible when the Show Symbols item in the Symbols tab of the Curve properties dialog is enabled.

Profiles may have Depth Sections associated with them. Sections contain layers derived from external processing results. Data is supplied either by defining layer depth tops and bottoms OR by point depths with associated physical properties such as conductivities (to produce CDIs (Conductivity Depth Images)) or magnetic susceptibility sections etc.

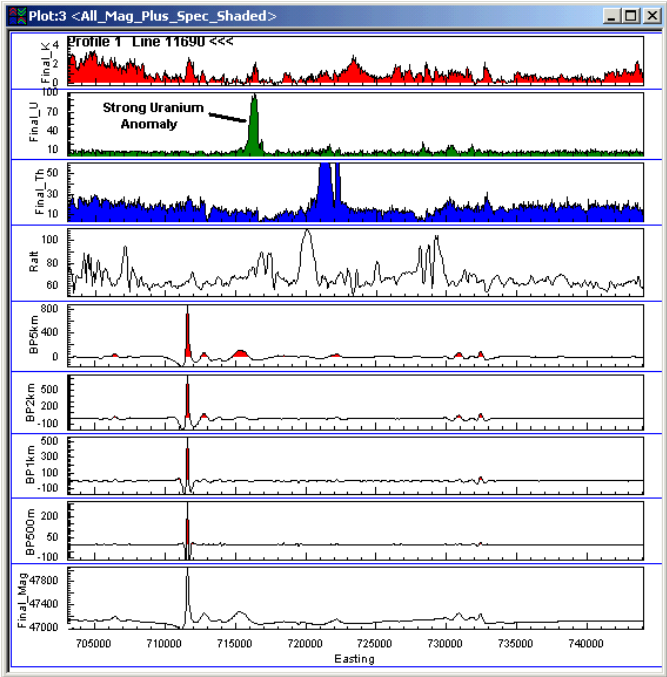
Many applications can make use of modulation. Geochemical presentations, amplitudes of geophysical data, relative rotational information such as EM components or downhole information.

Example: Multiple Fixed Scaling Profiles

The majority of geophysical data is collected along lines and quickly converted to grids for subsequent image analysis. Subtle targets may go undetected when we rely on imagery alone. Profile analysis allows an interpreter to derive additional information that cannot be resolved in images. Examples include simultaneous visualisation of more than 3 channels of data, subtle shape information in curves, comparison of unrelated data types, and fine detail that may be lost during gridding. Until the arrival of Encom Discover PA, the detailed analysis of profile data from large surveys was slow and image analysis provided the only practical method for analyzing large quantities of survey data.

A frequently used and powerful technique for examining complex interactions of data within large surveys is to display adjacent tracks with combinations of displays and then sequentially cycle through the survey, noting features as you progress. An example of this form of sequential line presentation is shown below.

Important in the display style shown is the fixing of vertical scales. This means that amplitudes of tracks can be related between traverse lines in absolute terms. This feature is controlled by the Fix Y Range option of the Y Scale tab of the Profile branch property dialog. Other relevant features are the dismissing of X axis labels, annotation and expansion of the profile height to fill the available track space. This track height effect is controlled by disabling the X Axis title and division labelling plus reducing the size of the Outer and Inner Margin sizes in the Appearance tab of each Profile branch.



Example of multiple profiles presented sequentially to highlight uranium features

Example: Adding a Depth Section to a Profile

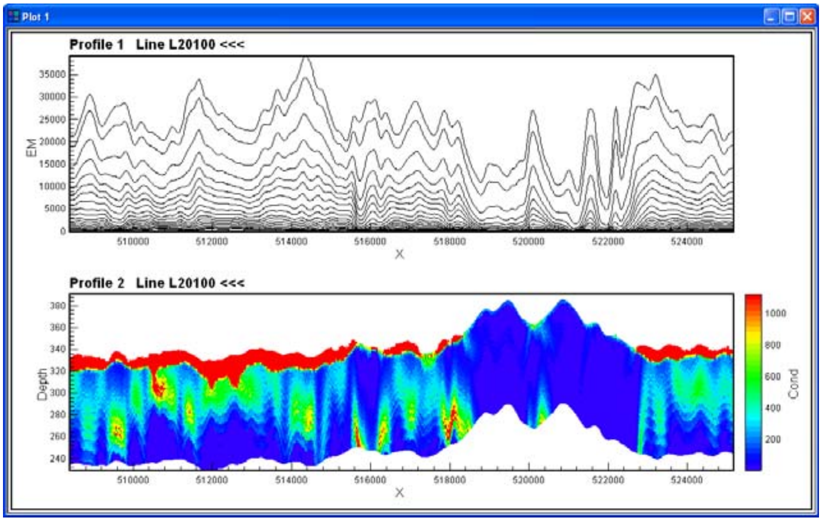
Some external processing procedures allow for data channels to be available that have a depth associated with a property (such as magnetic susceptibility, conductivity, or a solution result (from an inversion process). Encom Discover PA permits such data to be presented as a section. A section can be imagined as a slice of earth directly beneath a particular survey line.

Profiles may therefore have Depth Sections associated with them. Data is supplied either by defining layer depth tops and bottoms OR by point depths with associated physical properties such as conductivities (to produce CDIs (Conductivity Depth Images)) or magnetic susceptibility sections etc.

Depth and property data can be provided in two ways:

- Physical properties that are defined at discrete depth values. An example of this would be conductivity depth images derived from EM Flow.
- Layers with constant properties. In this case, layers require a top and a bottom boundary and the material between is assigned the physical property. Layer inversion programs such as AIRBEO produces data output in this form.

The example below represents an EM array data profile over a CDI (Conductivity Depth Image) and illustrates the example of conductivity properties defined at discrete depths.



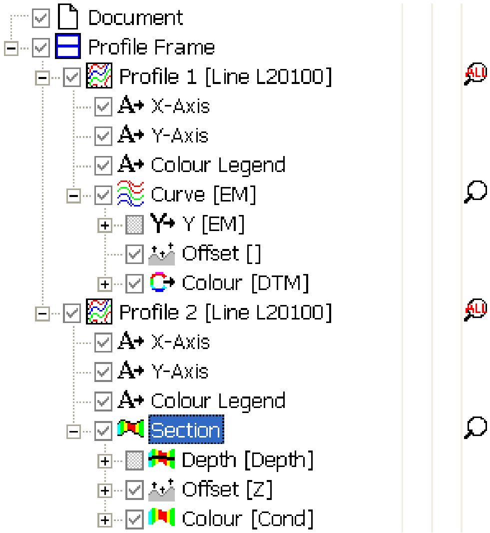
Two profiles with sections derived from two different external processing modules

To create a section in a new profile frame, select the Add Section button on the Data Objects toolbar or alternatively select the Profile button on the Documents toolbar and select the Section option. To add a section to an already created profile frame, click the right mouse button on the Profile Frame branch and choose Add>Section Profile. This action appends a Section Profile to the Profile Frame in addition to the original Curve Profile.

Profiles can be created in the normal way (see *Example: Two Separate Profile Tracks*). A number of methods can be used to include layer results. The layer data must be contained in the source database as:

Tops	Multi-band for Layers 1, 2 and 3
Bottoms	Multi-band for Layers 1, 2 and 3
Property	Multi-band for Layers 1, 2 and 3

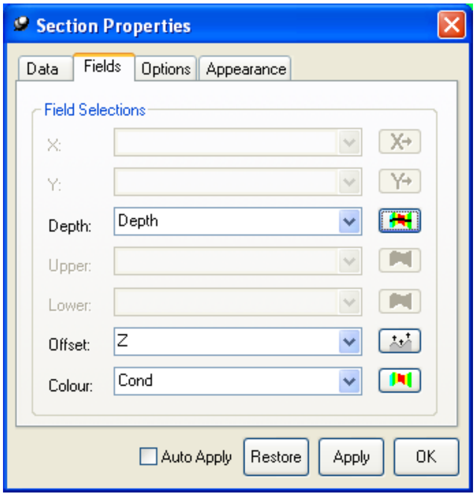
The Workspace tree indicates the Section sub-branch beneath the Profile. Associated with the Profile are Axis branches and these operate identically to those of a normal profile or graph axes.



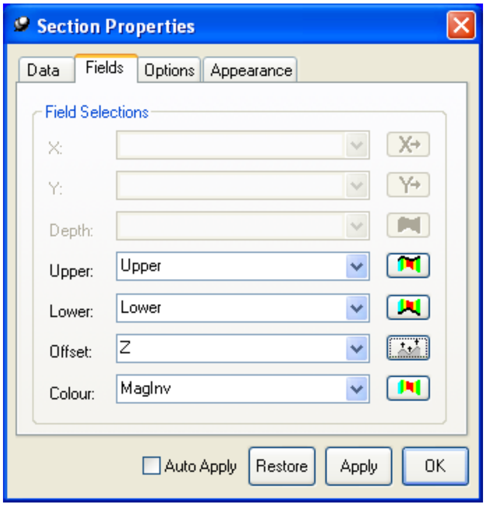
The Section branch and the data definition sub-branches beneath it control the source fields of the Top, Bottom and Colour attribute that is assigned to the layers for each reading and their physical properties to be displayed. Usually this is colour modulated. You can assign each of the required data fields for each of these layer requirements to display the section.

Assign the Depth field (as shown in the dialog below) to a multi-banded incrementing depth field and a multi-banded field for the property (such as resistivity or conductivity) to the Colour Field.

Note Two Section layer construction types are supported. These using either a Depth field or Upper/Lower Field data structures. See *Section Structure and Layers* for additional information.



Section Properties for a database that uses a Depth and Property field classification.



Section Properties for a database that uses a Upper and Lower and Property field classification.

If topography is to be placed along the top surface, this can be done by allocating a data field to the Offset branch. The effect of this is to move all the top and bottom data values for each reading up and down as defined by the amount of individual reading values within the specified Offset data field.

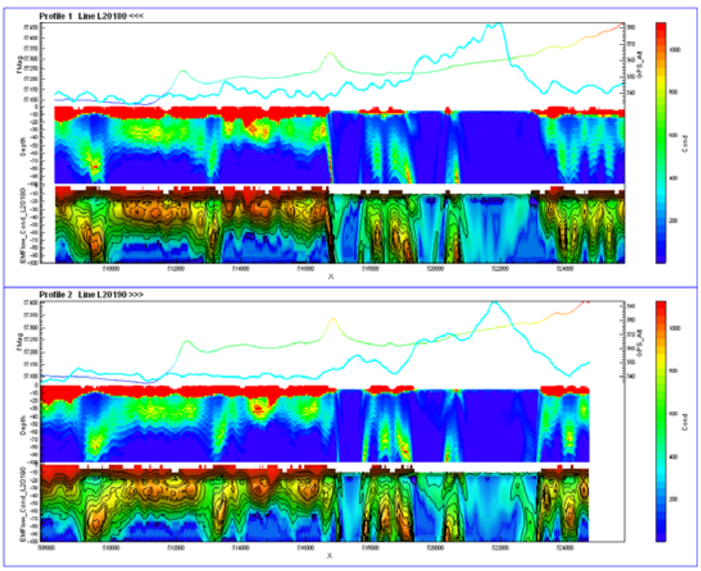
When displaying sections, each layer of each reading of the sections is separately drawn. These displays therefore are graphically intensive to draw and can be slow. Faster drawing methods can be used to create sections. These include:

- Using the Located Image facility of the *Batch Printing Wizard* that creates a located image of each section which can then be used in profiles or 3D displays (see *Adding Located Images*).

- Gridding the section so that standard grid formats can be used to display the section. These formats offer the advantage of allowing shading and contouring of layered sections. See *Adding Section Grids*.

Profiles can also have Section Grids added. These items are created as grids (using a Section Grid utility from the **Gridding** menu) and can be used instead of layer sections (CDIs). Section Grids have advantages over Sections in that they are faster to display, plus they can have real-time shading and labelled contouring.

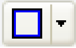
IP pseudo-sections can be displayed using Geosoft IP databases as input to gridding. The displayed sections retain the usual pseudosection style and can be used for resistivity, chargeability and metal factor plots in conjunction with other conventional data stored in the databases. Section Grids can also be used in 3D maps.



Like section profiles to display Section Grids in 3D displays they must first be converted to located images..

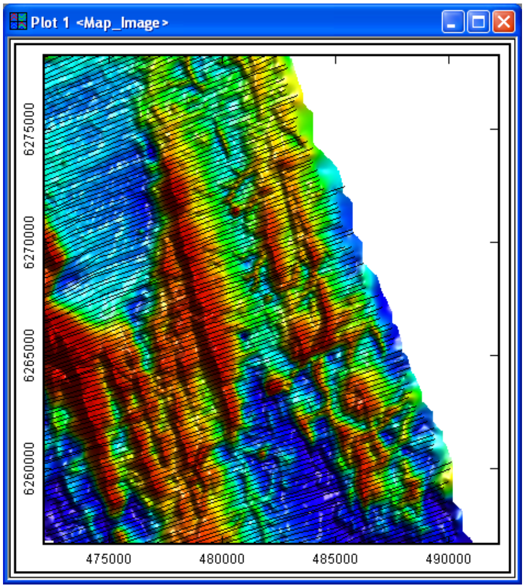
Example: Flight and Image Maps

Flight path and image maps are a valuable method of displaying data. The flight path map also has significant additional value in being used to navigate through a dataset with direct links to displayed profiles.



Initially, a Flight Path map is created. This is best done from the **2D Flight Path Map** pull-down button or from the **File>New>Map>2D Flight Path Map** menu item. Initially a flight path traverse map is displayed as derived from the initially accessed database. You can control the database to use from the **Dataset** tab of the Map properties or the **Flight Path** tab from the properties of the Flight Path branch. Configuring of the map can be done from the properties of the Flight Path branch of the Workspace tree.

To add an image to the map, highlight the Map branch of the tree and click the right mouse button. Select the **Add Data>Grid** menu item. A new Grids branch is added to the tree. From the properties of the Grid Group branch, you can specify Pseudocolour, RGB or contour layers and then add image layers from multiple grids or from individual bands within multi-band grid datasets. Even image enhancements derived from ER Mapper are supported and can be used by Encom Discover PA as ER Mapper algorithms.

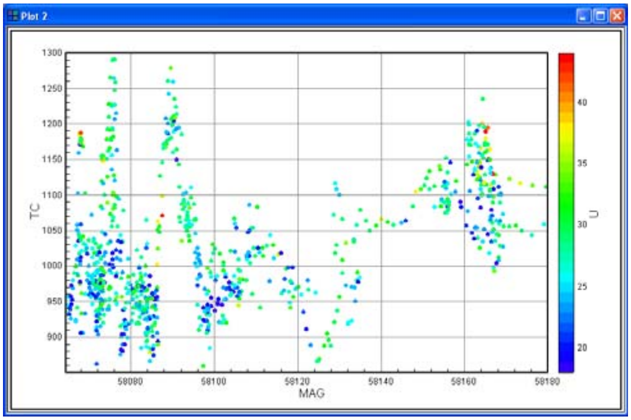


An example of a flight path map with a pseudocolour image of magnetics

Review the Display an Image Map item to review the controls of images with instructions for inserting and specifying images.

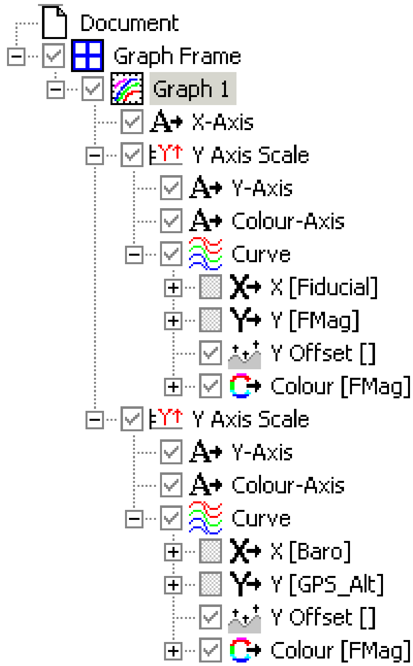
Example: Graph and Scattergram Presentations

Graph documents are similar to profiles except the horizontal axis has additional control to allow data fields with no spatial (distance) properties to be used. Examples of graph use could be for the plotting of airborne data profiles using time or fiducial as the horizontal axis. Another advantage of the graph style of display is the ease with which you can display multiple curves in a single track with each curve having different vertical scales.

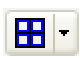


An example of a Scattergram document

This form of display can be done in the profile document but it is not as easy due to the inherent spatial dependency of the horizontal axis. The Workspace tree for the display above is shown below.



Workspace Tree of a 2D Graph with multiple Y axis scales applied

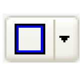


The initial graph is created from the pull-down button (and choosing the **2D Graph** option) or the **File>New>Graph>2D Graph** menu item. Initially a blank display is presented but the tree has field definition levels for **both X** and **Y** axes. You can individually specify these as desired using the Field selection tab dialog.

Once a single curve is created in the document, a second can be added from the pop-up menu (from the Graph branch) and selecting the **Add Axis Scale>Y Scale** option. With a new Y scale, create a new Curve by selecting the new Y Scale, display the pop-up menu and choose the **Add Data>Curve** option. The X and Y data fields for the second curve can then be specified to be different from the first curve.

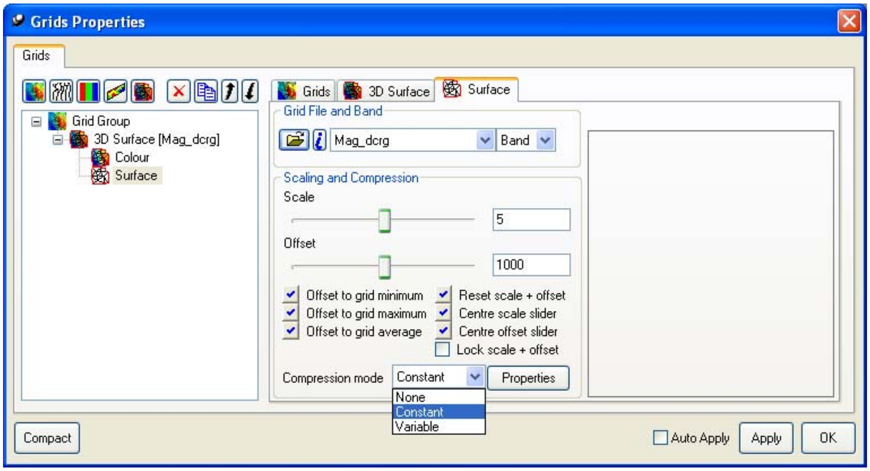
Some re-scaling of the Y-Scale levels may be required to apply any offset or positioning of the second track. This is from the **Y Scale** tab of the properties.

Example: 3D Grid Surface with Flight Path

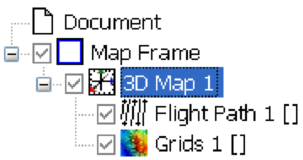


To display an image surface with a flight path above it, select the **File>New>Map>3D Grid Map** or select the **pull-down** button and choose the **3D Grid Map** option. From the Grid properties dialog a grid surface is specified and displayed. The vertical exaggeration (scaling) and offset of the surface can be applied if required from the **Surface** tab dialog of the Grid Properties dialog.

To add a flight path to the image, select the Map Frame and click the right button to display the pop up menu. Choose the **Add Data>Flight Path**. The resultant Workspace tree appears as:

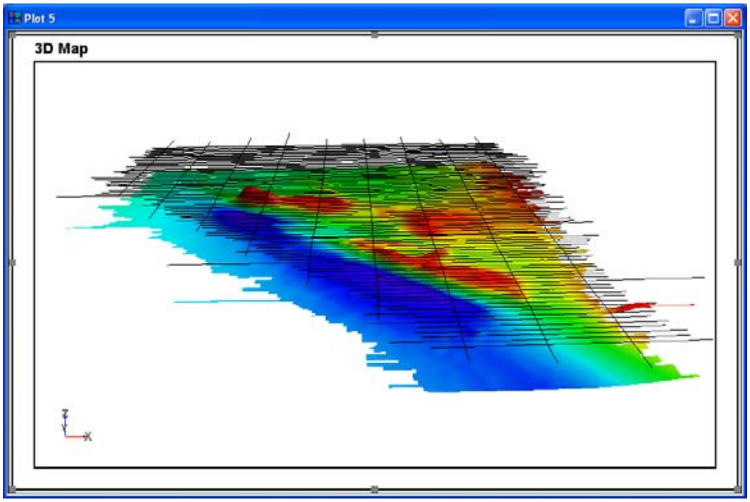


Grid properties dialog for 3D surface showing the Scaling and Compression parameters.



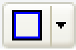
Typically, there needs to be some adjustment between the vertical scaling and offset of surfaces to best suit other surfaces or flight paths etc.

The resultant display appears as below. Note that navigation of the three dimensional view is described in 3D Navigation.

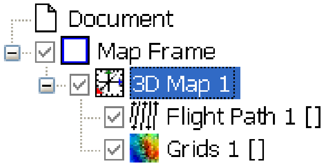


Three dimensional surface with flight path of airborne survey superimposed

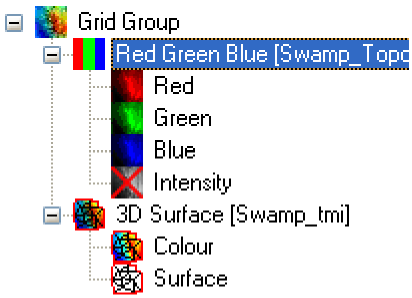
Example: 3D Grid Surface with Image Drape



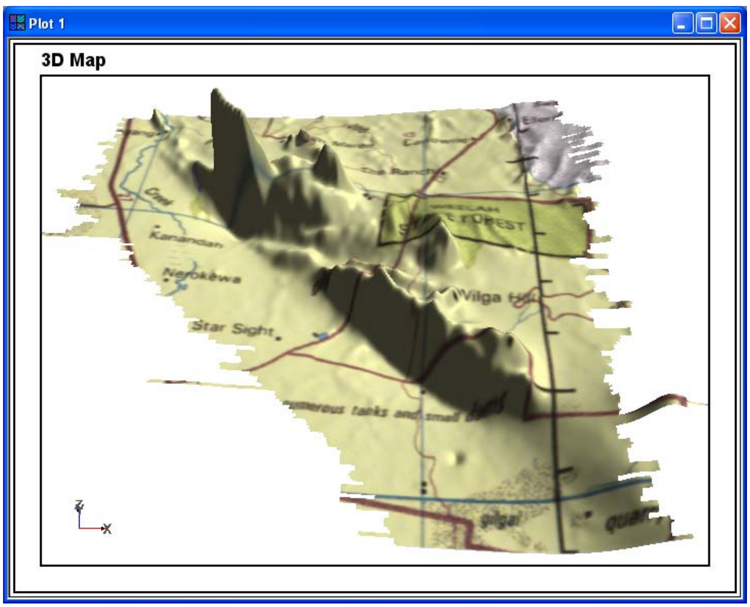
Combinations of two image surfaces can be used with one image creating the colour surface while a second is used for the elevation. In the case shown below, a Red:Green;Blue scanned image (Topo) is used to present the image surface and a second (magnetics) grid is used to apply the elevation effect. The entire display is derived from a single 3D Map layer in the Workspace tree created from the **File>New>Map>3D Grid Map** or select the **pull-down** button and choose the **3D Grid Map** option. When this is done the Workspace tree appears as shown.



In the Grid Properties dialog, the Grid Group tree should show a 3D Surface of a grid (e.g. DTM or magnetics) with a Red:Green:Blue surface located on top of this showing another image file (e.g. Surface Geology or Topography).



In the example three dimensional display below the Surface branch of the tree uses the magnetics grid for elevation of the image while the Red:Green:Blue references the scanned Topography image to render the coloured surface.



Two image surfaces, one used to control elevation and one the colour surface

Example: 3D Grid Surface with Contours Overlain

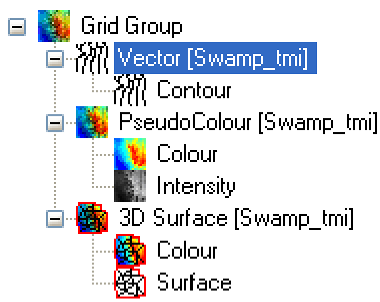
Contours (as vectors) can be added to a surface but a method must be used to do this that is not initially obvious. When a surface has any other entity above it, it reverts to a texture mapped surface. When texture mapped, it has no colour property and so when contours are added to a surface, initially the surface displays as grey. To make the surface coloured, a Pseudocolour must also be added.

To display contours on a surface:

1. Create a 3D map and nominate a surface image (either Pseudocolour, RGB or Located Bitmap Surface). See previous example.

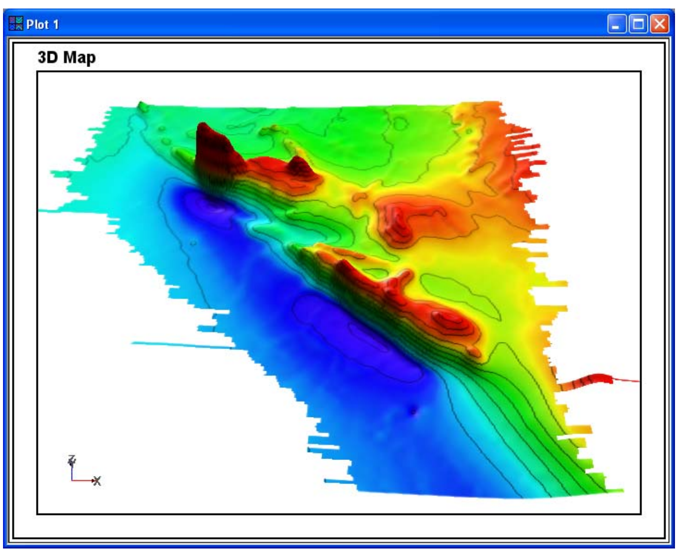
2. Duplicate the Surface twice and make the middle layer Pseudocolour and the top layer a Vector-Contour. This is done by using the **Duplicate** button and then clicking right on the **Surface** branch to alter it to **Pseudocolour** and **Contours**.

   The Grid Group tree should now appear as shown below:



3. Ensure each layer is accessing the image required by pathing to the relevant grid.

4. Scale or vertically apply exaggeration as required:



Contours displayed over coloured surface

Example: Flight Path over a Grid Surface

Frequently for quality control of a magnetic or geophysical survey it is useful to visualise the aircraft track over a surface such as a digital terrain model or grid of the recorded sensor (magnetic) surface. It is a simple matter of creating a Curve Profile display above a surface to illustrate this form of display.

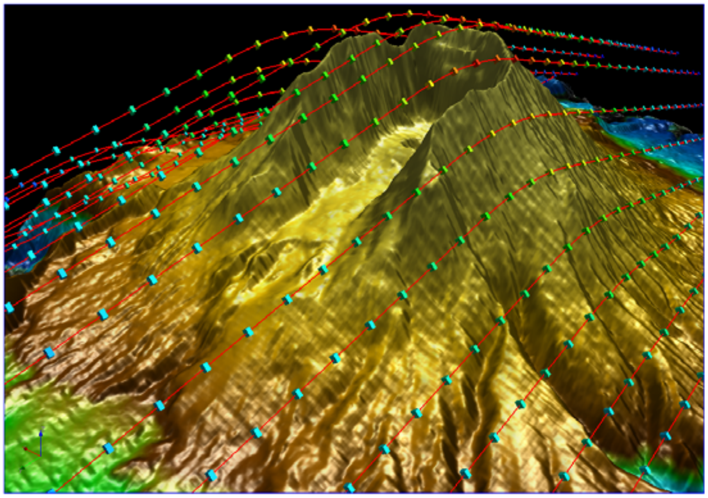
To create a flight path over a grid:

1. Create a 3D map and nominate a surface image. See examples above.

2. Select the **3D Map** branch of the tree and click right to display the pop-up menu. Choose the **Add Data** item and Lines. From the **Fields** tab nominate a data channel to provide vertical offset of the display trace.

3. Select the **Line Style** tab and control the method of colour modulation or the colour itself. Initially only the first data line is displayed with the image surface. You may need to adjust the vertical offset between the surface and the lines. This is done from the **Fields** tab of the Lines property dialog and setting an Offset entry.

4. With the setting of a single line specified, you can then select the **Lines** tab and specify which lines are to be displayed with the surface.



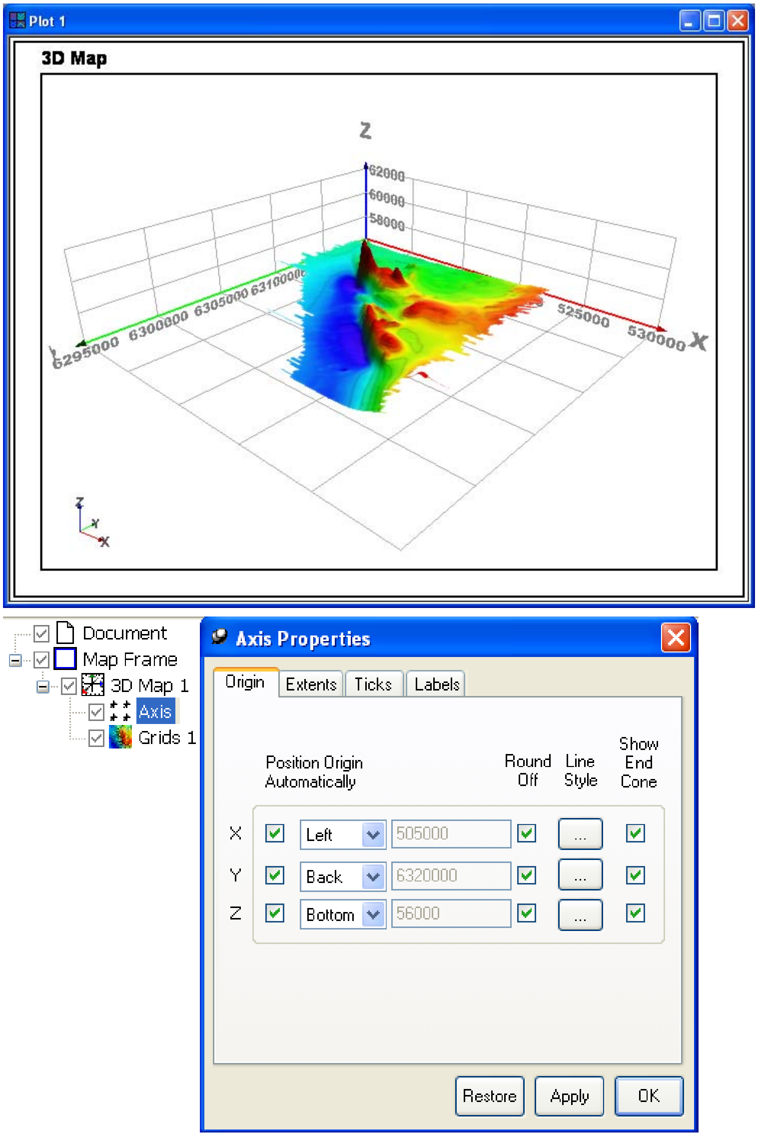
An example of a ground clearance colour modulated flight (Line Series) over part of a topographical surface

Example: Adding 3D Axes to a Display

Three dimensional displays are powerful tools in assisting the understanding of the positions of such things as drillholes, sections, geological features etc but they require a clear understanding of their location both in easting/northing and depth. Encom Discover PA allows you to add axis annotation in 3D displays to locate such features.

3D Internal Axes

To determine the location of features in a 3D display, add axes. With the 3D Map item highlighted in the Workspace tree, click the right mouse to display the pop-up menu and select **Add Axis>3D Axis**. With this done, the primary axes (East – X, North –Y and Depth – Z) are drawn into the display.



A 3D grid surface display with Internal 3D axis and associated Workspace Tree and Axis Properties dialog.

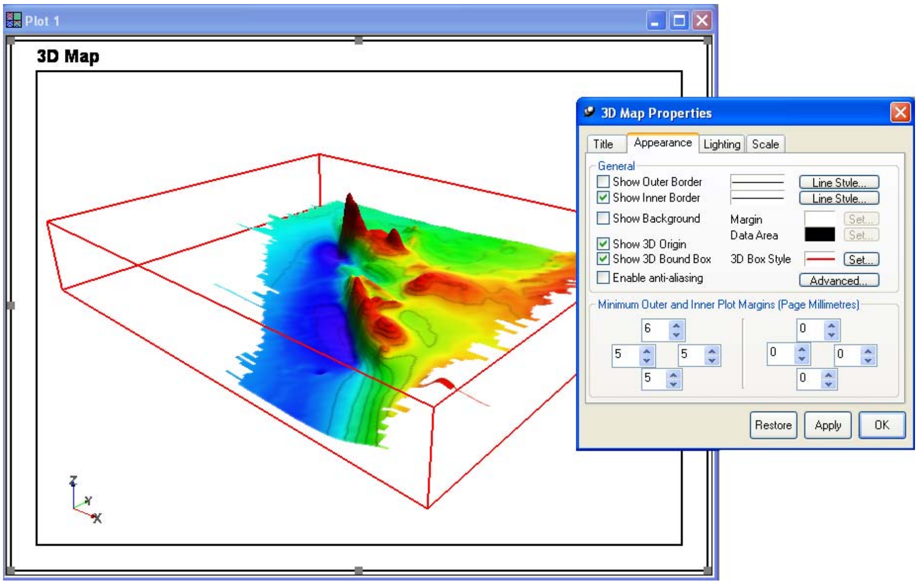
The 3D Axis dialog can be displayed from the properties of the tree level. This dialog allows you to control all aspects of the displayed axes. You have control for:

- Axis location (Left/Right/Centre), (Top/Back/Centre) or (Top/Bottom/ Centre)
- Axis labelling, font and frequency

- Axis line and increment style (major and minor linework)
- Axis origins (as related to axis locations).

Bounding Box

In a standard 3D display, you can add a bounding box from the Appearance Tab of the 3D Map branch of the Workspace tree. Enable the Bounding Box check **Show 3D Bound Box** in the **Appearance** tab. The colour, line style and other attributes of the display can also be defined.



Bounding Box enabled in a 3D display

Example: Displaying 3D Located Images

If a bitmap is to be used in a three dimensional display, the bitmap is required to be accompanied by an Encom Georeferenced Image file (EGB). This file defines the image source file by name, its type and its corner locations.

Externally created images of seismic sections, IP sections, geological interpretations etc can be located and placed within a 3D display.

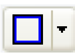
In this example, a magnetic surface is viewed with a series of parallel geological interpretation sections. These sections have been derived from drillhole interpretation in a GIS package. As for all the 3D views the display can be manipulated and moved to see the view from the best location. You can fly-through the surface and sections and obtain a perception of the relationship between the geology and geophysics readily.

Located images can be displayed within a 3D view by using the Located Image data option in the Workspace tree. Located images are positioned by using an ancilliary file called an .EGB file.

Creating a 3D Located Image Display

To create 3D displays with bitmaps it is initially required they be created within a map. Either a 2D or 3D image or flight path map display can be used. In this example a 3D Flight Path Map display is described.

To create a 3D located image display:



1. Create a 3D Flight Path Map or 3D Grid Map using the **File>New>Map>3D Grid Map** or **3D Flight Path Map** menu option (or from the appropriate pull-down buttons).

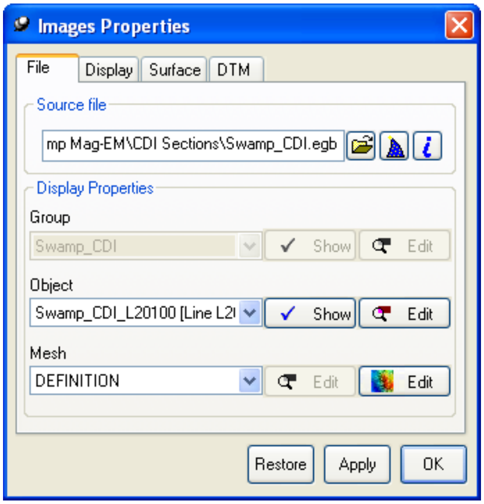
2. With a Map created, select the Map branch of the Workspace tree and click the right mouse button. From the **Add Data** menu item, select **Grids**. Specify a raster surface and then manipulate the surface as required with suitable vertical exaggeration and offset. Note that the data values of the displayed grid determine the vertical position of your display and this is important when displaying located images with a different vertical position.

   For example, if you wish to display a magnetic grid (with average data values of say, 55,000nT) combined with geological sections (with vertical reference of say 1-200 metres) as shown above, then an offset is required to bring the two data objects into a similar vertical reference frame. Typically therefore, you would need to reduce the vertical location of the magnetic grid down by about 55,000. The steps following describe this procedure.

3. You will have needed to prepare the necessary images before display and these require associated EGB descriptor files (see *Appendix B: Data File Specifications*).

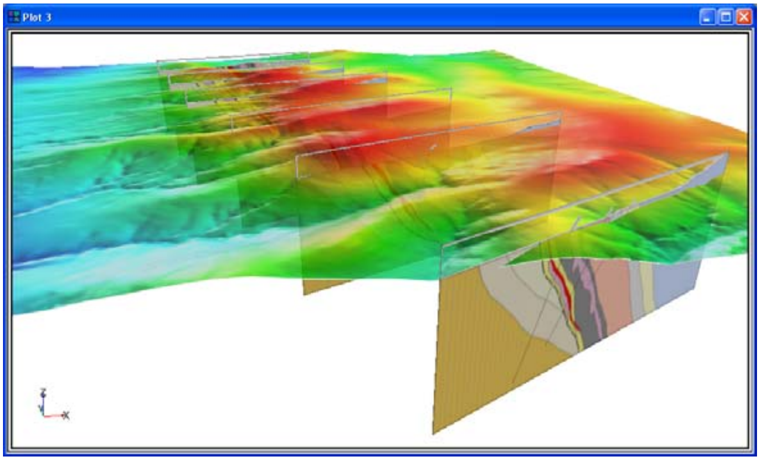
4. Select the 3D Map branch of the Workspace tree and click right. Select the **Add Data>Located Image** option. An Images branch will be added to the tree associated with the 3D Map.

5. Display the properties dialog of the Located Images branch. Initially specify the path and EGB file. You can use the Browse button to navigate to the required EGB file.



Located Bitmap properties dialog to specify the EGB file and display options

6. If the displayed map surface and bitmap are widely separated, their vertical offset may require adjustment. This can be done either by editing the vertical location in the EGB file (to offset the located image), or by interactively setting an offset amount in the surface display control. To do this, display the Grid Group properties dialog and highlight the Surface branch of the Grid Group tree. At the base of the dialog are controls to scale and offset the grid surface. Set this to an appropriate value by using the slider and increment entries.

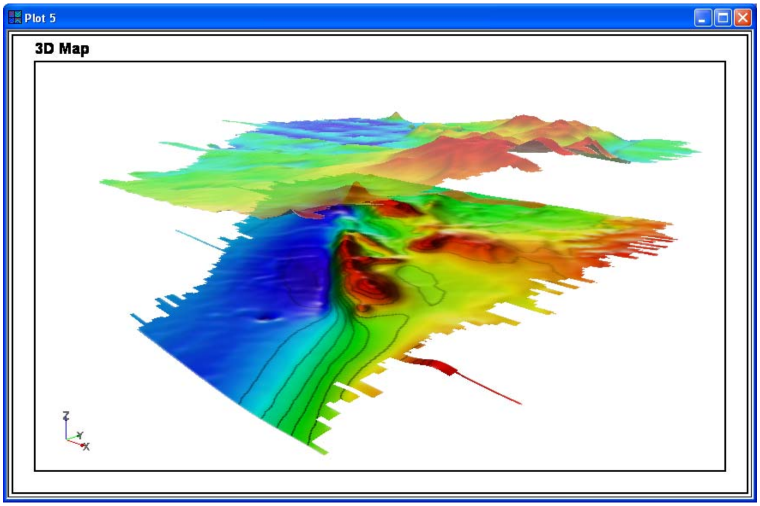


Multiple bitmaps displayed with a 3D magnetic surface

Example: Use of Transparency

Transparency is useful for 3D surfaces as it allows one layer to be seen through to view detail from a lower surface while still seeing information from the upper surface.

The Transparency control is located on the main Surface level of each Plot in the image tree. The control progressively reveals surfaces underlying a top surface.

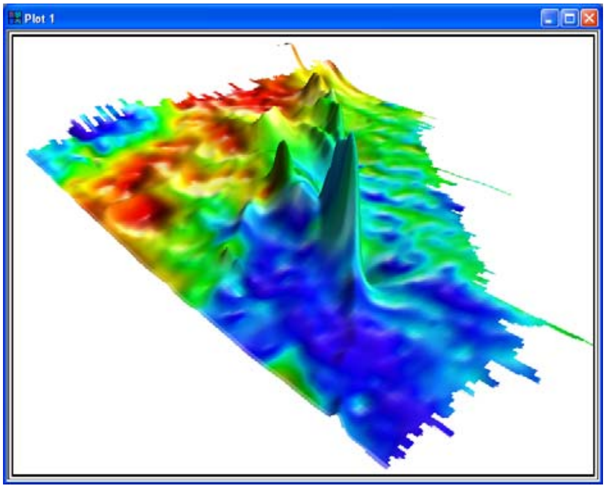


Transparent magnetics grid over elevation grid

Note that each surface can be individually scaled vertically and offset, but also, from the Map level of the Workspace tree, the Z Axis tab can be used to impose a vertical exaggeration over each surface equally.

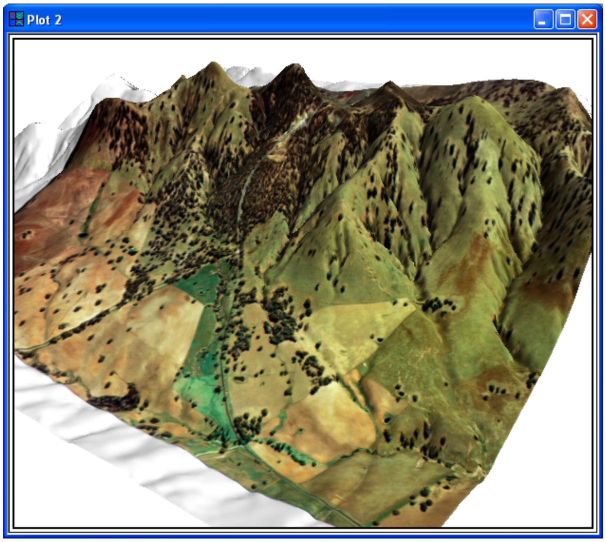
Example: Colouring a Surface with Another Surface

The colouring used for a surface need not come from the surface itself but from other surfaces. For example, if the effects of topography were to be investigated on magnetic response, the magnetic grid could be displayed (as the Depth layer) with the colour layer specified from the topography image.



Magnetics surface with colouring derived from topography

This style of display is especially appropriate for air photography or satellite imagery where elevation data is available. An example of this is shown below:

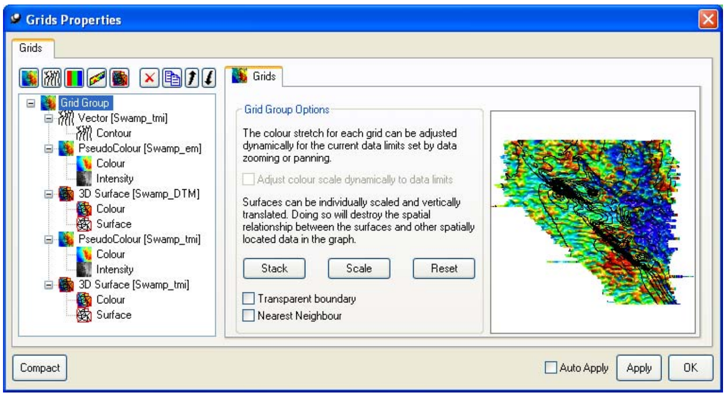


3D aerial photograph draped over a topographic surface

Example: Texture Mapping

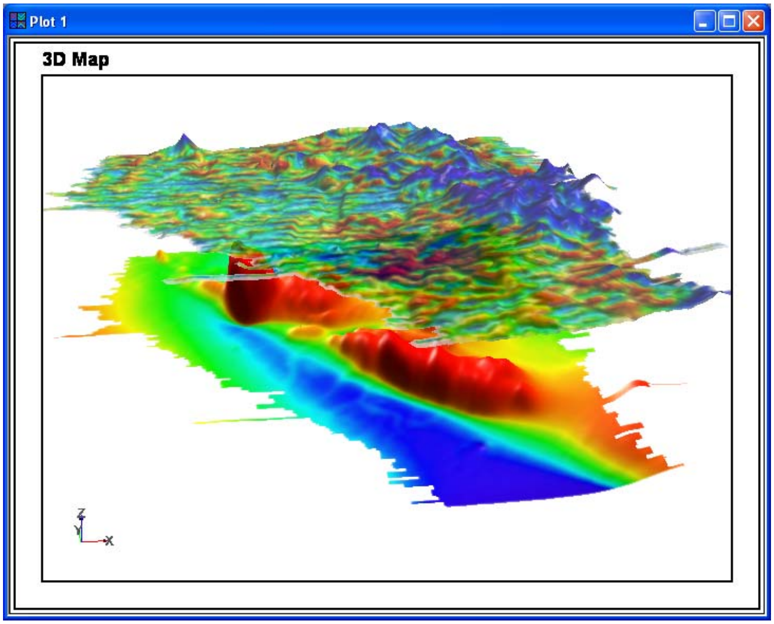
Note that displays of the type above use texture mapping. Any surfaces above a depth surface (in the Grid Properties dialog) are rendered as texture onto the depth surface. Importantly, the stack of texture mappings is capped when it hits another depth surface.

As an example, in a case with two depth surfaces, both mapped and offset, the bottom is a magnetics surface with the top having contours draped by texture mapping onto an EM image of the same area. The image properties dialog appears below:



Grids Properties dialog with two 3D surfaces and 2 draped pseudocolour surfaces plus contours.

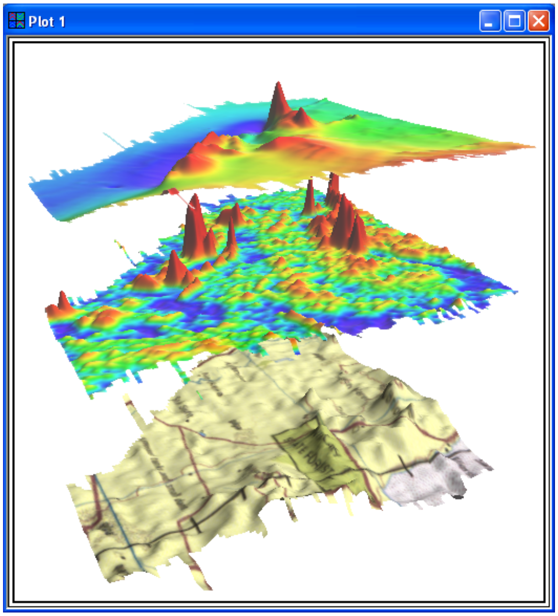
The resulting image shows the two surfaces with contours draped over the upper surface.



Two depth surfaces with contours draped over the bottom surface

Example: Multiple Surfaces

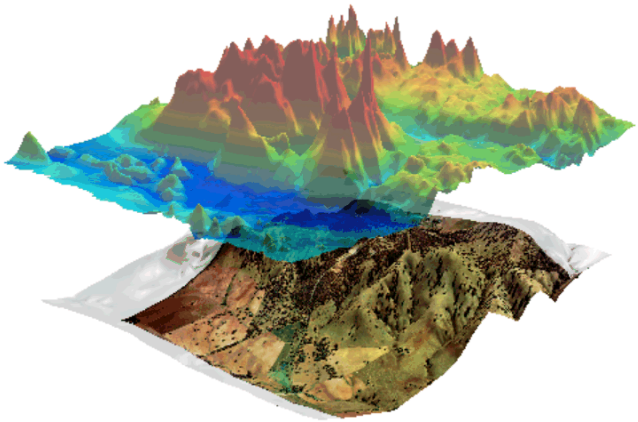
Multiple surfaces can also be displayed. This facility is especially useful for comparison of anomalous areas with different data content. Below is an example of a magnetics surface overlain by radiometrics (total count on one surface and uranium on a second).



Multiple image surfaces to allow correlation of features

Example: Merged Texture Surfaces

Combinations of merged texture surfaces can be used where different surfaces can be displayed to correlate information. Below is an example of a magnetics surface offset above an aerial photograph merged with topography.



Merged and texture mapped surfaces of magnetic above an aerial photograph draped on topography

6 Printing and Graphical Output

In this section:

- *Graphical Output*
- *Printing*

Graphical Output

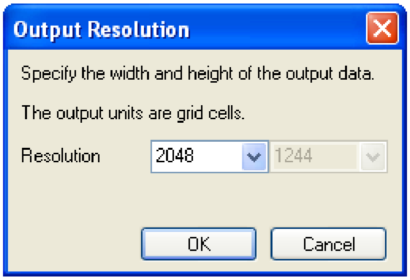
You can save the selected display in a variety of image formats:

- *Saving as an Image*
- *Saving as an RGB Raster Image*
- *Saving as a Located Image*

Saving as an Image

Any Encom Discover PA display and all its contents can be saved as an image. The supported image formats are listed in *Appendix E: Supported File Formats*. The output bitmaps can then used in external display software or reports.

Use the **File>Save View As** menu option to save the view as an image file. When selected, you are requested to define the output style of the saved image. This can be specified in image resolution (in pixels). Resolution of the bitmap being created is available in a range of pixels. Be aware that high resolution bitmaps can create large output data files.



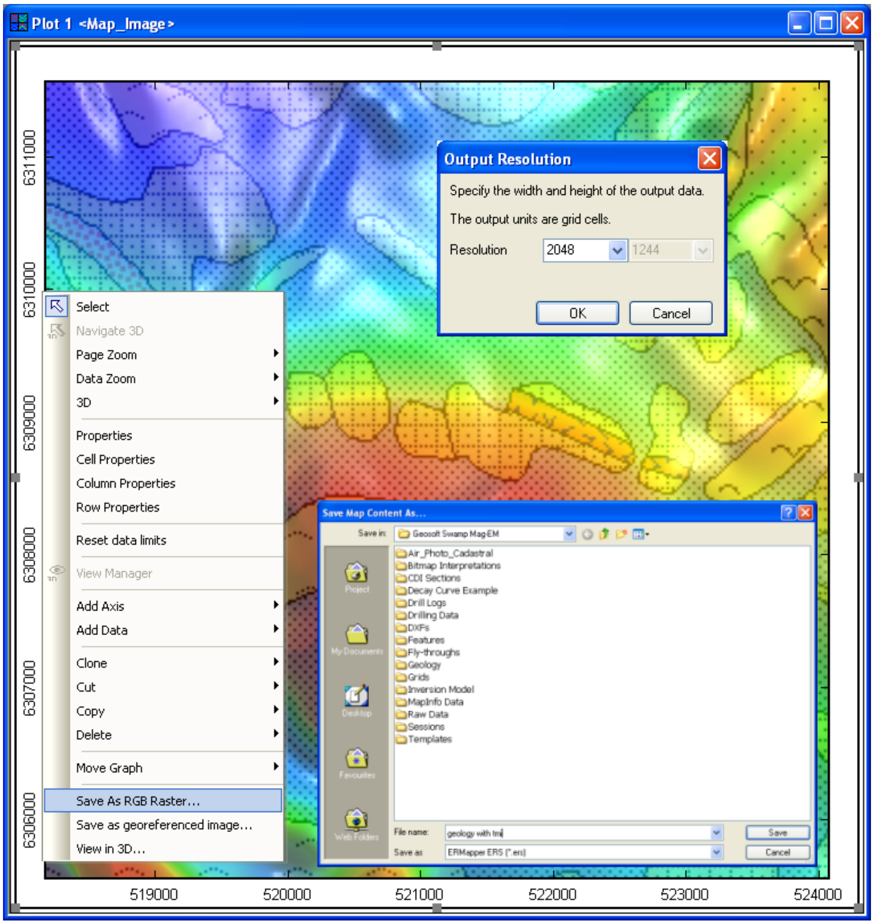
Specify the required bitmap size or resolution

You are then requested to specify a file location, name and bitmap format supported (available from a pull-down list at the base of the File Save dialog).

Saving as an RGB Raster Image

Encom Discover PA can create an external raster file in ER Mapper or GeoTiff formats. This option can only be used from a 2D image map.

To initiate this option, from a Grid Map, position the cursor within the window and click the right mouse button to display a pop-up menu. (Ensure the cursor is in **Select/Navigate** mode initially. Click the **Select/Navigate** button if not.) From the shortcut menu select the **Save As RGB Raster** option.



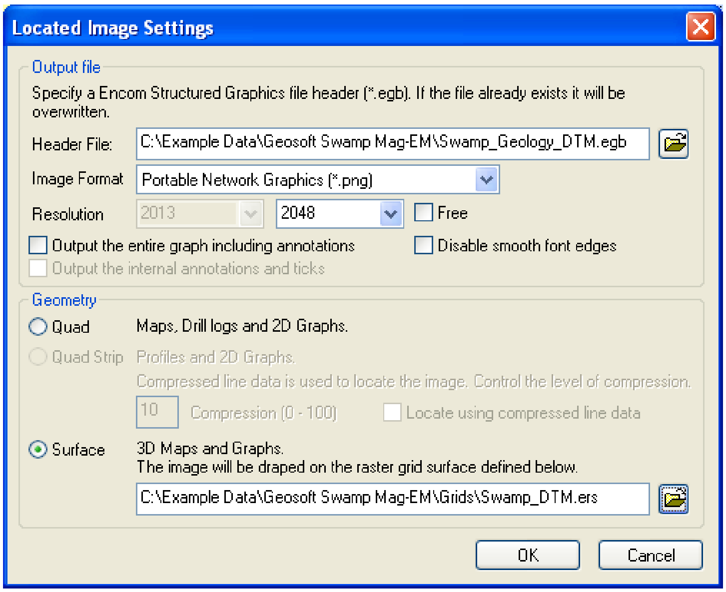
Saving a raster image of a displayed grid to ER Mapper output format

Output from this feature is to an ER Mapper .ERS file and associated image raster file or GeoTiff format. You can select the ouput resolution similarly to that described above in *Saving as an Image*. Note that any input grid format supported by Encom Discover PA can be output as a raster image using this feature (see *Appendix E: Supported File Formats*).

Saving as a Located Image

Encom Discover PA can save a display as a located image. Located images have a header file associated with the image file that is used by Encom Discover PA to geographically position the image when displayed. This form of presentation is fast and can be used as a convenient method of quickly presenting large amounts of data without required access to the data.

To initiate this option, from a 2D Map (or profile or section etc), position the cursor within the window and click the right mouse button to display a pop-up menu. Alternatively right mouse click on the Map 1 or Profile 1 branch in the workspace tree to view the same pop-up menu. (Ensure the cursor is in **Select/ Navigate** mode. Click the **Select/Navigate** button if not.) From the menu select the **Save As georeferenced image** option.



Dialog to save a located bitmap EGB file for a display

Located images can have different resolutions and formats. Resolutions range from 256 to 32768 dpi. The available image formats supported are listed in *Appendix E: Supported File Formats*.

Located images can also have three different types of geometries depending on their intended use. These geometries are:

Quad

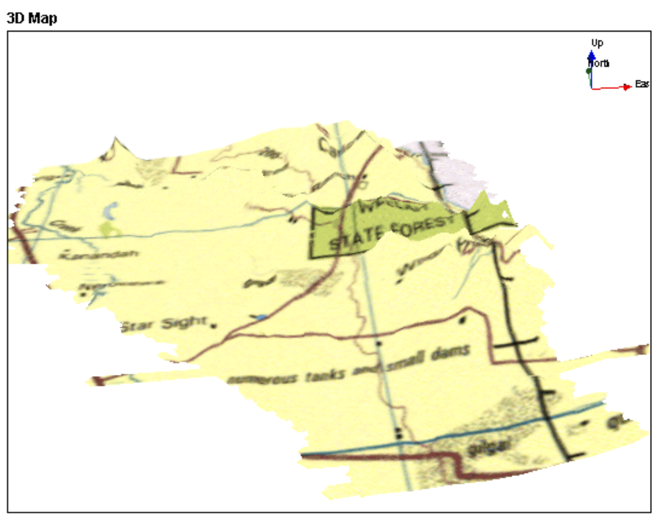
This geometry is used for display in 2D profiles and graphs and has an entry in the associated .EGB header that references the Line number from which the Located Image is created. This means that when this type of Located Image is used with profiles and graphs, the Line Iterator can be used to update and navigate through the bitmap displays.

Quad Strip

This type of Located Image also adds Line Number references in the associated .EGB header but also specifies the XYZ corner points of the bitmap such that it can be displayed in both profile/graphs AND in 3D line displays. To assist in fast 3D display, a Compression can be applied to these bitmaps. The higher the compression applied, the faster the 3D display can be redrawn but with a reduction in resolution.

Surface

This Located Image geometry places a grid surface specification in the associated .EGB header. When displayed in a 3D Map, the grid surface is then applied to the captured bitmap to simulate an undulating surface with the associated raster grids shape. An example of this is shown below where a scanned Located Image has a topography raster grid used to define its displayed shape.



3D display of Located Image as a Surface geometry using a topographic raster grid

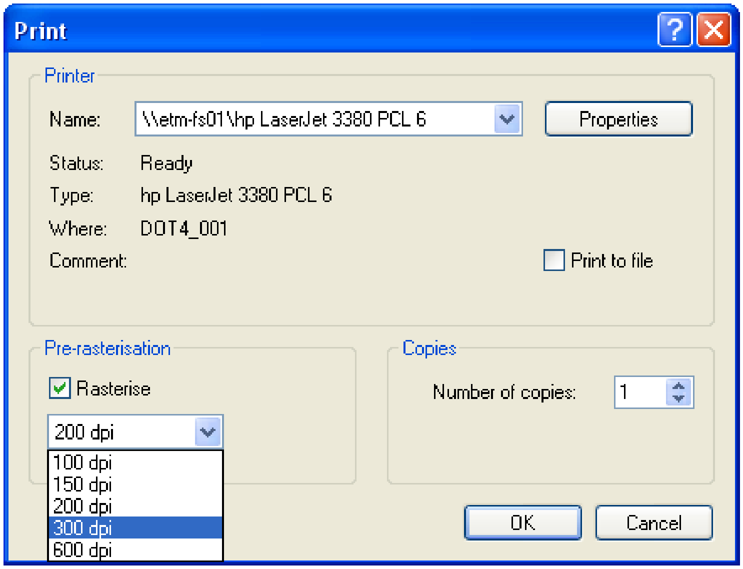
The located image produced from this dialog is not automatically loaded into Encom Discover PA. To view the located image in Encom Discover PA use the drag and drop functionality. Open Windows Explorer or use the Explorer floating window in the Information sheet and navigate to the location of the located image file. Select the located image (.EGB) file and holding the mouse button down drag the file to the display area of Encom Discover PA. Then release the mouse button to drop the file into the software.

To display the located image in a new display window position the cursor over the dark grey area of the display area before dropping the file. A Drag and Drop prompt dialog will appear in order to specify that the image is to be displayed in either a 2D or 3D window. If you wish to add the located image to a display window already open in Encom Discover PA position the cursor over the active window before releasing the mouse button and dropping the image into it.

Printing

The **File>Print** option enables Encom Discover PA to create hard copy output to a printer or plotter. Selection of the Print option automatically outputs a copy of the active window to a printer or to a print file (depending on your computer’s print settings). The printing function operates on the active (highlighted) display window only. The output can be scaled or simply made to fit a predefined page size. Output destination is controlled by the **File>Page Setup** options but a default printer setting can be defined in the **File>Print Setup** dialog. When invoked, the Print command determines the printer settings, device/paper size and window size for both Profile and Map displays. From these parameters, Encom Discover PA estimates a suitable scale to fit the print.

Note Encom Discover PA can use standard Windows drivers directly for printing or it can use a rasterising option to speed up and produce higher quality printed output. The rasterise option is included on the Print Setup dialog and is recommended for routine use in printing. It is recommended that a rasterising level of between 150 and 200 dpi is optimal for speed and quality.



Printer and Rasterise options

For more information, see:

- *Print Scaling*
- *Profile Scaling*
- *Map Scaling*
- *Page Margins*
- *Batch Printing Wizard*
- *Batch Printing to PDF*
- *Print Preview*

Print Scaling

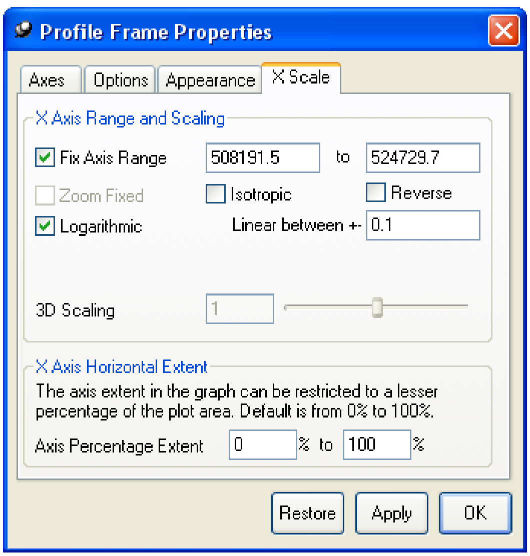
Hard copy prints can be sized to a user defined scale. Scaled output is limited in size by the physical size of the hard copy media being used. For example, it is not possible to print a complete, large scale map on a single A4 piece of paper if the scaling definition is too small. Encom Discover PA takes the size of the requested paper and scales the output accordingly. If the document being printed does not fit the paper (see *Page Margins*), a warning is presented. If the user requests to print anyway, the printing is performed with clipping.

Scaling control is slightly different between the two display types of maps and profiles.

Profile Scaling

Profiles are scalable in both the horizontal and vertical directions. It is possible to scale the height of a plot but it is not feasible to scale individual tracks since numerous tracks, channels and individual ranges can be used in profile displays. The height of individual tracks can be adjusted by dialog control or graphically by positioning the cursor above the separation line of a profile track, clicking the left button and dragging the line to the desired location. Note that to manually adjust the track size this way, the Workspace tree must have the Profile Frame level highlighted.

To create a profile of a specific horizontal scale, display the profile Property Frame dialog and select the **X Scale** tab. A specific scale entry can be entered as required.



Profile frame properties dialog

With the scale specified, select the **File>Print** menu item to print the profile.

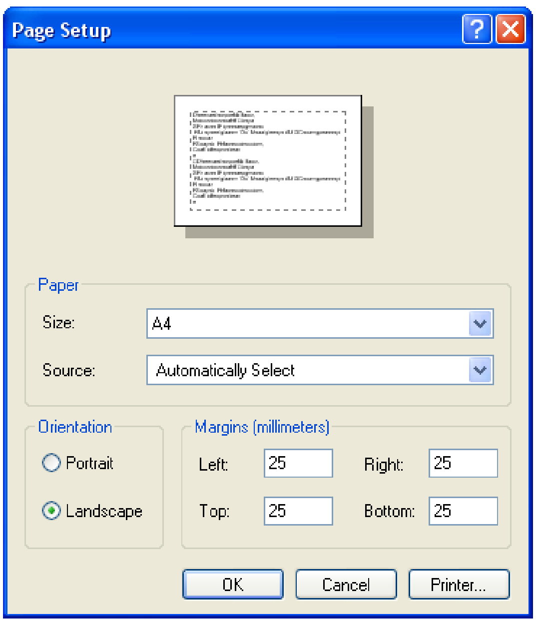
Map Scaling

Map scaling usually uses (although does not have to use) the same scale in both a vertical and horizontal direction. To create a map on a specific scale, display the Map Frame Property dialog and select the **X axis** and **Y axis** tabs to specify the scales.

To print the map, select the **File>Print** option. Printer setup and connection to hardcopy devices is controlled by the **File>Print Setup** item (or from the menu activated by a right mouse click within a display window). This item enables access to various printer types, the orientation, paper size and destination of the print. For a more detailed explanation of the Page Setup utility, refer to the *Microsoft Windows Users' Guide*.

Note A wide range of printer types are supported. Output can also be directed to plotters, ink jet and thermal printing devices for which Windows drivers are available from the manufacturer. Output for a particular hard copy device can also be directed to a file. This situation is often necessary on networked installations or where the printer is not physically connected to the operating computer.

The **File>Page Setup** option displays a window similar to:



Page Setup dialog to specify the printer, orientation, page size and margins

The Page Setup dialog enables you to define:

- Paper size (including custom page sizes)
- The source of the paper suited to the printer available
- The orientation of the paper

- Margins (in millimetres). See *Page Margins*.
- Printer or plotter from the Printer button.

If you select the Printer button the standard Windows printer definition and selection dialog is displayed.

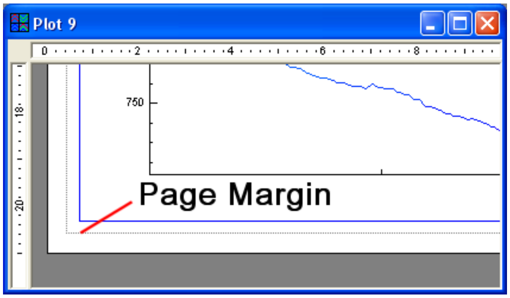
Specific printers can be selected by using this dialog. Depending on the printer type, the **Options** or **Properties** button displays various controls that can be set through subsequent menus. Different printers have different configuration controls.

Printing using a defined scale is a desirable feature, but occasionally a print scale might be selected too small for the size of allocated paper. In this case, the print process can either clip the output to the first page and print what it can, or the software can print on the first page, clip the remainder and continue printing on the second, third and subsequent pages.

The **File>Print** option clips output to the first page. Any remaining output due to selection of an inappropriate scale is clipped.

Page Margins

The printable area of a page varies with different printers and plotters. It is important to know where this area is to ensure your print is contained within the margins. The margins themselves can also be specified in the Page Setup dialog if you wish to change them. The margin is displayed in Encom Discover PA when the display is in Page Layout mode and with the Grid Design Mode (see *View Menu*) enabled.



Page margin displayed in Grid Design Mode indicating the outer print margin

The page margin is shown as a dotted line around the perimeter of the displayed paper.

Batch Printing Wizard

The Batch Harcopy Wizard feature (initiated from the **File>Batch Hard copy Wizard** item on the main menu) allows the printing of multiple plots of selected profiles to a printer, images or Located Images (see *Located Images*). Each chosen line of data prints to a separate piece of paper in print mode, or to a different file in image mode.

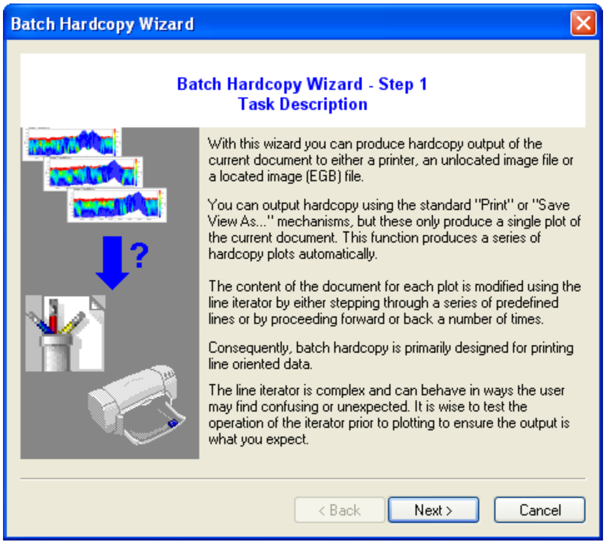
The Batch Printing Wizard allows you to control the output, format, line selection and resolution. The Wizard does not specify scale or printer setup. These must be determined prior to using the wizard.

Note Printing to a defined scale is a desirable feature, but occasionally a print scale is selected that is too small for the size of the chosen paper. In this case, the print process outputs what can fit on the first page and clips the remainder (as shown in the Print Preview). You must ensure that the scale and selected paper/printer selection is appropriate for your largest line hard copy output.

Five wizard screens are displayed to allow various choices. These are:

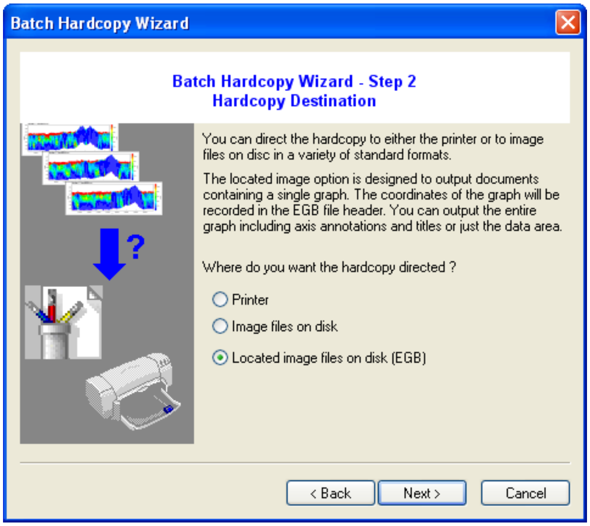
Step 1

Descriptive only with no choice available.



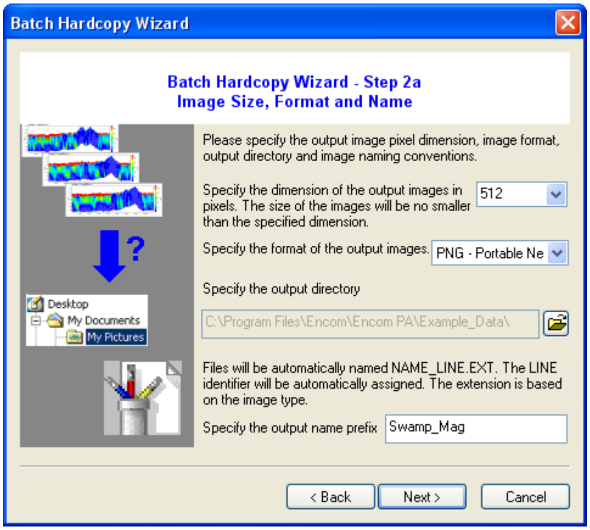
Step 2

Allows selection of output to a printer, output images or Located Image files. Located Image files can be written with a single .EGB or separate .EGB header files for each Located Image.



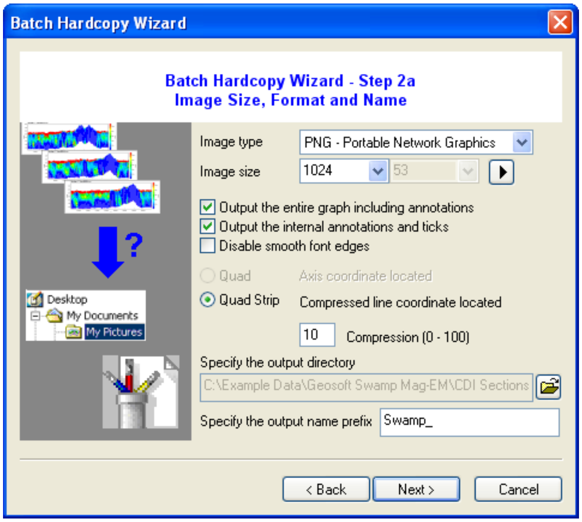
If the Printer option is chosen, you can output your images as hardcopy to an installed print driver or PDF document.

If the print as Image option is chosen, you can nominate the image resolution (512, 1024, 2048, 4096, 6144, or 8192), the supported image format, output location and appended default image name. The default file name appends the line number of output to the specified prefix (eg. `IMAGE10200.BMP`, `IMAGE10300.BMP` etc).



If the print as Located Image option is chosen you can nominate the image resolution (512, 1024, 2048, 4096, 6144, or 8192), the supported image format (including ECW), output location and appended default image name. With this option an Encom Georeferenced Bitmap (.EGB) header file is also created which contains all georeferencing information for each image that is printed. Additional parameters to apply include:

- The option to include all of the outer graph annotations, including labels, titles, etc.
- The option to include the internal annotations and internal ticks.
- The option to improve the appearance of text by enabling/disabling smooth font edges.

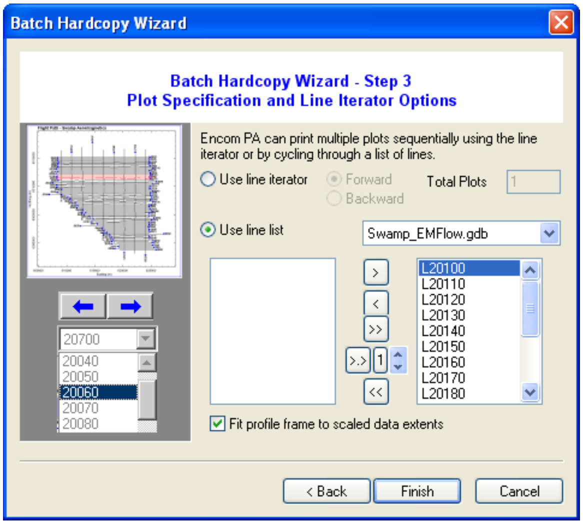


Print wizard screens for located image selection

Step 3

This final screen is common to both printer, image and located image outputs and allows line selection. Two methods of line selection are available:

- **Line Iterator**: This method uses the automatic line selection specification as defined by the Line Iterator settings. Note that any of the Spatial, Standard or Sequential settings can be used.
- **Line Selection Table**: By manually building a line list in the Selected column, you define the data lines that are to be output in the batch process.



You can use the Left/Right and Group selection buttons to move lines back and forth from the available and selected lists. You can also use the SHIFT and CTRL keys to specify multiple lines.

Warning While Encom Discover PA is batch printing and the required output has georeferenced images included, if you make the window redraw – for example by passing another window over the top of it – you get no output from that moment even though Encom Discover PA reports everything printed correctly. The printing of images takes the content of the screen display and therefore if it is interrupted, the print procedure will fail. It is therefore recommended that you do not touch your screen display until the batch printing has finished.

Batch Printing to PDF

When batch printing to PDF files the default operation is to print each page to individual PDF documents where each file name is manually entered after each iteration.

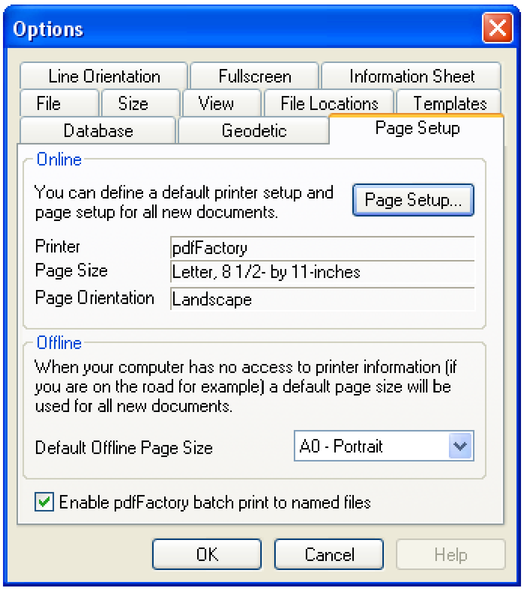
The PDF printing process in Encom Discover PA can however be streamlined by specifying the print driver to pdfFactory or pdfFactory Pro (for further information on this print driver please refer to *http://www.fineprint.com*)

To print documents to PDF using an automated process whereby the prefix output file name is preset and there exists the option to print all pages to the one PDF document the following process should be followed:

Step 1

Before printing access **Tools>Options** and view the Page Setup tab dialog. Check the status of the **Enable pdfFactory batch print to named files** option:

- To print all documents as individual PDF files make sure that this check box is selected.
- To print all documents to the one PDF file ensure that this check box is deselected.



The Options dialog with “enable pdfFactory batch print to named files” option enabled.

Click OK to return to Encom Discover PA.

Step 2

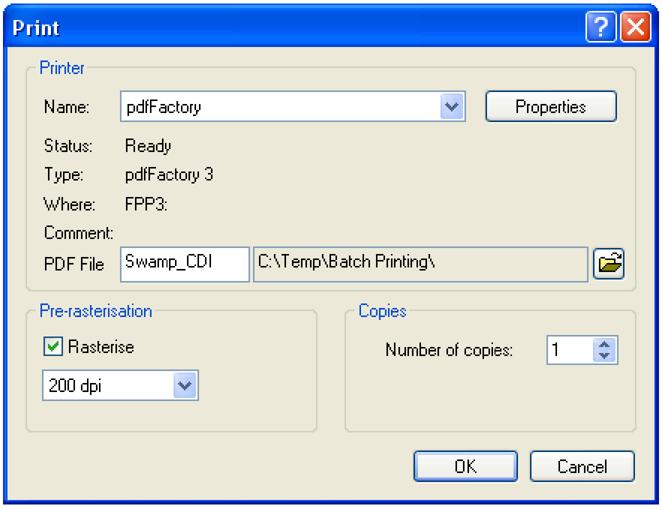
Go to **File>Page Setup** and choose pdfFactory or pdfFactory Pro as the default printer in the Page Setup properties.

Step 3

Choose **File>Batch Hardcopy Wizard** and proceed through the wizard to select the output to “Printer”, and choose the lines you wish to print.

Step 4

Click on **Finish** and if you selected the **Enable pdfFactory batch print to named files** option in the **Options>Page Setup** dialog (in Step 1 above), the Print dialog for pdfFactory or pdfFactory Pro will appear. Enter the file prefix and file destination into the dialog. Click OK to print and the status dialog will appear indicating that the document is printing.



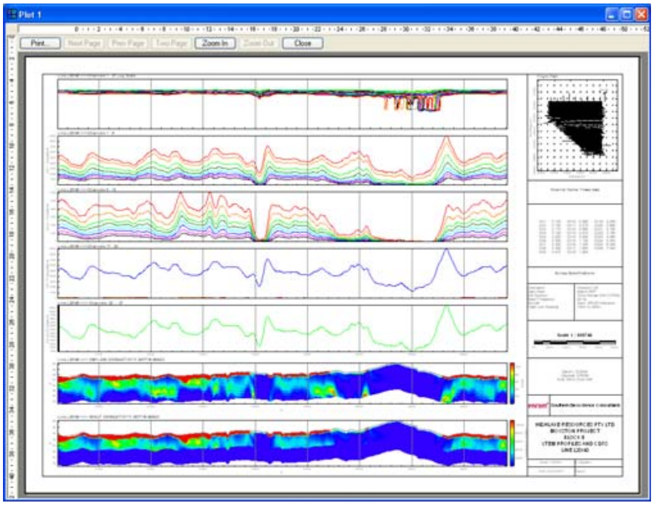
The print dialog for entering a PDF file prefix name and location.

Otherwise you will not be prompted to enter a file prefix or destination. Instead the file is by default called “Encom Discover PA1” or similar and will need to be changed at a later date if you prefer a different file name. The output file will be saved to the default location that can be changed by going into the Properties in the **File>Page Setup** dialog and specifying the AutoSave location in the Settings tab dialog.

The output of this process will consist of one PDF file which will be displayed as soon as the first line has been printed but the additional pages will get added to this as the batch printing proceeds.

Print Preview

Before printing, you can preview your document from the **File>Print Preview** menu item. From Print Preview, you can print directly using the **Print** button or zoom in or out of the print.



Print Preview window

7 Managing Session Files

In this section:

- *Session Files*
- *Packaging Datasets*
- *Templates*

Session Files

Sessions and project management is controlled from the **File** menu with the **Open**, **Save** and **Save Session As** commands.

Encom Discover PA uses binary session files to save and restore projects. Within the session files the following information is stored:

- Paths to the necessary database source information
- Paths to any used template files for profile configuration
- Paths to feature databases
- Configuration data for open windows and components, their attributes and properties

Session files are usually saved with the file extension .EGS.

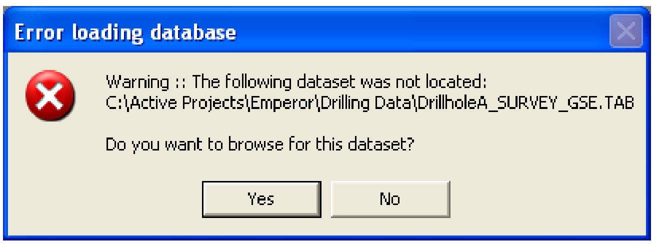
The session files do not contain data or feature information. Consequently, if a session is opened but the database access is not available, the software will not be able to restore the session. A message (see *When a Database Fails to Open*) is displayed in this circumstance.

Error message displayed typically when a dataset is open in another application.

Restoring a Session

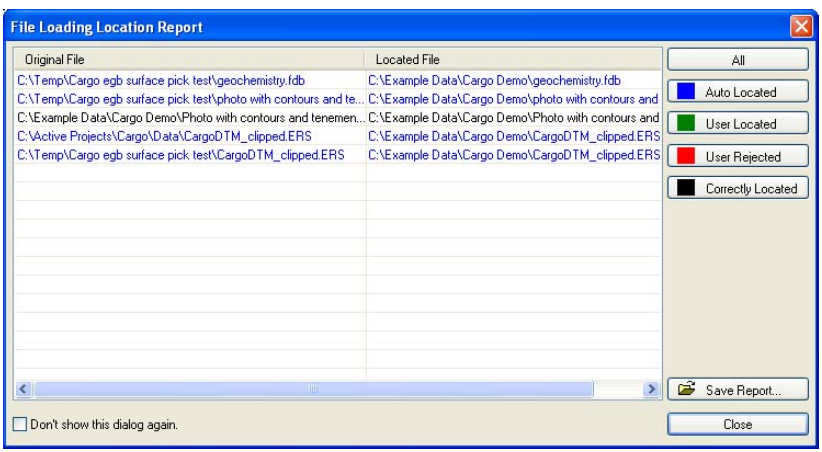
When session files are restored, Encom Discover PA attempts to access and open as many of the individual files, images, grids etc it possibly can. Note that data, image and grid files are not stored in sessions – only the pathing, names and necessary information to access them.

As the session files attempt to restore and display the saved data, there will be instances when some files have been moved, deleted or renamed. When this occurs, you are provided with the opportunity to browse for the missing file, or you can click Cancel and skip this file.



Encom Discover PA provides the opportunity to find a missing file

At the conclusion of a session restoration where some files have not been located, a report is provided that describes the files successfully and unsuccessfully loaded plus what has occurred to them.



Session or template recovery report

This report can be used to determine which files have not been located, which were rejected and which were loaded automatically.

Recent Sessions

The names of the last five session files saved while using Encom Discover PA are displayed in the File menu list. These files can be individually selected to quickly access previously saved sessions.

File Recovery

Session file restoration has a lost file recovery mechanism. If Encom Discover PA cannot locate a database or other necessary files, it alerts you and offers the option of browsing for them. This applies to:

- Geosoft databases
- Intrepid databases (by a browse operation to a folder)
- MapInfo .TAB files
- Drillhole databases (acQuire, Geosoft WHOLEPLOT databases (either single or split) plus Encom Discover)
- Grid files
- Feature databases
- Vector files
- Located image files (if unavailable, a red cross is shown at the location of the image).

Packaging Datasets

To allow you to archive data or to send a complete data package to a client or work associate, Encom Discover PA offers a data packaging facility. This function takes all data, display objects and databases and places them separately in a folder with a session file to allow restoration.

The folder used to store the data and objects can be placed in a new folder that can be burnt to a CD or sent to an associate. To restore the complete project with all information required, only the session has to be restored.

Follow these steps to package a dataset:

Step 1

Display the data, windows, displays as you want them restored.

Step 2

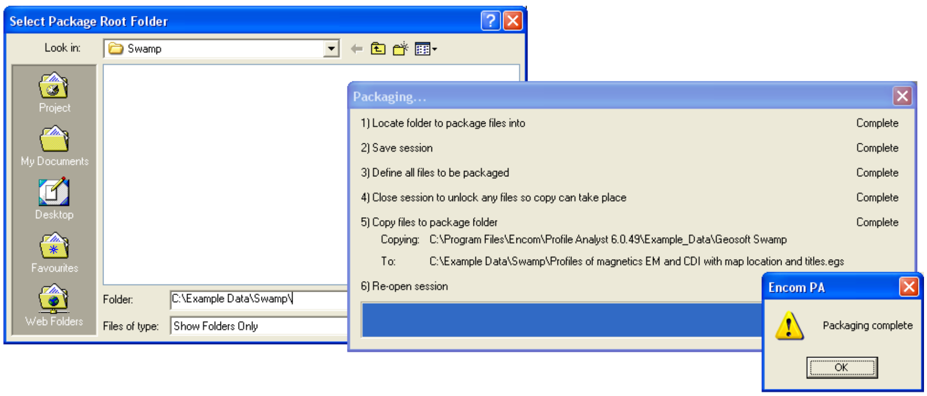
Use the **File>Save To Package** option.

Step 3

Specify a folder to store the package of data and data objects. You may need to use the **Create Folder** button of the Open File dialog when presented. Note that a session is required to restore the data package. If one has not already been created or associated with the data, you will be prompted to create one with a Session Open dialog.

Step 4

When OK is clicked in the above dialogs, the data packaging process commences the process of storing all the objects, databases and display information.



Data Packaging dialog and saving to a specified folder

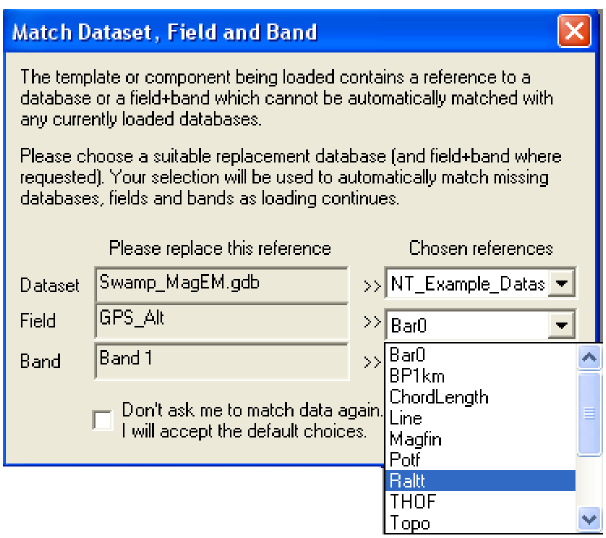
At the conclusion of the packaging operation, Encom Discover PA opens and exercises the session file to ensure the storing process is completed correctly.

Templates

Encom Discover PA enables the style, properties and display attributes of all windows to be stored in a template. Templates are useful when graphs or maps with similar appearances need to be reproduced. This makes comparison of results or hard copy output to be done simple and quick. Templates can be saved using **File>Save As Template** or restored with the **File>New>Open from Template** option. Alternatively, use the template button.

A wide range of templates and components are provided with Encom Discover PA. Refer to the *Encom Discover PA Templates and Components* document.

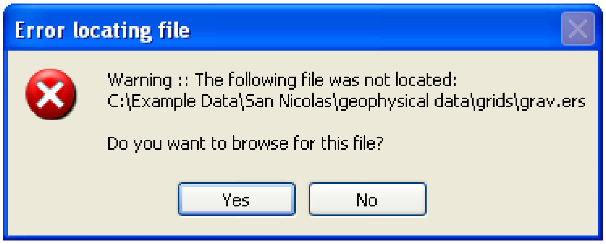
All aspects of a profile and the displayed trace properties can be stored in templates. Consequently, applying a template from the profile of one dataset, to the profile of a second dataset may cause a channel mapping problem if the same channel naming conventions are not used etc. Encom Discover PA has a number of logical search rules it applies when a template is used with different datasets. For example, if the dataset used to create the template has a channel with 14 bands of data, but the second dataset has only 10 bands, Encom Discover PA applies the template display attributes as closely as it can, but it might halt when the options are exhausted.



Dialog to alter the channel mapping of a database to suit a reused template

If different data field naming conventions are used for a second dataset where a template is to be applied, Encom Discover PA asks you which field and bands to use.

Where grid surfaces are required for a template to load, but the required grid file is not available (through a change or absence of the required folder or file), then the loading procedure requests you to either find the file or to substitute a different grid. The message displayed appears below:



Error in locating a grid file

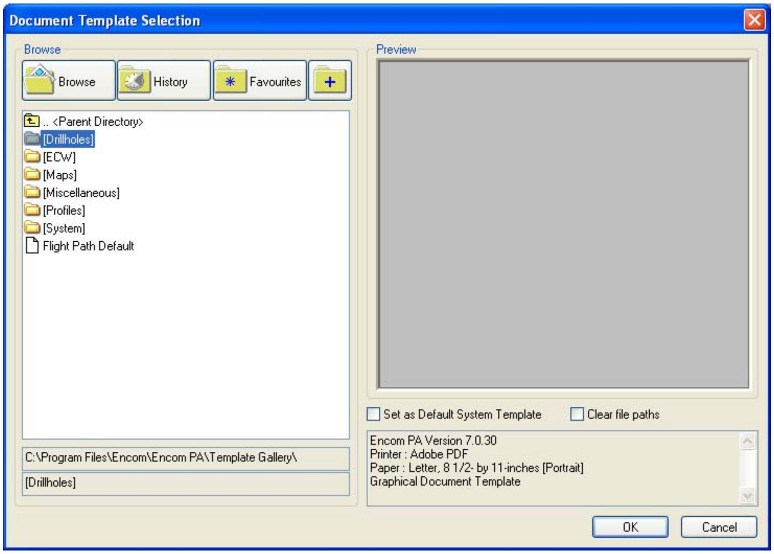
Application Templates

To assist you in displaying various presentations, Encom Discover PA comes with a set of application templates. Templates are simple scripts that are used to reproduce previously displayed presentations. Further information on templates is provided in Templates.

Each template has its own description and is accompanied by a small preview of its content. Templates are stored in a folder called Template Gallery. Also within this folder are sub-directories that are for advanced template use. These directories contain templates that are tailored for specific applications such as:

- Potential Field Methods
- EM Methods
- Uranium Search
- Diamond Search
- Map Outlines

To use the templates for creating immediate displays, access the **File>New>Open From Template** menu option or alternatively click on the **Open from Template** button in the Documents toolbar. You can also browse the Template Gallery folder or alternatively individual folder names are listed in square or <> brackets. You can click on these folder names to allow navigation to them.



Template selection with folder navigation, template preview and description

At the base of the selected template are details of the printer, page size and orientation as set when the template was saved. A description of the template content can also be saved. This description can be entered when saving a template from the property dialog of the Document.

Browse, History, Favourites and Add to Favourites buttons allow fast search and retrieval of frequently used templates. These options operate similarly to the Windows Internet Explorer utility.

With a template selected, click the OK button. Encom Discover PA then uses the content of the template to reproduce the presentation as previewed.

How Does a Template Work?

A template operates by storing the contents and properties of the Workspace tree not by storing details of the original window display. The tree contains all the relevant components and properties of a particular display. The tree can become complex with displays within displays etc. and hence the need to store and quickly reproduce the style of the original display.

Templates can operate from the available databases (not the databases used when originally saved) and therefore some data fields or components of the original data may not be present for the template to operate exactly.

For example, if a data field is not present in the accessed database, the template cannot use it for the display. In these cases, Encom Discover PA attempts to locate items but if they are not available it displays the dialog (shown above) and allows you to select replacement items:

Note In the case of grids, these are not part of a loaded database and as such may be available as specified in the template, even though a different, unrelated database may be used. For these situations you need to re-specify the replacement grid location (from the Grid Group properties) and if in a different geographic area, select the **Data Zoom** button after the new grids have been defined. This action re-coordinates the location of the dataset and the incoming grids.

The details of the printer, page size and page orientation are also saved in a template. This information is used when a template is transferred from one machine to another and the template re-applied. The correct application of these settings however can only be applied to the template if the same printer is available to the second machine. If it is not, the default printer settings of the machine are used. When a template is accessed through the Template Selection dialog (see above), the printer information is displayed.

8 Working with Data

In this section:

Accessing Data with Encom Discover PA

Encom Discover PA is designed to access data directly from industry standard databases. Unlike other geophysical packages, Encom Discover PA uses existing import and raw data reading facilities of external packages to provide access to the data. This approach has a number of advantages such as:

- A wide variety of data formats, data sources and file formats are catered for by other packages and can therefore be supported by Encom Discover PA.
- The manipulation and editing of data when obtained from a database provides access speed, power and increased flexibility.
- Reading of data from a binary database is much faster than from raw (text) data files and does not have the overhead of checking record or data validity.

- Navigation of data from one point (or traverse line) to another can take advantage of search and indexing techniques to increase speed and simplicity.

Encom Discover PA is designed to act as an interface between external databases (containing data), to display and provide interpreter interaction (through accessory processing or interpretation modules). Data links are transparent to the user as Encom Discover PA detects the type of database present in a folder and builds the link automatically.

When Encom Discover PA accesses a database, editing or changing the attributes of the data entries is possible through certain operations. These operations may involve data channels being renamed or new data fields being created by mathematical manipulation or operations such as filtering.

Encom Discover PA directly reads data from industry standard databases or uses external modules to create databases and import them. For list of the supported databases, see *Supported Databases*. Additional information about supported file formats can also be found in *Appendix B: Data File Specifications* (file specifications) and *Appendix E: Supported File Formats* (supported file formats). Access to a database from the **File>Open** option displays a folder access dialog that is used to navigate to the relevant folder containing the data. The folder rather than a specific file path requires specification.

When the OK button is clicked an opportunity to access other databases is provided and then Encom Discover PA interrogates the specified directories and ensures it can read the necessary files and ancilliary data. If the database is satisfactorily opened the Encom Discover PA buttons and access to the data are enabled. A message indicating a successfully opened database also appears in the Status Bar.

To close access to databases, select the **File>Close All** option. The link between Encom Discover PA and all selected open databases is broken and data contained in memory is lost. Note that after the Close All option is selected, you can re-open another database.

For more information, see:

- *Supported Databases*
- *Importing ASCII Data Files*
- *When a Database Fails to Open*
- *Selecting Channels and Tables*
- *Ancilliary Data*

- *Closing a Database Link*
- *Preferred Directory Access*

Supported Databases

Current industry databases supported are:

- *Geosoft Oasis montaj*

  Geosoft Inc., Toronto, Canada *www.geosoft.com*.

- *Intrepid*

  Des Fitzgerald and Associates, Melbourne, Australia, *www.intrepid-geophysics.com*.

- *MapInfo TAB File*

  Pitney Bowes Software, Troy, New York, *www.pbinsight.com*.

When data is linked to Encom Discover PA, certain additional data fields become available that are not in the database initially. Data fields such as CHORDLENGTH and PROJECTEDDISTANCE are required by Encom Discover PA for line distance and plotting control.

- A wide variety of ASCII file formats can also be imported from other sources. See *Importing an ASCII File*.

Geosoft Oasis montaj

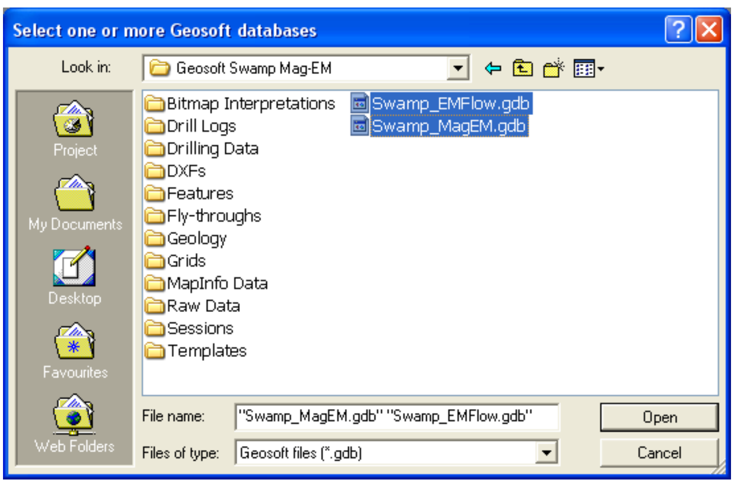
If you wish to access a Geosoft Oasis database (usually with file extension .GDB) you can individually open a single .GDB database file, or you can open multiple .GDB files. To open a single Oasis database file, select the **GDB** button (or **File>Open>Geosoft Oasis montaj**) and specify which .GDB database file is required. As an alternative, if you wish to access **multiple** .GDB files, use the SHIFT or CTRL keys and position the cursor over the selected files. Highlight them and click the **Open** button to provide access to these databases.

Warning Geosoft databases cannot be accessed by more than one application at a time. This means that if Encom Discover PA is accessing a database, Oasis montaj cannot read or display the same database at the same time. Similarly, if Oasis montaj is being used to view a database, Encom Discover PA cannot gain access to the same database.

Preparing a Geosoft Oasis montaj database for use with Encom Discover PA requires:

- Importing data into the dataset as a line or points dataset. Data may be imported in the customary fashion by creating an import template (.I3) and using the Geosoft Import Tool. An alternative to this is to use an external utility such as the ASCII Import tool (see *Importing ASCII Data Files*).
- Ensuring that multi-channel data (such as EM data) are imported as an array
- Easting and northing channels MUST be created using channel names of X and Y respectively.

See *Appendix B: Data File Specifications* for examples of import control files for use with Encom Discover PA and Geosoft Oasis. Additionally the Geosoft documentation describes the import of data and the necessary procedures. If you wish to access an Oasis database (usually with file extension .GDB) you can individually open a single .GDB database file, or you can open multiple .GDB files from multiple directories. To open an Oasis database file, select the Geosoft button and specify which .GDB database file(s) are required. Use the SHIFT or CTRL keys and position the cursor over the selected filenames to multiply select. Highlight them and click the Open button to provide access to these databases.



Select the desired GDB databases to be opened

If additional databases are to be opened after initially opening one or more in one folder, you can repeat the database opening procedure by selecting the **Geosoft Open** button navigate to the additional database(s), select and again click the Open button. Alternatively use the drag and drop operation from Windows Explorer for faster loading of data.

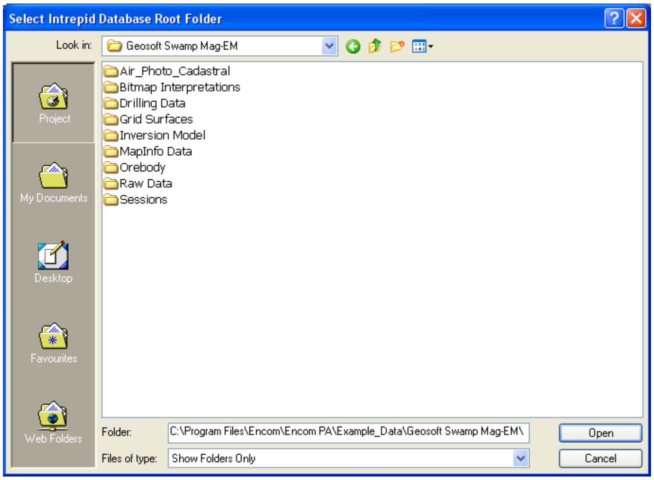
Intrepid

Preparing an Intrepid dataset for use with Encom Discover PA consists of:

- Importing data into Intrepid as a line dataset. Data may be imported in the customary fashion by creating a Data Definition File (.DDF) and using the Intrepid Import Tool. You must ensure that multi-channel data (such as Geotem) is imported as banded.
- Ensuring the SurveyInfo file is complete. This file is found in the data folder and has all channel aliases assigned. The file can be edited in a text editor or by highlighting the imported line data folder in Intrepid and using the **File>Edit SurveyInfo** option in the Intrepid processing window.
- Essential data channels that must be present in an Intrepid dataset (or using the SurveyInfo alias file) are X (Easting), Y (Northing), Fiducial, FlightNumber and LineNumber.

See *Appendix B: Data File Specifications* for examples of import control files for use with Encom Discover PA and Intrepid. Additionally the Intrepid documentation describes the import of data and the necessary procedures.

The nature of Intrepid databases is such that no one file specifies the database. Instead, they are stored as multiple files within a root folder. To access an Intrepid database, select the Intrepid button and specify the root folder containing the database.



Specify the folder holding the required database(s)

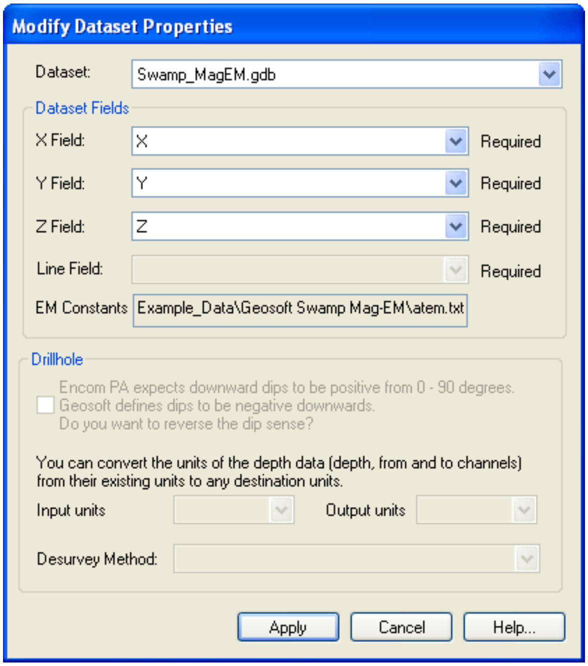
Select the **Open** button.

MapInfo TAB File

MapInfo .TAB file formats can be imported directly into Encom Discover PA but only if they contain data columns that record the location (Easting and Northing data fields). These two data fields are mandatory. Elevation data is optionally required if the imported data is to be displayed in 3D. Line information is also optional but required if the data are to be navigated using the line navigation facilities. The menu option **File>Open>MapInfo TAB** allows import of .TAB files.

An Open Folder dialog is displayed from the Browse button. Navigate the path to locate the required folder containing one or more .TAB files.

Once Encom Discover PA accesses the file, it may display a dialog to permit you to specify the Easting, Northing (mandatory) and optionally, the Z (elevation) and Line data fields. To access this dialog again at any time during the operation of Encom Discover PA select the **File>Modify Dataset Properties** menu option.



Dialog to specify the required and optional .TAB data fields as required

With a MapInfo Professional .TAB file accessed, Encom Discover PA can display the data using its available display objects that support the data available. For example, a .TAB file that does not have a Line definition could not be displayed in a profile display. However, if the .TAB data had located (X, Y) readings representing say a geochemical or gravity survey, they could be displayed as a Point display (see *Displaying Points*).

Importing ASCII Data Files

If a database of data is not available, one can be constructed using the ASCII Import option (from **File>Open>Text Import** or the **ASCII Import** button. This facility can read a wide range of ASCII text format files including:

- Comma delimited or CSV format (see *Delimited Data*)
- XYZ ASCII line-based data formats (see *ASCII File Formats*)
- Drillhole collar and downhole data formats (see *ASCII Drillhole Data*)
- Point data provided in column-oriented ASCII files (see *Point Data*).

The Import text tool is designed to read external multi-column ASCII data files, create a Geosoft database and then automatically have Encom Discover PA access the new database. Encom Discover PA can import and open the new database at any time during a session (similar to opening a database directly from the menu commands). In this case the new database is accessed and appended to any existing database links already established.

Other features of the text import tool include:

- Flexible import format definition that can accommodate any number of ASCII data columns and up to 32,000 characters per record
- Read ASCII files with either fixed width (defined by column size) or delimited by characters you choose
- In-built intelligence to interrogate a specified file and classify it as one of a range of types. A number of interpretations based on these types is then attempted to assist the decoding and recognition of the file.
- Easy interface for column naming can use a header line or interactive dialogs. Unwanted header records can be skipped during the import.
- Comment lines are accommodated as are various data formats such as real, and scientific notation.
- Data specification can include multi-channel, banded data (for example, with EM data) that is used extensively in Encom Discover PA
- Use or save Geosoft templates to describe the data format for re-use. Comments, headers and nulls are all handled in the templates.

After data is imported, it is automatically opened by Encom Discover PA.

For information about the types of data you can import and the features of the text import tool, see:

- *ASCII File Formats*
- *Importing an ASCII File*
- *Treatment of Null Values*
- *DMS Format*
- *Saving and Reloading a Layout*
- *Pre-set File Formats*

- *Text Import Options*
- *Text Import Procedures*

ASCII File Formats

The Text Import facility can load a wide range of ASCII multi-column formats. Typically the formats fall into the following classes:

No header or comments, multi-columns of constant width

Sample data:

```
1600   300200.0   6253800.000   56481.027     -0.010
1600   300220.0   6253810.000   56479.535     -999999
1600   300240.0   6253820.000   56478.371     -0.003
1600   300260.0   6253830.000   56477.547     -0.001
1600   300280.0   6253840.000   56477.035     -0.000
1600   300300.0   6253850.000   56476.781     -0.000
1600   300320.0   6253860.000   56476.707     -0.001
1600   300340.0   6253870.000   56476.723     -0.003
1600   300360.0   6253880.000   56476.734     -0.005
1600   300380.0   6253890.000   56476.660     -0.008
```

Header lines and comments with multi-columns of constant width

Sample data:

```
LINEEASTNORTHMAGGRAV
\ This is a comment line used for information
1600   300200.0   6253800.000   56481.027     -0.010
1600   300220.0   6253810.000   56479.535     -999999
1600   300240.0   6253820.000   56478.371     -0.003
1600   300260.0   6253830.000   56477.547     -0.001
1600   300280.0   6253840.000   56477.035     -0.000
1600   300300.0   6253850.000   56476.781     -0.000
1600   300320.0   6253860.000   56476.707     -0.001
1600   300340.0   6253870.000   56476.723     -0.003
1600   300360.0   6253880.000   56476.734     -0.005
1600   300380.0   6253890.000   56476.660     -0.008
```

Geosoft-style data files having line numbers embedded as standard XYZ file

Sample data:

```
Line 1600
```

```
300200.0  6253800.000  56481.027    -0.010
300220.0  6253810.000  56479.535         *
300240.0  6253820.000  56478.371    -0.003
300260.0  6253830.000  56477.547    -0.001
300280.0  6253840.000  56477.035    -0.000
300300.0  6253850.000  56476.781    -0.000
300320.0  6253860.000  56476.707    -0.001
300340.0  6253870.000  56476.723    -0.003
Line 1700
300660.0  6243880.000  56476.734    -0.005
300680.0  6243890.000  56476.660    -0.008
```

Delimited data with inconsistent column widths with or without headers

In this example are CSV files (from Excel etc), comma, tab or other character delimited.

Sample data:

```
1600,300200,6253800,56481,-0.01
1600,300220,6253810,56479.5,-1.00E+30
1600,300240,6253820,56478.4,-0.003
1600,300260,6253830,56477.5,-0.001
1600,300280,6253840,56477.0,0.005
1600,300300,6253850,56476.8,0.034
1600,300320,6253860,56476.7,-0.001
1600,300340,6253870,56476.7,-0.003
1600,300360,6253880,56476.7,-0.005
```

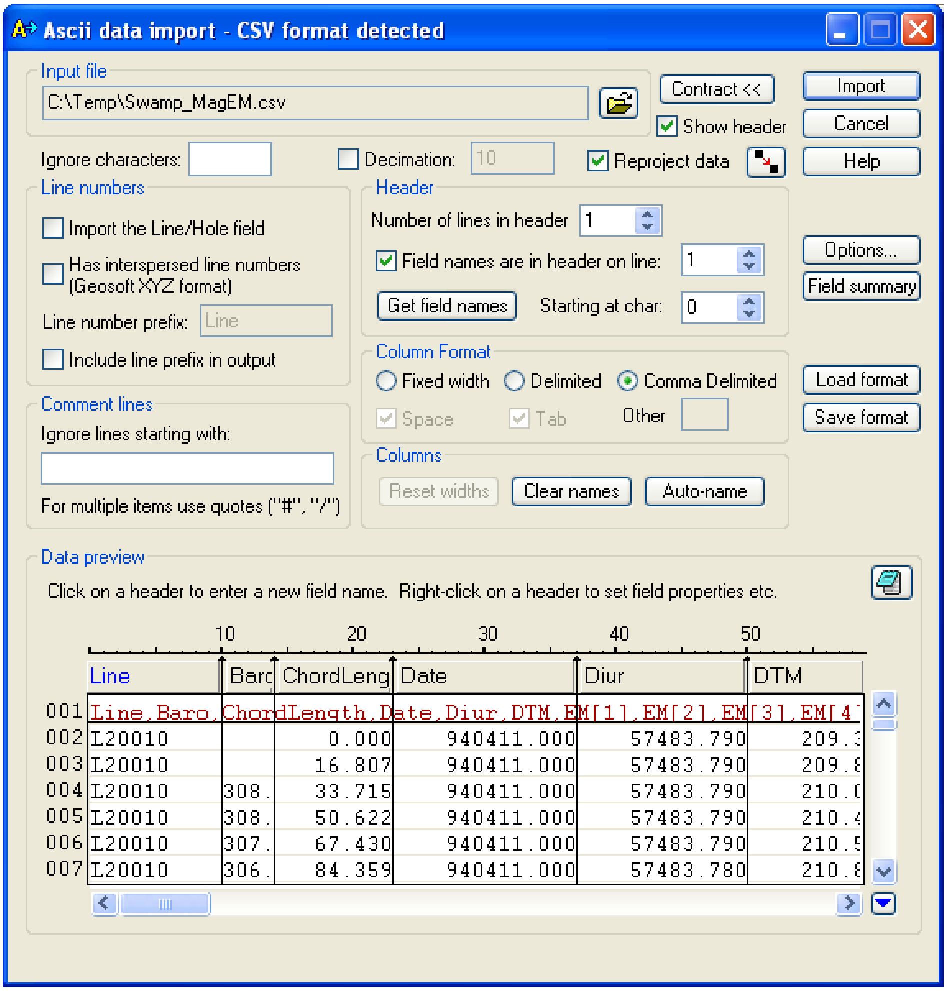
Importing an ASCII File

To import a text file or multiple text files in the same ASCII format, select the menu item **File>Open>Text Import** or the **ASCII Import** button.

Once a file is specified it can be altered from the initial dialog. The path and file name are echoed in the top left corner but you can search or alter the input filename if necessary.

Once the ASCII file to import is selected, the ASCII Data Import dialog screen is displayed. In the example that follows, a fixed width ASCII file with no headers, in multi-column format is used. If the data is not suited for loading (such as a binary file, or an inappropriate text file), a warning message is displayed. If no data is loaded, a message is displayed in the data preview area.

To import multiple ASCII files ensure that they are all selected at once in the browse operation. When multiple ASCII files are being imported in the one operation a preview of the first ASCII only will appear and the parameters specified in the import dialog for the first ASCII file will be applied to all subsequent ASCII files also being imported. After the Import button is selected the user will be prompted to enter a Geosoft database file name for each ASCII file being imported.



Text Import dialog for previewing data and defining the format, headers and column names

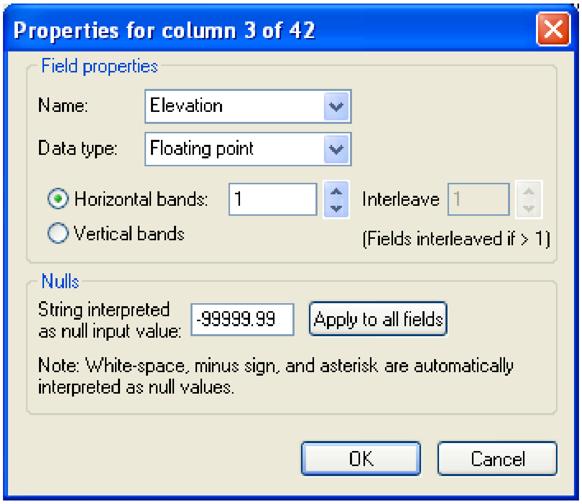
The dialog is divided into a series of controls with a data preview area at the bottom. In the preview area, the first 100 data records of the selected file are displayed. By default, the Text Import tool initially interrogates the file and determines if it is a **Fixed Width** or **Delimited** format. It then computes the column widths and displays these with vertical separator lines in the preview area. The dialog and its controls allow modification of the program's initial review of the import file.

Once column widths and format are determined, the second purpose of the dialog is to assign column names to each required column. This can be done automatically (using a header record), or manually by naming each column.

Treatment of Null Values

Input data fields containing an asterisk, a solitary minus sign, or just white space are automatically treated as null values and when imported are written out to the Geosoft database as the Geosoft null value (–1.0E32).

If you have input data fields that contain a different null specifier string, (e.g. –999.99), then you must enter this string in the **Nulls** area of the Field Properties dialog. To open this dialog, just double-click on the column/field label.



Properties dialog and the entry for nulls

If you wish to apply the same null string to all input fields, click the **Apply to all fields** button on the field properties dialog.

DMS Format

A DMS data format is automatically detected. This format is DDMMSS.SSC or DDDMMSS.SSC where C is a compass direction (N,S,E, or W) and the fractional seconds part (.SS) is optional and can be arbitrarily long if present.

Saving and Reloading a Layout

If you need to import a number of files with the same data layouts it may be useful to save your layout template for later use. To do this, simply click the **Save Format** button and you are prompted for a filename. Be sure to do this before you start importing the data, otherwise your layout information is lost.

When you want to import another data file with an identical layout, just reload your template using the **Load Format** button and all your field names, field properties, delimiters, column widths, etc are restored.

Note that the format used by Text Import for the format specification is the same as for a Geosoft database import template. Where used with Geosoft, your same files can be used with Text Import.

Pre-set File Formats

The Text Import has been designed to assist with formats of data files by automatically recognising certain file types. If a file type and structure are recognised, the format should automatically be decoded and presented for you. The available automatically decoded format types include:

- ER Mapper ASCII files
- AMIRA time domain .TEM files
- Datamine ASCII files
- GEMSys (GEM Systems) magnetometer raw data files
- Geosoft XYZ files
- LAS (Log ASCII Format) data files
- ModelVision Pro located data files
- SRG (Station Referencing) data files.

The above list shows a variety of standard industry data formats that are supported. An example of this is the Station Referencing format (.SRG) that allows the station field to determine where a new line starts. The station field must increment within a line. If the station field value decreases, this also indicates the start of a new line.

An example of this format is:

```
X,Y,Z,Station
100,200,50,1   // Line with 4 stations
105,200,50,2
110,200,50,3
115,200,50,4
100,205,50,1   // Line with 3 stations
105,205,50,2
110,205,50,3
100,210,50,1   // Line with 1 station
100,215,50,1   // Line with 1 station
```

In this case, set the field named Station to be a station field using the right-click menu. The SRG Classifier then interprets the remaining fields.

Text Import Options

To display the complete set of options on the ASCII Import dialog, click the **Expand** button.

There are three buttons in the Columns area of the Text Import dialog.

- **Reset Column widths**. is rarely needed as the column widths are usually automatically updated as the result of changes you make to other entry fields on the dialog. If you manually drag and add columns, then clicking this button resets your changes.
- **Clear names** button resets the names of all unnamed columns/fields back to Skip. Columns with this name are not imported.
- **Auto-name** button provides a quick way to select all fields for import. It names all columns with a prefix that you choose and the column number for a suffix (e.g. Field1, Field2, etc.)

Text Import Procedures

For details of how to import specific formats, see:

- *Fixed-Width Data*
- *Delimited Data*
- *Data with Interspersed Line Numbers*
- *Point Data*
- *File Headers Containing Field Names*
- *Multi-banded Fields*

Fixed-Width Data

A fixed-width data file is one where each column is confined to fixed character positions on each line of the file. To import a simple fixed-width file, follow these steps:

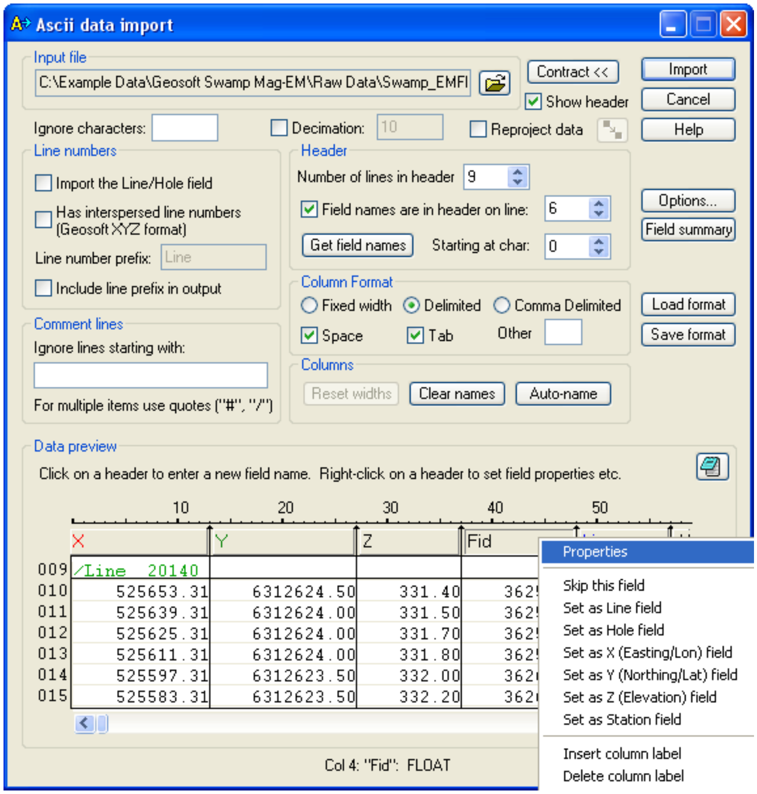
1. Choose an input file using the file dialog.

2. Check the **Fixed width** box in the column format area.

3. Ensure that the correct number of header lines are selected. Header lines are displayed in the data preview area in green and are separated from the data by a horizontal line.

4. In most cases, the Text Import tool automatically determines the column widths. If there are fields that are not separated by any white space, you may have to add and/or drag your own column lines in the data preview area. To do this, click in the data area to add a column break or double click to delete a column break. If you hold the button down you can select a column separator and drag it to a preferred position. If you hold the CTRL key plus move the mouse, it drags all columns to the right of the current column.

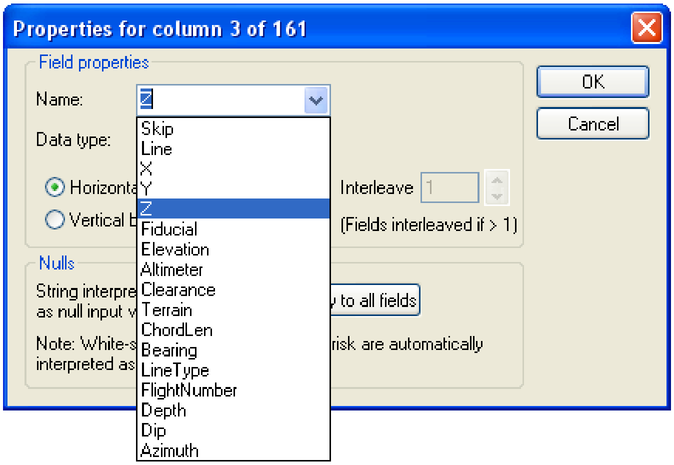
5. Choose a Line field by right clicking on a column label. This field should contain a value that is constant for each station along a line, but changes for each separate survey line or flight line in the data. If you do not choose a line field, when you click the **Import** button, you are asked if you wish to import point data.



Right mouse click pop-up window to set channel properties or assign a name

6. If required, choose an X (Easting) and Y (Northing) by right clicking on the appropriate column labels.

7. Name all the other fields that you wish to import. The simplest way to do this is to left click on the column label, type a name, and then press the TAB key to move to the next column. An alternative way is to double-click on the column label to bring up its Properties dialog. Fields named Skip are not imported.



8. Click the **Import** button in the top-right corner of the dialog. You are then prompted to select the name of the Geosoft database that is created to contain the imported data.

Delimited Data

A delimited data file is one where each column is separated from the others by some delimiter character or characters. Delimiters can be spaces, tabs, commas, or any other characters you enter in the Other field. To import a simple delimited file, follow these steps:

1. Choose an input file using the file dialog.

2. Check the **Delimited** or **Comma Delimited** item in the Columns format area.

3. Ensure that the correct number of header lines is specified. Header lines are displayed in the data preview area in green and are separated from the data by a horizontal line.

4. Choose the appropriate delimiter characters for your data. By default, space and tab characters are already selected, but if needed, you can enter other delimiter characters.

5. Choose a Line field by right clicking on a column label. This field should contain a value that is constant for each station along a line, but changes for each separate survey line or flight line in the data.

6. If required, choose an X (Easting) and Y (Northing) by right clicking on the appropriate column labels.

7. Name all the other fields that you wish to import. The simplest way to do this is to left click on the column label, type a name and then press the TAB key to move to the next column. An alternative way is to double-click on the column label to bring up its properties dialog. Fields named Skip are not imported.

8. Click the **Import** button in the top-right corner of the dialog. You are then prompted to select the name of the Geosoft database that is created to hold your imported data.

Data with Interspersed Line Numbers

In some cases, there is not a line field in your survey data. Instead, the file contains lines that specify the start of a new survey line. The simplest example of this is for the line simply to contain the string Line x, where x is a survey line name e.g. 1020.

To import data from files with such interspersed line numbers, simply check the box that says Has interspersed line numbers and select an appropriate line number marker string from the drop down list. If you need a string that is not in this drop-down list, type one in.

When the **Has interspersed line numbers** option is enabled, there is no need to select a field to be the line field.

Point Data

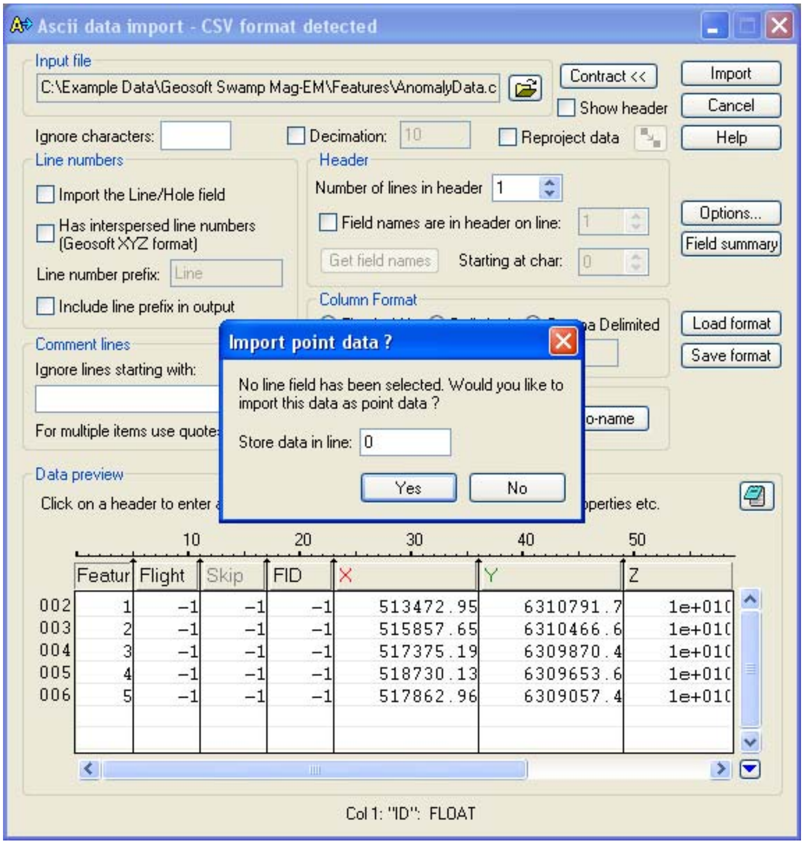
By definition, point data has no line-related column. Instead each record represents a point usually with some sample data acquired at an Easting and Northing position but randomly distributed.

An example of point data:

```
POINT   X         Y          Elev  ObsGrav   FreeAir  Boug2.67
601293  286861.3  6557450.5  287   9793745.0   29.3   -291.9
601298  285206.5  6565350.0  225   9793608.0   11.5   -240.5
601303  285111.2  6563139.5  243   9793674.0   29.4   -241.6
601308  285976.0  6553271.5  312   9793608.0   -3.5   -353.0
601313  286368.0  6567701.5  320   9793491.0  -37.6   -395.6
601318  286959.1  6569595.0  330   9793355.0  -107.0   -476.3
601323  287311.8  6571065.0  376   9793209.0  -62.8   -484.2
601328  287002.8  6572854.0  431   9793255.0  182.0   -301.2
601333  286495.7  6574801.0  680   9792787.0  490.1   -271.6
```

Note that each point has its own station number identifier (column Point), but this is not essential. Only the XYZ and measurement columns are necessary. When the file is accessed by the Text Import tool you are required to ensure each data column to be imported is displayed in the Column headers. You can use the software to do this or you can define them interactively as described in *Fixed-Width Data*.

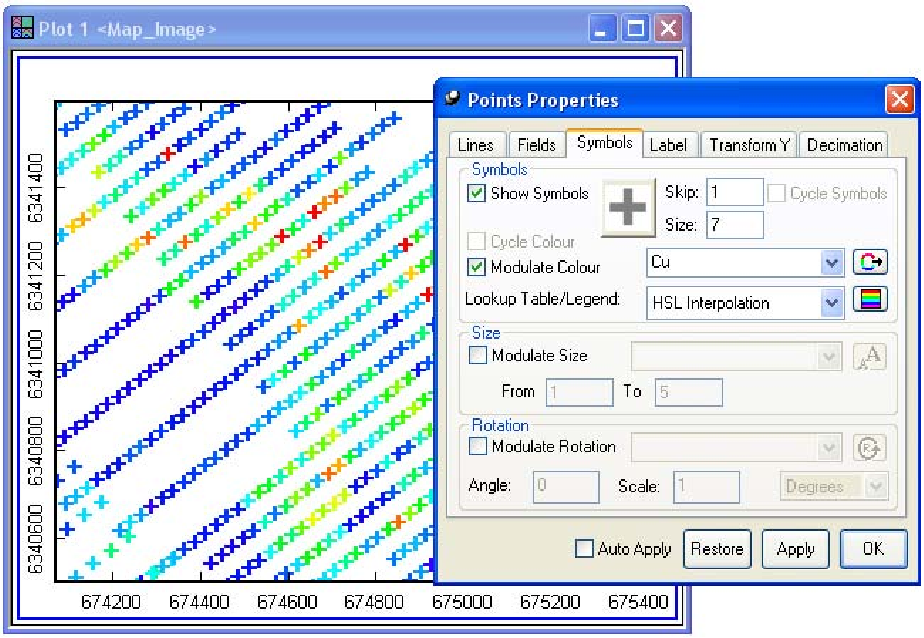
When the column headers and nulls are specified, select the **Import** button. The Text Importer recognises that a Line column has not been specified. It therefore displays a message asking if the data is point data and do you wish to continue. This message is shown in the dialog below:



Text Import dialog with message displayed if point data is being imported

If you respond that the data is point data, the software creates a database in the conventional way and then opens it within Encom Discover PA.

Point data would not normally be displayed in line profiles or graphs since this would usually not be meaningful. Points can however be shown in maps or scattergrams. An example of a point display with modulated colour and size (dependent on a nominated data field) is shown below using a **Curve with Line Styles** disabled and **Symbols** enabled.



Example display of point data within a map as modulated symbols

File Headers Containing Field Names

In some data files, the names of the fields are contained in a comment line in the file header. If you wish to use these field names without re-typing them, ensure that the appropriate number of header lines are selected, and check the Field names are in header box. Next, choose which line of the header contains the field names. Finally, click the **Get field names** button. Note that only field names starting with alphabetic characters or underscores are allowed.

In rare cases, you may need to offset the character position of the start of the field names, using the **Starting at char** entry field.

Multi-banded Fields

Multi-banded fields are extremely useful for data import when requiring data of like information. That is, if the import data contains multiple data columns of information derived from a similar source, we can use a banded-array import specification. Examples of such data include time delay measurements of EM or depths from a geophysical inversion. With these multiple data channels contained within a single data field, it makes their manipulation much simpler.

Multiply defined column data can be presented in import files:

Horizontally

This style of data definition is more common than vertically defined. Each individual reading is recorded by one line in the dataset. An example using Depth and Conductivity is shown below. Note that the record length can be long in these datasets. In this case Conductivity data is recorded to the right in the records.

```
Line   East      Nrth       Fid  Dpt1 Dpt2  Dpt3
10010  302500.2  5682568.5  6237  100  110  120
10010  302500.4  5682579.6  6238  100  110  120 etc
```

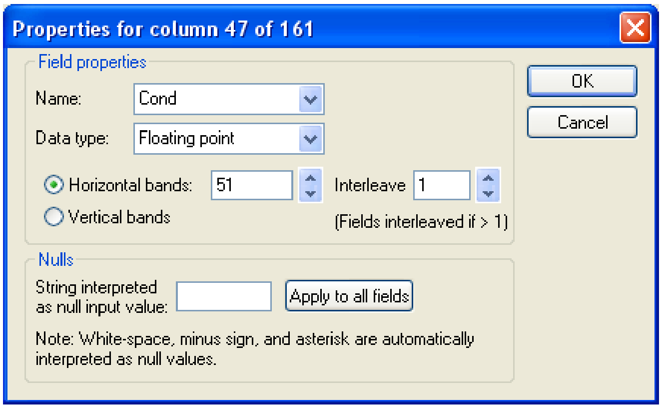
Vertically

If the Vertical bands option is selected, it means that one station’s data is spanned over multiple input text lines. If your input data is vertically banded, you must select the vertical bands option for every field which changes within a station. Other fields which have the same value for each text line within a station must not have the vertical bands option selected.

Below is an example space delimited file which illustrates this. This example contains multi-banded data for 3 stations.

```
X       Y       Depth  Density
1000.0  5000.0    0.0  3.0
1000.0  5000.0  -10.0  3.2
1000.0  5000.0  -20.0  3.3
1000.0  5000.0  -30.0  3.2
1000.0  5000.0  -40.0  3.1
1015.0  5000.0    0.0  3.2
1015.0  5000.0  -10.0  3.3
1015.0  5000.0  -20.0  3.4
1015.0  5000.0  -30.0  3.5
1019.0  5005.0    0.0  3.1
1019.0  5005.0  -10.0  3.2
1019.0  5005.0  -20.0  3.3
1019.0  5005.0  -30.0  3.3
```

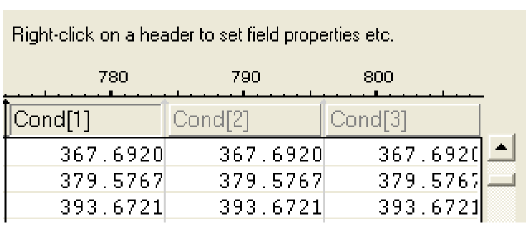
To import this file select the vertical bands option for only the Depth and Density fields. Note that there is no requirement for each station to contain the same number of text lines. The Text Import tool determines the maximum number of bands required and pads the data for any stations which have less bands than this with nulls.



Properties dialog for entry of multiple-channels using either horizontal or vertical bands

To set a field to have multiple bands (for data such as EM channels), double-click on the first column/field label and choose the number of bands in the field properties dialog. Specify if the banded format of the data is **Horizontal** or **Vertical** (see above). Also enter the number of bands that are in the channels of the file to be imported as a banded array data field.

Once defined, multi-banded fields are displayed in the Text Import column headers with square brackets after their names. The number in the square brackets contains the index of each band. Only the first band of each multi-banded field is then selectable.



Importing Data from Grids

If a database of data is not available, one can be constructed using the **File>Open>Grid database** menu option. This option can read a wide range of industry-standard grid formats which include:

- Arc ASCII grid files (.ASC)
- Arc Binary grid files (.ACF)
- Arc Gridfloat Grids (.FLT)
- ASEG GXF (.GXF)
- BIL files (.BIL)
- Encom grids (.GRD)
- ER Mapper (.ERS)
- Intrepid (.ERS)
- Geopak grid (.GRD compressed and uncompressed)
- Geosoft Oasis montaj (compressed and uncompressed) (.GRD)
- GeoTIFF georeferenced TIFF files with file header (.TIF, .TIFF)
- Landmark Seismic grid format (.GRD)
- MapInfo grid files (.MIG)
- Minex grid files (.XYZ)
- Surfer ASCII grid files (.GRD)
- Surfer binary grid (.GRD)
- USGS grids (.USG)
- USGS DEM grid file (.DEM)
- USGS SDTS grids (.TAR)
- Vertical Mapper grid files (.GRD, .GRC)

When loaded in Encom Discover PA the data appears in a spreadsheet window and each row in the grid is converted to a survey 'line', referenced by the Northing coordinate.

Once loaded and displayed in a spreadsheet window Encom Discover PA allows grid data, including multi-banded gridded data to be displayed using the various data plotting series including curve profiles, points, lines and as directional vectors.

It also allows you to examine grid data via the *Statistics Explorer* and to export grid data from the spreadsheet to an ASCII file as located stations.

For information about these display types, see:

- *Profiles*
- 2D Flight Path Maps – see *Creating a 2D Grid Map*
- 2D Lines – see *Displaying Lines*
- 2D Points – see *Displaying Points*
- *Stacked Profiles*
- Directional Vectors – see *3D Vector Displays*

Converting to MapInfo TAB

- *Import ASCII Data to MapInfo TAB*
- *Geosoft GDB to MapInfo TAB*

Import ASCII Data to MapInfo TAB

This menu option initiates the same Text Import utility used for converting ASCII data to a Geosoft database (.GDB) but the ASCII data is converted into a MapInfo .TAB file instead. This is useful in the cases where a need exists for labelling a line or point with information existing in a field consisting of only text. e.g. description tab of a rock unit.

Geosoft GDB to MapInfo TAB

Available from the **Vector** menu. See *Geosoft GDB to MapInfo TAB* in Vector Utilties.

Converting to Geosoft GDB

- *MapInfo TAB to Geosoft GDB*
- *Seismic SEG2 Data to Geosoft GDB*
- *ASCII Data to Geosoft GDB*
- *ASEG-GDF2 to Geosoft GDB*
- *UBC 2D Model to Geosoft GDB*
- *UBC 3D Model to Geosoft GDB*

MapInfo TAB to Geosoft GDB

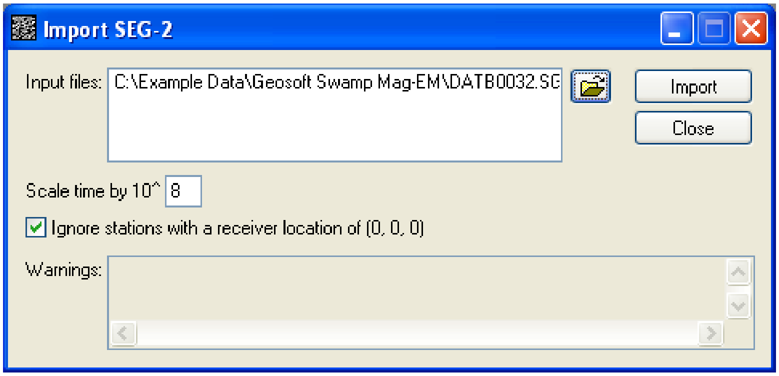
Available from the **Vector** menu. See *MapInfo TAB to Geosoft GDB* in Vector Utilties.

Seismic SEG2 Data to Geosoft GDB

It is possible to import SEG2 format data files into Encom Discover PA for display. The SEG2 data format is typically used in the acquisition of Ground Penetrating Radar (GPR).

Refer to the *Appendix B: Data File Specifications* for additional information on the SEG data format.

Access the Seismic SEG2 plugin utility from the **Database** menu to display the Import SEG-2 dialog and specify a SEG2 format data file (usually these files have a file extension of .SG2). If you wish to process multiple data files and lines consecutively, select these .SG2 files while holding down the SHIFT key.



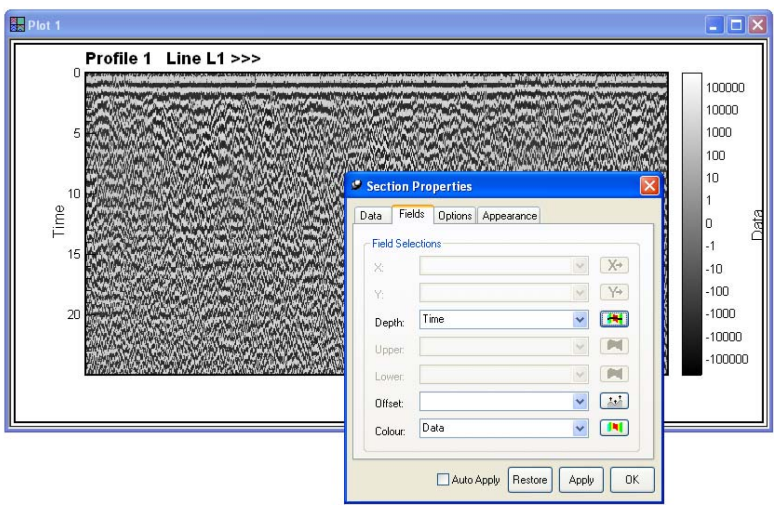
The Import SEG2 utility dialog

The time can be scaled if necessary to account for non-standard recording intervals. By default the time scale is 108 seconds. The import procedure produces a Geosoft database which requires naming. Encom Discover PA automatically accesses this file after creation.

To display a Section Profile of imported SEG2 results:

1. After the SEG2 Geosoft database has been created, open a Section Profile (refer to *Adding Sections*)

2. Select the **Fields** tab and assign the **Depth** entry in units of time (usually either seconds or microseconds) and the **Colour** entry to the Data or amplitude field in the pull-down channel list. Also assign any **Offset** to account for elevation if this information is available.



Section Properties to assign the correct data channels and the resultant Section Profile

3. Click the **Apply** button and the Section Profile should display the resultant SEG2 results. Typically the amount of data is significant and so the redraw time may be quite long.

You may need to alter the colour look-up table to be a greyscale. Additionally, since the Y Axis of the display is required to increase numerically upwards, you may need to **Reverse** the vertical scaling. These two controls are available from the **Colour Table** and the **Y Axis** tabs of the Profile Properties respectively.

Tip The data volume being created in a seismic section (using the Section Profile facility) is very large. Consequently, it is more efficient for faster displays to use the Batch Display Wizard after having created the Section Profile. The Wizard can capture the requested lines of data as bitmaps and these can then be used for rapid display (refer to *Located Images*).

ASCII Data to Geosoft GDB

This menu option is an alternative to the **File>Open>Text Import** menu option for importing ASCII data into Encom Discover PA. The imported ASCII file is converted to a Geosoft database (.GDB) file. Please refer to *Importing an ASCII File* for more information on using this utility.

ASEG-GDF2 to Geosoft GDB

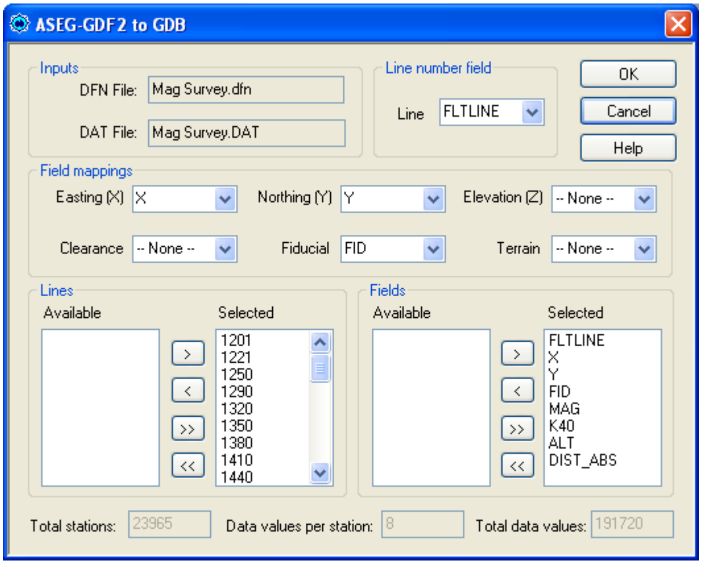
This tool reads ASCII data files in the ASEG-GDF2 standard format.

The ASEG-GDF2 format is defined and approved by the Society of Exploration Geophysicists (SEG) and the Australian Society of Exploration Geophysicists (ASEG). Refer to Appendix A for a description of the ASEG-GDF2 Format. A definition file (.DFN) accompanies a data file and this provides a self-defining format that the utility interprets and imports to a Geosoft database. The database is then automatically opened by Encom Discover PA.

Operation of ASEG-GDF2 Import Tool

When executing ASEG-GDF2 import the software initially requires you to specify a definition (.DFN) filename. When identified, it requests you also identify the file containing the associated data. By default it uses the .DFN filename and appends the file extension .DAT but other filenames can be used.

Once specified, the program interrogates the .DFN and data files and displays a dialog to confirm what it has interpreted plus some information about the data.



PAgdf2Read dialog and confirmation of data channels and lines

At the top of the dialog are the definition and data file names and line number field. The association of field names to the main channel requirements of X, Y, Z, Clearance, Fiducial and Terrain can be individually specified from their respective pull-down lists.

At the base of the dialog you can nominate which data lines and data fields to import into the Geosoft database. Use the left and right arrow buttons to select individual lines or the entire dataset.

At the bottom of the dialog is some information relating to the number of records, channels and data values found in the file. Items such as nulls are automatically defined in the DFN file.

Select the OK button to initiate the import. You are then prompted to select the name of the Geosoft database that is created to hold your imported data.

UBC 2D Model to Geosoft GDB

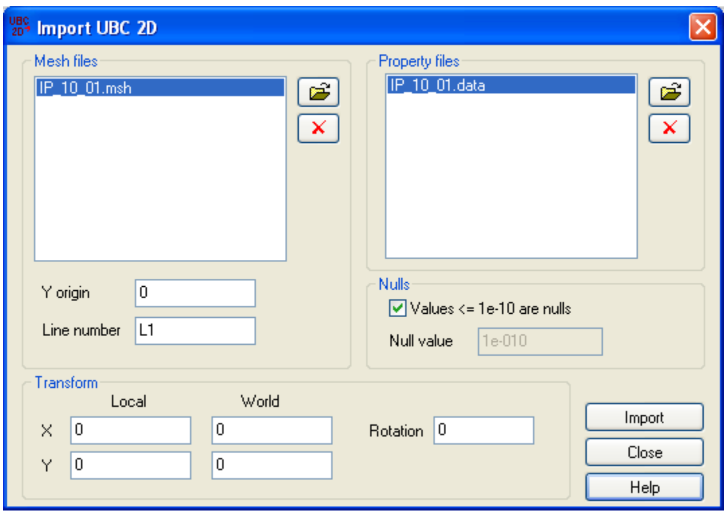
The University of British Columbia (UBC) have produced a range of modelling and inversion programs (eg DCIP2D and DCINV2D) for two dimensional analysis. The output data files from this processing can be displayed by Encom Discover PA in Section Profile format displays.

The output data files comprise a geometry mesh file (with file name extension .MSH) plus one or more property files (for conductivity, resistivity or chargeability). The file format for these files is different from the three dimensional UBC format described in UBC File Formats. A detailed description of these file formats is provided in:

www.geop.ubc.ca/ubcgif/documentation/dcipdocs/fdmesh.html

www.geop.ubc.ca/ubcgif/documentation/dcipdocs/model.html

When initiated the UBC 2D Mesh dialog is displayed. You can define one or more .MSH files (on the left side of the dialog) and various property files to be used. The import procedure creates a Geosoft database with Upper and Lower layer thickness definitions that can be used in Section Profile displays. For additional information on this, refer to *Section Structure and Layers* and *Upper and Lower Boundary Layers*.



Import a UBC 2D Model file dialog

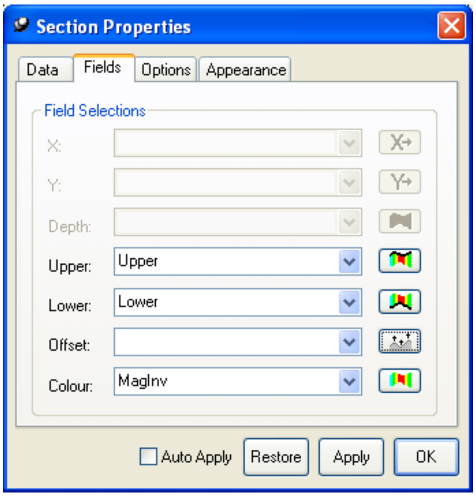
When the Import button is selected, the specified files are read and a datasheet displayed with the data.

To display a Section Profile of imported UBC results:

1. After the UBC Geosoft database has been created, open a Section Profile (refer to *Adding Sections*).

2. Display the Section Profile properties tab and select the Options tab. Enable the **Build from Upper and Lower Boundaries and Layer Property** item.

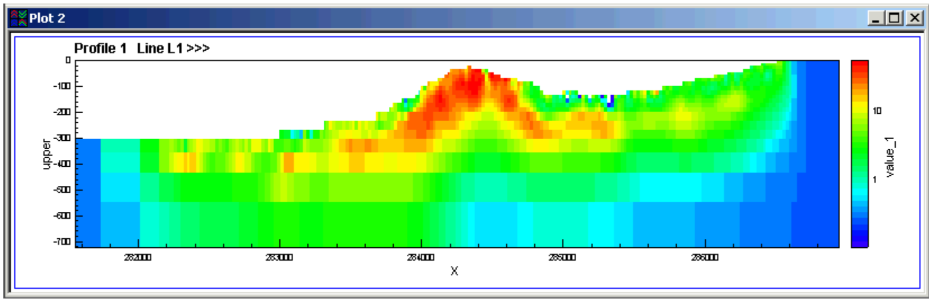
3. Select the **Fields** tab and assign the appropriate **Upper** and **Lower** data fields in the pull-down channel list. Also assign the correct properties channel to the **Colour** entry.



Section Properties to assign the correct data channels

4. Click the **Apply** button and the Section Profile should display the resultant UBC model results. You may need to alter the colour distribution to be logarithmic as by default, a linear colour stretch is applied. This can be altered from the **Colour Scale** tab of the Profile properties dialog.

The resultant UBC profile would appear similar to the section below:



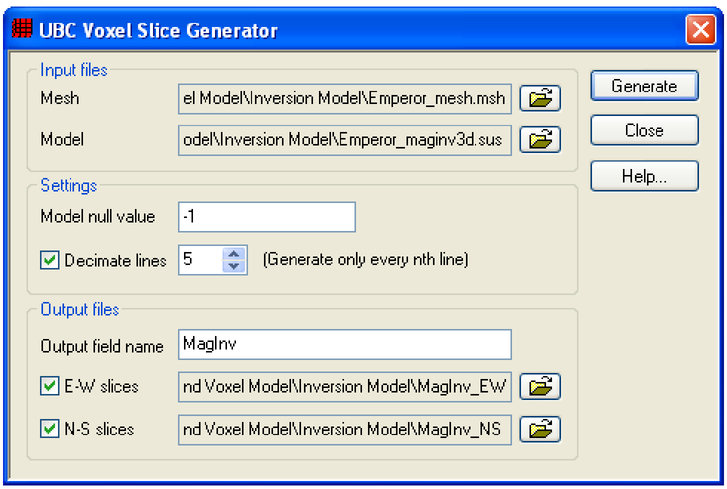
Example Section Profile derived from a UBC 2D inversion

UBC 3D Model to Geosoft GDB

The conversion of a UBC 3D model to a Geosoft .GDB database is a utility tool designed to use a UBC format model and create two Geosoft Oasis montajTM databases. These can then be used for slice or section displays in Encom Discover PA. The slices are formed along the row and columns of the original UBC model but cannot be created transverse or cutting across the primary row:column directions. As a consequence, the orientation of the slices is usually described as being East-West and North-South.

Slices created and stored in two separate Geosoft databases (East-West and North-South) are displayed in Encom Discover PA using the Section Group display formats. The sections can be defined using either the Upper:Lower boundary layer style or the Depth:Value style (refer to Layer Styles of Sections).

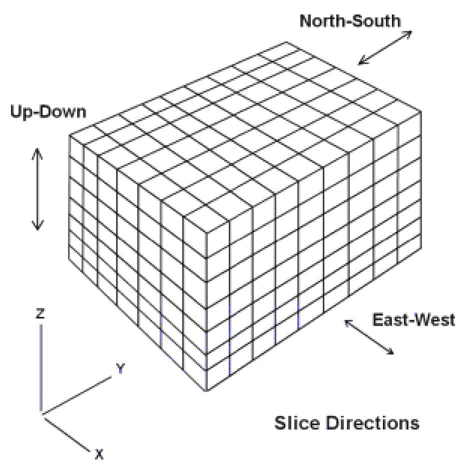
The UBC 3D model to Geosoft database interface appears as:



Slice Generator module interface

Referred to as a Voxel Slice Generator, this utility requires you to specify a UBC model. These model formats have an ASCII header file plus a second data file describing the properties of the various voxel cells (refer to *Appendix E: Supported File Formats* for a specification of the File Format). These file names are entered into the Mesh and Model entry fields of the dialog. You can use the Browse button to navigate to the relevant files.

The output of the Slice Generator is two Geosoft databases corresponding to the East-West and North-South directions. The UBC model is composed of rows and columns within specific layers, as shown below:



North-South and East-West slices of a UBC model

After Slice Generation, the Geosoft databases contain arrayed data corresponding to the following data channels derived from the model:

- **Depth** – the centre point depth of each layer (with n-depths for n-layers)
- **Upper** – the upper depth for each bounded layer (with n-Upper values for n-layers)
- **Lower** – the lower depth for each bounded layer (with n-lower values for n-layers)
- **Output** – the property value associated with the layer, defined by its depth or its Upper-Lower bounds.
- **X, Y and Z** – the locations of the top centre of the various voxels.

Geosoft databases in the East-West and the North-South orientation are created and their names are as specified in the Slice Generator dialog.

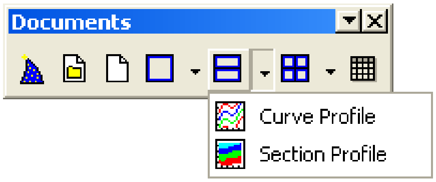
To create the databases with a UBC model specified, click the **Generate** button.

Two Dimensional Profile Display of Slices

To create a 2D Profile of a sliced voxel model, the process is identical to conventional section displays (refer to *Profiles*).

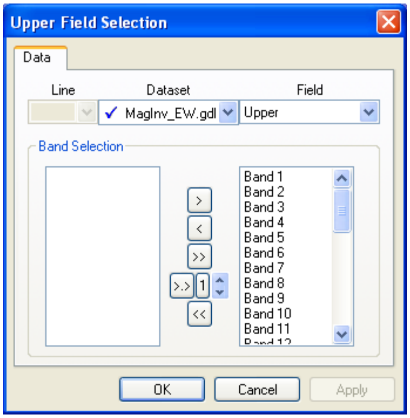
1. Open the East-West and/or the North-South sliced voxel model data.

2. Create a Section Profile from the **New>Profile>Section Profile** menu item or the **Section Profile** pull-down item on the Documents toolbar.



3. Either the Layer Boundary or the Depth:Property layer style of section creation can be used (see *Adding Sections*).

   In this example the **Depth:Property** specification is used, but by altering the Section branch Options tab setting, either is available. In the Workspace tree select the **Depth** item and set the data source to the Depth item of the required database. You can use every layer or you can use a subset of layers to speed up the display.



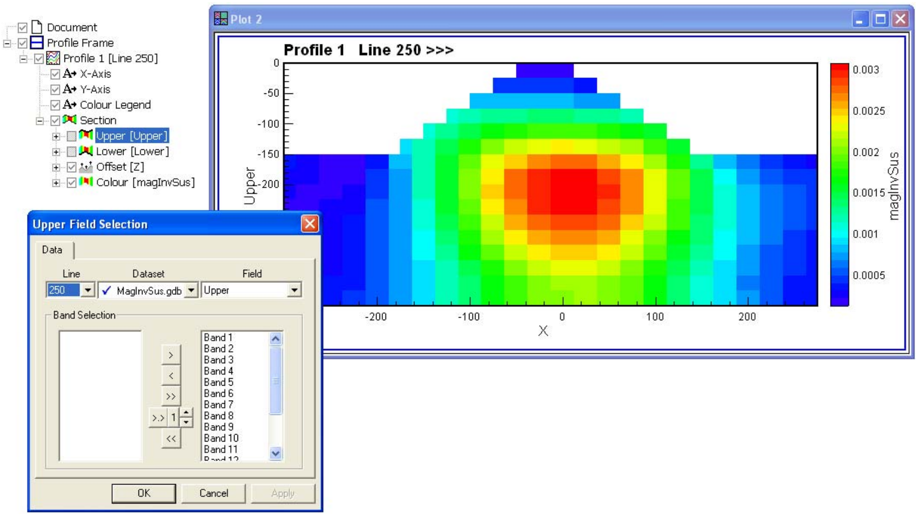
Selection of Depth bands for the slice display

4. Repeat the band selection but for the **Colour** branch of the Section. Ensure that the same bands are selected for the Colour branch to match the bands chosen for the Depth branch.

5. Once both the Depth and Colour bands are applied, a section should be displayed. You may need to offset the upper surface (using the Offset level of the tree) to compensate for topography.

6. The **Line Iterator** can be used to cycle through the slices from the source database. Note that the slice line numbers are annotated by numbers that increment in accord with their east-west or north-south location.

   In addition to using the Line Iterator you may wish to modify the look-up table of the colour rendering of the cells displayed. Use the **Profile Colour Axis** tab to do this.

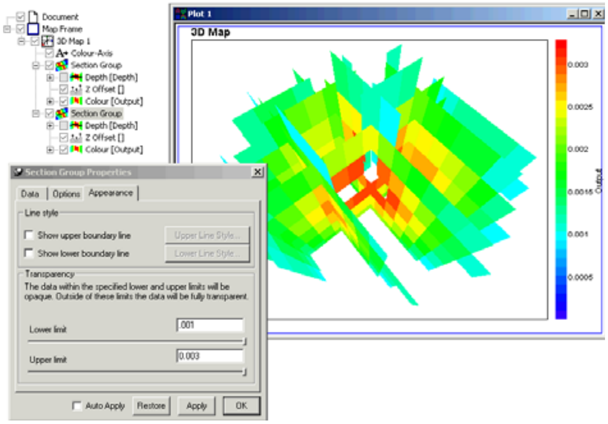


Example of a slice displayed as a 2D Profile section

Making Slices Transparent

It is possible to apply specific colour-look up tables to make slices appear transparent. This allows you to view anomalous zones of a slice without them being obscured by the background layers of slices.

To do this, select the **Section Group** branch of the Workspace tree and from its property dialog choose the **Appearance** tab. Using the two slider bars available, specify the upper and lower data ranges required to be displayed. The sliders are interactive and update the display when the **Auto-Apply** option is enabled.



Display of slices with thresholding of the data values between specific ranges

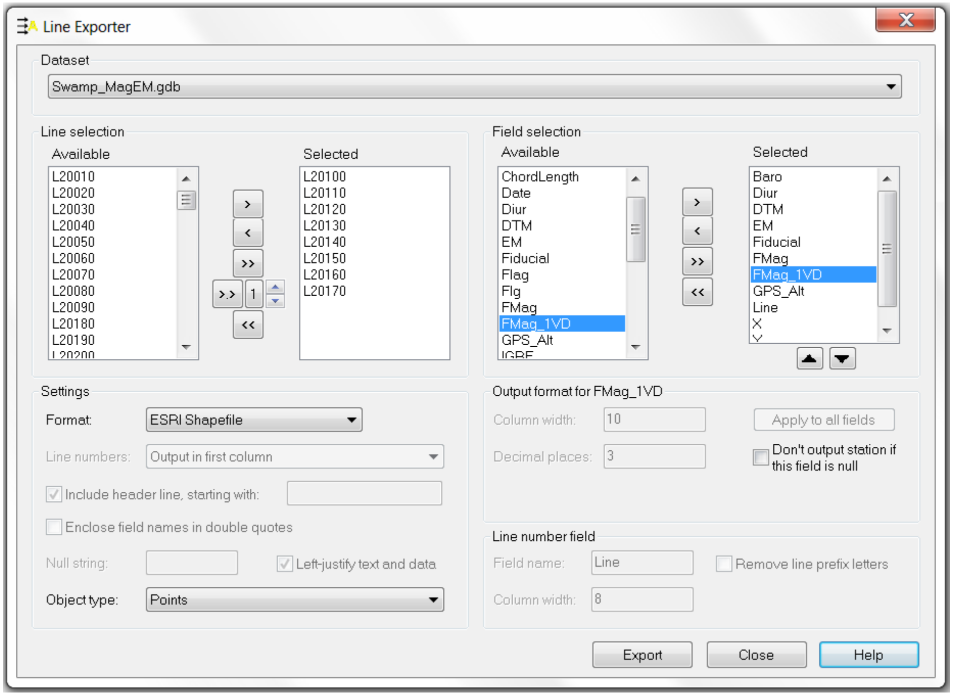
Exporting Data

It is often useful to export a selection of line and channel data in a loaded database to a predetermined ASCII format file. The **Line Exporter** utility will perform this requirement and can be accessed from the **File>Export data** menu option.

The output file format can be any of:

- ASCII format with space delimited, comma separated (.CSV) or fixed width
- CSV format with or without line numbers
- Geosoft XYZ format with line numbers and line prefixes
- ESRI Shape file format (.shp)
- ASEG GDF2 format (.DFN and .DAT) with line numbers and line prefixes

The dialog to control the export is:



Line Export utility for the ASCII output of databases

On the left of the dialog, you can nominate the data lines to be exported. On the right, the required data channels are specified. Various settings permit:

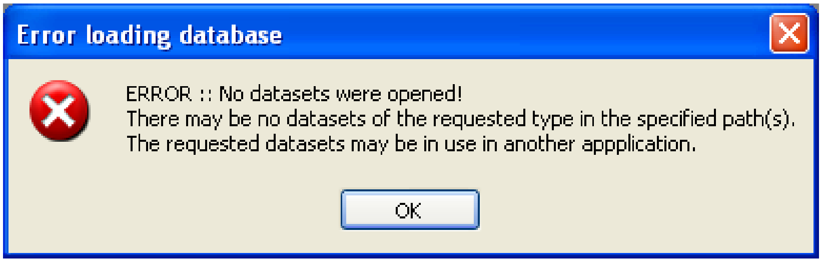
- Format - to be Space Delimited, Comma delimited or Fixed Width
- Line Numbers – can be appended in the first column, on a Separate line (such as Geosoft XYZ format) or with No line numbers.
- Include a header line
- Include field names derived from the channels of the input database
- Define a Null string if required
- Specify Point or Polyline objects to be exported
- Fixed width format enables the Column Width and Decimal places
- Line number field can be specifically named with a prefix if required

Once the various parameters are specified, click the **Export** button. Name the output ASCII data file in the **Save As** dialog and click the **Save** button.

When a Database Fails to Open

There are several reasons why a database may not open. The most common are:

- The database is already open in another application.
- The file you are attempting to open is not a valid database.
- The database file cannot be found at the specified folder location.



Error message displayed when a dataset is open in another application.

If an error message similar to the one above is displayed, do the following:

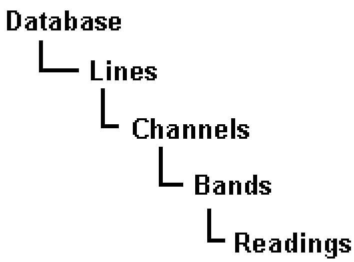
1. Check that the database is contained in the specified folder.
2. Ensure the database is not opened by another application.
3. Close Encom Discover PA and open the database using its native application.
4. If the database can be opened successfully in its native program, check that the following data fields exist – Linenumber, X, Y, Fiducial (mandatory fields) and Flightnumber (for Intrepid databases, refer to the SurveyInfo File). Note that Flightnumber is an optional field.

After performing these checks, if you are still unable to open a database, do the following:

- Try to open one of the sample databases supplied with the installation.

If you are still unable to resolve the problem, then contact Encom support.

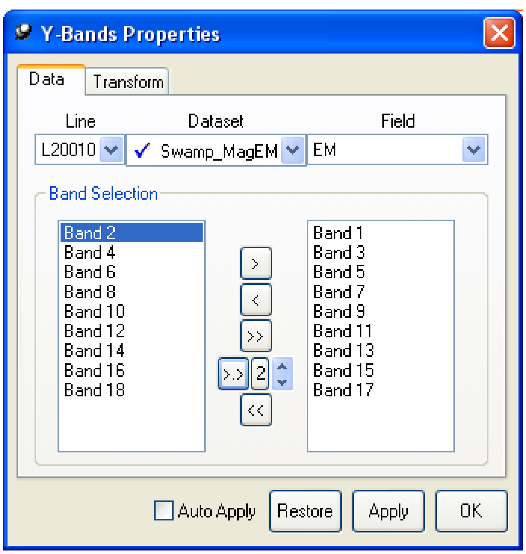
Selecting Channels and Tables



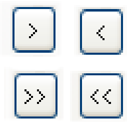
Once data is accessed by Encom Discover PA it can be selected at a number of levels. Usually the databases are structured such that they contain discreet traverse lines along which data fields are found. In turn, each data field may have a single or multiple set of data bands. Each band of the data has the individual data readings corresponding to the data sensor used to acquire the original data.

Both Oasis and Intrepid databases are capable of storing data using this database structure. In some cases, a Geosoft database may contain points or drillholes rather than lines. For these datasets, Encom Discover PA reads the individual data points and drillholes as if they existed on a single line.

Selection of the required data within Encom Discover PA is controlled by a single dialog type that is common throughout the software and enables data selection to the band level.



Data channel/field selection dialog



For a selected database/line/field combination, the bands of a multi-band dataset can be individually nominated using the individually selected band buttons to select and to deselect or alternatively, you can select or deselect all bands using the **select** or **deselect** buttons. You can highlight selected bands using the SHIFT or CTRL keys.

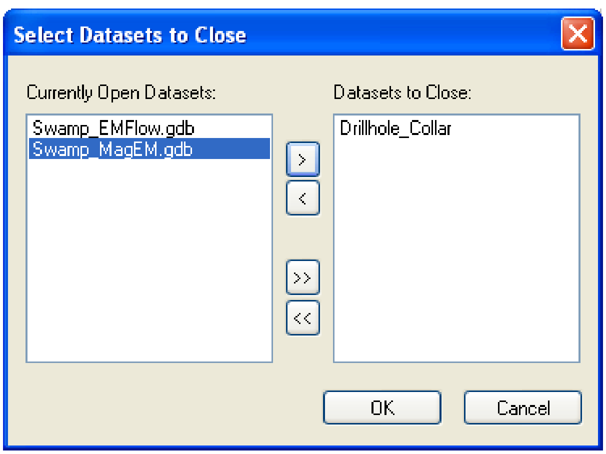
Ancilliary Data

In Encom Discover PA, banded data and especially banded geophysical EM data can be displayed and analysed. For time domain EM, this form of data has channels in the various bands that are representative of time delay windows. Each channel represents a period of time in which received data is recorded from an EM sensing receiver. As Encom Discover PA has some operations that can make use of the time information for analysis operations (for example, the decay analysis tool), a file called `ATEM.TXT` is used to define the time channels.

The format of an `ATEM.TXT` file is provided in *Appendix B: Data File Specifications*

Closing a Database Link

To selectively close databases, from the **File** menu, choose **Close Datasets** and nominate which files to close. To close all databases, choose **Close All**.



Closing selected datasets

Preferred Directory Access

Encom Discover PA uses a Global Directory to access data. The Global Directory determines the folder that is automatically navigated to when a file of any type is opened.

By default, this location is updated to be the location of all subsequent file locations until it is changed by:

- Opening any data file in a different folder with the **Use Global Directory for all file locations** option enabled,
- Selecting the **Global Directory** in the **File Locations** tab of the **Tools>Options** menu item and clicking the **Modify** button, or
- Selecting a **File>Set Project Folder** option and specifying a particular directory path.

Note If you wish to specify a project directory for use but sometimes want to access files from different locations, define the Global Directory but disable the **Use Global Directory for all file locations** option. This allows the Global Directory to always remain at the defined folder but does not prevent you accessing files at different locations.

For additional information, see *Setting the Global and Project Directories*:

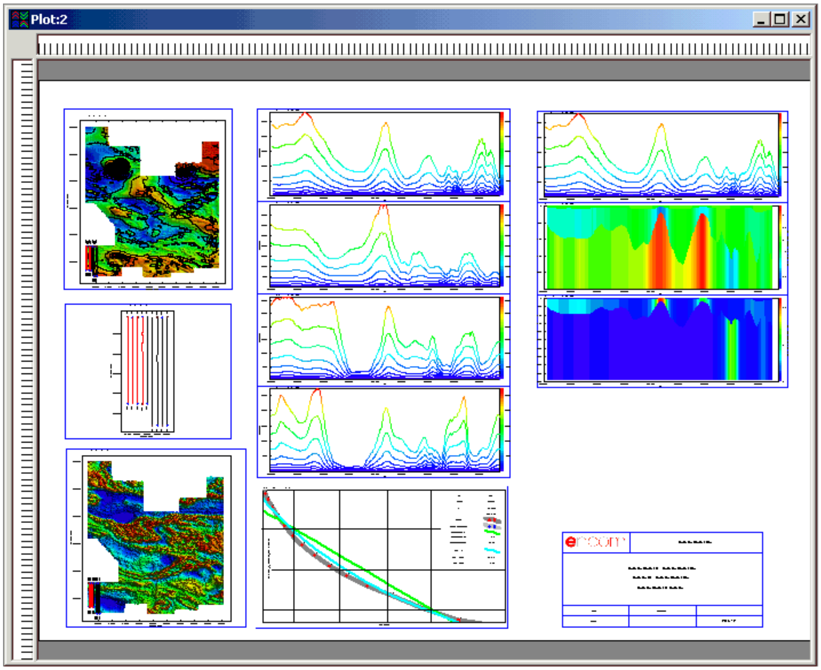
9 Page Layouts

In this section:

About Page Layouts

The Page Layout display is used to combine various display frames for on-screen viewing and a preview of hard copy. Any frame can be copied into a layout. The Page Layout provides a WYSIWYG (what-you-see-is-what-you-get) view. Initially after a new document (or any display window) is requested a blank window is displayed with a page of paper corresponding to the **File>Page Setup** paper size. The other document views, title blocks and objects are then added to the paper if a finished layout is required.

Use the **File>New>Empty Document** or the button to create an empty document. Any already displayed frames in existing documents can be appended to the layout using Copy/Paste operations. An example of this using map, image, profile, layer and title block components is shown below:



Composite graphics objects in a Layout Window with maps, images, layers Profiles and title block

Document Window Properties

A Document in layout mode inherently has no properties but inherits the properties from the frame objects added to it. The Workspace tree expands to reflect whatever frames are added and the various properties are derived from these frames. Refer to the various frame property descriptions for information on controlling property dialogs.

Adding Frames to a Document

Two methods are available to add existing frames to a new, empty or partially occupied document. The methods available relate to the alignment required in the end product. No Alignment and With Alignment are available. The copying procedure is identical in both methods.

- *Adding Frames Without Alignment*
- *Adding Frames With Alignment*

Adding Frames Without Alignment

To add existing frames to another document, configure the source window display as required. Any frame can be used (profile, map, scattergram or graph). Follow the steps below to copy and paste a frame:

1. Highlight and select the required frame (e.g. Map Frame, Profile Frame etc).

2. Click the right mouse button and from the pop-up menu select the **Copy** item. This operation places the frame on the Windows clipboard.

3. Select the target document to receive the copied frame. Ensure its paper size has the required dimensions (using the **Page Setup** option). Using either the **Edit>Paste** menu item or the **Paste** button, position the cursor into the new document and either click the left mouse or position the cursor and with the left mouse button pressed drag out a rectangle to hold the copied object.

4. Re-size and position the pasted object as required.

Note Certain pasting operations are not permitted by Encom Discover PA. For example, it is inappropriate to paste and then embed a Map Frame into a Profile Frame. Encom Discover PA prevents you doing these types of operations by not displaying the **Embed** option.

You can repeat the above operation and align objects manually as necessary. Any combination of frames can be copied using this approach, including saved components such as title blocks.

A disadvantage of this method is that the positioning of objects is by graphical alignment and not by a program controlled manner. Note that the frames are positioned according to the Snap control for the document (see *Snap Operation*), but alignment can still be inaccurate. A more accurate method of aligning objects is to use frame cells and frames. This method is detailed in *Adding Frames With Alignment*.

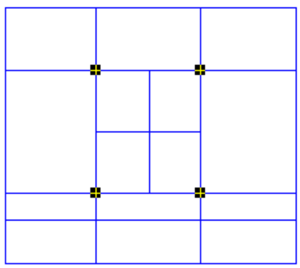
Adding Frames With Alignment

You can add frames to a document such that they are automatically aligned and involves creating an empty, container frame in the target document before inserting the frame items. While this method provides good alignment of the objects it restricts flexibility in sizing of objects within the container frame cells.

To align objects using container cells in a Document:

1. Initially create a Document ensuring the **Page Size** is as required (using the **Page Setup** option for the document).

2. Insert an empty frame using the Frame tool. You can use the **Insert>Frame** menu option or the **Insert Frame** button (see *Frames* and *Split and Merge Frames*). Display the Properties dialog of the Frame and specify the rows or columns as required and click the **Apply** button. Configure the cells by row and column as necessary to hold the various objects. The Properties dialog of the created frame allows you to specify the number of rows and columns required. Alternatively, hold the cursor over a horizontal (for rows) or vertical (for columns) and click the right button. Select the **Split** option to create additional cells or **Merge** to remove cells.



Tip To make cells within cells, use a second empty frame and insert this into an existing cell and manipulate the number of rows:columns in the same way.

3. Highlight the various configured document frames that are to be transferred to the layout. From the Workspace tree (or by right mouse clicking in the active document window) select the **Copy** option. This copies the frame contents of the active document to the Windows clipboard.

4. Select the target document and position the cursor over the container cell to contain the copied object. Click the right mouse button and select the **Paste** item from the pop-up menu. The contents of the Clipboard are pasted into the selected cell and sized to fill the entire cell area.

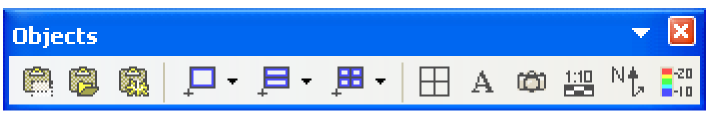
You may need to adjust the positioning of the various objects by moving the cell boundaries or adding/subtracting margins to the object in the property of the relevant layer.

Adding Objects to a Page

To any display document, Encom Discover PA can add a wide range of ancilliary objects. These can be:

- **Text** – Completely configurable for font, colour, size and content. See *Adding Text*.
- **Images** – Loaded from a file or drawn in the document display. See *Adding a Picture*.
- **Preformed Components** – Title blocks, legends etc that are supplied externally or composed by you.
- **Graphical Objects** – *Scale Bar*, *North Arrow*, *Floating Colour Bar*, etc.
- **OLE Objects** – Created by the applications that support OLE (Object Linking and Embedding) technology - such as MapInfo Professional maps, pictures, MS Word documents, MS Excel spreadsheets etc.

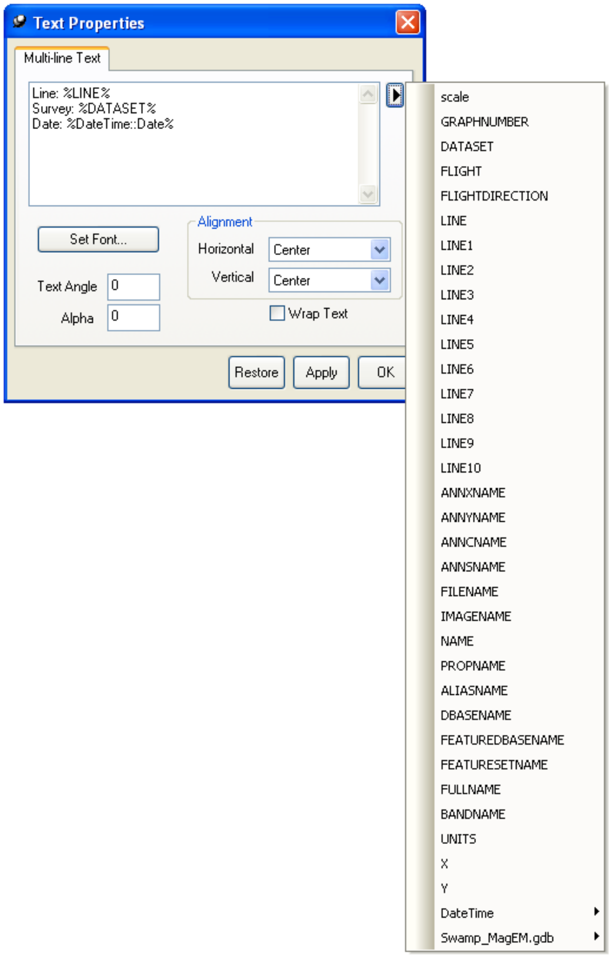
Individual objects are placed within a display document from the Object Palette toolbar.



Adding Text

Text can be added to a display as a title, interpretation, legends and for any instance where character strings are required. To add text, select the button of the Object Palette toolbar or from the **Insert>Text** menu item. Position the cursor over the active window area required and either click the left mouse button or drag a rectangle out by holding the left mouse button down while dragging.

A default text string is displayed. A **Text** branch (with the first few characters of the text string) appears in the Workspace tree. Display the **Text Property** dialog to access the following controls:



Text Properties dialog for content, font and justification control.

- **Text Content** – Single or multiple line text entry is supported. To make the text wrap onto a new line, press the CTRL+ENTER key combination.

- **Font Control** – Invoked from the **Set Font** button, it allows selection of a font from those currently installed on the system, specify the style (bold, italic or underline) , size and colour.

- **Wrap Text** – Text that is longer than can be displayed in the assigned (graphically controlled area of the display will wrap to the next line. This is especially useful for formatting legends or blocks of text.

- **Angle Control** – The text angle can be entered to make text to appear rotated. Increasing positive angles are anti-clockwise.

- **Alignment** – Both vertical and horizontal alignment (within the drawn bounding rectangle as initially defined) is provided.

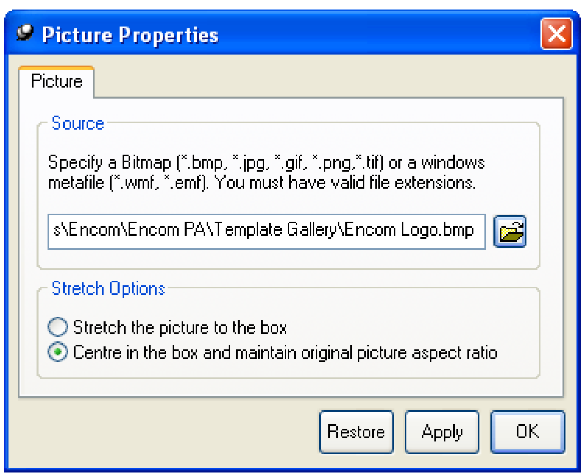
A push button arrow on the right of the text dialog allows a drop down list of **Variable Substitutes**. Selecting one of these items will substitute the variable selected. The syntax of the substitute variable is embedded between two % characters (eg %LINE%.).

Note When adding new text, holding down the SHIFT key when inserting will not automatically embed it into the encapsulating frame. It will stay floating until you embed it.

Adding a Picture

Pictures (company logos, clip art etc) can be added to any display document. Supported picture formats are listed in *Appendix E: Supported File Formats*. Windows metafiles (.WMF and .EMF) are also supported. To add a picture, select the **Insert>Picture** menu item or the button. Drag out a rectangle or pick the location of the picture in the document. Once the picture is placed, the Property dialog allows you to navigate to a picture file.

You can size the picture within a defined area and fit to the area initially defined or maintain the aspect ratio



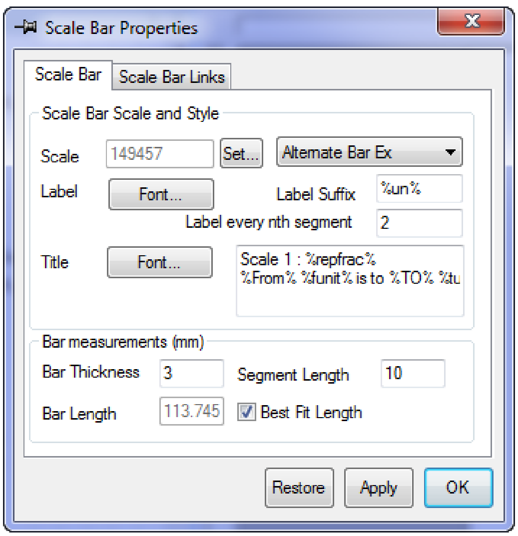
Picture Properties dialog

If a picture is not accessible or the filename has been changed etc, Encom Discover PA displays it as a red crossed area. This should be remembered if pictures are used in templates but files are changed when the template is re-used.

Note When adding a new picture, holding down the SHIFT key when inserting will not automatically embed it into the encapsulating frame. It will stay floating until you embed it.

Scale Bar

Scale bars are useful to complete a map or indicate the output scales of a profile or a map. Automatically sized and scaled scale bar objects are available from the Objects toolbar (using the button). Once the button is clicked, position the cursor at the desired location and either click the left mouse button (to draw a default scale bar), or drag and size a rectangle area. Scale properties are available from the Workspace tree.

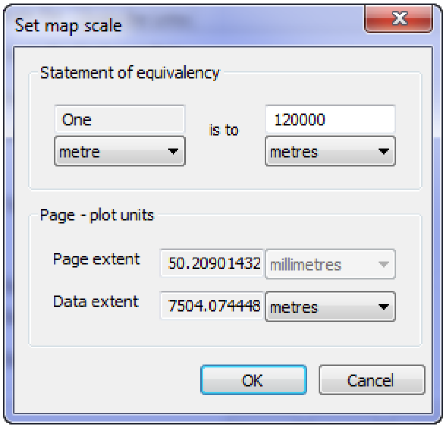


Scale bar Properties dialog

The scale used in the drawing of a scale bar is automatically determined by the object into which it is inserted. This scale can be overridden in the **Scale Bar Links** tab (refer to *Changing the Scale of Scale Bars*).

Note When adding a new scale bar, holding down the SHIFT key when inserting will not automatically embed it into the encapsulating frame. It will stay floating until you embed it.

To edit the scale from the **Scale Bar Properties** dialog press the **Set**... button to view the **Set map scale** dialog. Alternatively you can set a map scale via the **Map Frame Properties**. Refer to *Profile Frame Properties* for more information on this. The Map Frame scale and the Scale Bar should show the same value.



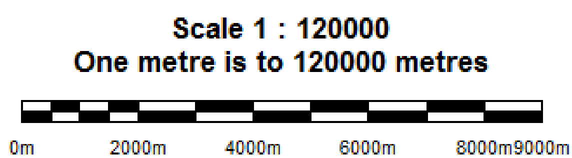
Set map scale dialog.

The appearance of scale bars is controlled by font, label and available types (**Solid**, **Alternate** and **Mixed Alternate** breaks). Labelling and intervals of text along with length and thickness controls are available including both metric and imperial (feet) measurements.

To set the scale a statement of equivalency is used. For example ‘One inch is to 10 miles’ or ‘One metre is to 100,000 metres’. In the latter example, the text substitution for this, which can be added to the scale bar **Title** in multi-line format, is:

```
Scale 1 : %repfrac%
%From% %funit% is to %TO% %tunits%
```

And the resultant display for this in the scale bar is:



There are three measures of units:

- from
- to
- data

The ‘from’ and ’to’ units refer to the statement of equivalency and the user can set them to whatever they want.

The data units are the units of the actual data in the plot. If possible, this is obtained from the frame coordinate system. This can also be manually set in the **Set map scale** dialog.

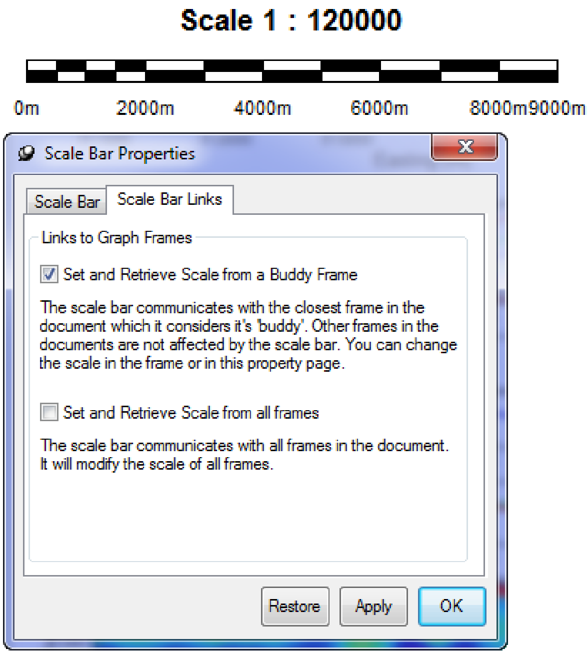
The scale bar will be rendered using the ‘to’ units.

To help with all this there are a number of text substitution codes. By default the scale bar title shows the representative fraction and the statement of equivalency below it.

```
//      From unit
//      %fun%, %funit%, %funits%            (abbreviated, singular, plural)


//      To unit
//      %tun%, %tunit%, %tunits%            (abbreviated, singular, plural)
//      %un%, %unit%, %units%               (abbreviated, singular, plural)


//      Data unit
//      %dun%, %dunit%, %dunits%            (abbreviated, singular, plural)


//      Statement of equivalency
//      %from% %From%
//      %to% %TO% %To%                      (integer, decimal, one decimal point)


//      Representative fraction 1 : N
//      %repfrac%  %REPFRAC%  %Repfrac%     (integer, decimal, one decimal point)


//      Plot scale (referenced to data units)
//      %scale% %SCALE% %Scale%             (integer, decimal, one decimal point)
```

Changing the Scale of Scale Bars

Although the scale of a scale bar by default is determined by the data object associated with it, the Scale Bar Links tab allows you to change this so that the scale bar can control the scale of associated grid objects.



Scale Bar Properties to control the scale linking to frames.

Of the two options provided, the first one forces the scale bar to obtain its scale from the nearest displayed object. Since an Encom Discover PA document may have numerous objects contained (each with their own scale), this is an important method of operation. The second option forces all contained objects to have the scale of the scale bar.

North Arrow

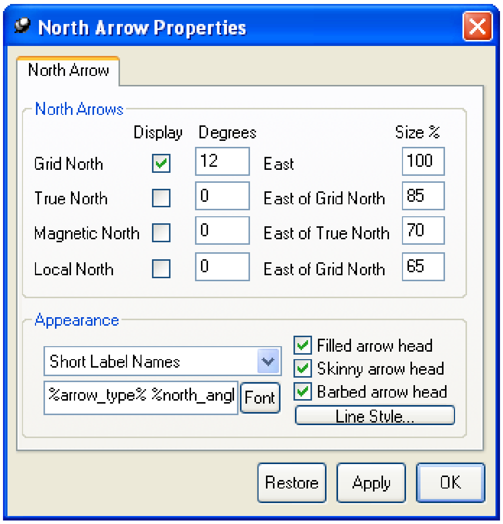
The north arrow is used in many map displays to orient the data and complete a layout presentation. The **North Arrow** button on the Object toolbar can be selected and a north arrow location can be placed within any document.

The properties of a North Arrow allow for the presentation of arrows of different types:

- **Grid North** – Used as the reference north direction (from which all other north angles are measured). Grid North is typically used for local grid directions and projections.

- **True North** – Used for deviation from Grid North but usually is the same. True North is used typically in relation to Magnetic North.

- **Magnetic North** – Indicates the direction of a magnetic compass with measured declination of the Earth's magnetic field.
- **Local North** – Indicative of a local grid and any deviation in direction for local surveying from Grid North.

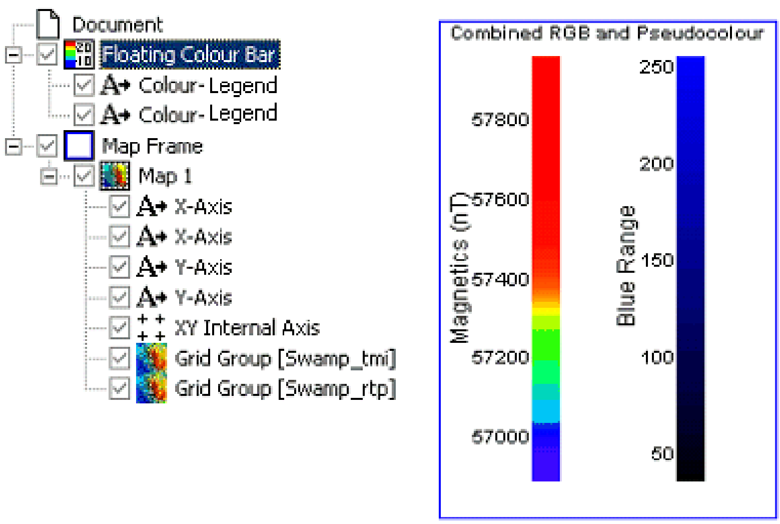


North Arrow properties dialog and controls

Note When adding a new North Arrow, holding down the SHIFT key when inserting will not automatically embed it into the encapsulating frame. It will stay floating until you embed it.

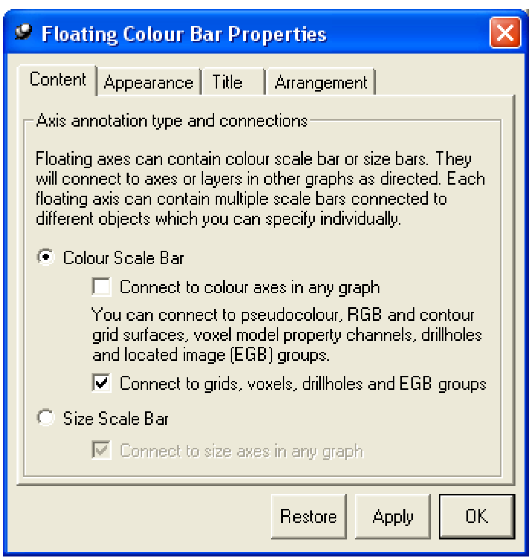
Floating Colour Bar

For images, graphs, maps, voxel models and colour presentations where no colour axis is automatically created, a Floating Colour Bar can be added to a display. The object acts similarly to a text or north arrow item in that it can be positioned and sized anywhere within a display. The axis can be connected and so obtain its colour from a grid image or graph. An example of a Floating Colour Bar is:



Example of Floating Colour Bar with colour bars derived from two image displays

Note from the Workspace tree that a **Colour-Legend** branch is associated with each vertical colour bar of the axis. The connection of the bar can display colours or sizes and connect to different axes or layers individually. This connection is specified from the **Content** tab.



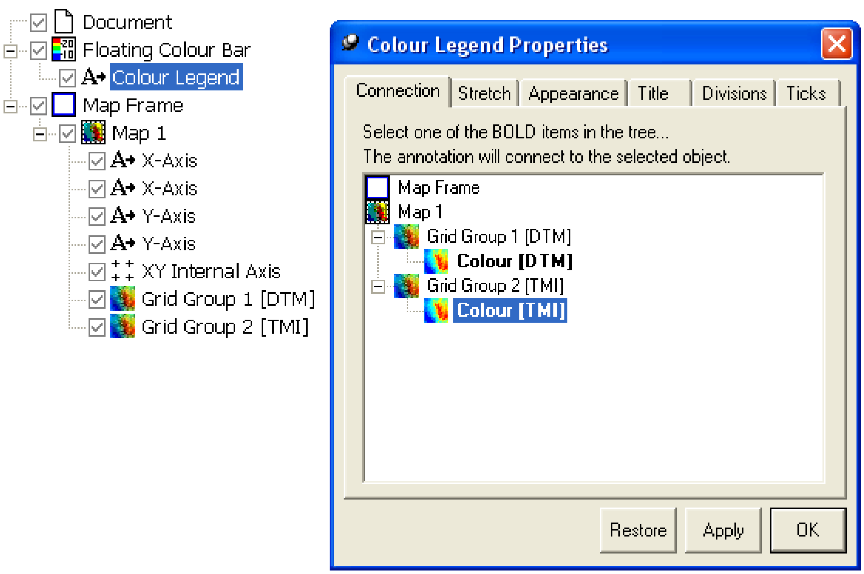
Controlling the Content of the Floating Axis

Controls of the size, title, centreing etc of the colour or size bar are specified in the **Appearance**, **Title** and **Arrangement** tabs. Their operation is identical to axis controls described in *Axes and Annotations*.

Individual control for colour and size bars is controlled from the properties of the various Colour-Legend branches of the Workspace tree beneath the Floating Colour Bar (see the tree above). Refer to *Floating Colour Bar Properties* for further information.

Floating Colour Bar Properties

Control of the individual colour or size axes of the Floating Colour Bar is derived from selecting the properties dialog of the **Colour-Legend** branch of the Workspace tree associated with the Floating Colour Bar.

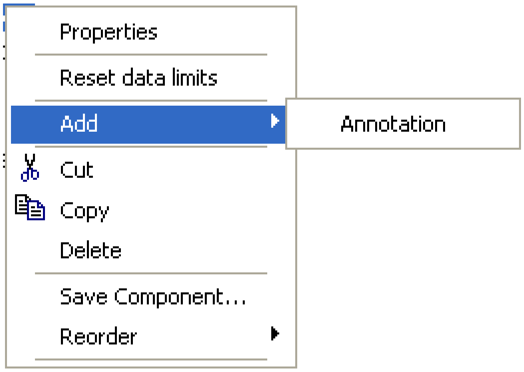


Control dialog of Floating Colour Bar Colour layer

Using the **Connection** tab (which displays a tree of the available imagery), nominate which image layer (or graph in the case of a size axis) is required for the Floating Colour Bar. When you click the **Apply** button, the axis updates the display.

You can **Stretch** the distribution of the colour display as well as control the **Colour Scale** and **Colour Table**. If an alternative colour table is to be used or the LUT is to be edited, you can access the table using the LUT Editor. Annotation of the displayed colour legend can be controlled for **Ticks**, **Division** and **Title**. Controls of the position, title, divisions and ticks are described in *Axis Properties*.

If multiple floating colour bars are to be used (as shown in an example in Floating Colour Bar), select the Floating Colour Bar layer of the Workspace tree and click the right mouse button. From the pop-up layer, select the **Add>Annotation** item.

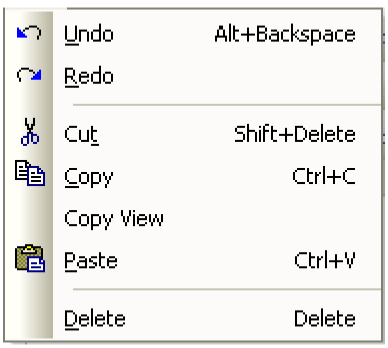


Adding additional Colour Bar annotation

Editing Documents

Document within Encom Discover PA can be selected, copied, pasted, cut and deleted from the Workspace tree. Considerable control in moving a document from one location to another is available while retaining all the attributes and properties of the object.

Certain editing controls are available from specific layers that are not available from other layers. This relates to the object type and its edit properties.



The Edit menu options

For more information, see:

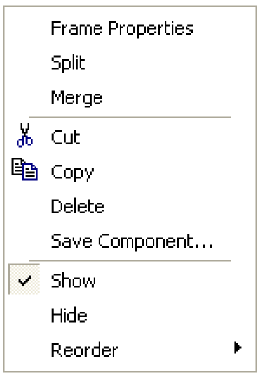
- *Copy, Delete, Cut and Paste*
- *Split and Merge Frames*
- *Embed and Float*

Copy, Delete, Cut and Paste

In most instances the **Edit** menu items have the Copy, Paste, Cut and Delete options enabled. Also on most occasions the pop-up menu appearing on right mouse button click on a layer has the **Copy**, **Cut** and **Delete** options. Paste may not be applicable since it may require a specific cursor action. In these cases the Paste option is available from the Edit menu or the button of the Objects toolbar.

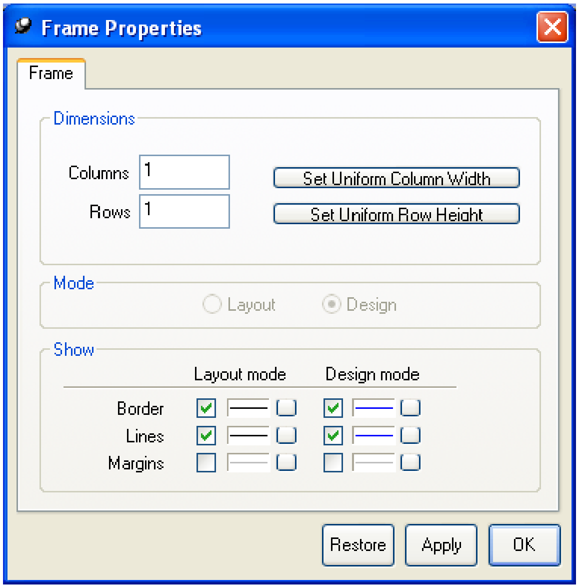
Split and Merge Frames

The Spilt and Merge menu commands are used in combination with frames. The pop-up menu displayed with a selected frame can display a grid’s properties but also allows you to insert columns or rows. When these are added to a grid you can position them by placing the cursor over the cell boundary and dragging it while the left mouse button is clicked.



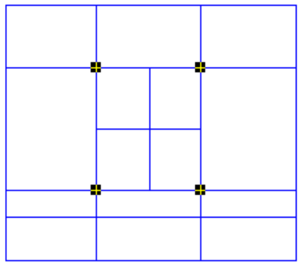
If a group of sub cells (cells within a cell) is required, you can insert a new container frame (using the **Insert Frame** button or the **Insert>Frame** menu item). To do this, click the **Add Frame** button, place the cursor in the cell to have the sub-grid and then click the left mouse button.

Alternatively, you can specify the number of Rows and Columns to be contained within a Frame from the Frame Properties dialog.



Enter the number of Rows and Columns required in a frame

If you need to remove a cell row or column, use the Merge menu item. Similarly, a fast method of adding a new cell is to Split a cell.



When objects are to be placed in grid cells, you need only point to the relevant cell and Paste the contents of the Windows clipboard. When the object is pasted, it assumes the outer size of the containing cell and is locked to the cell size. It is possible to unlock the object by breaking the cell-object link. This is done by the Float menu item (refer to *Embed and Float*).

Embed and Float

Another menu option that is available from the pop-up frame menu allows objects inserted in a cell to be moved and re-sized. A link is created between an object (such as a document or text etc) when it is inserted in a cell. This link can be broken by floating the object. To float an object, activate it (use the CTRL and left mouse button if required) and then select the Float item from the pop-up menu. Once floated, an object is free to be moved to any new location.

To re-insert the object into a grid cell, it can be moved and dropped to a new cell position. To re-establish a link it to the new cell location the pop-up menu Embed instruction can be used. The Embed command locks the relocated object to the new cell and re-scales it accordingly.

Note When adding new objects such as text, floating colour bars, pictures, scale bars etc. holding down the SHIFT key when inserting will not automatically embed the object into the encapsulating frame. It will stay floating until you embed it.

Note Certain pasting operations are not permitted by Encom Discover PA. For example, it is inappropriate to paste and then Embed a Map Frame into a Profile Frame. Encom Discover PA prevents you doing these types of operations by not displaying the Embed option.

---

10 Display Properties

In this section:

Documents, Frames, and Objects

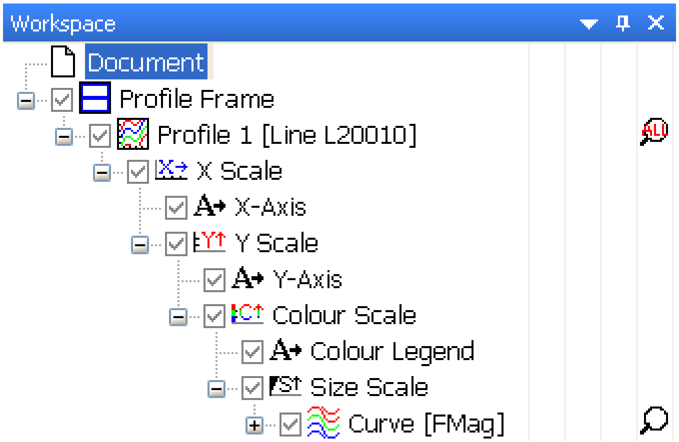
This section describes the properties of documents, frames, and graph objects. In Encom Discover PA, all profiles and graphs are based on the same fundamental graph object. Consequently, the property page and dialog descriptions for a profile are equally applicable to all graphs - including those created in map, profile, scattergram, decay curve and graph displays.

A profile is a graph object, except that the X axis range is scaled according to a cumulative distance channel automatically computed by Encom Discover PA (called ProjectedDistance). This channel is computed individually for each line of a survey database. This range and scale is shared with all profile graphs contained in a common profile frame. The user is not given the opportunity to select an X data channel for horizontal scaling but he is given control over annotation of the X axis (eg Fiducial, Easting etc). A profile can only display data from a single traverse line in a single track. Multiple tracks can be added to display data from multiple lines.

Encom Discover PA uses the multi-branched Workspace tree to control profiles. Each branch of the tree has its own properties. Note that if the tree is displayed as concise or minimised, then it is possible that not all components of the document are displayed. In this case, the properties of hidden components may be accessible higher in the tree – often at the Profile branch.

Properties dialogs are available for each level of the Workspace hierarchy used for a Document that contains a Profile. Property dialogs have been designed to have similar size, appearance and common features. Various tabs at the top of each dialog allow you to select the property required.

In the Workspace, if the full document tree is exposed (click right and select **Tree Content Verbose**), the following expanded structure is displayed.



Profile verbose tree structure showing relationship of all axes

See *Working with the Workspace Tree* and *Branch (Object) Properties* for information on how to display the property dialogs.

Document Properties

The Document is the highest level of the Workspace and provides only the options of **Page Setup** and **Default Layout**. The Page setup option duplicates the **File>Page Setup** menu item or the button.

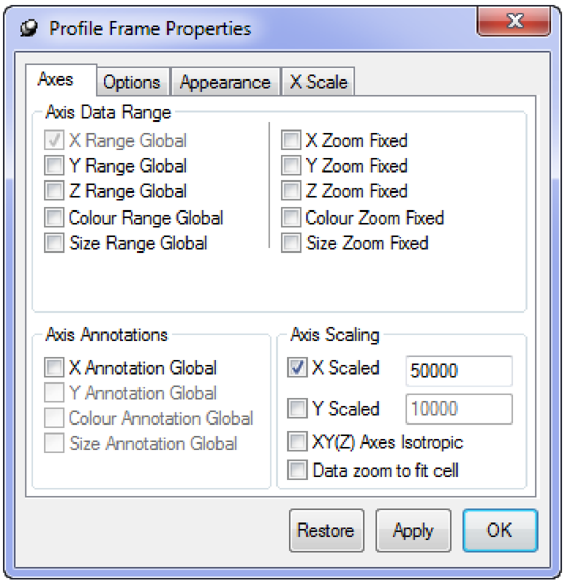
The Default Layout returns a display to fill the available default page size defined by the printer setup.

Profile Frame Properties

The Profile Frame allows multiple profiles to be contained in their own frame cells or tracks within the display. The Profile Frame Properties dialog has four tabs:

- *Axes Tab*
- *Options Tab*
- *Appearance Tab*
- *X Scale Tab*

Axes Tab



Axes tab for Profile Frame Properties

The Axes tab of a Profile Frame is identical to the *Axes Tab* of a profile graph object but is designed to override the axis settings in all profiles in the frame because of its level in the Workspace hierarchy. This provides you with an easy way to control the axis settings in all profiles in a frame. The Axes tab is used to control the range and scaling of the X, Y, Z, Colour and Size axes. It also enables you to control axis annotations for all profiles in a frame and to specify advanced graph properties like anisotropy and data range fitting. Note that for some types of frames and graphs some of these options may be read only, that is, not user modifiable. There are five major control groups in this tab.

The first, under **Axis Data Range** allows you to specify that the axis data ranges are global. When enabled for an axis (either X, Y, Z, Colour or Size) this implies that the limits of that axis will be set to the minimum and maximum of all data in all graphs in the frame. Note that when these global options are set the axis property pages refer to all axes of the same type. Also the location of the property page shifts from the graph properties to the frame properties.

Also under **Axis Data Range**, are the controls for rescaling of axes. If the zoom fixed option is specified for an axis, then the minimum and maximum limits of the axis will not be modified when you data zoom in a graph. The normal behaviour when this option is disabled is for the axis range to be reset to the range of the data within the zoom window whenever you data zoom or pan.

The options under **Axis Annotations**, allows you to control all axis annotation styles for all graphs in a frame. If an axis range is global, then it is possible to specify that the axis annotations are also global for that axis. When annotations are global, the axis in the workspace tree will appear underneath the frame rather than underneath the graph. Now, you can control the appearance of all axes in all graphs by editing the properties of one global axis.

The options under **Axis Scaling**, enables you to set the X and Y scale for all graphs in the frame. You can scale one or both axes independently. Note that when a document is being viewed in Normal mode scaling will be switched off – you need to switch to Page Layout mode to have your scale setting applied.

Axis Scaling also enables you to set advanced anisotropy and data fit properties in all graphs in the frame. When the XY(Z) Axes Isotropic option is switched on, the ratio of the axis ranges in X,Y (and Z) will be equal to one. For example, a 2D map is isotropic by default so that objects in maps do not appear stretched or distorted. By contrast, a profile is almost always anisotropic as the range of the Y axis is usually not of the same order as the X axis. A 3D graph is isotropic in X, Y and Z by default. Note that when the isotropic option is set then it is only necessary to specify the X scale as the Y scale will match it by default.

The scaling and isotopic options can affect the physical size of a graph on the page. Usually, a graph is as big as the frame in which it is embedded. If it is scaled or isotropic then its size is determined by the range of the axes and the scaling - the frame no longer controls the physical size of the graph. The Data zoom to fit cell option, when switched on, allows the software to modify the axis ranges so that the graph exactly fits the frame. For example, a 2D map with an image in it may be scaled at 1:50,000. With Data zoom to fit cell switched off the image may be larger than the frame in which the map is embedded and it may overlap other objects on the page. When Data zoom to fit cell is switched on the scale of the image does not change but the XY limits of the map is modified so that the map displays the image centred about the requested coordinates but clipped so that it does not overlap the frame.

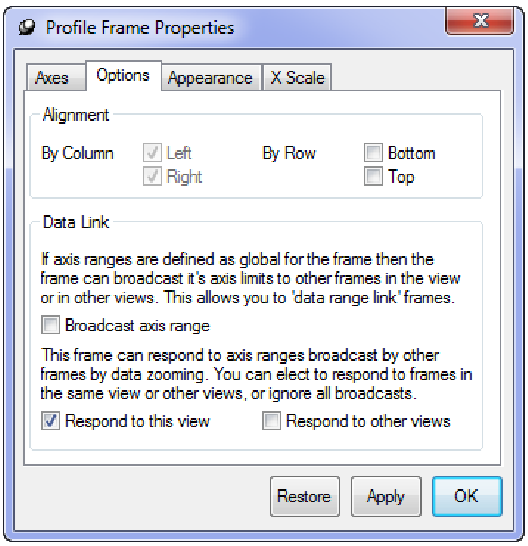
Group	Option	Description
Axis Data Range	**X Range Global**	Range of all X axes is fixed to the minimum and maximum of all X axes in all graphs in the frame.

Group	Option	Description (Continued)
	Y Range Global	Range of all Y axes is fixed to the minimum and maximum of all Y axes in all graphs in the frame.
	Z Zoom Global	Range of all Z axes is fixed to the minimum and maximum of all Z axes in all graphs in the frame.
	Colour Range Global	Range of all Colour axes is fixed to the minimum and maximum of all Colour axes in all graphs in the frame.
	Size Range Global	Range of all Size axes is fixed to the minimum and maximum of all Size axes in all graphs in the frame.
Axis Data Range	**X Zoom Fixed**	Fixes the X axis limits so that data zooming does not rescale the axis.
	Y Zoom Fixed	Fixes the Y axis limits so that data zooming does not rescale the axis.
	Z Zoom Fixed	Fixes the Z axis limits so that data zooming does not rescale the axis.
	Colour Zoom Fixed	Fixes the Colour axis limits so that data zooming does not rescale the axis.
Axis Annotations	**X Annotation Global**	When the X axis range is global, allows properties of all X axis annotations in the frame to be set in one place.
	Y Annotation Global	When the Y axis range is global, allows properties of all Y axis annotations in the frame to be set in one place.
	Colour Annotation Global	When the Colour axis range is global, allows properties of all Colour axis annotations in the frame to be set in one place.
Axis Scaling	**X Scaled**	Define a specific scale in the horizontal direction.
	Y Scaled	Define a specific scale in the vertical direction.
	XY(Z) Axes Isotropic	Force the ratio of width to height in all graphs to be equal.
	Data Zoom to fit grid cell	Force graph data to expand or contract to exactly occupy a frame cell and prevent frame overlaps.

Options Tab

The Options tab is used to control alignment of frame columns and rows as well as the data links that may exist with other windows. Data links control the method of updating related data in another display. For example, if a profile document has another window containing a map, if the data links exist, positioning the cursor and clicking in the profile document has a pointer presented in the map window to indicate the location of the same point.

Data range linking allows you to have graphs in different frames in the same or other documents in the application to respond to data zoom and pan events in an event source graph/grid. The Options tab is shown below:

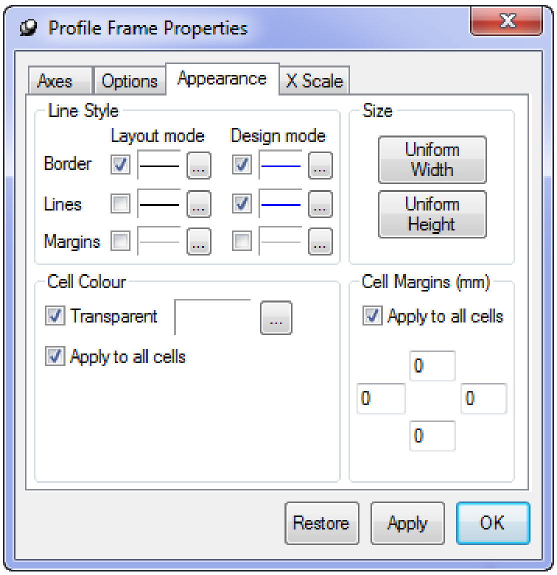


Options tab and properties for data links

Group	Option	Description
Data Link	**Broadcaster Axis Links**	The Broadcaster is used to transmit zoom and update messages to other window displays (documents). Either horizontal and/or vertical updates can be transmitted.
	Respond to This View	Response broadcast is to the Local, current document only.
	Respond to Other Views	Broadcasts updates and communicates to all documents in the application.

Appearance Tab

Control properties of frames are provided by the Appearance tab of the Profile Frame properties. The Appearance tab displays property controls that are used to view and manipulate the cells, columns and rows of a frame. A detailed description of the modes used for cell appearance and their meanings is provided in Cell Normal and Design Modes.



Profile Frame Appearance properties tab dialog

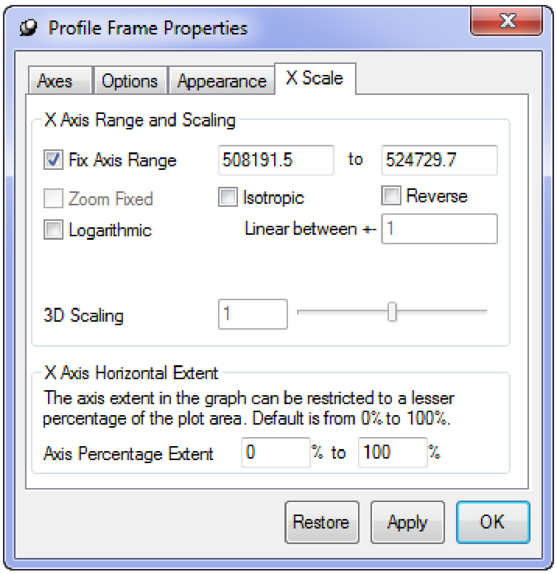
The Appearance tab is used repeatedly wherever a frame is used. Controls available include:

Group	Option	Description
Line Style	**Normal/Design mode**	These modes define the presentation style of the border, lines and margins of cells of a frame. See *View Menu*.
Cell Colour	**Transparent**	The background colour of a graph can be specified (using the Browse button. If Transparent is off, the graph uses the colour setting specified.
Cell Margins	**Top, Left, Right and Bottom**	Used for offsetting the cell margin from the outer border by the amount specified. Units are millimetres.

Group	Option	Description (Continued)
Resizing Mode	**Columns/Rows**	Size the columns and rows according to a Fixed amount, a data controlled amount (By Content) or freely for dynamic sizing.
	Uniform Cell Width/ Height	If cells of a grid are to be kept equi-spaced either vertically or horizontally, select the Uniform Cell dimension.

X Scale Tab

The X Scale tab properties for a profile frame are identical to the Y Scale properties if a Y Scale tab is available.



X Scale dialog tab properties

The axis properties control the scaling of the axis both in on-screen (Normal) and Layout modes. In particular the setting for scaling the axis is important for printing output at a specific scale. The options provided include:

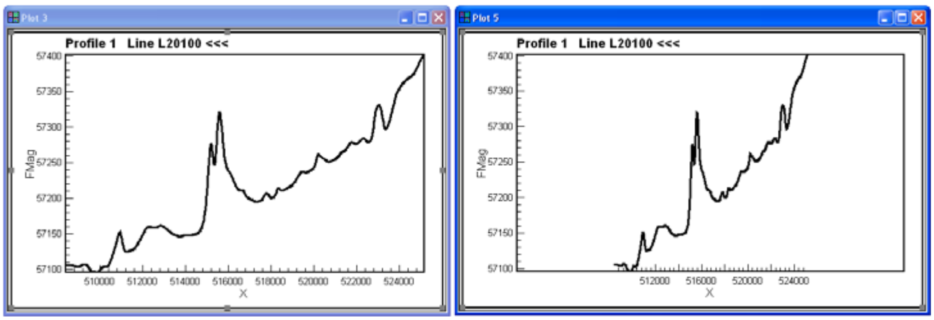
- **Fix Axis Range** – Define a specific range over which the data is displayed. This range can be used in combination with the scale to produce an output of a specified size using a predefined range.

- **Logarithmic** – Scale the axis data logarithmically. Since it is not possible to have the Log of zero, a linear portion of the logarithmic scale is used in the transition of positive to negative values. The linear section can be specified as a power raised by base 10. Therefore, a power entry of 0 has a linear section from +/- 1.0. The drawn height of the linear section is set equal to one decade of logarithmic height.

- **Scale the Axis** – If enabled, displays the Layout display at the specified scale. This option has no effect if in Normal mode. If an inappropriate scale is used (that is, too small for the Page Setup), the graph display is extended off the page and indicates the printed join margin of multiple sheets of paper that are used for the required plot.

- **3D Exaggeration** – If the profile is displayed 3 dimensionally, the vertical scale can be specified by the data entry or the slider bar.

- **Horizontal Extent** – The minimum and maximum values represent percentages of the space to be used for the plot within the internal boundaries of the plot area (see below).



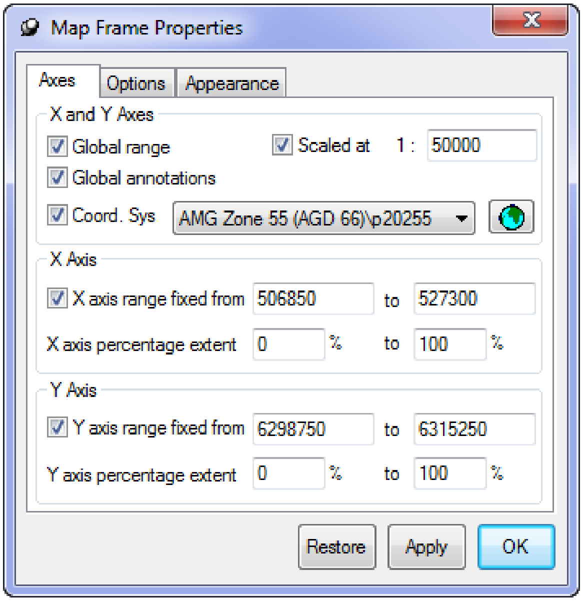
2D Map Frame Properties

The Map Frame in a 2D Map display allows multiple maps to be contained in their own frame cells within the display. The **Map Frame Properties** dialog for a 2D Map display has three tabs:

- *Axes Tab*
- *Options Tab* (see Profile Frame Properties)
- *Appearance Tab* (see Profile Frame Properties)

Axes Tab

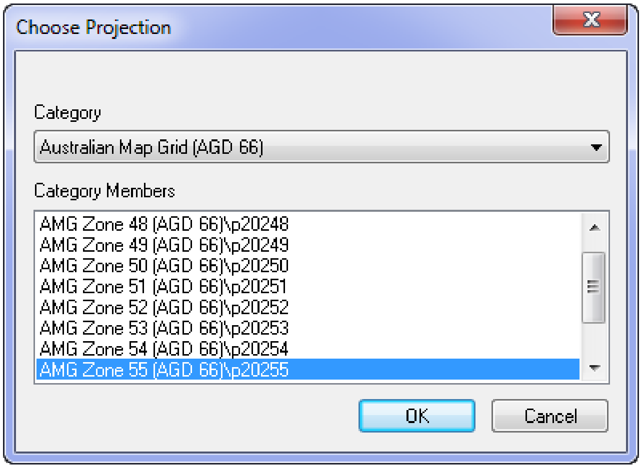
The **Axes** tab of a **Map Frame** is designed to override the axis settings in all maps in the frame because of its level in the Workspace hierarchy. This provides you with an easy way to control the axis settings in all maps in a frame. The **Axes** tab is used to control the range and scaling of the X and Y axes. It also enables you to control axis annotations for all maps in a frame. Note that for some types of frames and graphs some of these options may be read only, that is, not user modifiable. There are three major control groups in this tab.



Axes tab for 2D Map Frame Properties.

The first, under **X and Y Axes** allows you to specify whether or not the axis data ranges are global, the axis annotations are global, the coordinate projection for the data contained within the Map Frame and the X and Y map scale.

To define the coordinate projection of the map frame, enable the **Coord Sys** option and then either choose a projection previously used in Encom Discover PA from the drop-down list available or else press the **Set Projection** button to select the coordinate projection system from the list of available projections as shown below.



Select a coordinate projection system from the available list.

Under the **X Axis** section you can define a specific X (Easting) range over which the data is displayed. This range can be used in combination with the scale (set at the top of the property dialog) to produce an output of a specified size using a predefined range.

X Axis Percentage Extent – The minimum and maximum values represent percentages of the space to be used for the horizontal plot within the internal boundaries of the plot area.

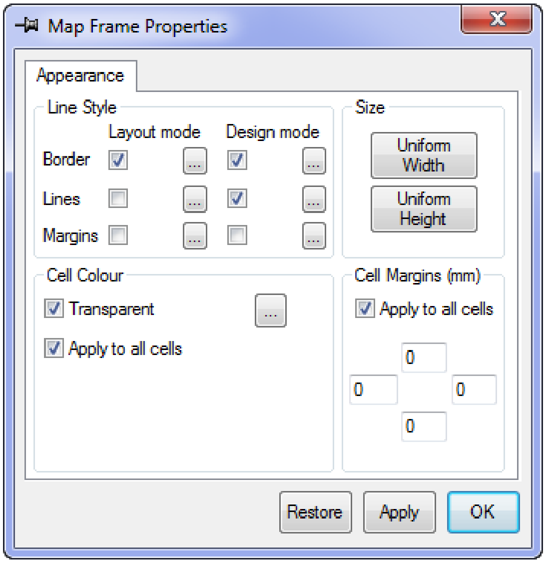
Under the **Y Axis** section you can define a specific Y (Northing) range over which the data in all maps within the Map Frame is displayed. This range can be used in combination with the scale (set at the top of the property dialog) to produce an output of a specified size using a predefined range.\

Y Axis Percentage Extent – The minimum and maximum values represent percentages of the space to be used for the vertical plot within the internal boundaries of the plot area.

3D Map Frame Properties

The Map Frame in a 3D display allows multiple 3D Map displays to be contained in their own frame cells within the document window. The Map Frame Properties for a 3D display has only one tab dialog:

- *Appearance Tab* (see Profile Frame Properties)



Appearance tab for 3D Map Frame Properties.

See also:

... Creating 3D Displays
... 3D Map Properties
... Profile Frame Properties

Profile Properties

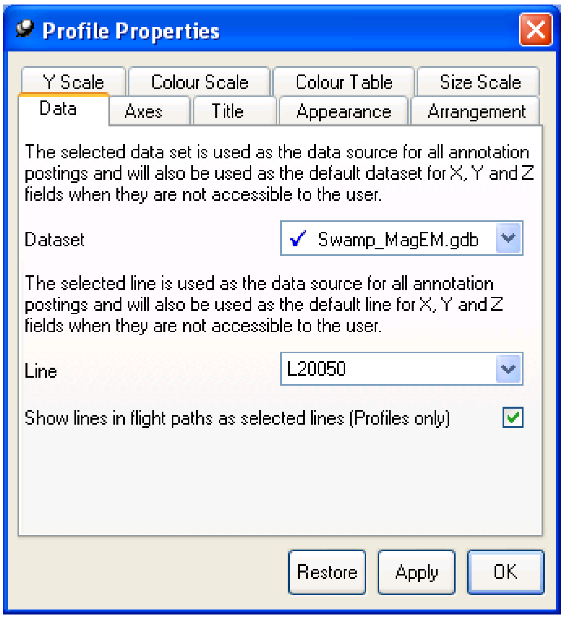
The **Profile** branch of the Workspace toolbar represents the graph within a frame of a document. The tree item shows the line being used (in this case line 3721). It also indicates the number of the graph in the frame.

Data, axes, title, graph appearance, axis arrangement and selection properties are available within a profile. In addition to these six property pages, the Y axis, Colour axis and Size axis properties are often available from this branch.

- *Data Tab*
- *Axes Tab*
- *Title Tab*
- *Appearance Tab*
- *Arrangement Tab*
- *Y Scale Tab*

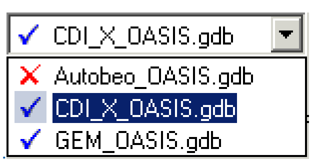
- *Colour Scale Tab*
- *Colour Table Tab*
- *Size Scale Tab*

Data Tab



Data source tab dialog for profiles

The Data tab provides a method of setting the line and dataset source for all components of the profile. See *Accessing Data with Encom Discover PA* for information on accessing different data sources. The Dataset specification refers to the source database. Encom Discover PA can access any number of databases and is able to display data fields from multiple datasets within a profile or profile frame. Selection of the dataset is via a pull-down list. Datasets may not always share common lines. If the selected line for the profile is not found in a dataset, that dataset is marked with a red cross. Otherwise it is marked with a blue tick. If the profile contains data from datasets that do not share the specified line, the profile becomes invalid and no data content is displayed.



The Line setting can also be specified from a pull-down list. It displays all lines in the selected dataset. The nominated line is used as the source for all data, scaling, annotation etc.

Axes Tab

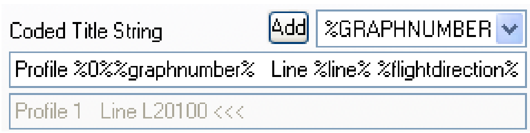
The Axes tab used by the Profile level of the Workspace hierarchy is almost identical to the Profile Frame Axes Tab. Generally, if an option is set in the profile frame axes tab, then the equivalent option is set and read-only in the profile axes tab. In other words, if an option is set for the frame, then it is set for all graphs in the frame. If an option is set for a graph, it is set for all applicable axes in the graph. If it has not been set in this way, the option is available to the user in each individual axis property page.

From this property page you can set the following axis options:

- **Axis range can be global** – All axes in the profile are forced to have the same range. Usually, this range is the global range of all data plotted against the axes.
- **Axis zoom can be global** – When switched on, the data zoom range is calculated from the whole line regardless of the user specified data zoom. So, when the user data zooms to a portion of the line, the axis range remains fixed to the range of the whole line.
- **Axes can be global to the graph** – If set, the annotations for the specified axes become children (at a lowevel of the hierarchy) of the profile in the Workspace tree. It is a time saver option that enables you to set axis properties only once when you have multiple axes to annotate. It is only available if the axis range is set to global for the specified axis type.
- **Scaling** – X and Y axis scaling can be individually specified.
- **Isotropic X and Y axes** – Ensures that one unit in X equals one unit in Y on the page.
- **Data zoom to fit cell** – When enabled, the profile range is modified to ensure that the graph fills the cell in which it is embedded. If not enabled, the profile shrinks and stretches to fit the data zoom and scaling.

Title Tab

The Title tab allows control over the positioning, style and content of a single graph title. You can show or hide the title (**Show Graph Title**), select a text style and colour (**Font** button) and set the title content. The title can be located where desired. The title text can be directly typed or you can request Encom Discover PA to enter text strings as *Variable Substitutes*. A wide range of variable strings can be used and these are substituted for logical text (if available) at the time of drawing. A preview of the final title text is displayed beneath the coded title string as shown:



Variable Substitutes

Variable Substitution assists you to label graphs by substituting text items that may be variable in nature. For example, the line name for a profile may need to be altered for each update of the profile.

Substitution strings are automatically created for you if you use the Add button to append a variable. The substitute name is enclosed in % characters. The case is not important. Many other substitution variables are also available.

Variable	Description
NAME	Data field name. NAME is the field name as reported in the name channel in the spreadsheet.
PROPNAME	Data field name derived from properties specifiers. It is the field name recorded in the properties file.
ALIASNAME	Some data channels can have aliases and these can be used to name the data presented.
DBASENAME	The database file name. the true field name stored in the database.
FULLNAME	The full field name contained in the source database. It is a constructed name combining the database name and the field.
UNITS	Units of the data channel if available. UNITS is a string derived from information in the properties file.

Variable	Description (Continued)
LINE	The line name of the source profile.
BANDNAME	Data channel used as the source for the profile.
DATASET	The name of the database source.
GRAPHNUMBER	The incremental number used for each profile graph.
ANNXNAME	X axis assigned name.
ANNYNAME	Y axis assigned name.
ANNCNAME	Chordlength axis assigned name.

Title Positioning

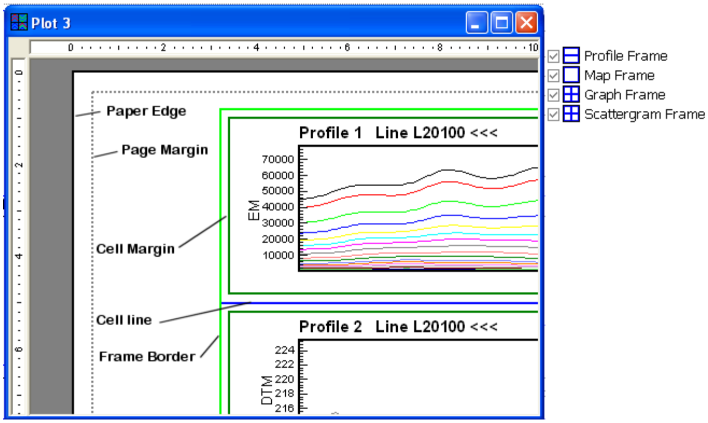
The title can be positioned anywhere in a profile area. It can be automatically positioned using the Offset switches, justification flags and text anchor. When the Offset switches are enabled, the title is positioned relative to the data plotting area. Otherwise, it is relative to the edges of the profile. For final adjustment, a manual page offset setting is available which moves the title X and Y millimetres from the specified plotting position.

Graph title position

Appearance Tab

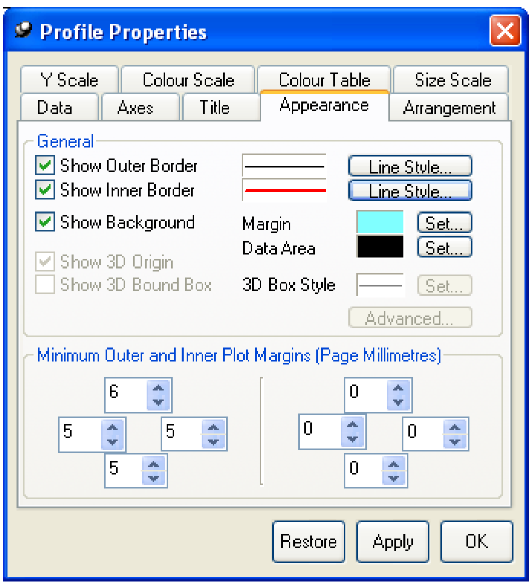
Each profile graph exists within a profile container (or frame). It occupies one cell of this frame (see *Working with Graphical Objects*). A frame cell has an inner margin specified in millimetres, that can be set in the Appearance property dialog of the Profile Frame. By default, this inner margin is set to zero.

Surrounding the profile frame is its own outer and inner borders. The outer border is used for registration with the enclosing cell and the inner border forms the enclosing extent of the data limits (see the diagram below). If the frame cell inner margins are zero then the plot outer border plots coincident with the frame border. For this reason, the plot outer border is not shown by default.



Cell boundaries relative to profile inner and outer boundaries

The appearance of the frame borders, background and margins within its frame cell are controlled by the Appearance property page.



The Profile Appearance tab dialog

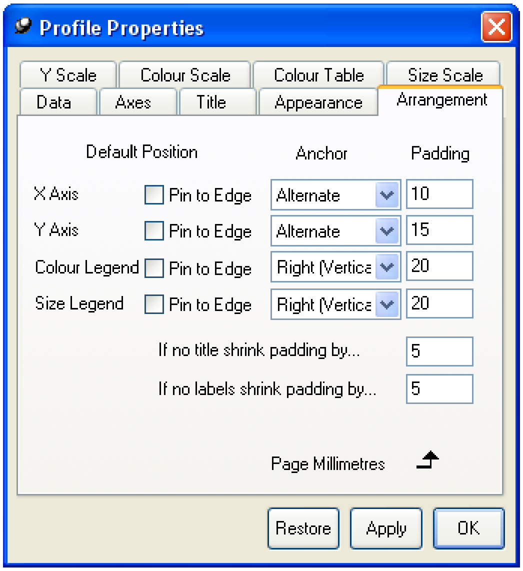
You can show or hide the inner and outer plot borders of the frame and alter the line styles of these borders. If **Show Background** is enabled, the profile has background colour. By definition this means the profile is no longer transparent and over-plots underlying objects. You can specify the colour of the inner plot area and outer plot margins separately.

The size of the outer plot margin (specified in millimetres) is generally determined by what axis annotations are displayed in the margin. However, you can specify the minimum size of the outer margins from this property page. The inner plot margin is a redundant feature that allows you to define a margin within the plot data area. These offsets can be used in combination with the Grid Cell margins that are definable in the Appearance tab.

For 3 dimensional displays (other than profiles) you can also have a 3D box surrounding the area of the presentation.

Arrangement Tab

The profile Arrangement property page defines what rules are used to position axis annotations when those annotations are automatically positioned by Encom Discover PA.



Profile axis Arrangement tab dialog

A description of the various axes used in Encom Discover PA is provided in *Axes Tab*.

Each axis type (X, Y Colour and Size) is assigned a Padding value, specified in millimetres. This is the amount of page space that is automatically be assigned for this axis. Encom Discover PA will adjust the profile outer margins appropriately to make room for this axis.

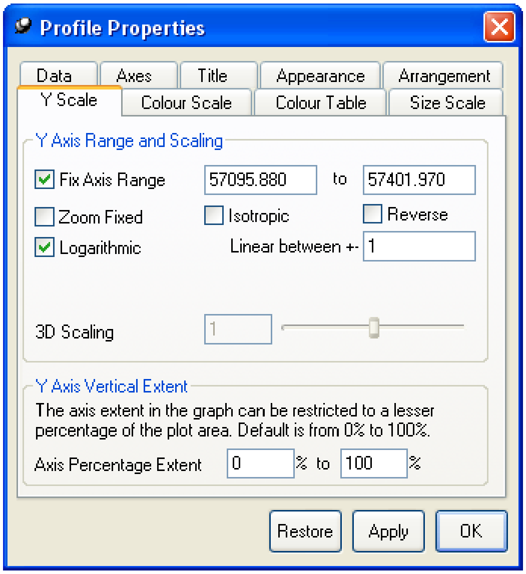
Each axis can be positioned according to the rule specified in the Anchor setting. The possible settings are top, bottom, left, right and alternate. If top or bottom options are specified for an X axis then all X annotations are placed in the specified position relative to the plot area. If there are multiple axes then they are stacked from the inside out. Similar rules apply for the Y, Colour and Size axis annotations except that these annotations are all vertical and can therefore be placed at the left or right. If alternate is selected then annotation will be placed on the top and bottom, or left and right, alternately.

If the **Pin to Edge** setting is enabled for an axis type, then annotations will be pinned to the outer plot border rather than the inner plot data border.

Padding between labels and axes can also be controlled from the **Shrink padding** options.

Y Scale Tab

The Y Scale Properties tab is similar to the X Scale tab that is generally accessible from the Profile Frame object. Typically, this tab is used to specify a fixed data range for the Y axis or to set the axis to linear or logarithmic mode.



Y Scale profile properties tab

Other items available are equivalent to options available on the Scales tab of the profile or profile frame. **Global Range from Unclipped Data** is equivalent to **Y Zoom Global** and if enabled ensures the data range of the axis spans the entire data range of the traverse line displayed in the profile, regardless of any data zoom.

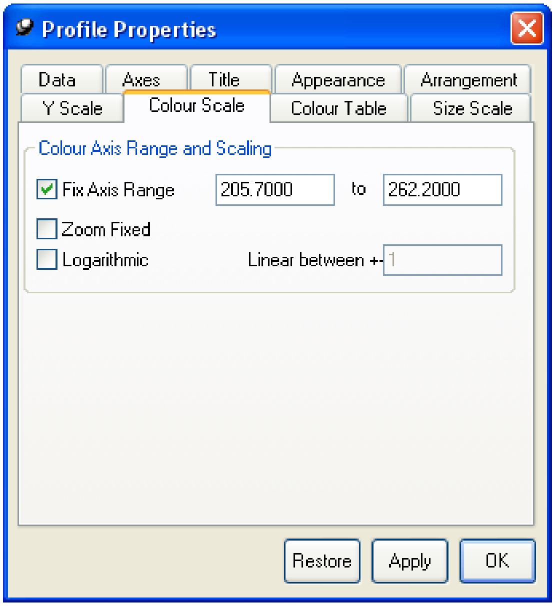
The axis can also be scaled, however usually it is set at a more global level either in the profile or the profile frame. Note that only one axis is used for scaling and it is the last axis in the profile for which scaling is enabled. Logarithmic scaling is also available. Since it is not possible to have the Log of zero, a linear portion of the logarithmic scale is used in the transition of positive to negative values. The linear section can be specified as a power raised by base 10. Therefore, a power entry of 0 has a linear section from +/- 1.0. The drawn height of the linear section is set equal to one decade of logarithmic height.

The **Y Axis Vertical Extent** is specified as a percentage of the plot data area. Usually, it covers 0 – 100 % of the data range and extends over the entire plot area. However, you can set the upper and lower extents to restrict the size of the Y axis. All annotations of this axis also have a restricted extent. Use this option to spatially separate multiple Y axes in a profile. Note that the same option is available for X axes but for Colour and Size axes the option is set for Colour and Size Scales individually.

The axis can be **Reversed** such that the minimum and maximum ends of the drawn axes are swapped. The axis can also be made **Isotropic** in conjunction with its parent X axis. This option is not used often with profile displays since an isotropic setting forces equal horizontal and vertical scaling. This is a useful feature where the vertical:horizontal exaggeration needs be unity for cases such as drillhole targeting with distance versus depth graphs. Again, only one axis can be isotropic in a profile and the axis used is the last axis found to have the option enabled.

Colour Scale Tab

The Colour Scale tab is used to control the colour data range within a profile. Typically, this tab is used to specify a fixed data range for the Colour axis or to set the axis to linear or logarithmic mode.



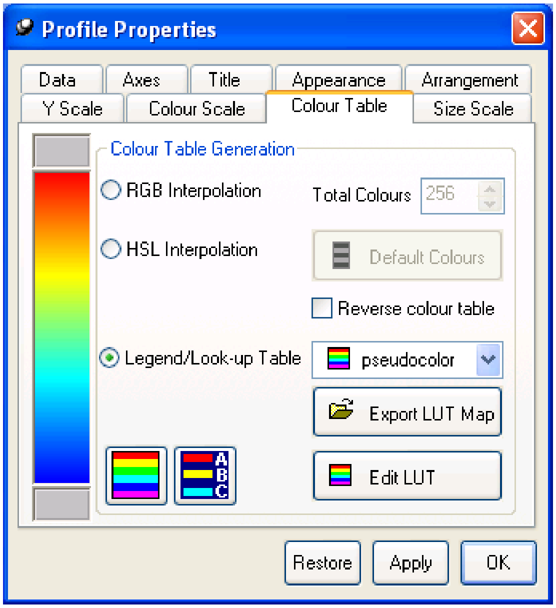
Colour Axis scaling tab

Options available include:

- **Fix Axis Range** – Define minimum and maximum colour limits.
- **Zoom Fixed** – Allows data zooming of the profile but does not alter the colour range.
- **Logarithmic** – Display the assigned colour range logarithmically with a linear portion around zero.

Colour Table Tab

The Colour Table tab is used to modulate the actual colours displayed in a profile.

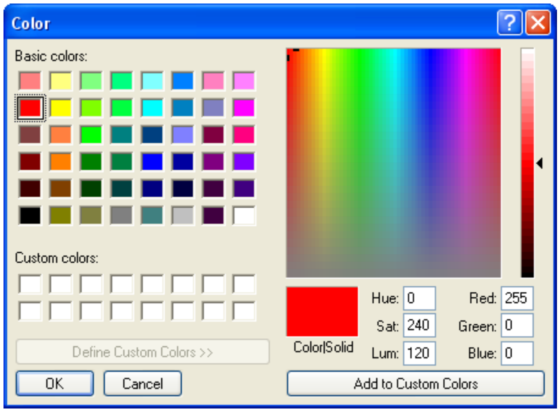


Colour Scale control dialog tab

The colour scale uses three methods of colour definition across a definable number of colours (defined in the **Total Colours** item).

- **RGB Interpolation** – Uses the hexadecimal Red:Green:Blue specification of colour.
- **HSL Interpolation** – The Hue:Saturation:Luminosity colour model is used.

In both colour mapping schemes a predefined colour palette is available. This palette can be reset by clicking the **Set Default Colours** button. You can control the upper and lower colours of the colour scale by selecting the **Colour Browse** buttons at the top or bottom of the displayed colour bar. This option displays a standard Windows colour selection dialog.



Windows colour selection palette

- **Legend/Look-up Tables** – The third method of assigning colours is to use predefined look-up tables OR, as is used in certain circumstances, patterns and colours. Such circumstances include assigning legend patterns and colours to drillholes, drawn polygons (features) or legends.

  To maintain colour look-up and legend tables, two editors are provided with Encom Discover PA. Descriptions of these are in the *Colour Table Editor* and *Legend Editor*. Access to the Colour Editor is from the Edit LUT button.

  - **Look-Up Tables** – Supported LUT formats include ER Mapper, Geosoft montaj, MapInfo and Surfer. The file formats are listed in *Appendix E: Supported File Formats* and described in *Appendix B: Data File Specifications*.

  - **Legend Tables** – These tables use a combination of line styles, colors and patterns. A range of legend files are provided when Encom Discover PA is installed, but you can modify, create or manage your own. Modification of legend files is done using the Legend Editor (see *Legend Editor*).

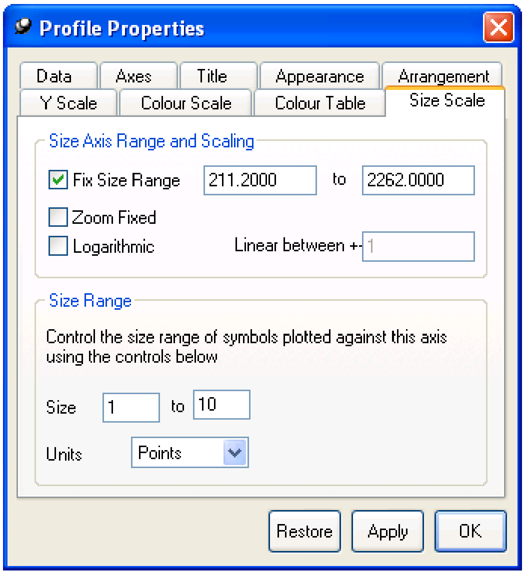
  Colours can be limited in the Colour Mapping entry to any number supported by the selected Look-Up Table. Additionally if you wish to **Reverse** the Colour Table, this has the effect of re-assigning colours and their data values in reverse order. This option is useful if data values are to be displayed in a reverse sense, for example if resistivity is to be displayed as conductivity.

  It is useful to also import and export LUT tables. This can be done with the Colour Editor or the Export LUT Map options.

Note Look Up Tables and Legend Tables are installed into the folder `\PROGRAM FILES\ENCOM\ENCOM\COMMON\LUT`. If patterns are used with the Legend Tables, these are also stored in a common folder called `\PROGRAM FILES\ENCOM\ENCOM\COMMON\PATTERNS`.

Size Scale Tab

The use of the Size Scale is described in *Axis Properties*. The Size Scale is used to control the size of modulated data symbols. The properties associated with the Size Scale are similar to those of the X Scale Tab and Y Scale Tab.



Size Scale tab dialog

You can define the Size Range of all symbols plotted against this size scale and also the units of the symbol size. Available units include Millimetres, Page Points, Screen Points and Percentage (see *Size Units*).

Axis Properties

Each axis in a profile can be graphically visualised by adding one or more appropriate axis objects. They appear in the Workspace tree as lower branches of the associated axes.

Axes can be global to the profile or to the profile frame to simplify specification of axis properties across multiple axes or multiple graphs.

The properties associated with each axis type are essentially identical. There are five property pages:

- *Data Tab*
- *Appearance Tab*
- *Title Tab*
- *Divisions Tab*
- *Ticks Tab*

The Axes tabs control the style, position and title of the axis as well as tick divisions, labels and tick style.

There are four basic types of axes – X, Y axes, Colour axis and Size axis.

A fifth type of axis is an X Y internal axis. These axes are used to draw internal ticks or lines in a graph and can be used to display geographic and local grid axes. It can be added to a graph from a Y axis (or visible Y axis parent if the Workspace tree is concise). The Internal Annotation option is described in *Internal Axes and Annotation*.

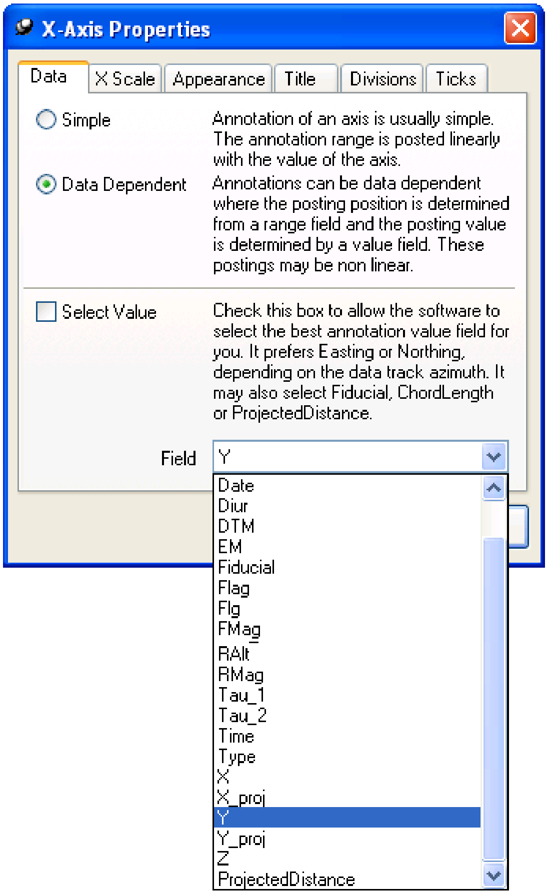
Data Tab

The Axis Data tab (accessed from either the X-Axis or Y-Axis levels of the Workspace tree) specifies the source of the data used for axis annotation. The annotated data field can be automatically chosen by the software, or can be explicitly specified. This balance of user and software control is specified by the options on the style property page.

- The **Simple** setting uses the X channel for both the data positioning and label annotation. For some axes and graphs the data used may be system defined. For example, in a profile the X data is always Projected Distance. In a map the database X channel (Easting) is always used.

- The **Data Dependent** setting allows the user or the software to assign a channel for axis annotations (such as Fiducial, Easting or Northing). If this is selected and the modification applied, up to two new objects will appear in the workspace tree under the Axis object – Range and Value data objects. Ticks will be drawn according to Range posted according to Value.

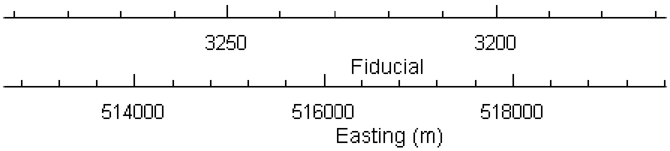
- The **Select Value** setting allows the system to choose appropriate Range and Value channels from the database. For example, it may choose EASTING as the Value channel in a profile displaying a line that is approximately east-west.



X-Axis Data dialog tab

If a specific Data Dependent channel is to be used for axis labelling and plotting (such as Fiducial), deselect the Select Value option (so Encom Discover PA does not choose it for you) and then choose the Field you wish to use for axis annotation. Note that this option is only visible when the Data Dependent item is selected. Also note that the specified field is shown in the properties of the X-Axis level of the Workspace tree.

Using this control it is also possible to have multiple horizontal axes for the one profile. For example, along a profile track may be a location axis (eg Easting) and a second axis that indicates Fiducial or timing.

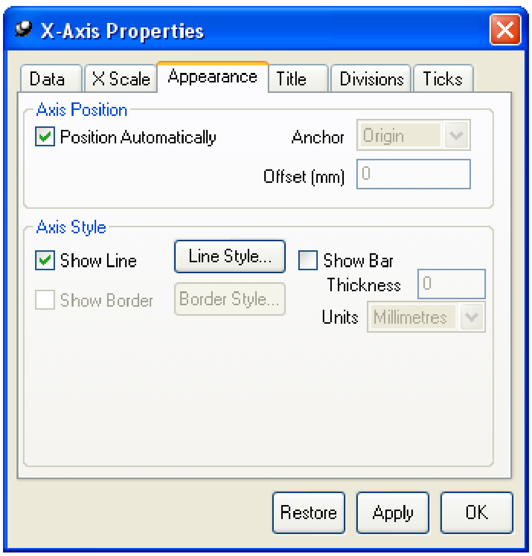


Combined axes for one profile showing Easting and Fiducial annotations.

Appearance Tab

The Axis Position controls the location and style of the axis. These controls are identical for all axis types.

- **Axis Position** – This can be automatically defined by the program, or if you wish, an anchor position with an associated offset can be used. The anchor is relative to the profile boundary and can have a percentage offset (see *Size Units*).



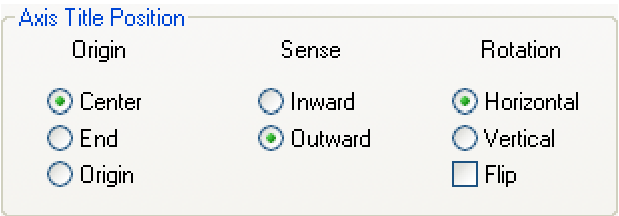
The Axis Position tab dialog

- **Axis Style** – This controls the appearance of the axis bar. All axes are comprised of a bar, ticks, and tick labelling. The bar may be a line or have finite thickness. Colour, style, width and thickness units can all be controlled. You can also control the vertical extents of Colour and Size axes from here.

Title Tab

The title of an axis is controlled by the Title tab. By default, Encom Discover PA uses the data field name of the displayed data using a Variable Substitute. A description of the use of titles is provided in *Title Tab*.

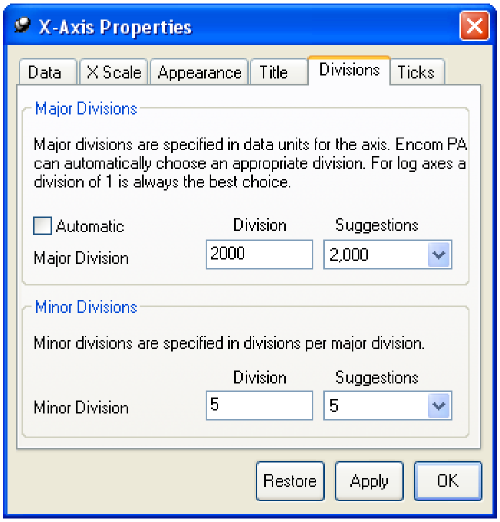
The position of the title text relative to the axis is available from the options in the **Axis Title Position** entries.



Axis Title Positions of the Title Tab

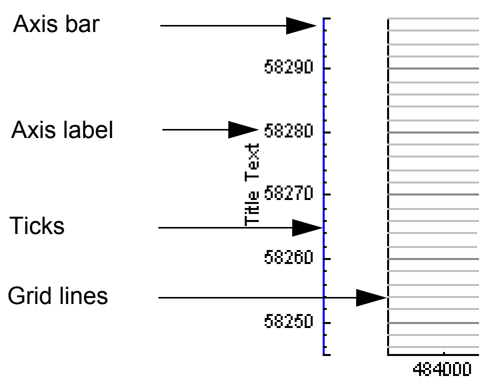
Divisions Tab

An axis has ticks and tick labels. The tick marks are subdivided into major and minor types. The Division tab specifies the interval used for both Major and Minor intervals. These intervals control both the labelling interval (if enabled) and the tick interval (if enabled using the *Ticks Tab*).



The Axis Division tab

The property page specifies both Major and Minor divisions. Specific major division interval can be specified or use the optimised suggestions provided by Encom Discover PA. Minor divisions are specified as divisions per major division.

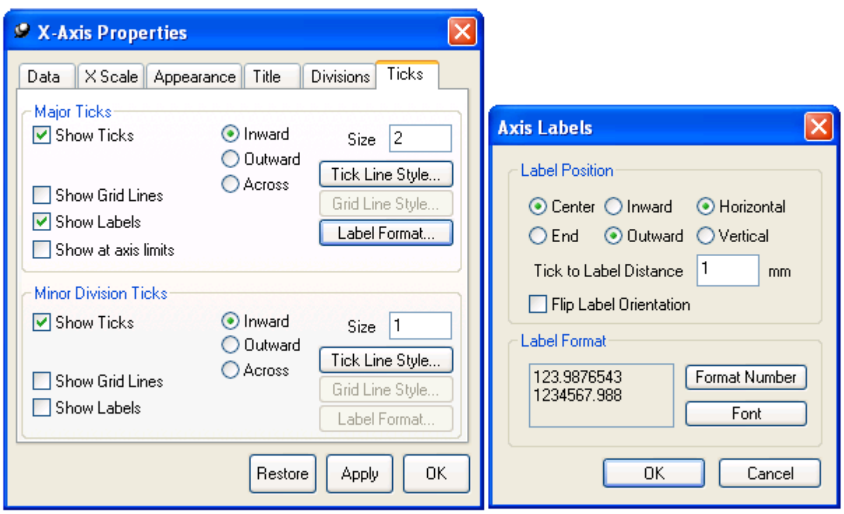


Ticks Tab

The Axis Ticks tab is used to control major and minor tick style as well as grids and labels. An example of each of these is shown.

Ticks can be drawn **Across**, **Inward** or **Outward** (from the data) and have a specific length. The line style can also be chosen.

Grids extend across the plot area and can have styles similar to ticks. Labelling for both major and minor divisions has a format dialog (as shown).



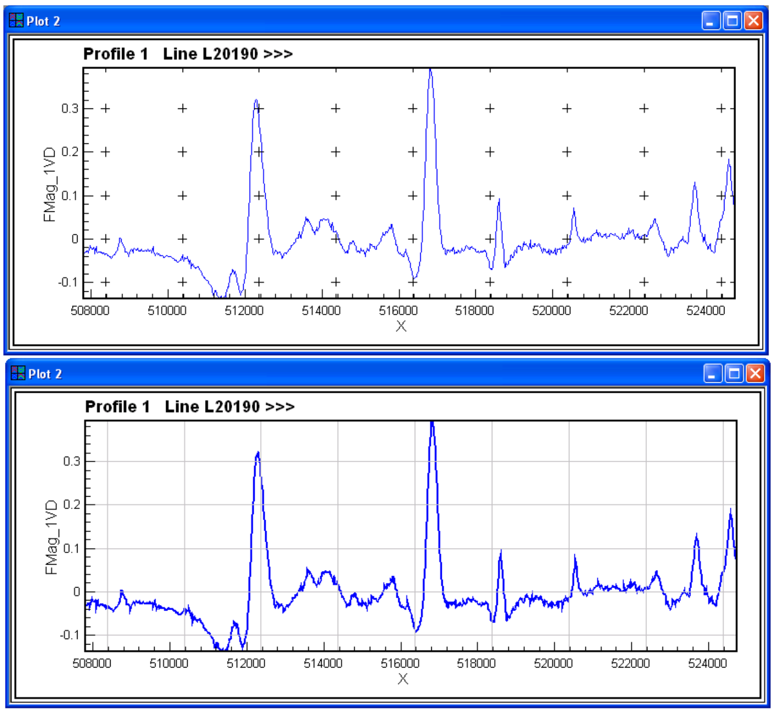
X-Axis Ticks tab dialog and label format dialog

The control enables labels to be written as required using formatting and position control. The label orientation and style (**Format** button – see *Label Tab*) can also be specified. The **Font** button allows specification of the text style.

The location of the labels relative to the axis allows combinations of centre or end, **Inward** or **Outward** from the plot area, and horizontal or vertical text strings. The option, **Show at Axis Limits**, forces annotation values to be displayed at the top and bottom of the axes.

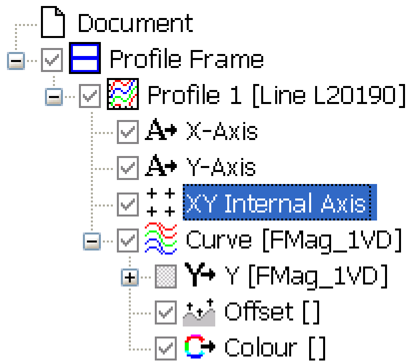
Internal Axes and Annotation

Any supported Encom Discover PA display can have internal axes. Usually this takes the form of tick marks.



Two example profile displays with internal tick mark annotations of ticks (top) and lines (bottom).

The tick marks for internal axes can be controlled and local or geodetic projections applied. The ticks can be also annotated. Internal ticks or lines can also be used in profiles, graphs or scattergrams. In these cases, geodetic projection is available but generally is not applicable.



For any of the display types, internal axes can be added by:

1. In the Workspace tree, highlight the Map, Profile, Graph or Scattergram item and while selected, press the right mouse button.

2. From the pop-up menu, select the **Add Axis>XY Internal Axis** option.

3. A new XY Internal Axis level is added to the Axis (see tree diagram)

From the highlighted XY Internal Axis branch, the properties dialog can be displayed. Four tabs are provided and these are:

- *Internal Annotation Tab* – Controls data conversion between graph coordinates and tick coordinates.

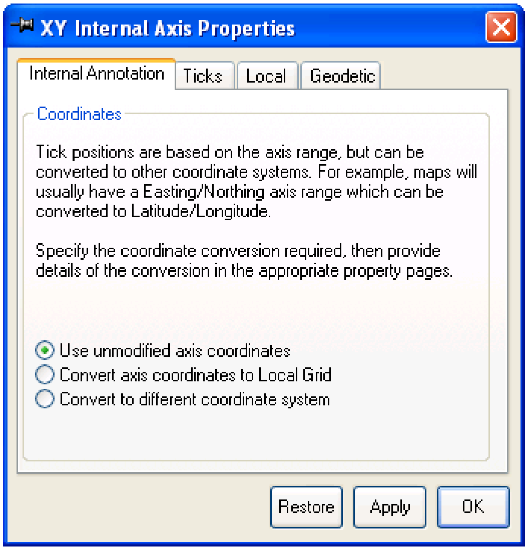
- *Ticks Tab* – Defines the tick and line format, divisions, style and annotations.

- *Local Tab* – Defines a transformation from graph coordinates to a local coordinate system or vice-versa.

- *Geodetic Tab* – Allows specification of a projection and a spheroid to define a transform from graph coordinates to geodetic coordinates or vice-versa.

Internal Annotation Tab

This property page specifies what type of data conversion between graph coordinates and tick coordinates is to be applied prior to plotting the ticks. For example, maps usually have an Easting/Northing axis range that can be presented and displayed as Latitude/Longitude once converted to a relevant projection and spheroid.



Internal Annotation tick control dialog

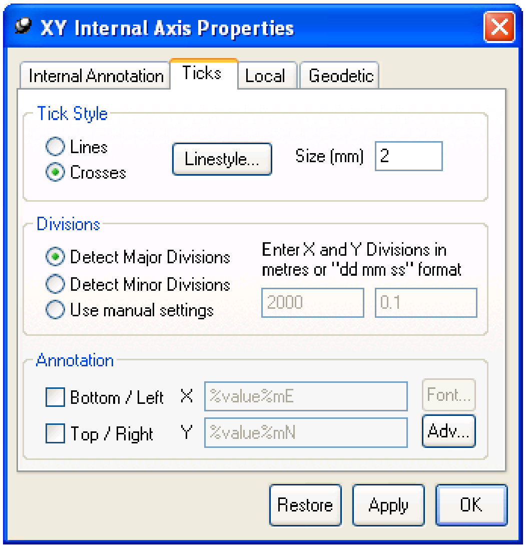
The option you select depends on the type of data in the X and Y fields of the graph (eg Eastings/Northings, Latitude/Longitude, Local Grid) and the coordinate system you want the ticks to be plotted in. The options include:

- **Use unmodified axis coordinates** – This option uses the X and Y data retrieved directly from the selected database. No location transformation is performed.
- **Convert axis coordinates to Local Grid** – This option uses the X and Y data retrieved directly from the selected database and then applies local grid transformation defined by the data in the *Local Tab*.
- **Convert axis Easting/Northing to Latitude/Longitude** – This option uses the X and Y data retrieved directly from the selected database and then applies a transformation defined by the data specified in the *Geodetic Tab*.

- **Convert axis Latitude/Longitude to Easting/Northing** – This option uses the X and Y data which must be in a geodetic coordinate system within the selected database and then applies a transformation to a grid coordinate system using the data specified in the *Geodetic Tab*.

Ticks Tab

Control the appearance of Internal Annotations from this property page.



Tick control tab for Internal Annotations

- **Tick Style** – Either ticks or lines can be displayed. Lines are often used in profile, graph or scattergram displays. The size of the ticks (in millimetres) and linestyle can also be specified.

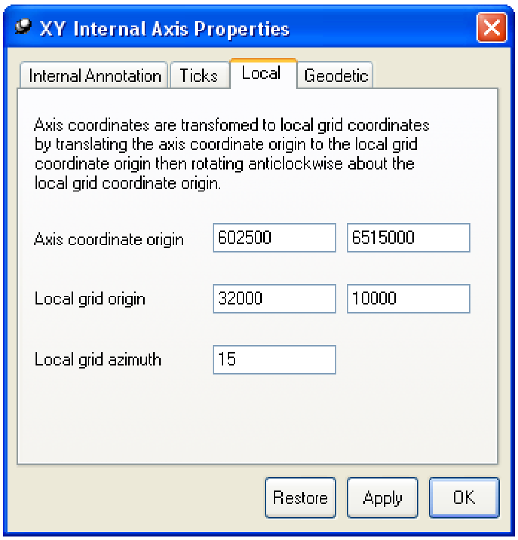
- **Divisions** – Major, minor or specific divisions with a definable increment can be chosen. Note that if a Latitude/Longitude output is expected the format must be in dd mm ss format (e.g 147o 58’ 21” entered as 147 58 21).

- **Annotation** – The annotation text can be located at the Bottom/Left, Top/ Right, both or neither. Text style can be modified by clicking the Font button.

Local Tab

Internal annotations can be plotted after a transformation from axis coordinates to a local grid. Axis coordinates are transformed to Local grid coordinates by translating the axis coordinate origin to the local grid origin and then rotating anticlockwise by the defined azimuth angle (in degrees).

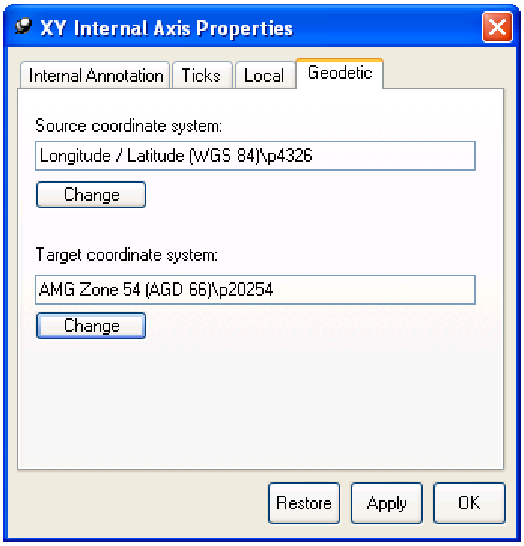
The property page for the Local grid transformation is shown below:



Local coordinate and offset of Internal Annotation ticks

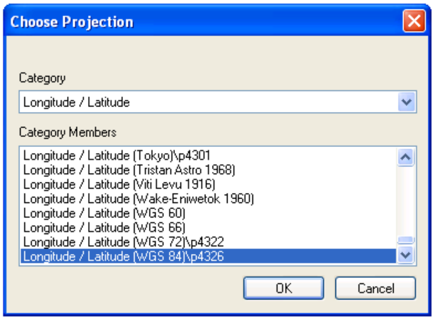
Geodetic Tab

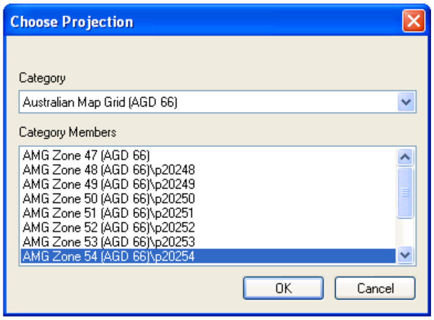
Encom Discover PA can display maps using most projections of the Earth. Geodetic projections are used to control the axes of map displays. The Geodetic options are available on the Geodetic property page. Here they can be set for each axis individually. In addition, there are default geodetic coordinate system settings which can be accessed via the **Tools>Options** menu. It displays the following property tab.



Options tab (in Tools>Options menu item) that is identical to the Geodetic tab of Internal Axes

The operation of the Geodetic tab is to convert the source data coordinates and display the output in an alternative, Target coordinate system. Use the **Change** button to define the Projection (Category) and Datum (Category Members) required from the available lists.





Set the Category member information for the required Projection

See Map Projections for additional information. Using the geodetic options with map internal axes is described in Internal Axes.

Map Projections

Encom Discover PA has built-in Map Projection information that supports a wide range of projections in use across the Earth. To fully specify a Map Projection, both the PROJECTION (called a Category) and DATUM (Category member) for a map must be specified. Grid Groups that have the same Map Projection (that is, the same Projection and Datum) can be automatically registered if they are derived from an ER Mapper algorithm.

Note too that a utility is available to transform data from one projection to another, which is described in *Database Coordinates Reprojection*.

Projections support is provided in Encom Discover PA from an ASCII file (called `ENCOM.PRJ` stored in the folder `\PROGRAM FILES\ENCOM\COMMON\PROJECTIONS`).

You can add your own coordinate system definitions to the end of this file. Categories must begin with the string "--- Name ---" where Name is the name of the category. Each member within the category is a single line with the following format:

```
"Name",projection_code,datum_code[,units_code][,origin_lon]
[,origin_lat][,std_parallel_1][,std_parallel_2][,azimuth]
[, scale_factor][,false_easting][,false_northing][,range ]
```

The parameters following the projection code are specified in the following table:

Code	Name	D	U	O	L	1	2	A	S	E	N	R
1	Longitude/Latitude	•	·	·	·	·	·	·	·	·	·	·
3	Lambert Conformal Conic	•	•	•	•	•	•	·	·	•	•	·
4	Lambert Azimuthal Equal·Area (polar aspect only)	•	•	•	•	·	·	·	·	·	·	•
6	Equidistant Conic (also known as Simple Conic)	•	•	•	•	•	•	·	·	•	•	·
8	Transverse Mercator (also known as Gauss-Kruger)	•	•	•	•	·	·	·	•	•	•	·
9	Albers Equal-Area Conic	•	•	•	•	•	•	·	·	•	•	·
10	Mercator	•	•	•	·	·	·	·	·	·	·	·

Code	Name	D	U	O	L	1	2	A	S	E	N	R
11	Miller Cylindrical	•	•	•	·	·	·	·	·	·	·	·
12	Robinson	•	•	•	·	·	·	·	·	·	·	·
13	Mollweide	•	•	•	·	·	·	·	·	·	·	·
14	Eckert IV	•	•	•	·	·	·	·	·	·	·	·
15	Eckert VI	•	•	•	·	·	·	·	·	·	·	·
17	Gall	•	•	•	·	·	·	·	·	·	·	·
18	New Zealand Map Grid	•	•	•	•	·	·	·	·	•	•	·
20	Stereographic	•	•	•	•	·	·	·	•	•	•	·
24	Transverse Mercator (modified for Finnish KKJ)	•	•	•	•	·	·	·	•	•	•	·
25	Swiss Oblique Mercator	•	•	•	•	·	·	·	·	•	•	·
26	Regional Mercator	•	•	•	·	•	·	·	·	·	·	·
27	Polyconic	•	•	•	•	·	·	·	·	•	•	·
29	Lambert Azimuthal-Equal·Area	•	•	•	•	·	·	·	·	·	·	•
30	Cassini-Soldner	•	•	•	•	·	·	·	·	•	•	·

Where:

Projection codes
D = Datum
U = Units
O = Origin of Longitude
L = Origin of Latitude
1 = Standard Parallel 1
2 = Standard Parallel 2
A = Azimuth
S = Scale Factor
E = False Easting
N = False Northing
R = Range

Unit codes:
0 = Miles
1 = Kilometres
2 = Inches
3 = Feet
4 = Yards
5 = Millimetres
6 = Centimetres
7 = Metres
8 = Us Survey Feet
9 = Nautical Miles
30 = Links
31 = Chains
32 = Units_Rods

Predefined datum codes and ellipsoid codes are as defined in the MapInfo Professional documentation.

User-Defined Datums

If you wish to define your own projection, spheroid and datum coordinate system the following applies:

- If `datum_code` = 999 then it is a user-defined datum and these parameters must follow it:

  ```
  ellipsoid_code, dX, dY, dZ
  ```

- If `datum_code` = 9999 then it is a user-defined datum and these parameters must follow it:

  ```
  ellipsoid_code, dX, dY, dZ, Ex, Ey, Ez, m,
  prime_meridian
  ```

- If `datum_code` = 99999 then it is a user-defined datum and these parameters must follow it:

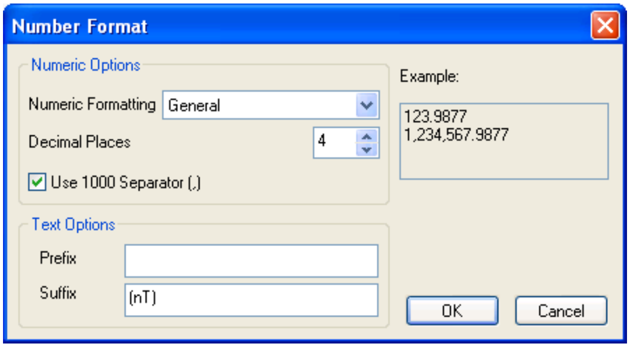
  ```
  ellipsoid_A, ellipsoid_InvF, dX, dY, dZ, Ex, Ey, Ez, m,
  prime_meridian
  ```

Examples of User-Defined Datums

```
"AMG Zone 55 (AGD 66) 3-param datum shift", 8, 999, 2,  -133, -48,
148, 7, 147, 0, 0.9996, 500000, 10000000

"AMG Zone 55 (AGD 66) 7-param datum shift", 8, 9999, 2,  -117.808, -
51.536, 137.784, -0.303, -0.446, -0.234, -0.290, 0, 7, 147, 0, 0.9996,
500000, 10000000

"AMG Zone 55 (AGD 66) 7-param datum shift", 8, 99999, 6378160.0,
298.25,  -117.808, -51.536, 137.784, -0.303, -0.446, -0.234, -0.290,
0, 7, 147, 0, 0.9996, 500000, 10000000
```

Label Properties

Common to many dialogs, it is possible to format the appearance and style of text labels. These labels are used for automatically assigning the presentation style of number labels in locations with axes, titles and line numbers etc. Usually the label style dialog is displayed from a Format button. The dialog appears as below:



Number and text label format specification

The dialog gives control for the Numeric Formatting to be:

- **General** – Catering for most numeric values (both negative and positive)
- **Scientific** – For example, 1.2345679877e+006
- **Automatic** – Creating the best suited available number representation
- **DMS (Decimal)** – For example, -34.2756 latitude
- **DMS (Latitude)** – For example, -34:27:53.2
- **DMS (Longitude)** – For example 143 56 16.8

The number of **Decimal Places** can be specified and a comma can be used (instead of the full stop) as a **1000 Separator**.

A **Prefix** and **Suffix** (for example, nT, dd:mm:ss, Feet) could be added to the numbers. A preview example in the dialog provides a sample of the output format.

Curve Properties

A Curve branch can display multiple curves from a multi-banded data source. It can display the data as a colour modulated line, filled above or below a specified level. It can also display symbols at each data point. The symbols can be colour, rotation and size modulated. Symbols can display a label that can also be data modulated and rotated. The curves can be added to a reference data channel to allow for topographic offset.

A graph can contain as many Curve objects as required. The Curve properties dialog contains property pages, which control data source (globally), line styles, symbols styles, symbol rotation and symbol labels.

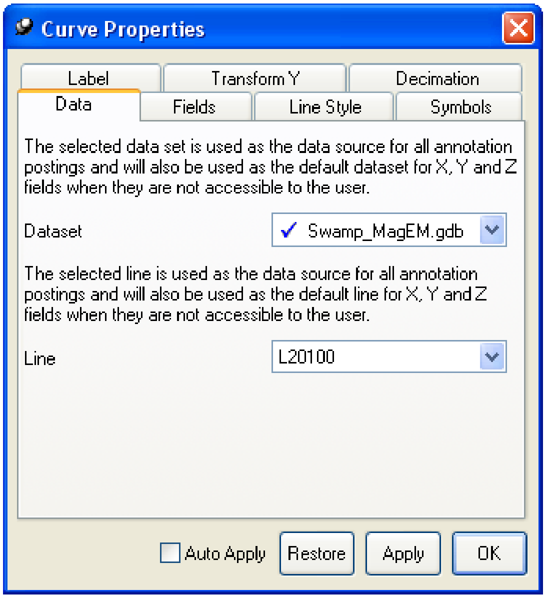
- *Data Tab*
- *Fields Tab*
- *Line Style Tab*
- *Symbols Tab*
- *Label Tab*
- *Transform Tab*
- *Decimation Tab*

For additional information on defining curve objects, see:

- *Data Specification*
- *Modulating Data*

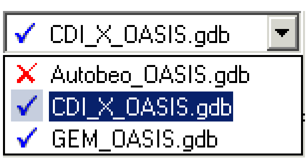
Data Tab

The Data tab allows the user to select the dataset and line for all data fields in the Curve. Note that an identical Data source is available in the Profile properties (see *Data Tab*). The difference between the tabs is one of scope – the profile Data tab applies it settings to all objects in the profile whilst the Curve data tab applies it settings only to the Curve. Note that in a Profile there can be only one line and so changing the line at any level changes the line for the profile globally. See *Accessing Data with Encom Discover PA* for information on accessing different data sources.



Data source dialog tab for Curve properties

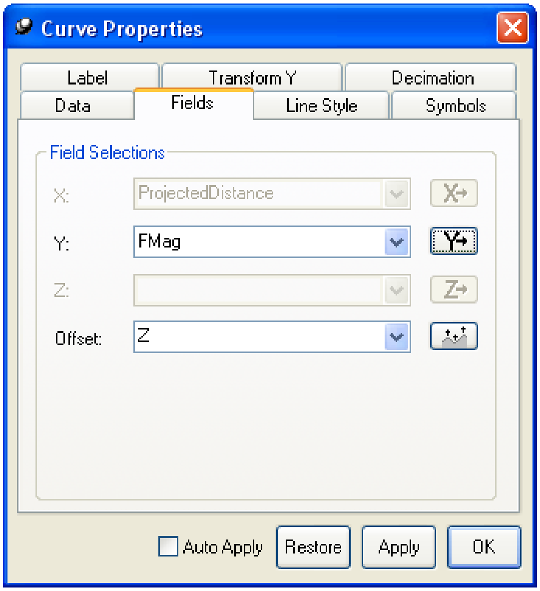
The Data tab provides a method of setting the line and dataset source for all components of the Curve. See *Accessing Data with Encom Discover PA* for information on accessing different data sources. Also see *Fields Tab* for channels specification of data. The Dataset specification refers to the source database. Encom Discover PA can access any number of databases and is able to display data fields from multiple datasets within a profile or profile frame. Selection of the dataset is via a pull -down list. Datasets may not always share common lines. If the selected line for the profile is not found in a dataset, that dataset is marked with a red cross. Otherwise it is marked with a blue tick. If the profile contains data from datasets that do not share the specified line, then the profile will become invalid and no data content is displayed.



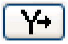
The Line setting can also be specified from a pull-down list. It displays all lines in the selected dataset. The nominated line is used as the source for all data, scaling, annotation etc.

Fields Tab

When a database is accessed, it usually contains a number of data fields (channels) that can be used for display. These channels can be specified from either the Fields tab or the Y data band specification described in *Data Specification*. Both data specifications do the same thing but the Y data band specification allows additional control in multiple channel situations.



Specification of a data channel in the Fields tab

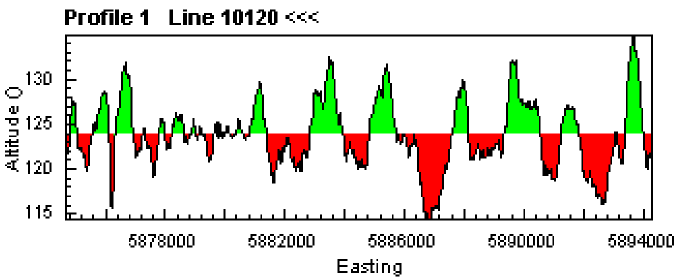


You can select the relevant data channel field from a required database. In circumstances where you have more than one database open, you may need to use the Data tab to select a different database before having access to the correct data field. In the case of multiple bands within a data field, you can use the **Browse** button to select the required channel range (see *Data Specification*).

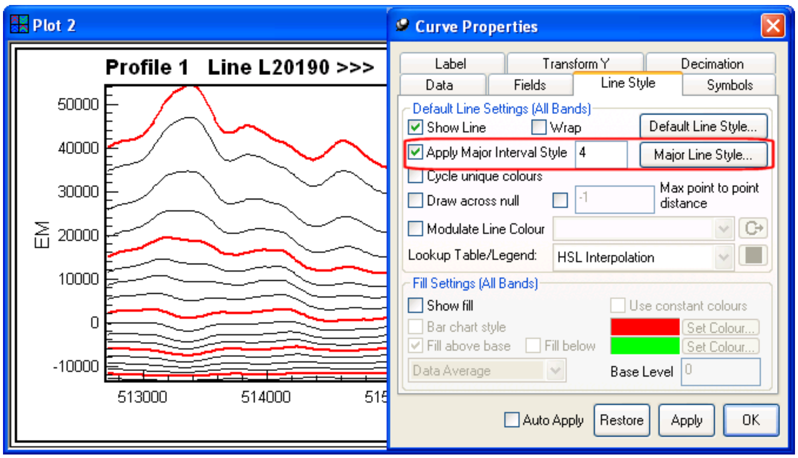
Where appropriate, additional data fields (such as Offset) can also be specified from the Fields tab. Note too that any changes to the data channel specification update the Curve immediately if the **AutoApply** option is enabled.

Line Style Tab

The Line Style tab controls the appearance of all curves in the Curve profile. Curves can have different colours, style, thicknesses and modulated colour.

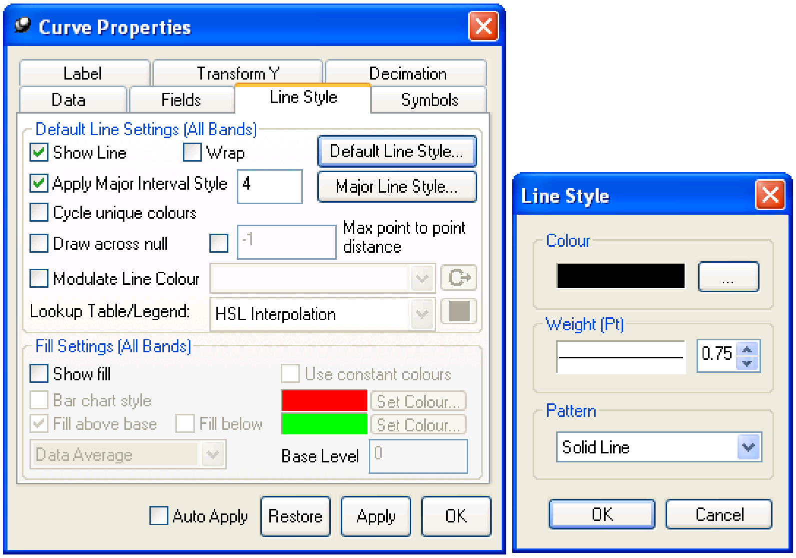


Data profiles can be filled by colour. The fill can be above and/or below a specified level. The level to control a fill can be specifically defined, data minima, maxima, median, mode or average.



An additional control of this dialog (**Apply Major Interval Style**) provides the ability to automatically divide multiple band datasets into major and minor line styles at a controlled interval (see example above). This means that one line style can be used for major data traces, but say, every 4th trace in the selected bands has a different style.

Line styles can be specified with a thickness, pattern and colour. Use the **Default Line Style** button of the Line Style dialog to specify the required colour. The line patterns include Solid, Dashed and Dotted.



Curve profile Default Line Style controls

Rather than individually select each band to have a specific colour, the software can be used to automatically Cycle unique colours if required. Alternatively, the colour data channel used for the data display can Modulate line colour of the data line. If appropriate, different colours (using the Lookup Table/Legend selection) can also be used to vary curve types. Colours can also be edited to adopt a specific look-up table modulation. See *Colour Table Editor* for additional information.

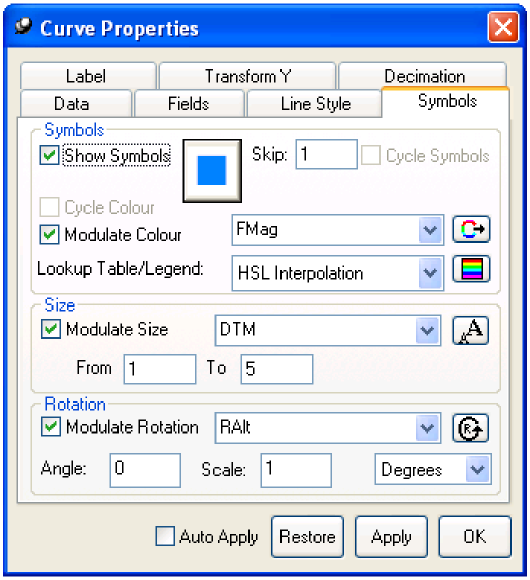
Symbols Tab

The representation of the data source (see *Data Tab*) by symbols can often be used to advantage.

The Symbols tab enables each (or a sub-sample of) data points along the profile curve to have a symbol displayed. Available controls include:

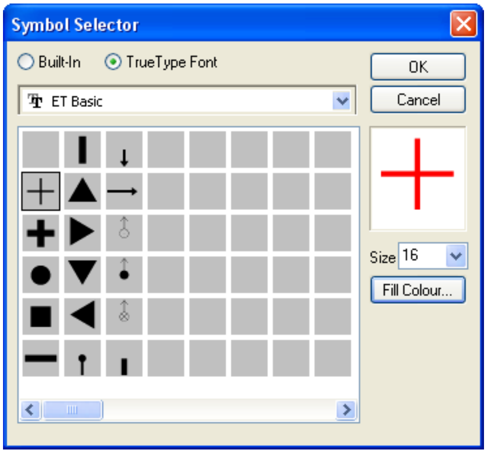
- Selection of a single symbol type from the available fonts installed.
- Cycle the available symbols (if displaying multi-banded data curves).
- Define the size of the symbols using one of the various supported Units.
- Cycle the colour of symbols automatically.
- Skip a number of readings to locate symbols are displayed. This is especially useful for densely sampled datasets.

- Use an external data field to control the appearance of the symbols (for colour, size, rotation and labelling). This is described in *Modulating Data*.



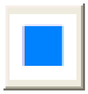
Symbol control dialog tab

To access this dialog ensure the **Show Symbols** option is ticked.



Symbol Selector dialog

To customise the symbol display, click in the **Symbol** box. This opens the Symbol Selector dialog. A range of symbol libraries is available from the pull down list at the top of this dialog, with the selected symbol previewed to the right. The symbol colour and size can also be altered here.



The symbol colour and border can also be changed using the **Line Colour** (for Built-In symbols) and **Fill Colour** buttons in the Symbol Selector dialog, accessed by clicking on the **Symbol** button shown to the left. The **Skip** value entered determines what interval of point data samples is displayed in the curve. For example, to display every 5^{th} point data sample enter a **Skip** value of 5.

To Modulate Colour activate the tick box, select the data field from the drop-down list to use for the modulation and a colour look-up table or legend to use. See *Colour Table Editor* for more information.

To **Modulate Size** activate the tick box, select the data field to use for the modulation from the drop-down list and enter a range for the minimum and maximum modulation size.

To **Modulate Rotation** select the data field to use for the modulation and enter the angle and scale. The rotation of modulated symbols can use either radians or degrees anti-clockwise.

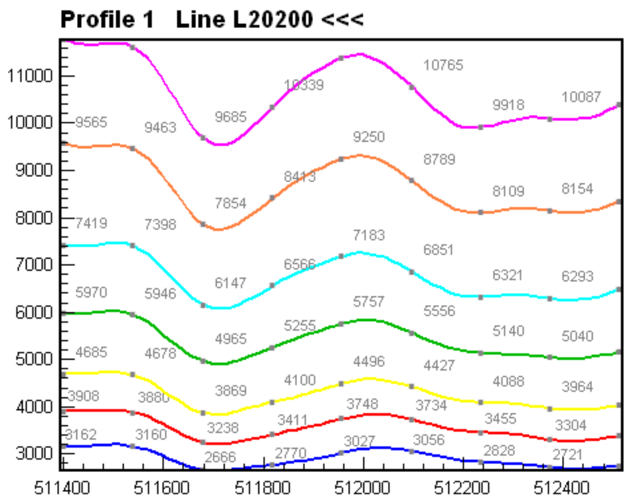
If the data channel specification is left blank, an un-modulated rotation is applied. In this case all symbols are rotated by an equal amount as specified. If Modulated rotation is used, it is necessary to specify the data field to use for modulation. This is done by clicking the **Browse** button or from the pull-down list. The rotation of modulated symbols can use either radians or degrees anti-clockwise.

Displaying Line Stations

From the Symbols tab enable the **Show Symbols** tick box to add Station Points to a Profile display. This option is used when data points are to be displayed in a Profile or Graph. Symbols Tab has many of the properties available for the Line Style tab, such as Modulate Colour, modulate Size and some additional parameters such as applying a Rotation angle and skip factor. To display only station points without the joining line of the curve profile disable the **Show Line** tick box in the Line Style tab dialog of the Curve Properties.

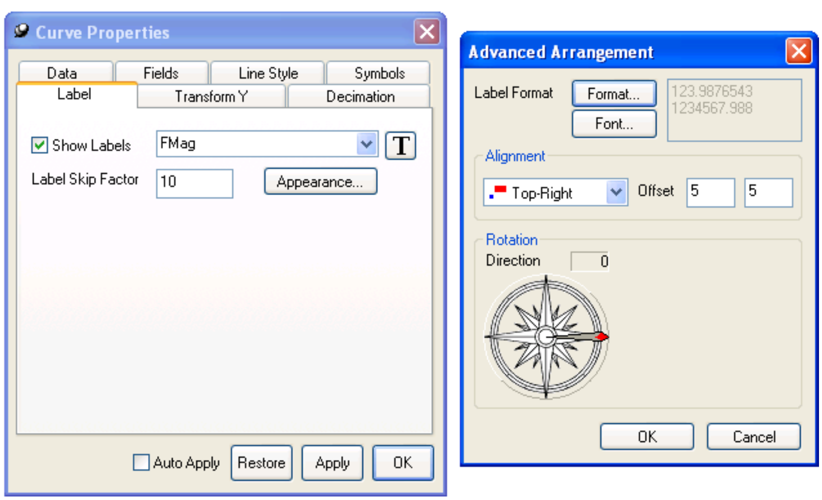
Use this option when data is randomly distributed or needs to be shown with only symbols displayed by default.

Label Tab



Labelling data points in a graph is useful in applications where actual data values are significant. The data source used for labels is specified in the properties dialog. If no symbol is desired, an empty symbol is available for selection. The dialog provides control for the labels to be formatted, applied with a skip interval, anchored relative to the plotted data point or offset from this. Rotation of the label text is also possible.

The dialog used for the Label properties is shown below. If modulating of labels by another data field is required (see *Modulating Data*), select the data channel from the pull-down list after enabling the Show Labels item.



Label control and formatting properties dialog

The format of the posted labels can be specified from the Format button. See *Label Tab* for information on specifying number formats.

The format of the data value (selected from the **Set Data** button) can be **Decimal** (of a set number of characters) or **Compact**. Compact formatting allows the software to choose the most appropriate format (such as scientific notation) if the allowable text size causes excessive over posting.

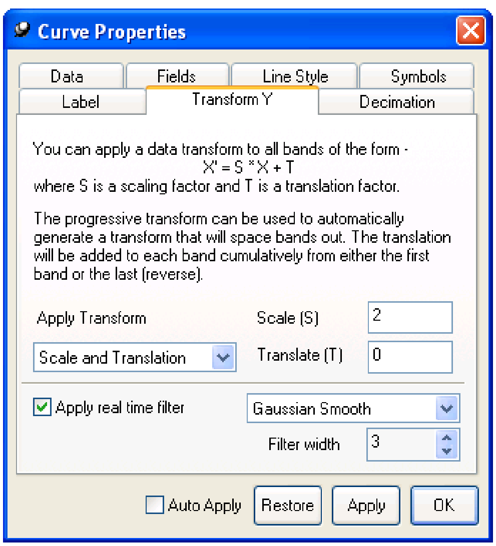
Transform Tab

The Transform tab of the Curve dialog allows you to apply a single data offset (additive) or multiplicative factor or real-time filtering and transformation. This function can be of use to scale data before it is displayed or to match other data channels without having to apply different scaling.

To enable the Transform operation, select it and then apply one or both of the Scale (S) and Translate (T) values. Ensure the **Apply Transform** setting is **Scale and Transform**. The transformation is applied using the formula:

```
X' = X*S + T
```

Note the transform can apply to multiple bands of a data channel.



Transform dialog allowing the specification of an offset and transform factor

Additional transformation types include Progressive and Reverse Progressive Transforms. These transformations are especially useful where an offset is used to better display data across multiple data channels. For example, where profile traces overlap or have different scaling, you can apply a **Progressive Transformation** and this will add an offset to the data. If, for example, a **Reverse Progressive Transformation** was used with say 10 data channels in an arrayed field with a Translation factor of 1000, then:

Channel 10 would have zero added to it

Channel 9 would have 1000 added to it

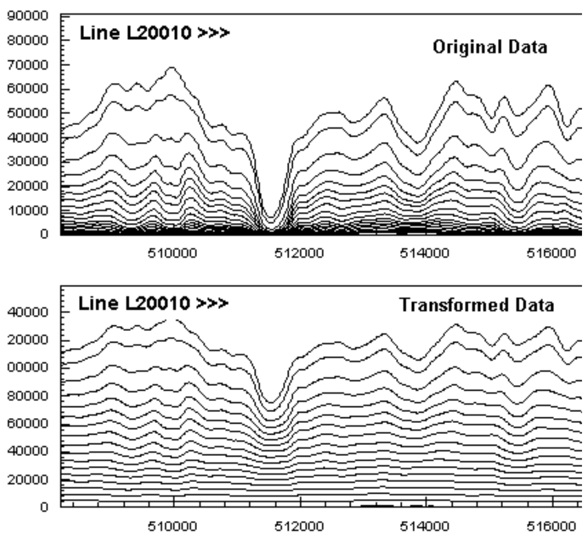
Channel 8 would have 2000 added

...

Channel 1 would have 10,000 added to it.

The opposite order would be true for a Progressive Transformation. An example of using a Reverse Progressive Transformation used with EM data and a Transformation factor of 1,000 is shown below.

Note The vertical axis of a transformed display becomes affected by the transform applied and as such, may be meaningless when used to directly scale data values.

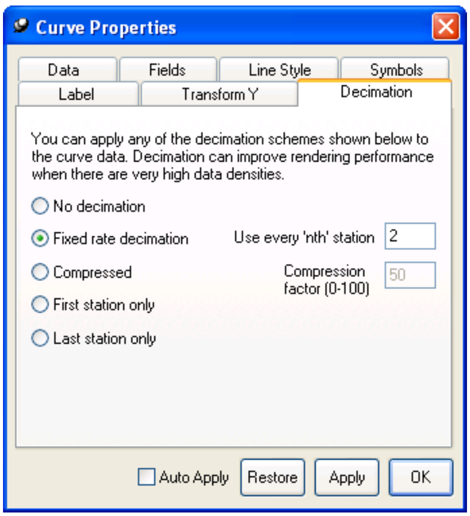


The same profile with untransformed, original data plus transformed data below

Real-time filtering can also be applied to specified data. You can select **Moving Average Smoothing**, or **Gaussian Smoothing**. In both cases, a filter width is required to specify over how many data samples the operator is applied. Filtered displays can be useful when displayed as two graphs superimposed so the filtered and unfiltered results can be directly compared.

Decimation Tab

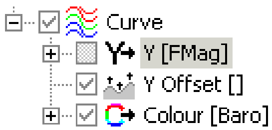
A range of decimation schemes are available to compress the amount of data displayed in a curve. This option allows high resolution data to be displayed without every readings used in the plotting. By default, no decimation is applied but a fixed rate of decimation or an arithmetic compression factor can be applied.



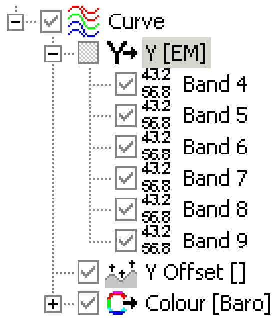
Decimation tab of the Curve Properties

Data Specification

In general it is possible to specify X, Y, Colour, Size, Rotation, Label and Offset multi-banded data fields for a Curve. However, depending on what type of graph the Curve is embedded in, some of these fields may not be available for selection. In a Profile, the X field is not selectable. In a Map, the X and Y fields are not selectable. In these cases the system chooses the data field.

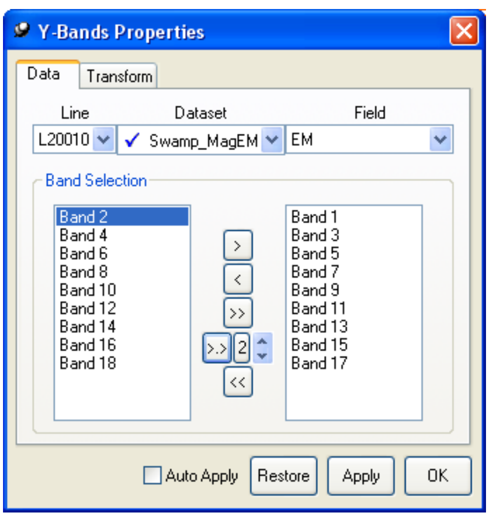


The data displayed in a profile is controlled by the Y branch entry of the Curve. The data field used to specify the data used in the profile uses a property dialog as described in *Data Tab*. If colour modulation is required, the colour data field must also be defined. By default, it uses the first available data field and is initially identical to that of the Y data specification.



If a multiple band field is selected for the data input to the Curve, each individual band can be controlled by expanding the Y field.

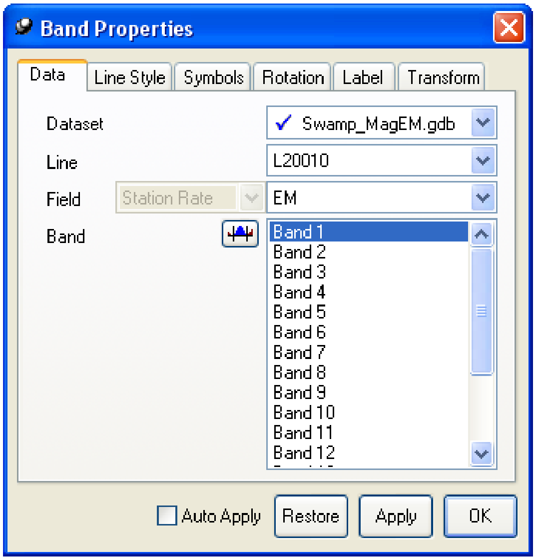
The properties of each band can be individually controlled if required. This is not usually necessary since sufficient controls are available at higher levels of the Workspace tree to preclude this.



Properties dialog for individual bands of a multi-band data field

Note that the specified data can also have a Transform (including real-time filtering) applied. See *Transform Tab*.

If you wish to individually control the properties however, data source, line, symbol and label styles can each be defined for each individual data band.

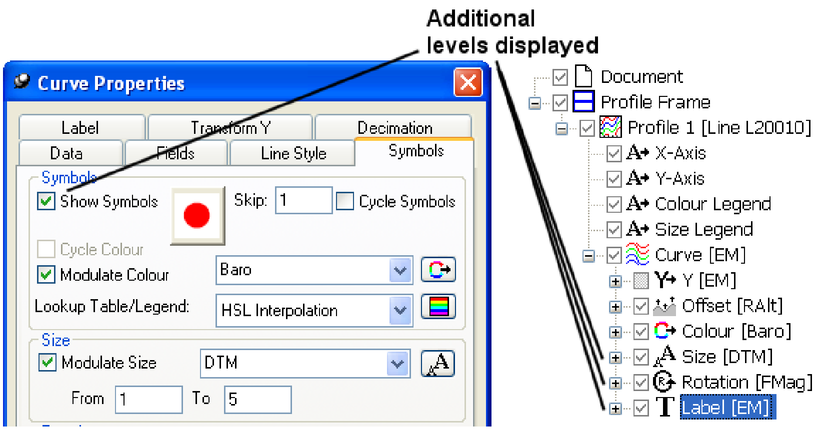


Properties dialog for individual bands of a multi-band data field

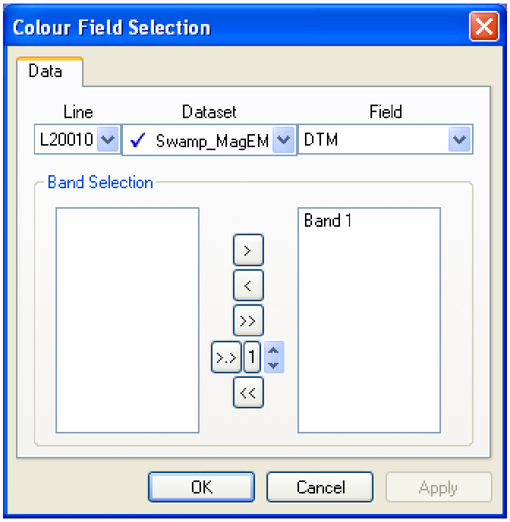
Note that the properties tab dialogs for each of Line Style, Symbols, Rotation and Labels are all identical in function and description to those referenced earlier in this chapter.

Modulating Data

With no symbols displayed, a Curve in a Profile only shows Y, Colour and Offset data sub-branches. When the **Symbol** option is enabled, the Rotation, Size and Label modulation sub-branches are added beneath the Curve branch of the Workspace tree.



Each additional branch has a property dialog that permits selection of a database data field and associated bands. These branches can be accessed through short-cut buttons – **Browse** on the Symbols page and **Field Selection** on the Rotation and Label tabs. An example of a multi-banded data selection dialog is shown below.



Band selection from a database channel

Operation of line, database, channel and band selection is described in *Selecting Channels and Tables*.

Controlling the appearance of symbols, lines and labels is a powerful feature of Encom Discover PA. Any compatible data field in the source database may be used to control the selected property of Colour, Size, Rotation or Labelling. Individual bands of a multi-banded data field may be used.

11 Spreadsheets

In this section:

- *Displaying Numerical Data*
- *Database Structure*
- *Using the Encom Discover PA Spreadsheet*

Displaying Numerical Data

Encom Discover PA uses a variety of presentation formats to display data contained in databases (see *Accessing Data with Encom Discover PA*). Graphical displays such as profile windows and maps are used to present data on-screen or for hard copy plots. To provide a numerical display of the content of a database, Encom Discover PA provides a data spreadsheet facility. The spreadsheet can present the numeric content of a database, statistics or data ranges. The databases accessed by Encom Discover PA typically contain flight information, lines and the various data channels of sensor information. The spreadsheet can also be used to export data or close a database.

Database Structure

The structure of a survey database is assumed to have certain components:

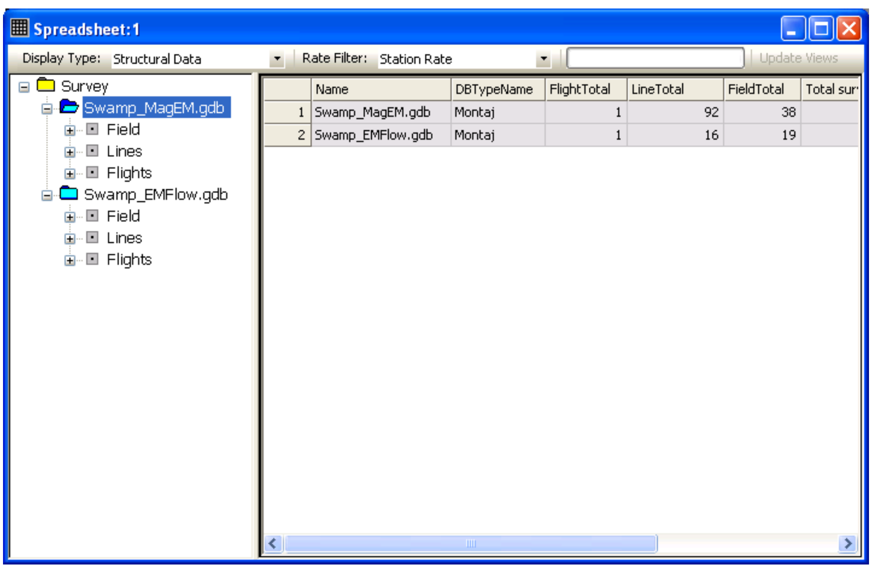
- **Survey** – This is a reference to the entire database and is assigned a name, usually reflecting the data content or project.
- **Datasets** – A dataset name is associated with a survey. Usually, this name is the content name of the opened database.
- **Flight** – Survey flights subdivide a survey and correspond to individual acquisition periods. In the case of an airborne geophysical survey, flights refer to individual aircraft recording sorties usually within a defined time frame. For ground surveys, flights may be comprised of equivalent divisions of a survey corresponding to, for example, a day by day acquisition period.
- **Line** – A traverse, of any orientation, collected within a flight. Lines have a finite start and end location and must have corresponding navigational or positional information. Any number of lines that are of any length can be accommodated by Encom Discover PA.

- **Field** – Fields comprise the various data values associated with readings made along lines. An alternative term frequently used for field is channel. Data fields are derived from geophysical sensors such as magnetometers, induction coils, radar altimeters etc. making measurements in a recording platform as it moves along a survey line. These measurements form individual data channels (fields) of the survey database.

- **Bands** – It is convenient to keep multiple readings of similar type for a given field together. An example of this is a series of decay measurements for an airborne electromagnetic survey. Rather than have EM1, EM2, EM3… fields it is more convenient to have an EM_Data field that is comprised of the individual reading bands. Some databases, such as Intrepid, can create band-oriented data fields on import.

Using the Encom Discover PA Spreadsheet

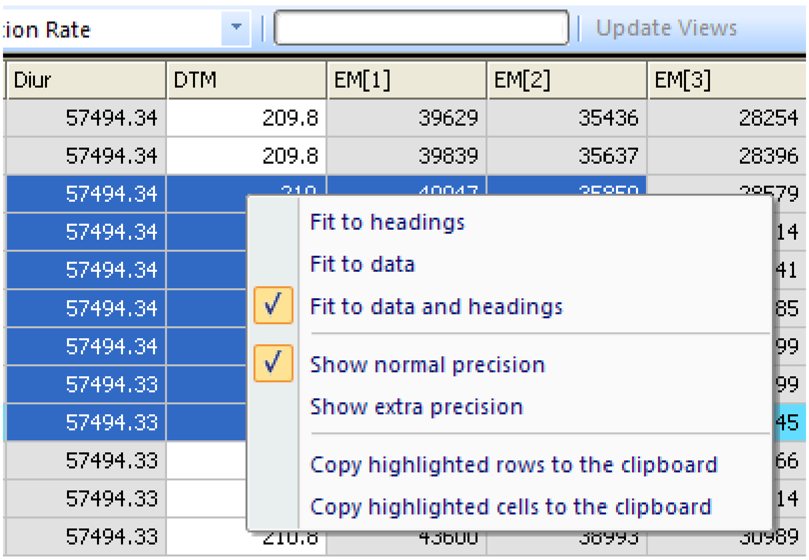
The Encom Discover PA spreadsheet is designed to provide flexibility and fast access to all aspects of the data in a database. After initial access links are created between Encom Discover PA and selected databases, the spreadsheet is displayed to indicate which files and data are available. At other times, to access and display an Encom Discover PA data spreadsheet, select the **File>New>Database Spreadsheet** option or use the spreadsheet button. When selected, the spreadsheet presents two open areas plus pull-down list items in a control bar at the top of the window.



Initial spreadsheet view after opening

The left portion of the spreadsheet view is used to select a hierarchy of the database components, as described in *Database Structure*. The right area of the spreadsheet displays the results of the selected components and the method of viewing the data.

Right clicking on the row header of a selected record header displays a pop-up menu. Options in the menu allow automatic heading and/or data spacing operations. Note these spacings can also be made manually by selecting and moving the header column widths.



You can select a row, column or group of cells by positioning the cursor over a cell, depressing the left button and dragging the mouse (as you would in other applications, such as Microsoft Excel). Once a block of cells has been selected, you can copy the highlighted rows of data or the highlighted cells of data to the clipboard and then paste the copied data into other spreadsheet or text based programs, e.g. Microsoft Excel..

Data presentation is controlled by the settings of the drop-down lists contained in the Display Type and Rate Filter options.

- *Display Type*
- *Rate Filter*
- *Database Operations*
- *Spreadsheet/Profile Display Data Linking*

Display Type

The choices available in the Display Type to control the spreadsheet data are:

No Data – Shows no information from the database.

Structural Data – Presents information on the structure of the selected data component. For example (see below), for a selected Dataset, the data displayed for the open database is path, name, flight, line, field and Total Survey Length. To view this information ensure that the branch containing the database name is highlighted in the left hand margin of the Spreadsheet window.

	Name	DBTypeName	FlightTotal	LineTotal	FieldTotal	Total survey length	Path
1	Swamp_EMFlow.gdb	Montaj	1	16	16	274620.86	C:\Example Data\...

Alternatively, if a Line is nominated, the same Structural Data selection displays information detailing the number and content of fields.

	Number	FlightNumber	ChordX	ChordY	ChordDir	Length	FieldTotal
1	20100	0	525178.88	6313464	270.0031	16771.952	16
2	20110	0	508132.59	6313208.5	89.772563	17203.44	16
3	20120	0	525680.38	6313025.5	270.15107	17579.203	16
4	20130	0	508295.41	6312794.5	89.809076	16418.463	16
5	20140	0	525653.31	6312624.5	269.90753	17229.573	16

Statistical Data – Summary information of data ranges is useful for assessing data quality and possible database problems. In particular, when selecting individual fields, data range information can be useful. The total number of data points, minimum, maximum, mean, median and null values are shown.

	FieldName	BandTotal	StationTotal	NullTotal	Minimum	Maximum	Mean	NullValue	
1	Alt	1	18128	0	107.7	150.1	124.98966	-1e+032	N/S
2	ChordLength	1	18128	0	0	18695.337	8609.0337	-1e+032	N/S
3	Cond	51	924528	26210	2.524	1125.3765	261.20575	-1e+032	N/S
4	Depth	51	924528	0	-100	0	-50	-1e+032	N/S

Field Data – This selection provides the capability of examining individual readings within a database. Note also that for Field settings, the contents of individual bands are presented.

	Dist	Fiducial	X	Y	Z
1	0	4753	525178.88	6313464	344.5
2	14.2	4753.25	525164.69	6313463.5	344.6
3	28.39	4753.5	525150.5	6313463	344.6
4	42.59	4753.75	525136.31	6313462.5	344.7
5	56.79	4754	525122.13	6313462	344.8

Note The Field Data selection provides the capability of examining individual readings of a database. As a consequence, display of data using this option can take excessive time for large databases.

If the interrogation of a database (for statistics or other reasons) is attempted but the response is taking too long (as indicated by a progress bar within the spreadsheet), press the **Esc** key. After a short time, the processing halts and return control to you.

Rate Filter

The Rate Filter is designed to control the data that is passed to the spreadsheet for processing. Fields in a database are defined at a rate that can be Survey, Flight, Line or Station. The Rate Filter modifies how many records are stored in the field. The spreadsheet displays fields of the selected rate only. Options available from the Rate Filter include:

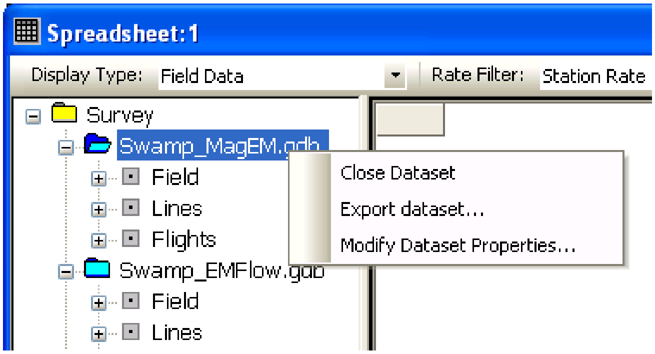
- **All Rates** – All components of data presented and used in statistics calculations.
- **Survey Rate** – A single record is displayed for each survey.
- **Flight Rate** – A single record for each flight.
- **Line Rate** – A single record for each line.
- **Station Rate** – A single record for each station. This is the smallest and the most common data rate.

Database Operations

Some operations can be performed directly from the spreadsheet. By highlighting a particular database in the spreadsheet and clicking the right mouse button, a pop-up menu allows you to Close, Export or Modify the default X,Y or Z data assignments:

- *Close a Database*

- *Export Dataset*
- *Modify Dataset XYZ*



Pop-up menu allows operations on a selected database

Close a Database

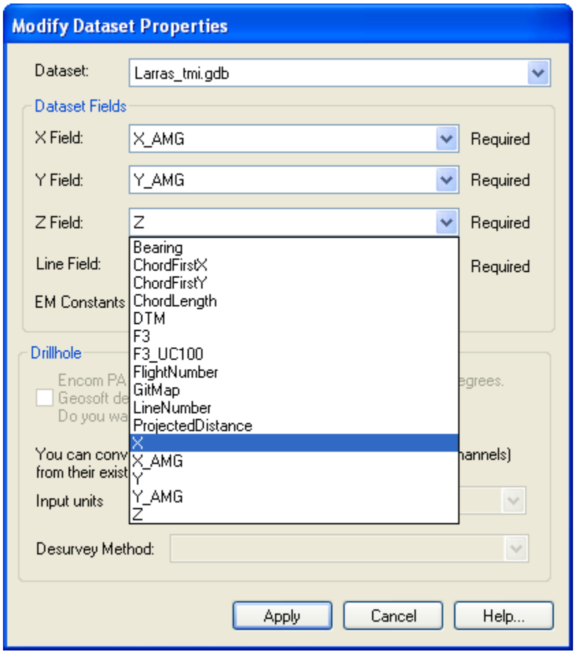
Individual databases can be closed using the spreadsheet **Close Dataset** option. This command is identical to the **File>Close Datasets** menu item. Individually closing a database affects only the selected dataset and leaves the remaining databases open for use in Encom Discover PA.

Export Dataset

Database content can be exported as ASCII CSV format files. The entire database or a subset, based on selected lines can be exported. Both Geosoft and Intrepid databases can be exported in this manner.

Modify Dataset XYZ

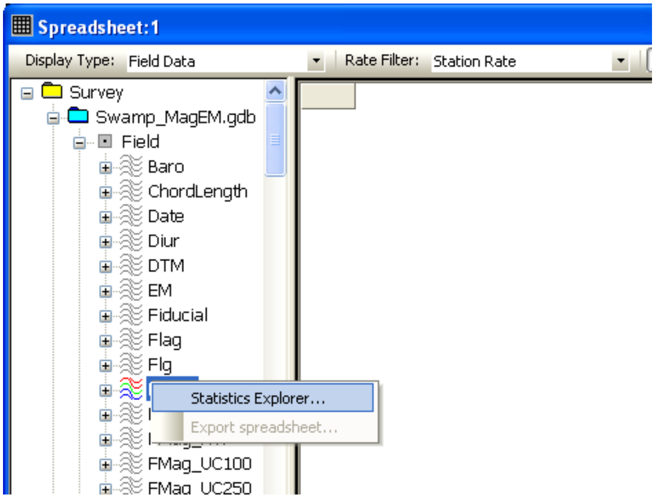
The **Modify XYZ** option is identical to the **File>Modify Dataset XYZ** menu item. This option allows you to specify the data channels of a database to use for data position. You can modify the easting (X), northing (Y) or vertical (Z) data channels. In particular, the vertical (Z) channel determines how a dataset is presented in three dimensional displays. If you need to alter this setting, select the option and change the Z data channel in the dialog displayed.



Alter any of the X, Y or Z data assignments from the pull-down list

Statistics Explorer

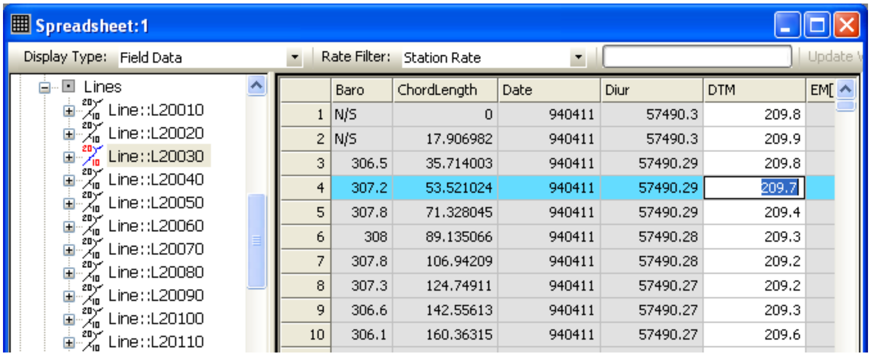
The Statistics Explorer can be accessed in the Spreadsheet view by right mouse clicking on any field listed under the Field sub-folder for a loaded database. This pop-up menu option will execute the complete Statistics Explorer utility. See *Appendix F: Statistics Explorer* for more information on using this utility.



The Statistics Explorer can be accessed from the spreadsheet view or a database field listed.

Editing Data

It is possible to edit line data from Geosoft and Intrepid databases displayed in the spreadsheet view of Encom Discover PA. For data editing to be possible the **Rate Filter** drop down option located above the right portion of the spreadsheet must be set to either **Station Rate** or **All Rates** and a specific line must be highlighted from a survey database listed in the left portion of the spreadsheet view showing the database heirarchy.



A spreadsheet view showing a cell value ready for editing

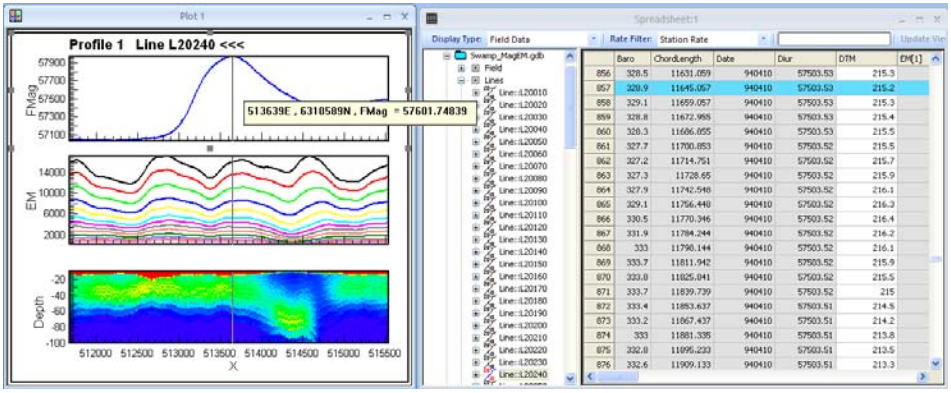
Any fields in the database that are not station rate or are protected in the database will be displayed with a grey cell background indicating they cannot be edited. However, to make a protected column editable right mouse click on the column header and choose **Unprotect field** from the pop-up menu that appears.

To edit a data value double-click on the cell in the spreadsheet to display a flashing cursor. Then either highlight and overwrite or press the BACKSPACE or DELETE key to edit the value. Press ENTER or click outside the cell to confirm the new value.

If data is modified via the spreadsheet then the **Update Views** button will become active in the spreadsheet view. Clicking **Update Views** will redraw any window open containing that database with the new data values. An update of the views will also automatically happen when the user closes the spreadsheet view if they have modified some data.

Spreadsheet/Profile Display Data Linking

It is possible in Encom Discover PA to link a cell in a Spreadsheet window (while in Field Data display view) with the position of that station in an opened Profile Display. Clicking anywhere on a row of a particular station will automatically display a vertical cursor in the corresponding profile. This data linking also works when a station reading is highlighted in a Profile Display while the Spreadsheet window is open. In this case the Spreadsheet display will update to highlight the row of the selected station.



When a station is selected in a profile display or a spreadsheet display the position of this station is highlighted in the related display (spreadsheet or profile) window.

To select a range of stations in the profile display or a range of rows in the spreadsheet window, view the context menu of the window by right-clicking on the window and choose the **Select Range** menu option.

12 Profiles

In this section:

About Profile Analysis

The majority of geophysical data is collected along lines and quickly converted to grids for subsequent image analysis. Subtle targets may go undetected when we rely on imagery alone. Profile analysis allows an interpreter to derive additional information that cannot be resolved in images. Examples include simultaneous visualisation of more than 3 channels of data, subtle shape information in curves, comparison of unrelated data types, and fine detail that may be lost during gridding. Until the arrival of Encom Discover PA, the detailed analysis of profile data from large surveys was slow and image analysis provided the only practical method for analysing large quantities of data.

Encom Discover PA uses a variety of presentation formats to display data contained in externally accessed databases (see *Accessing Data with Encom Discover PA*) and the most important of these is the Profile. For these displays, the most common drawing attributes such as line colour, thickness, axes, titles etc. can be individually controlled. Additionally, Encom Discover PA is designed with a number of intelligent processes to make logical decisions when you add graphs, change scales or add channels to profiles with different scales etc. The in-built intelligence of Encom Discover PA, especially in graph manipulation, makes it extremely powerful and versatile.

When the various display attributes have been specified and edited, Encom Discover PA uses templates or session files to save the presentation you have designed. Templates can be re-loaded and used for other datasets and projects. A range of templates for various applications are provided for your use without having to enter or modify the display properties.

To start using profiles, refer to the topics in this section. For more detailed information about changing the appearance of profiles, see *Display Properties*.

Introduction to Profiles

The Profile display format is displayed in a Encom Discover PA Plot window. The Profile is a specialised form of a Graph document. It is different from a graph because the horizontal axis of a profile is scaled by distance (and plotted using a computed data channel called ProjectedDistance). This is why when a Profile or Curve entry is seen in the Workspace tree, only the Y axis data entry is required – The X axis is ProjectedDistance. This difference between profiles and graphs however is fundamental in that multiple lines of a dataset can be compared directly in the same window and therefore important feature and interpretation benefits are gained through correlation of responses across lines. In all other respects Profile controls are identical to Graphs.

Profiles are the primary method of displaying line-based data in Encom Discover PA. Since this display format is so important to the operation of Encom Discover PA, extensive user control and flexibility have been provided. A document with profiles is a plot with a number of graph traces or layers superimposed on it. Consequently, the properties of a document with a profile are not solely to control the graph presentation format, but also those of a plot when output to hard copy.

When describing profiles and graphs, reference is frequently made to scales and axes:

- The **Scale** is a graphical setting that determines how large the object is drawn.
- The **Axis** is a graphical object that indicates the drawing scale.

Some of the options available for documents with profiles are:

- Multiple Profile windows may be open together or in combination with minimised profile windows.
- Any Plot window with Profiles may contain multiple graphs.

- Each individual graph track contained in a Profile may contain multiple Y-axes where each axis has its own scale to control the vertical plotting of each data trace.

- Each Y scale may contain many data channels. For example, a singe track may display magnetics with diurnal and gravity responses. Each channel can be individually scaled and have its own display attributes.

- Each graph track can be used to display trace information (a single channel value at a single reading location), or layers that provide a filled colour area between a top and bottom boundary.

- Each document with a Profile can cycle through the display of multiple lines and consequently a single horizontal scaling is used for all graph traces contained in the window. The use of a single, common horizontal X-scale is fundamental to all documents containing Profiles. See *Profile Horizontal Scale*.

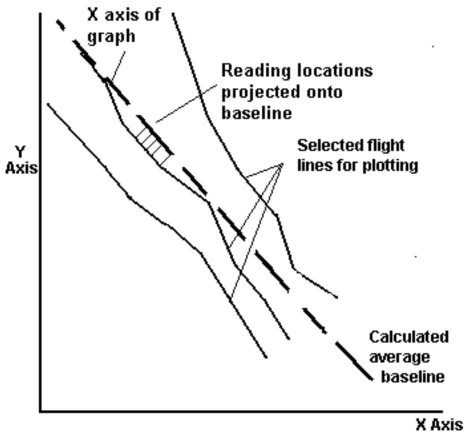
The Profile Frame

An Encom Discover PA document is similar to a document in, for example, Microsoft Word. It represents a piece of paper on which you can place a number of graphical displays. In the case of profiles, the graphical displays are contained in tracks (with their own axes, annotations, data traces etc). Tracks and their contained profiles are enclosed in a Profile Frame.

Profile Frames can have one or many profiles. The frame has its own set of properties (see *Profile Frame Properties*). Properties specified in the frame have overall control of track margins, borders, size, data source etc. Individual controls over specific Profiles are then modified by the controls of the contained Profile levels.

Profile Horizontal Scale

Encom Discover PA has the capability of displaying multiple lines in single or multiple graphs of one document. Each line must therefore have a common X-Scale. This scale is calculated by applying a number of rules to create a common projected distance baseline. The rules are based on the azimuth of the selected lines and the chosen scale to be used for the X-Axis.

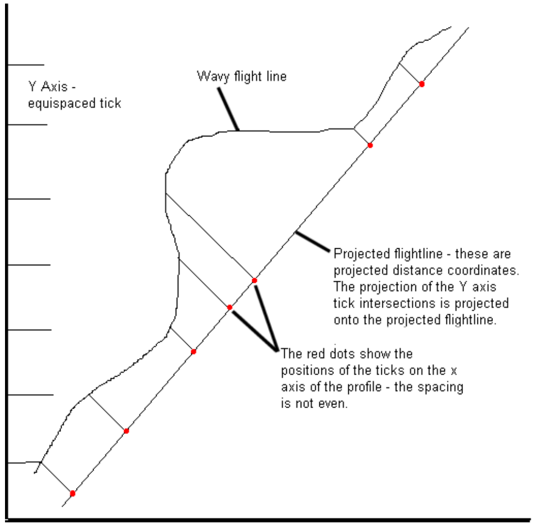


In the diagram, a group of three flight lines, each with different average azimuths are chosen for plotting. The azimuth is computed from north using the first and last point of the relevant line. From the computed azimuths, an average baseline, is calculated. The plotting of each flight line is individually projected to the average baseline by a perpendicular translation for each reading. The average baseline with the projected readings are then used to create a chord length (the cumulative distance from the origin of the baseline), to form a horizontal (X-axis) for the plotted graph.

This procedure, used for projected plotting means that in some instances, a flight line does not fill the entire graph area available, whereas other displayed lines do. Encom Discover PA is designed to maximise the display for the longest line such that shorter lines appear with a blank area. The benefit of this style of plotting however is a common horizontal scaling that has uniform annotation that can accommodate multiple flight lines with comparative scale and position. The interpretational advantage means that correlation of features from one line to the next or previous is simple.

Certain azimuth tolerance controls are available in the Line Iterator to provide specific rejection criteria of lines. Further information on this is available from the *Line Iterator Tool* and *Line Navigation*.

Flight Line Positioning and Scales



Under certain circumstances, the axes displayed in profiles may have what appears to be aberrant behaviour if flightlines are not straight or near-straight. This diagram is of a small part of the flight path track of a map where the X axis to be plotted for the profile is at an angle paralleling the flight path.

Note that the distances between the coordinates on the X-axis are not equal. The flight line track is quite "wavy". In this case the northings are being plotted on the X axis of the profile and the spacing between ticks is variable. This is a correct consequence of the wavy flight line.

The X axis ticks are equi-spaced in northing coordinates. The point where the flight line intersects the Y value is projected onto an ideal flight line. The coordinate of this projected point is in ProjectedDistance data channel coordinates as computed by Encom Discover PA. As the diagram illustrates, the spacing of the ticks on the X axis will only at ideal increments if the flight line is always aligned at the same angle to the ideal flight line.

Annotation of the Horizontal Axis

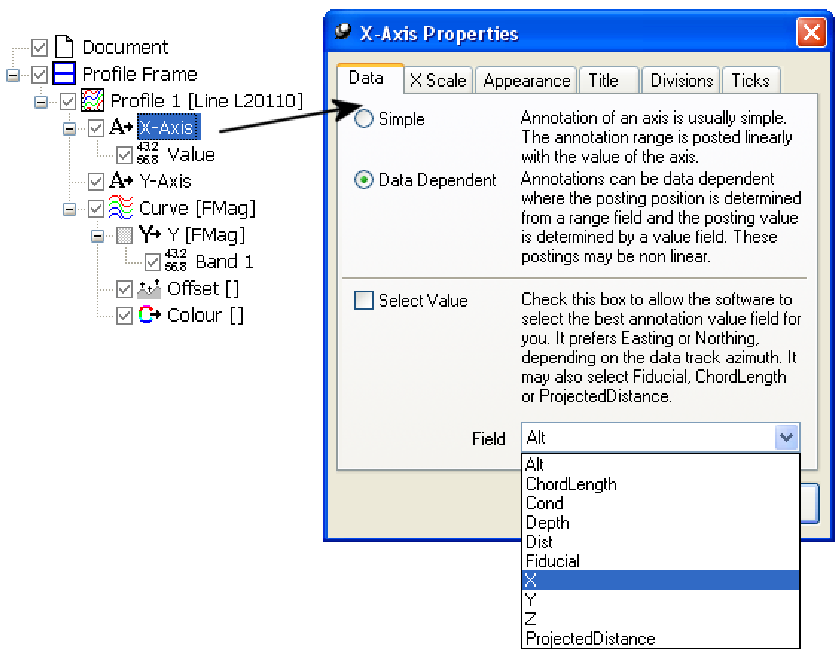
The horizontal axis and scale of a profile uses a common distance. The distance used is derived from a Encom Discover PA computation to derive two channels called ChordLength and ProjectedDistance. By default, profiles use the ProjectedDistance channel (as described in *Profile Horizontal Scale*) for plotting the position of individual reading values. The X-axis annotation text can be automatically chosen by the software, or can be specified from another channel. Which of these two options is used is determined by a **Simple** or a **Data Dependent** setting in the **Line Style** tab of the Axis Property dialog.

- The **Simple** setting uses the ProjectedDistance channel for both the data plotting and the axes. These axes are displayed linearly.

- The **Data Dependent** setting uses an assigned channel for axis annotations (such as Fiducial, Easting or Northing) and may not be linear.

A third option available for X-axis annotation is to allow the software to choose which channel is most appropriate to be used for labelling. The line channel is used for this selection and easting or northing is generally selected.

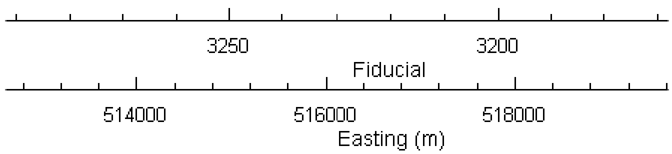
The X Axis property dialog controls labelling of the X-Axis.



Annotation style control of the horizontal X-axis

If a specific Data Dependent channel is to be used for axis labelling and plotting (such as Fiducial), deselect the Select Value option (so Encom Discover PA does not choose it for you) and then choose the Field you wish to use for the axis. Note that this option is only visible when the Date Dependent item is selected. Also note that the specified field is shown in the properties of the X-Axis branch of the Workspace tree.

Using this control it is also possible to have multiple horizontal axes for the one profile. For example, along a profile track may be a location axis (eg. Easting) and a second axis that indicates Fiducial or timing.



Combined axes for one profile showing Easting and Fiducial annotations

Creating a Document for a Profile

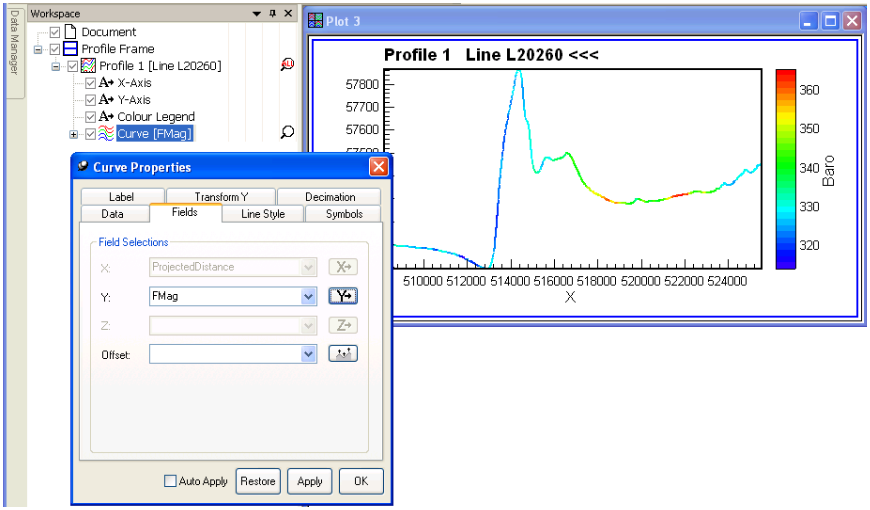
To create a new document containing a profile, either:

- Choose **File>New>Profile>Curve Profile**, or

- Click the **Curve Profile** pull-down button from the Documents toolbar.

The default document with a profile selects the first line and data field information of the first open database. Often the content is not initially appropriate and requires re-configuring.

The displayed profile uses data from the initially opened database, first line, first data channel. Associated with the profile is the basic hierarchy of the profile displayed in the Workspace toolbar. By default, the Property dialog for the displayed channel is also displayed (see *Document Properties*).



Default Encom Discover PA Document window with Profile

The Workspace toolbar describes the content of the Document. The tree contains a Profile Frame, a Profile and three associated axes (colour, vertical (Y Axis) and horizontal (X Axis)). Beneath the Profile branch is also a sub-branch called Curve. This tree level controls the data trace derived from a data field scaled vertically by the Y axis. By default, Encom Discover PA colours the data by itself but this can be controlled.

To change any of the profile contents, position the cursor in the Workspace tree and click the right mouse button. A pop-up menu appears with different options depending on which item is highlighted (see *Working with the Workspace Tree*).

Axes and Annotations

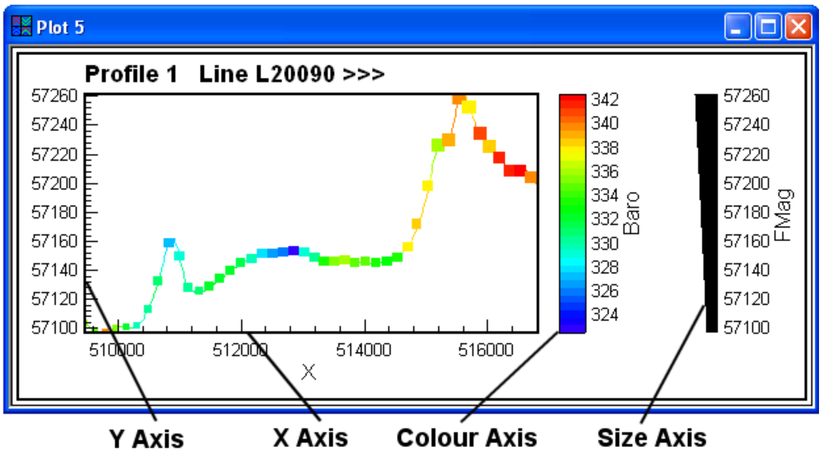
All presentations in Encom Discover PA have axes associated with them. The axes serve the purpose of providing scale and absolute referencing of data values plotted. Within Encom Discover PA it is important to understand the structure of axes.

There are five types of axes:

- **X-Axis** – Used for horizontal scaling, axis and reference
- **Y-Axis** – Used for vertical scaling, axis and reference
- **Colour Axis** – Usually hidden, this axis is used for colour modulation and reference to a data channel

- **Size Axis** – Usually hidden, this axis is used for symbol point modulation and scaling. It exists in the hierarchy of a profile above the data input (Curve). See below for further detail on the Size Axis.

- **Internal** – This is a form of axis display rather than an axis itself. Internal axes are used to create cross marks within the boundary of a map (or other display document).



The Size Scale is used for modulating the size of symbols used in a profile. The Size Scale can be displayed (as shown) controlling the symbol size and in this example, the Colour Scale modulates the symbol (and line) colour as well.

Displaying Other Objects in Profiles

Encom Discover PA can display items within Profile tracks, other than Curves. These items use the same horizontal scaling controls and are usually also related to the various lines of a survey. Example of these include:

- Sections derived from multi-banded layers

- Section grids that are gridded equivalents of sections

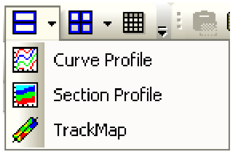
- Images associated with lines that may contain scanned or bitmap files

- Trackmaps that are derived from a 2D raster coverage of a survey

- Drillhole data

- Features that may plot anomalies and zones of interest.

Documents containing profiles suitable for displaying the above items can be created by:

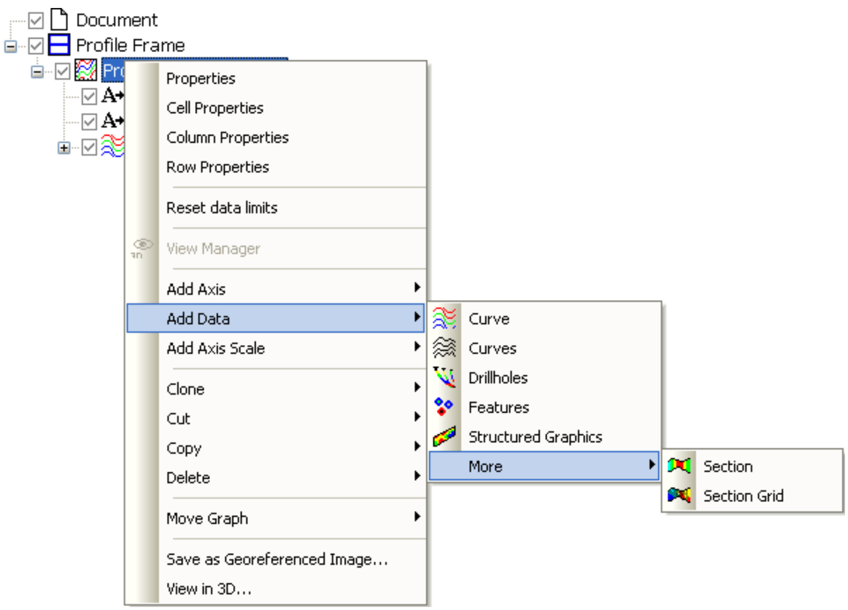


- Using the **File>New>Profile>Curve Profile** or **Section Profile** menu option
- Using the Profile pull-down button and select **Curve** or **Section Profile**.

- Use the **wizard** button and select a 2D Profile or Section.
- Any pre-existing document that has a Profile Frame available.

Once the Document containing a Profile Frame and Profile is available in the Workspace tree, the required item can be added by either:

- Highlighting the window and using the appropriate button on the Data Objects toolbar or
- Highlight the Profile branch in the Workspace tree and click the right mouse button. From the pop-up menu, select **Add Data** and the item required.



Select one of the items to add to an existing Profile branch of the Workspace tree.

For more information , see:

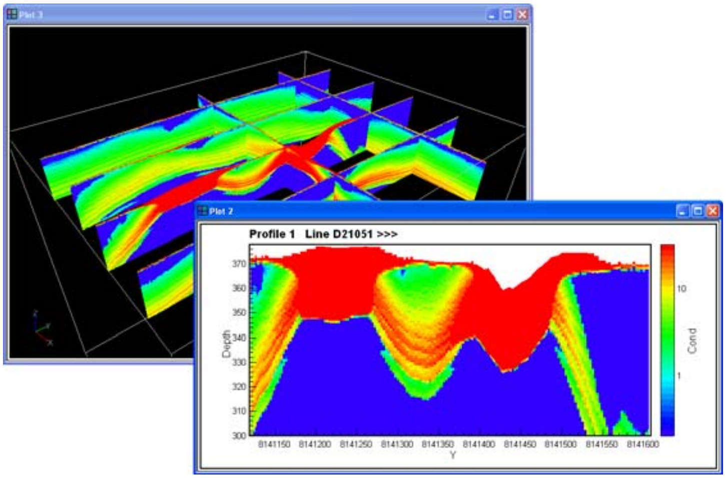
- *Adding Sections*
- *Adding Drillholes*
- *Adding Features*
- *Adding Section Grids*
- *Adding Located Images*
- *Adding TrackMaps*

Adding Sections

Section displays can be added to Profiles or they can have their own data track (using the **Add Data>Section** option). Section displays are used to present data that has property values associated with depth. Examples of this type of information are:

- Depths and seismic velocities
- Depths combined with EM inversion results
- Depths associated with models of some type

An example of sections and CDI (Conductivity Depth Image) presentations are shown below. Sections can be displayed in either one (profiles etc) or three dimensions.



Section profile displayed beneath a profile of EM data and in multiple 3D displays

Sections can be displayed individually (as Sections) or multiply in three dimensions (as Located Images). When displayed individually, each section can have individual properties. When using Located Images of the sections in 3D views, the properties (such as look-up colour and scaling etc) are applied to the original section profiles prior to creating the located images. For more information on creating located images see *Located Images in 3D Displays*.

- *Section Structure and Layers*
- *Creating Sections*
- *Section Properties*
- *Topography and Offset Field*

Section Structure and Layers

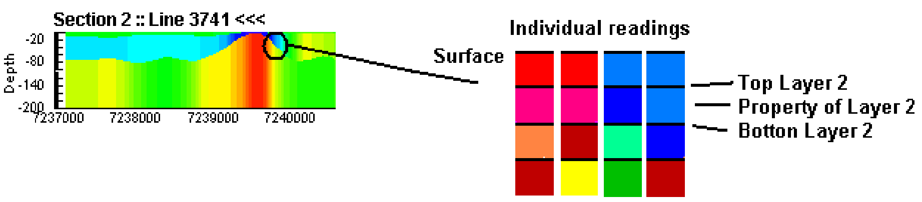
Data to be displayed as sections are derived from fields within a database that relate a property (such as conductivity or magnetic susceptibility) with depth. Different software packages provide layered/depth interpretation results in different ways. Two fundamental methods of defining layers of sections are supported by Encom Discover PA. These are:

1. Where a layer can be specified by a top and a bottom depth boundary and filled with material having a specific property. See *Upper and Lower Boundary Layers*.

2. Where a layer has a specific depth value and an associated property value. See *Point Depths and Property Values*.

Upper and Lower Boundary Layers

The presentation of layers using upper and lower boundaries requires data values to define the top and bottom layer depth. Within the depth limits is an assigned property value. The value may relate to a measured or derived quantity such as conductivity, resistivity or magnetic susceptibility. This value is scaled by a colour look-up table and then presented in the appropriate colour.



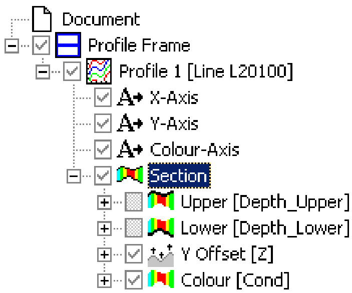
For each station, sections are specified by three primary components. These are:

1. An upper layer depth relative to a surface reading of zero (0) metres depth.

2. A bottom, or lower depth relative to the top surface depth (0 metres).

3. The property to be colour scaled.

With this section definition type, each reading requires three fields within the database to hold the required components to specify the layer. Each of these fields must be multi-banded with each band representing the data for a single layer. Note that depth is actually height – for correct display it is necessary that depth increases upwards.



Note that this method of layer definition is inefficient since the bottom depth value of a layer is usually exactly equal to the top boundary depth of the layer beneath it.

The **Upper** layer data field specification. This branch specifies the multi-banded field that contains the upper layer boundaries. Note that the number of bands in the field determines the number of layers displayed.

The **Lower** layer data field specification. This branch specifies the multi-banded field that contains the lower layer boundaries. Note that the SAME number of bands should exist in the Upper and Lower fields.

The property multi-band data field is specified by this branch. The same number of bands as for the Upper and Lower levels should be included in this field.

Point Depths and Property Values

In the second type of section definition type, the data is provided for each reading as a depth-property pair. Note that for each reading, two possibilities of this data type are possible.

- In the first type, each reading can have a constant depth with associated properties such as:

```
Band DepthBand Property
Reading 1Reading 2Reading 3Reading 4
0  3.16 4.324.72
-102.453.163.46
-202.182.893.12
etc
```

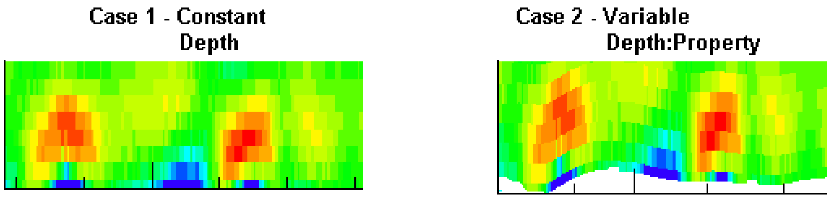
and all readings have constant depth with varying properties.

- In the second case, for each reading the depth can vary with the property:

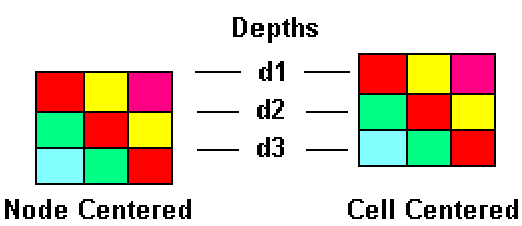
```
Band Depth:Property
Reading 1Reading 2
0, 3.160, 4.32
-10, 2.45-11, 3.16
-20, 2.18-18, 2.89
```

Encom Discover PA creates the point depth:property plot by simulating and artificially creating layer tops and bottoms similar to the case above.

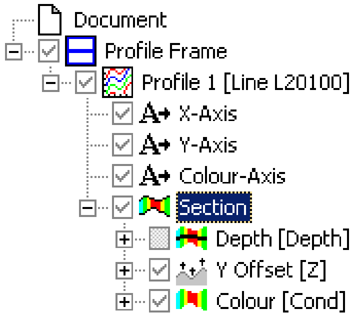
The following diagram illustrates the plotting procedure:



An additional variation on the point depth:property method of specifying sections is that each depth (beneath each reading) can be located either as a node (corner of the drawn layer cell), or the centre of the drawn layer cell. This specification is important as it makes a difference in the plotted depth of cells (by half the resolution of the data).



In the Workspace tree, specifying depth:property sections uses only two required multi-arrayed channels. These are the Depth branch and the Colour branch since the colour displays the property being displayed. The Offset branch is used for topographic corrections (see *Topography and Offset Field*).



Creating Sections

Two methods are available to create layer sections.. Their use depends on whether the sections are to be presented as two dimensional or three dimensional displays.

- Two dimensional displays use profile sections
- Three dimensional displays use Located Images of profile sections.

Both of these methods require the section to be displayed initially in a section profile.

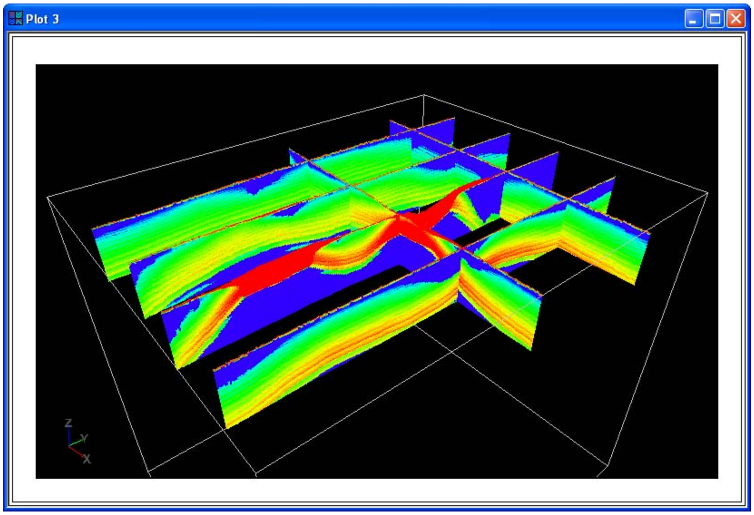
Section Profile Option

For section profile displays, the Section Profile option is available (from the **File>New>Profile>Section Profile** option or its associated toolbar equivalent). In this case, a single data line is selected and as for other profiles the Line Iterator and individual line controls are provided. For property specification of a section profile see *Section Properties*.

Located Image of Section Profiles Option

For 3 dimensional displays, the display section profile data must be set up as for the Section Option and then converted to a located image data object type before display in 3D.

The Located Images form of section creation allows individual or multiple traverse lines to be specified for display in 3D. It also means that a single line section or multiple line sections can be presented without any loss in speed efficiency.. For further information on this display type see *Section Profiles in 3D Displays*.



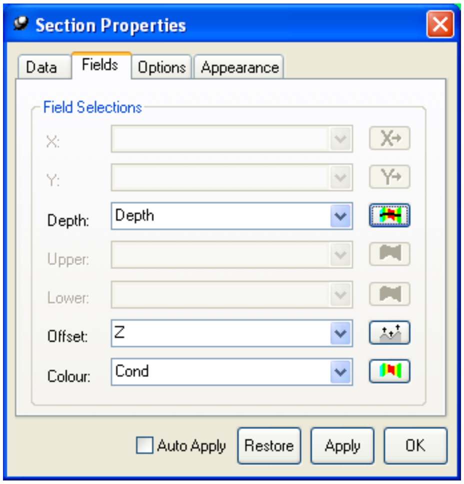
Example of the Located Image section display format

Section Properties

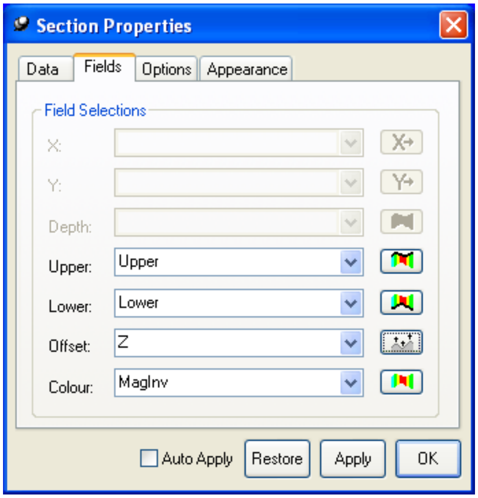
To access the properties of a Section, right click on the Section object in the tree and select Properties or double click the Section branch in the Workspace tree.

The Section property dialog has a Data tab that operates identically to the Curve Properties *Data Tab*.

Selection of data bands is from the **Fields** tab described in *Selecting Channels and Tables* or from the band selection tabs of the Upper/Lower or Depth and Colour layers of the Workspace tree. An example of the dialog for selecting Conductivity bands is shown below. Note band selection is from the buttons on the right side of the dialog.



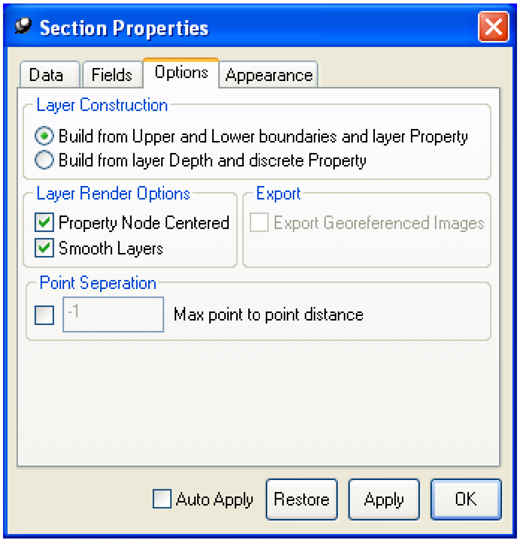
Depth and Property Layer definition field specification in the Fields tab.



Upper boundary, Lower Boundary and Property Layer definition field specification in the Fields tab.

Section Options Tab

The Options tab in the Section Properties dialog controls which layer structure is used (see *Section Structure and Layers*) and alters the Workspace tree to accommodate the specified format.



Options tab of the Section properties

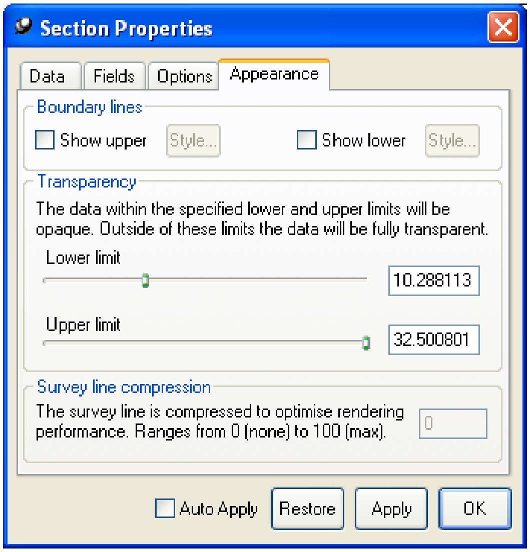
The Layer Construction specification defines whether layers are provided from the database as Upper, Lower and Property style or having Depths with discrete Properties. Depending on the mode, the tree displays Upper, Lower, Colour and Offset multi-banded objects or Depth, Colour and Offset multi-banded objects. In both cases Colour refers to the layer property. From the property pages of these objects, choose the appropriate multi-banded fields that contain the depth and property information.

Depth/Property style layers almost always are displayed using a node centred rendering approach (see *Point Depths and Property Values*). This approach assumes the property value has been computed at the depth value that is the centre of the layer.

If a property is node centred it can be smoothed. If depth varies from station to station, this will result in a more accurate rendering of the data.

Section Appearance Tab

The Section Appearance tab allows breaks between upper and lower boundaries of layers to be displayed. The breaks have a line (of a specified Line Style) drawn between the layers.



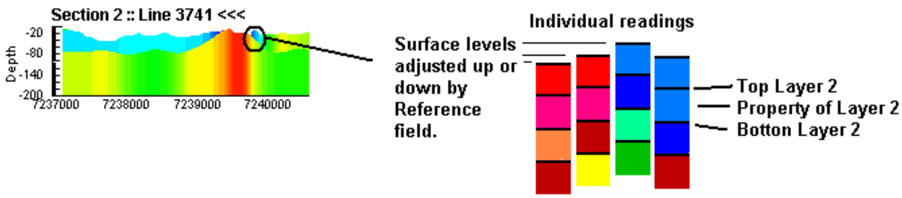
Dialog to control the Section boundaries

Initially the section is displayed so that it is opaque. This is sometimes a problem if particular trends or zones are to be viewed from one section line to another. To overcome this, you can alter the transparency of section values between a lower and upper limit. The number of bands for depth and property can sometimes be slow due to the volume of data. A level of compression can therefore be specified to assist the redraw speed.

Topography and Offset Field

A branch provided in the Workspace tree for Sections to be adjusted for topography is Offset . This branch is used to define a data field that, for every individual reading is added to the depths of the displayed section. The data value thereby offsets the position of the readings layers relative to each other and adjusts the displayed depths to account for topography, elevation or required offset to a reference level.

Tip If the offset of a data channel is in the wrong sense (that is, shifts the data up instead of down, use the Transform tab to reverse this by multiplying by -1.0)



The Offset level is usually selected to be an altitude, digital terrain model or barometer/ GPS field

Adding Drillholes

Drillholes can be added to a section within a buffer zone on either side of the associated survey line. See *Displaying Drillholes in Encom Discover PA*.

Adding Features

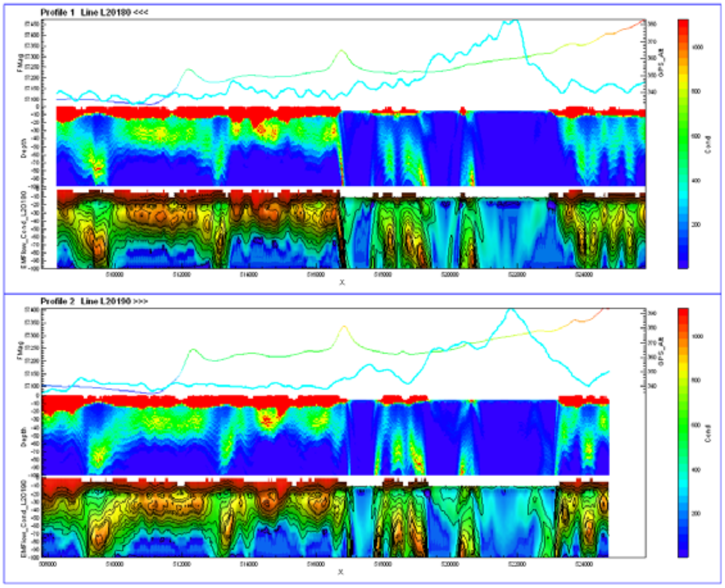
Features (points, polylines or polygons) can be added to a profile to mark locations or zones of interest. Adding Features is routinely used within sections or profiles for interpretation.

For a detailed description of using Features with Profiles and Sections, see *Managing Features*.

Adding Section Grids

Encom Discover PA can display gridded data in profiles. These items are referred to as Section Grids which are designed to display grids (in ERMapper or Geosoft format) as images. Section Grids can be used with their line association such that the Line Iterator can be used to cycle through survey data.

The Section Grid Groups can be added to documents containing either Profiles or 2D Graphs via the normal menu **Add Data>Section Grid** method. An example of a Section Grid Group is shown below:



Example of Grid Section Group with conventional section of the same data line

By displaying the grids as images, the advantages of image controls can be supported. These include shading, contouring, annotation of contours, advanced colour look-up table control etc. The property pages for the series are exactly the same as those for a standard image Grid Group with the following exceptions:

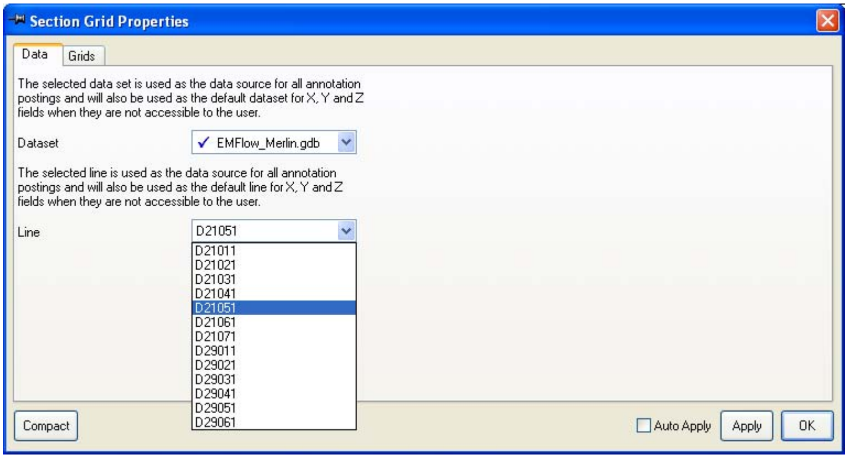
- You cannot display an ERMapper algorithm. This capability is disabled for this display format.
- You can only load special, extended ERMapper or Geosoft grid files that have line number associations – other grid formats are not supported
- In any one pseudo-colour or RGB surface you can only use one grid as a data source, although you can use different bands from that grid for each of the layers.

For more information about creating section grids and their properties, see:

- *Section Grid Data Properties*
- *Iterating Section Grids*
- *Creating Section Grid Files*
- *IP Data Display*

Section Grid Data Properties

The Section Grid also has a Data property page not seen in the conventional image dialog that displays the dataset and line number for the group. A Section Grid must be associated with a dataset/line just like any other line-based dataset in a profile. Note that you can only source data from a single line in a profile.



Data tab of the Section Grid property dialog

Iterating Section Grids

A Section Grid responds to the Line Iterator assuming the correct line definitions are specified in the extended ERMapper ERS file header. An extended header contains an information block similar to the one shown below:

```
SectionInfo Begin
   CompressedLineTrace = TRUE
   SectionXDefined = TRUE
   SectionYDefined = TRUE
   SectionZDefined = FALSE
   SectionDataset = "Swamp_EMFlow.gdb"
   SectionFilePrefix = "EMFlow_Cond"
   SectionFileSuffix = ".ers"
   LineTraceCoordinate = 0, 525178.88, 6313464, 0
   LineTraceCoordinate = 878.8970996, 524300.38, 6313441.5,0
   LineTraceCoordinate = 1431.946369, 523747.41, 6313437,0
   LineTraceCoordinate = 1958.351257, 523221.19, 6313427.5,0
   LineTraceCoordinate = 3014.05899, 522165.81, 6313446,0
   LineTraceCoordinate = 3828.409337, 521351.69, 6313433.5,0
   LineTraceCoordinate = 16771.95194, 508411.69, 6313446,0
SectionInfo End
```

This information block defines the Chordlength position, XY and Z coordinates of the line upon which the grid was created. It also records the source dataset and the grid file name. The grid file name is constructed from the prefix plus the line name plus the suffix (eg `EMFlow_L10200.ERS`, `EMFlow_L10210.ERS` etc).

When a section grid is iterated, it constructs a new grid name from this information and attempts to open that grid file from the appropriate directory. If it succeeds, the current grid file is replaced with the new. Image processing parameters and contouring parameters are retained although the colour stretch and first/last contours may be modified to reflect the data range of the updated grid.

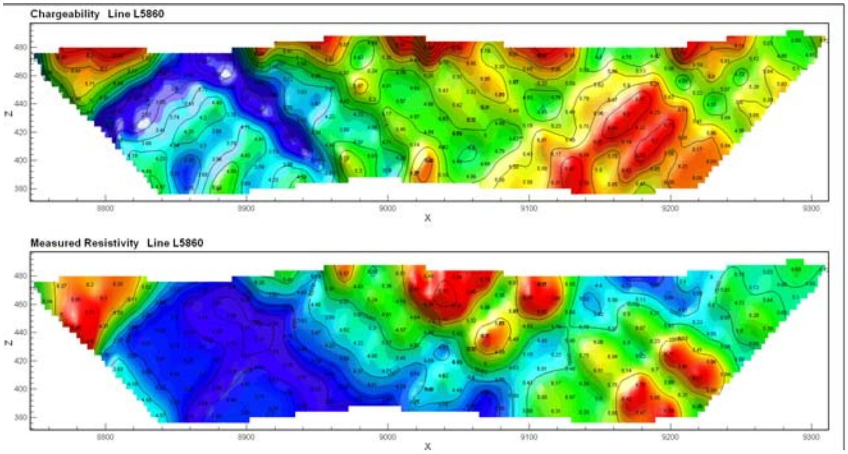
Creating Section Grid Files

The special grids and associated .ERS header file can be generated using a gridding tool that is accessed from the Grid menu option (**Grid>Section Gridding** option). See the *Section Gridding* in the Gridding Utilities section for details on creating Section Grids.

IP Data Display

Induced Polarisation data can be displayed in Encom Discover PA using the Grid Section utility. Standard Geosoft IP databases can be used. These datasets routinely have a multi-banded data field for IP, but multiple records for each data location corresponding to a different n-spacing (or receiver dipole pair).

An example of the output gridded and contoured result is shown below:



Examples of IP pseudosections with Chargeability and Resistivity

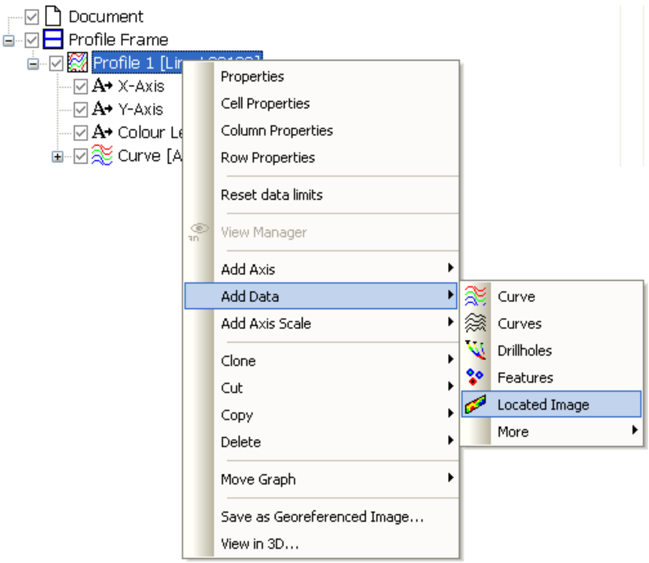
Adding Located Images

Located Images are used frequently in Encom Discover PA and these can be added to Profiles. These items are bitmaps of a particular display but with an associated file that is used to locate the image in real-data coordinates.

Located Images can be created in a number of ways:

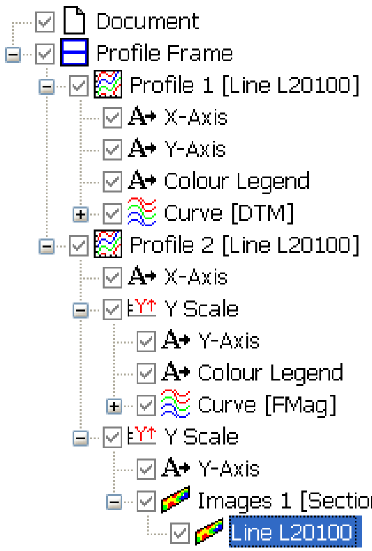
- From a pop-up menu option when a map or profile is displayed
- From the **File>Batch Hard Copy Wizard** menu option when multiple, line-oriented Located Images are required (see *Batch Printing Wizard*)
- Using the **Tools>Image Registration Wizard** (see *Image Registration Wizard*).

Detail of creating Located Images can be found in the relevant sections.

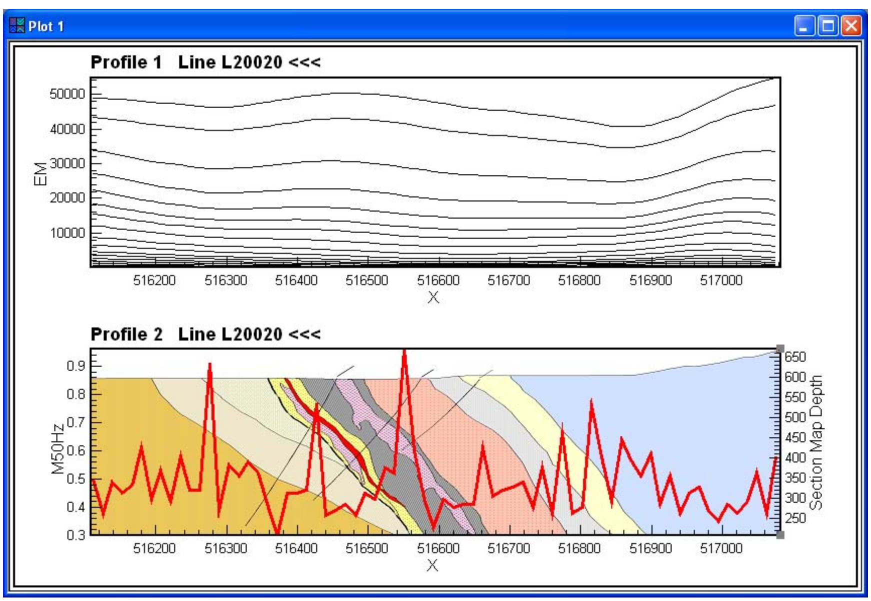


With a Located Image available, it can be added to a Profile using the **Add Data>Located Image** method. Alternatively, from a highlighted Profile branch in the Workspace tree, the Data Objects toolbar can be used with the **Add Image** button.

The Located Image can be used as a backdrop to data (usually using a separate vertical (Y) scale), or within an adjacent track.



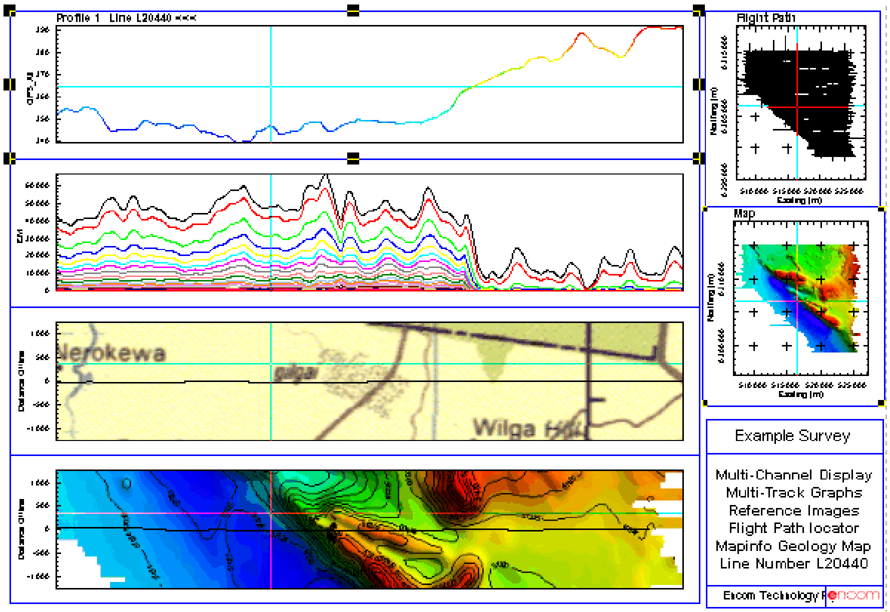
An example of a separate track Located Image plus a curve plotted within the same track is shown below. Note that in the case of the data and the bitmap within the same track, the vertical scale of the Located Image is using the section vertical data range (as defined by the Located Image corner points, but the data channel presented as a profile, uses a separate vertical Y axis. The Wokspace tree to produce this display is shown.



Example of a Located Image within a Profile with additional line data plus overlain data

Adding TrackMaps

A TrackMap is similar to the use of a Located Image, but it is created automatically from a 2D raster image. Details of the use and creating TrackMaps are described in *TrackMaps*.



TrackMap example showing the flight path location over a scanned image and over a map of the magnetic response

13 Two-Dimensional Maps

In this section:

About Two-Dimensional Maps

Encom Discover PA can display two dimensional maps that can be used to:

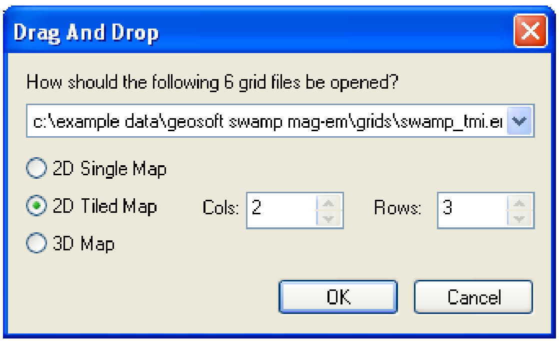
- Display flight path information for traverse lines of a database including line numbers and direction of data acquisition.
- Indicate the location of selected or displayed lines.
- Select lines for display in other windows (such as a profile window).
- Present gridded information in conjunction with the flight path maps. The grids can be displayed as shaded or unshaded images, with and without contouring.
- Display colour and symbol modulated curves and stacked profiles.
- Locate features and present their associated information for interpretation.
- Allow surfaces to be created for use in three dimensional displays.

- Interact with the user for map and profile navigation.

Drag-and-Drop Maps

A large number of file formats are available to be loaded into Encom Discover PA using the drag-and-drop facility. Wiith Windows Explorer open (or with the Explorer tab displayed in the Encom Discover PA Information Sheet) navigate to the appropriate file locations and either individually or multiply select the files. Then hold the left mouse button down and drag them into the Encom Discover PA interface window, before releasing the mouse button to drop the files into the application.

If a document window is already displayed in Encom Discover PA the files will be added to this window. If no document window is displayed an option dialog will appear allowing you to choose how to display these files. In the example of multiple grid files being loaded using the drag-and-drop facility the Drag and Drop dialog will appear:



The Drag and Drop dialog for displaying multiple grid files in a map mosaic window.

The following options are available from this dialog:

- To display a map mosaic of these grids, select the 2D Tiled map option and specify the desired number of Columns and Rows.

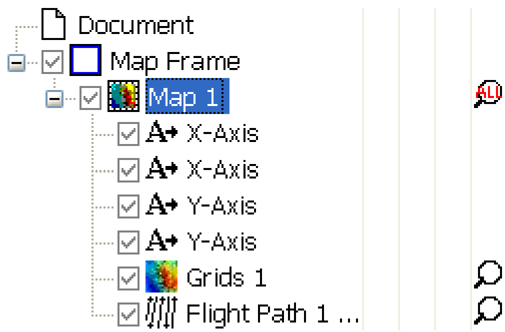
- To display all grids in one 2D Map window select the 2D Single map option. This option is desirable for producing a stacked grid display with the option of applying transparency to the upper grid layers or for using the Grid Flipper tool to alternate between the current upper grid layer.

- To display all grids in one 3D Map window select the 3D Map option. The latter option will produce a 3D map window with multiple Surface branches in the Workspace Tree each with a single grid file allocated.

Note The drag-and-drop facility is also available for importing Geosoft .GDB databases, located images, 2D and 3D vector objects and other data files.

Map Workspace Tree

When a Document containing a two dimensional Map is initially created, a simple Workspace tree is displayed. Beneath the Document is a Map Frame similar to profiles.



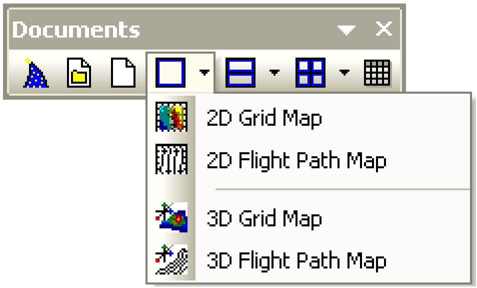
The Map Frame has properties including Options (see *Options Tab*) and Appearance (see *Appearance Tab*). The generic dialogs used to control the same features of frames in other display documents are described in *Display Properties*.

Below the Map Frame branch of the Workspace is the Map. Nearly all the properties of the Map are identical to profiles. Refer to the *Display Properties* for a description of these properties.

The properties of axes sub-branches of the tree are identical to those of profiles and are described in *Axes Tab*. Note that typically a map has four axis annotations – two for the X axis on the top and bottom and two for the Y axis on the left and right.

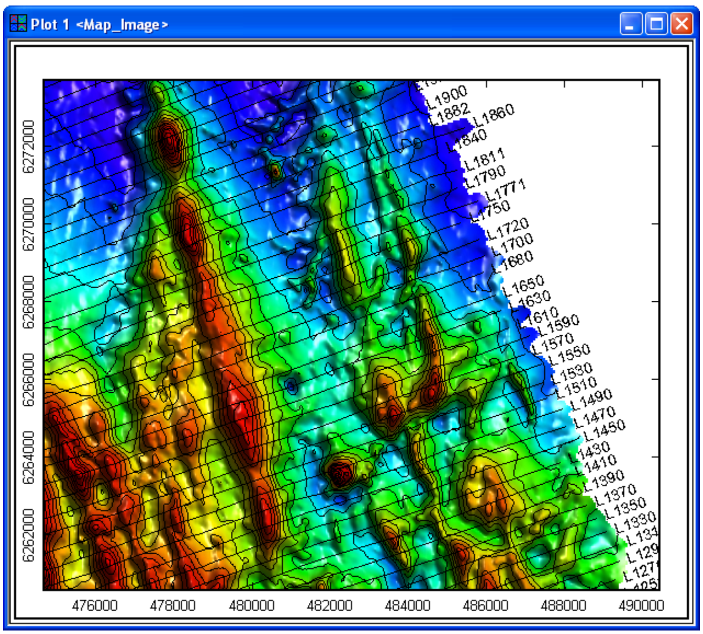
Creating a 2D Grid Map

The Grids Map format is used to display gridded or raster data derived from supported grid formats that can be displayed as images (pseudocolour or RGB) or as vectors (contours). ER Mapper algorithms (including compressed wavelet – ECW files) that access raster or grid data can also be used as sources for images.



To create a Grids Map you can use the **File>New>Map>2D Grid Map** menu option or the map button on the Documents toolbar. Once created, the displayed Workspace tree is used to configure and access the required image data.

An example of an image map (combined with a flight path map) is shown below:



A Map Document with annotated flight lines, image, contours and annotations

There are four types of surface and each has one or more layers. These types include:

- Pseudocolour with Colour and Intensity layers
- RGB surface has Red, Green, Blue and Intensity layers
- ERMapper surface contains a layer that is assigned an ER Mapper algorithm

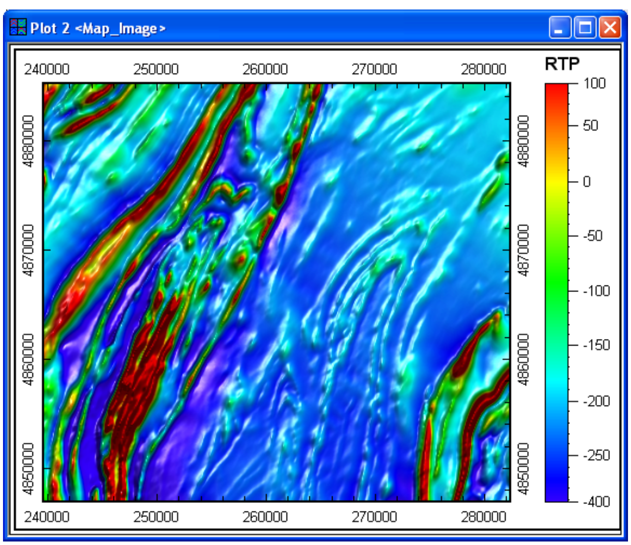
- Contour surface with a Vector layer.

For more information, see:

- *Adding Grids*
- *Querying Grids*
- *Display Order and Visibility*
- *Supported Grid Formats*
- *Located Image and ER Mapper Support*
- *ECW Image Support*
- *Vector Contours*
- *Grids Properties*

Adding Grids

To display raster or grid images in a two dimensional map, choose the **Add Data** pop-up menu to add a new Grid branch to a 2D Map (by right clicking the Map branch of the Workspace tree). When this is done a new **Grids** branch is added to the tree.



Example of flight path map with grid image and contouring

Querying Grids

Grid extents and coordinate projection information for opened grids can be viewed in the Grid Information dialog of Encom Discover PA. Please refer to *Grid Information* in Grids Properties for further information on this.

Note Control changes made in the Grids 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.

Display Order and Visibility

A plot is rendered by drawing all surfaces from the bottom level of the tree upwards. Surfaces at the top of the tree obscure surfaces lower down the tree (although the transparency control can modify this).

Surfaces can be moved up or down the tree in the plot order from the *Grids Properties* dialog. Individual surfaces and layers can be switched on or off from the tree item pop-up menu.

Supported Grid Formats

A wide range of industry-standard grid formats are supported and can be imported for use in Encom Discover PA. The grid formats supported by Encom Discover PA are listed in *Appendix E: Supported File Formats*.

Encom Discover PA loads the requested grid band(s) into memory to display images and contours of a grid. Consequently, the ability to view very large grids is restricted by your available system memory. As an alternative, to allow the viewing of very large grids, Encom Discover PA decimates the grid at load time to ensure the number of rows and columns in the grid does not exceed a user specified limit. You can set this limit in the Options dialog, accessible via the **Tools>Options** menu item. On the General tab you can set the Grid Loading column and row limit. It is set to 2000 by default.

Located Image and ER Mapper Support

The Located Image option allows display of a variety of positioned raster data formats. The raster formats supported by Encom Discover PA are listed in *Appendix E: Supported File Formats*.

Encom Discover PA makes use of an ER Mapper software plug-in that displays ER Mapper algorithms and imaging including ER Mapper Compressed Wavelet (.ECW) imagery that can be accessed across the internet. Also provided with the ER Mapper software support are a wide range of look-up tables that can be used in colouring images in Encom Discover PA.

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 correctly displayed within Encom Discover PA as map images.

Note that:

- ER Mapper images support Projection and Datum referencing and this is specified in the ASCII .ERS file accompanying the image data file. Within the header file, if the DATUM or PROJECTION is RAW and the COORDINATETYPE is LL or EN.
    - No datum or projections are written if the COORDINATETYPE parameter is not LL. In this case a UNITS parameter is written).
    - If coordinate type is LL the registration cell coordinate is written as Latitude, Longitude. If coordinate type is EN the registration cell coordinate is written as Eastings, Northings.
- Registration cell coordinate is always 0,0. If the NULL value is infinite, Encom Discover PA always records a NULL value of -5e75.

To use the Located Bitmap option and ER Mapper plug-in for algorithm presentation:

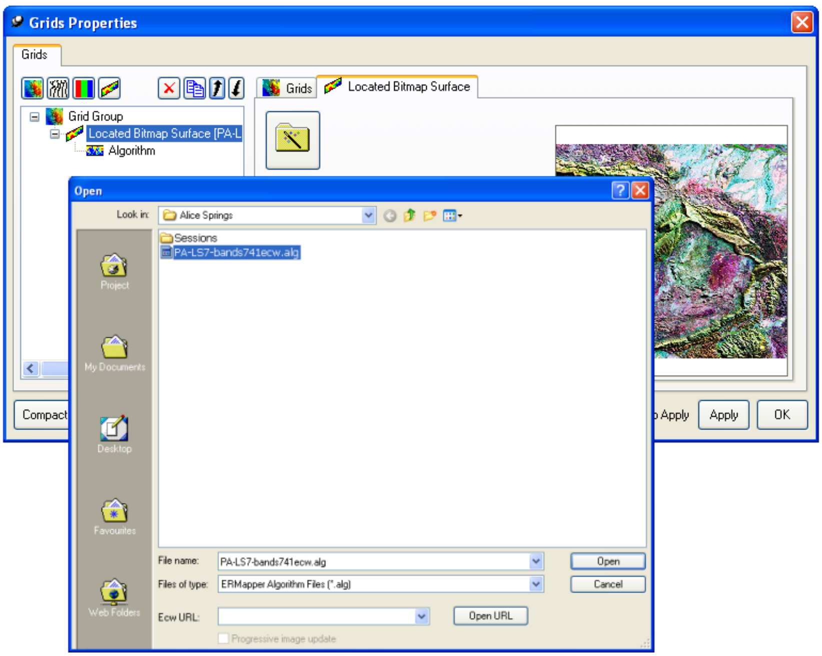
1. Create a Grids branch in the Map of the Workspace tree. Do this by highlighting the Map item of the tree, click the right mouse button and select the **Add Data>Grids** from the pop-up menu.
2. Highlight the created **Grids** branch and display the **Properties**.

3. Initially a PseudoColour layer is appended by default. Select the Grids branch in the tree and click the right mouse button to reveal a pop-up menu. Select the **New Surface>Located Bitmap Surface** option. Alternatively, click the **Add Located Bitmap Surface** button and a new Located Bitmap Surface branch is added to the Grid Group tree.

4. Click the **Browse** button and from the **Files of type** pull-down entry choose the file to be used. The formats provided in this display allow:
    - ER Mapper .ALG algorithm files

- ECW compressed wavelet format files (.ECW). See *ECW Image Support*.
- Encom Located Bitmap files (.EGB)
- ESRI triangulation files (.TIN)
- MapInfo Professional raster files (.TAB)
- GeoTIFF format files (.TIF).

5. For an ER Mapper algorithm file, select the .ALG algorithm file as shown below:



Algorithm selection and preview of the image

Once an algorithm or ECW file 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 change 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.

ECW Image Support

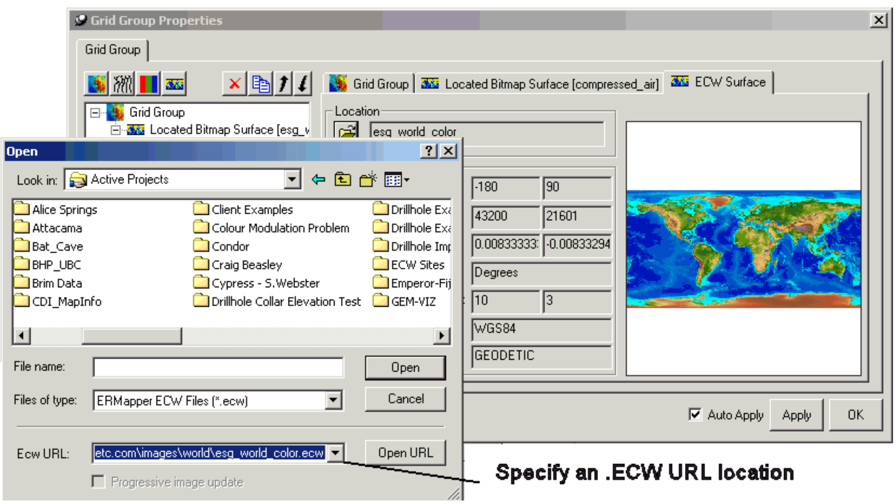
Encom Discover PA imaging can display ECW (ERMapper Compressed Wavelet) algorithms and files. ECW support can use an algorithm that specifies an .ECW file or alternatively, you can specify an .ECW file directly. ECW files can contain extremely large raster datasets and can provide access across the Internet from a wide range of sources. The images can be accessed from local websites with appropriate .ALG algorithm files and through Image Web Server (ER Mapper).

Some example ECW algorithm files are provided in the `\TEMPLATE_GALLERY\ECW` folder of your Encom Discover PA installation. To create your own ECW algorithms for use in Encom Discover PA, you need to obtain the URL of an ECW image. Some examples of URLs and freely available imagery are available at *www.earthetc.com*.

For example, you can add an ECW file definition in the Open dialog using an **ECW URL** such as:

ecwp://www.earthetc.com/mages/world/esg_world_color.ecw

Ensure you initially alter the **Files of Type** entry to be ERMapper ECW Files (*.ECW). This is shown in the example shown below:



Specify the ECW file URL address in the Open file dialog

An example of using a supplied ECW algorithm is with an image of the USA and Mexico that can be zoomed to use a resolution of 25 metres. The ECW algorithm for this image is:

ecwp://www.earthetc.com/images/geodetic/world/landsat742.ecw

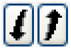
Landsat satellite image of the USA and Mexico with a spatial resolution of 25m The image is accessed in Encom Discover PA via the internet You can zoom and pan to any part of this image

Note That is not the ECW SDK which, although temporarily withdrawn for legal reasons, will continue to be available in the future.

Warning The support for the ERMapper algorithm image processing SDK has been withdrawn by ERMapper and as a consequence of this Encom Discover PA will continue to include the ERMapper data functionality but the algorithm support is no longer available by Encom or ERMapper. The ECW SDK is however not affected by this change and will continue to be available in the future.

Vector Contours

Grid files can be displayed as contour vector layers in Encom Discover PA. Contours can be displayed in conjunction with images.



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 and to ensure vectors are visible, they should lie above images in the plot tree. The **Move Up** and **Move Down** buttons (or pop-up menu options) should be used to ensure the correct order is created.

Note that **Transparency** and **Colour-Intensity Balance** slider controls are not available for contour surfaces.

To create a Vector Contour layer:

1. Create a Grid object in the Map of the Workspace tree. Do this by highlighting the Map item of the tree, click the right mouse button and select the **Add Data>Grids** from the pop-up menu. An alternative is to select the **Add Surface** button in the Data Objects toolbar.

2. Highlight the new Grids branch and display the **Grids Properties** dialog.

3. Create a new Contour surface using the **Add Contour Surface** button. Change the surface to a Vector – Contour surface via the pop-up menu for the surface. Alternatively, click the **Add Contours** button on the Data Objects toolbar.

4. Highlight the Vector branch in the tree of the Grids Properties dialog and click the **Load New Grid** button to choose the required grid file to be loaded to view the contours of (see *Supported Grid Formats*).

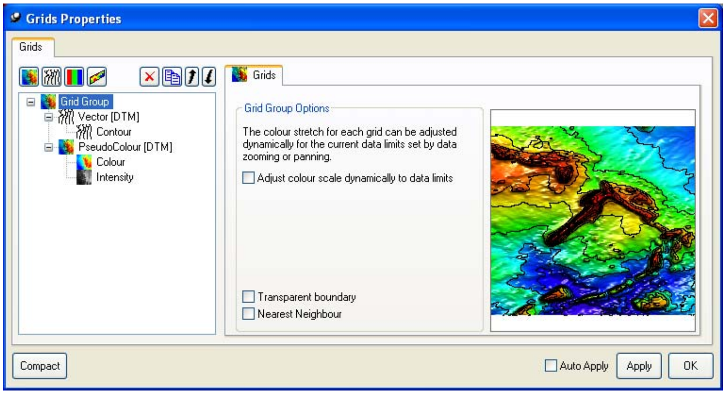
5. Once a grid is specified, select the Contour layer underneath the **Vector** branch in the tree of the grids Properties dialog to display the properties of the contours..As contours can take some time to generate, ensure that the **Auto Apply** tick box for this dialog is turned off.

For information on controlling the appearance of contours, see *Contours Properties*.

Grids Properties

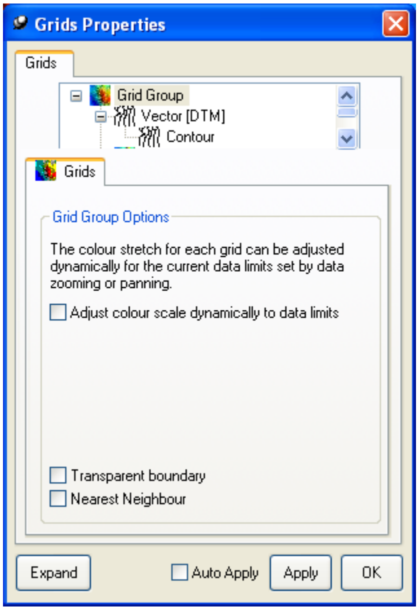
The properties dialog of the **Grids** branch in the Workspace tree is obtained from the **Properties** pop-up menu item or double clicking the Grids branch item. It is unlike other property dialogs in Encom Discover PA because it can provide previews of images before they are presented in the Map. Two modes of operation are available for this dialog, which is controlled by the **Compact/ Expand** button:

- **Expand** – The expanded view dialog is divided into an image tree, control and preview area. Although providing greater control over the displayed imagery, the dialog in this format occupies a larger area on the screen and therefore may obscure underlying imagery or workspace.



Expanded view of the Grids properties dialog

- **Compact** – The same dialog can be compressed to only display the controls and smaller image tree area. No preview area is displayed.



Compacted view of the Grids Properties dialog

A grid map consists of a variable number of surfaces that constitute a plot. The structure of the plot is shown in the tree view of the dialog. By selecting on any item in the tree, the displayed controls change to show the properties of the selected item.

For more information, see:

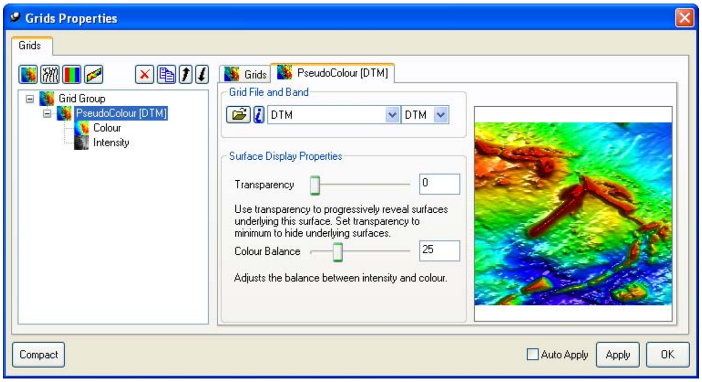
- *Transparency and Colour-Intensity Balance*
- *Pseudocolour Layer Controls*
- *Controlling a Colour Layer*
- *Grid Information*
- *Automatic Colour Scaling*
- *Controlling an Intensity Layer*
- *RGB Image Properties*
- *Contours Properties*

Transparency and Colour-Intensity Balance

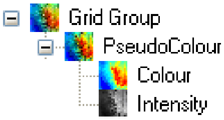
On the PseudoColour property page there are transparency and Colour–Intensity balance slider controls. Transparency is not available for contour surfaces. Colour-Intensity balance is not available for ER Mapper algorithm surfaces and contour surfaces.

Each surface is combined with the underlying surfaces using a Transparency transform that allows underlying surfaces to be seen through overlying surfaces. To ensure a surface completely obscures underlying surfaces, set the transparency value to minimum (opaque). Conversely, to reveal underlying surfaces and hide the selected surface, set the transparency value to maximum (invisible). An intermediate value combines the selected surface with the underlying surfaces to the specified degree.

Pseudocolour and RGB surfaces have colour-intensity balance slider controls on the surface property page. This allows you to easily adjust the balance the contributions of the colour and intensity layers to the final image. When set to minimum, colour and intensity are in balance. As the Colour Balance level is increased the intensity is progressively washed out and the colour saturation increases.

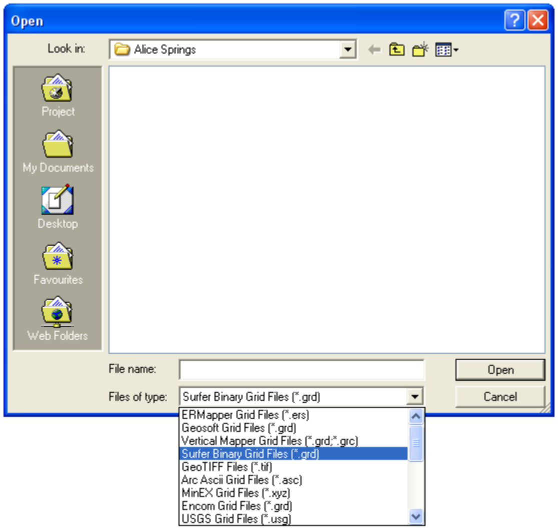


Grids Properties dialog used for control, access and previewing images



Select the button to add a Pseudocolour surface. By default, a pseudocolour surface is already present in the plot tree of a Grids. Note that **Colour** and **Intensity** sub-branches appear beneath the **Pseudocolour** branch.

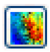
Access to external grids is provided by the **Browse** button that appears in the centre of the dialog beside a file entry field. Click this button to access a file selection window and specify any necessary path and grid surface. Loading a grid into a surface using this technique loads the grid into each Pseudocolour item. If required, grids can also be loaded into layers (Colour and/or Intensity) on an individual basis.



Grid image selection dialog.

Note in the above dialog that access to a range of different grid formats is possible from the Files of type pull-down item. Once a grid file is specified and the Open button is clicked, a preview of the image is displayed in the right of the expanded mode dialog.

A range of buttons at the top of the dialog allow the following actions:



Adds a new pseudocolour branch to the image tree.

Adds a grid branch which is displayed as contours

Adds a new Red:Green:Blue branch to the image tree.

Adds a Located Bitmap Surface file to the image tree and displays it. The supported file formats for this object type are listed in *Appendix E: Supported File Formats*

Note that in the case of ECW algorithms, images can be accessed across an internet web link.

Other buttons in the control area provide the following functions:

Adds a new grid to a plot.

Delete the selected grid branch.

Duplicate the selected grid branch.

Move the selected surface up or down.

Display information (location, extent, projection and row/column details) about the selected grid.

For interactive control of the various image levels, select a branch in the Image tree and click the right mouse button. A pop-up dialog allows you to directly add one of the supported display types or to adjust its position (for display superimposition), duplicate or remove a branch.

Similar to the Workspace tree, the image tree provides access to properties of the various levels of image plots. If you select the Pseudocolour layer and right click a pop-up menu is presented as shown.

Options provided by the pop-up menu are:

Item	Description
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 	These options duplicate the functionality of the buttons. 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 to the button.
Located Bitmap Surface	Change the surface type to a located bitmap surface such as an ER Mapper Algorithm.
Pseudocolour	Change the surface type to Pseudocolour. The Pseudocolour image representation uses a look-up table coupled with a colour and intensity controlling layers to render an image. This form of image display is the default style. Refer to *Pseudocolour Layer Controls*.

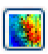
Item	Description
Red:Green:Blue	Change the surface type to RGB. This form of display uses multi-band grid files (or multiple grid files) to present images. Examples of this form of display include satellite or digital photography. Refer to *RGB Image Properties*.
Vectors – Contours	Change the surface type to Contour. Contours can be added to image maps to enhance displays. Refer to *Vector Contours* for additional details.

Pseudocolour Layer Controls

A pseudocolour surface consists of one or both of:

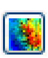
- A **Colour layer** in which a predefined RGB or HSL colour table is used to render the data from a single banded grid or a single band of a multi-banded grid. Colours can be assigned from user specified RGB or HSL colour space stretches or from colour look-up tables which can display any colour combinations including greyscales.
- 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.

To create a Pseudocolour image:



1. Create a Grids branch in the Map of the Workspace tree. Do this by highlighting the Map item of the tree, click the right mouse button and select the **Add Data>Grids** from the pop-up menu. An alternative is to select the **Add Surface** button in the Data Objects toolbar.
2. Highlight the created Grids branch and access the Properties of the image.

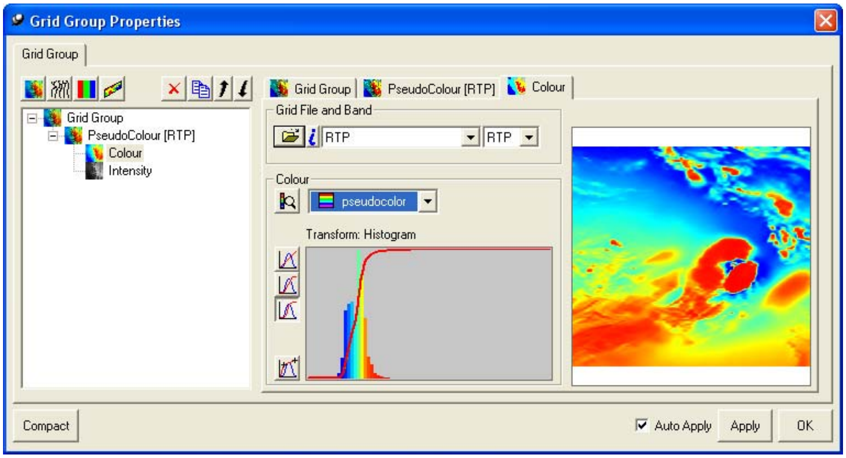
3. If required, click the button to create a new Pseudocolour surface. Select the Pseudocolour branch in the image tree and click the **Load New Grid** button to specify a grid for both the colour and intensity layers (see *Appendix E: Supported File Formats*).
4. Once a grid is specified, a Pseudocolour preview image appears in the right side of the Properties dialog (ensure the **Preview** checkbox is enabled).



5. A second (and faster method), is to simply select the **Pseudocolour** button. This automatically adds a colour level to the image tree.

Controlling a Colour Layer

Select the Colour sub-branch in the image tree to show the properties.



Colour layer controls for the Pseudocolour image

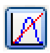
The properties available include:

- **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
  - 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. This is the default method.
  - 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.

You can select a colour look-up table from the drop-down list in the Colour section. By default, no table will be selected. If there are no colour tables in the list, Encom Discover PA has failed to identify any LUT files on your system. To change to RGB or HSL interpolation or to manually specify the number of colours in the table, click the shown button.

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 not shown on the histogram. By modifying the transform you can change the colour distribution across the image. Three options are available to do this.

Note You can use the LUT Editor to maintain, edit and create colour look-up tables. Refer to the *LUT Editor* for additional information.



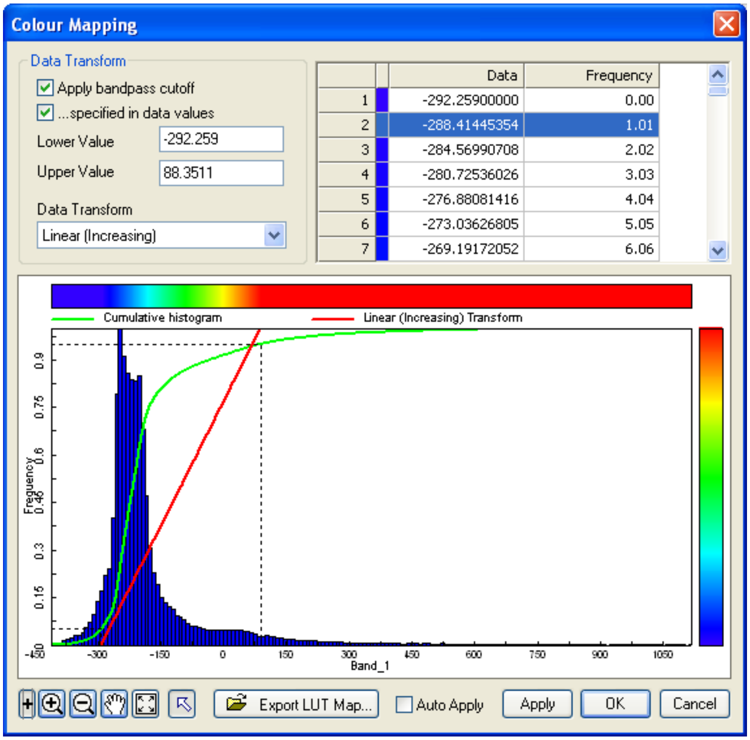
- **Linear Colour Mapping** – Click the button to enable linear colour mapping. The transform linearly distributes colour between the minimum to the maximum data values.

- **Band Pass Mapping –** Click the button to apply a bandpass filter to the transform. The percentage values of the bandpass range are specified below the bandpass button. 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. This transform produces an image with an equal amount of colour area across an image.

If you select the **Band Pass Mapping** button, you can also apply a Histogram mapping by again clicking the **Histogram** icon after setting the Band Pass limits. The histogram equalisation is computed only within the bandpass limits.

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.



The Colour Mapping dialog configured for an increasing linear 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 (which 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).

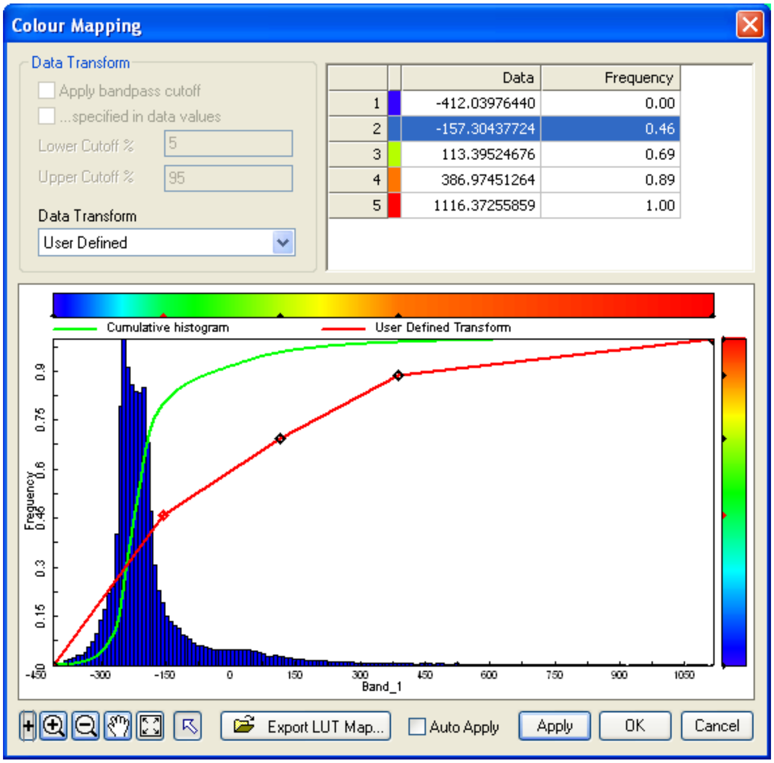
The following transform options are available in this dialog from the pull-down list:

- **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.

More advanced data transform controls are also accessed by the **Data Tranform** drop-down list. These include:

- **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.



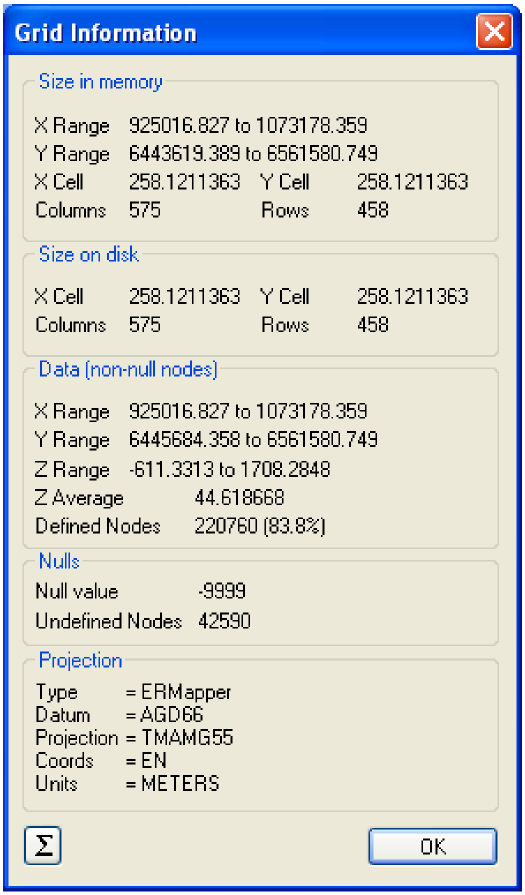
The Colour Mapping dialog configured for an user-defined data transform.

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.

If you have modified the colour look-up table using the colour mapping controls, you can then export the table (LUT) for use in other software or re-use in Encom Discover PA.

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 from selecting the button on the grid display dialog

This information is accessed from the Grids Properties dialog for a 2D map display, the section Grids Properties dialog for Profile displays or the Surface Properties dialog for a 3D display.

Grid formats supported by this query tool are:

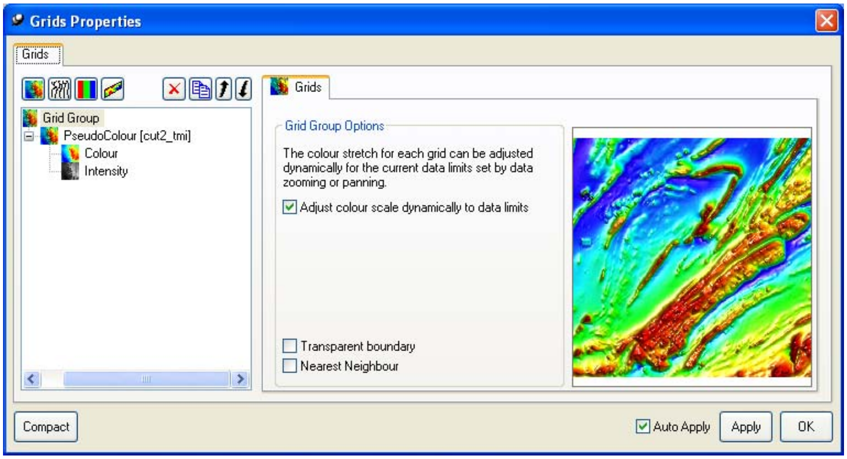
- ER Mapper (.ERS)
- MapInfo-Vertical Mapper (.GRD)
- Geosoft (.GRD) when a .GI header file is present

Automatic Colour Scaling

A powerful feature available in Encom Discover PA allows the colour look-up table to be dynamically scaled (using a dynamic histogram equalisation) within a data zoomed area of an image window. The dynamic colour scaling is demonstrated as you roam the image since the colours applied to features are re-scaled as the image is panned around in and therefore different amplitude levels of data move into and out of the window. This is especially useful when investigating subtle features within an image that has a data range that would otherwise make a feature hardly visible (such as diamond exploration).

The Automatic colour scaling is enabled from the Grids level of the image tree. Highlight this level and check the feature on.

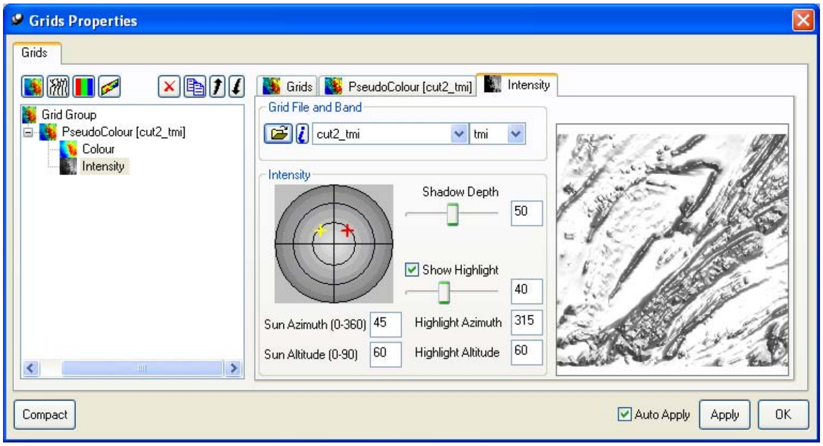
To enable the dynamic colour stretching, highlight the Grid Group branch onf the tree in the grids Properties dialog (or select the grids tab). The check box to Adjust colour scale dynamically to data limits then needs to be enabled for the feature to operate.



The dialog to enable dynamic colour stretch to a zoomed portion of an image

Controlling an 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 layer control dialog.

Properties available include:

- **Data Source** -You can select and load a grid using the 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.

- **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. 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.

- **Show Highlight** (wet look) - In addition to the red cross in the shade ball is a yellow cross. The position of this cross determines the illumination of specular highlights (changes in image gradient) to produce a wet look image. You can position the yellow cross by entering specific Highlight Azimuth or Altitude values, or alternatively you can graphically be positioned by placing the cursor over the yellow cross, pressing the right mouse button and moving the mouse to position the cross. When the cross is re-positioned, the intensity image alters to indicate the new highlighted gradients.

Note If the **Auto-Apply** option is enabled, adjustments to the shading intensity can be viewed in real-time. As you move the cursor and the sun azimuth and altitude are adjusted, so the image will alter.

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.

RGB Image Properties

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. Thus, 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 an RGB image:

1. Create a Grids layer in the Map of the Workspace tree. Do this by highlighting the Map item of the tree, click the right mouse button and select the **Add Data>Grids** from the pop-up menu. An alternative is to select the **Add Surface** button in the Data Objects toolbar.

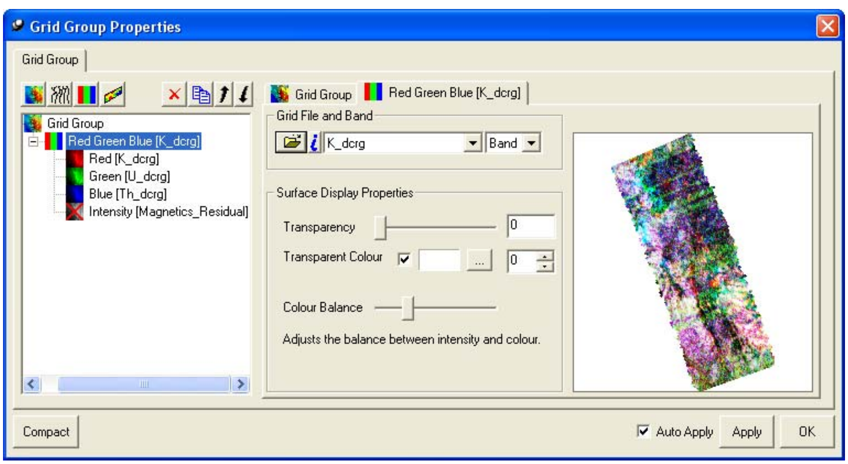
2. Highlight the created Grids branch and display the Properties.

3. Select the **Add Red Green Blue Surface** button and highlight the Pseudcolour default branch. Right mouse click in this branch and from the pop-up menu select **Delete Surface**.

4. Click the **Load New Grid** button to choose the required input multi-band grid (see *Appendix E: Supported File Formats*). Encom Discover PA assigns grid bands to the Red, Green and Blue layers by default.

5. Once a grid is specified, select the Red, Green, and Blue layers to display the properties of the relevant layer. A preview image appears in the right side of the Properties dialog (ensure the **Show Preview** checkbox is enabled).

The Red:Green:Blue layers have Red, Green, Blue and Intensity layers associated with it. By default, the Intensity layer is switched off.



Example of the Green Layer controls of a Red:Green:Blue image

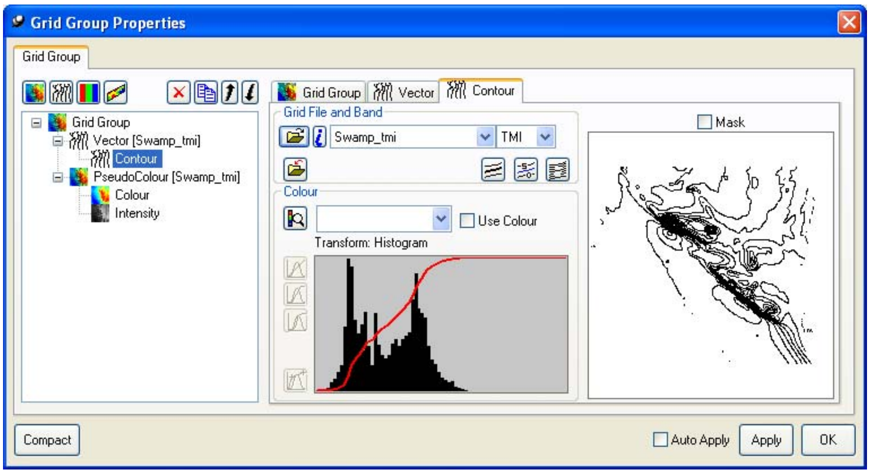
- **Data Source** -You can select and load a grid using the Load button. In most cases it is sufficient to load a grid from the RGB surface property page that loads a multi-banded grid and assigns grid bands to the red, green, blue and intensity layers. If the grid has multiple bands, the required band can be selected.

- **Branches** – Each of the Red, Green and Blue layers need to have their individual colour distributions adjusted according to a Linear, Band Pass or Histogram Equalisation process. Refer to *Controlling a Colour Layer* for details on this. In each case the look-up table appropriate to the colour layer is fixed.

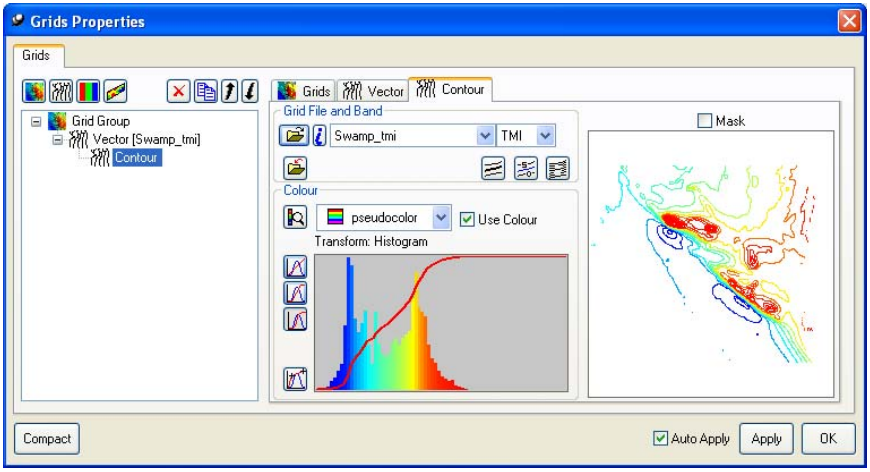
Note You can only preview the combined RGBI surface from the RGB surface property page (or the main plot window if **Auto Apply** is enabled).

Contours Properties

A large range of controls for contours is available. You can have them in monochrome or coloured (according to a colour mapping scheme) and the contour interval and dropout can be specified.



Contour control dialog with property buttons.

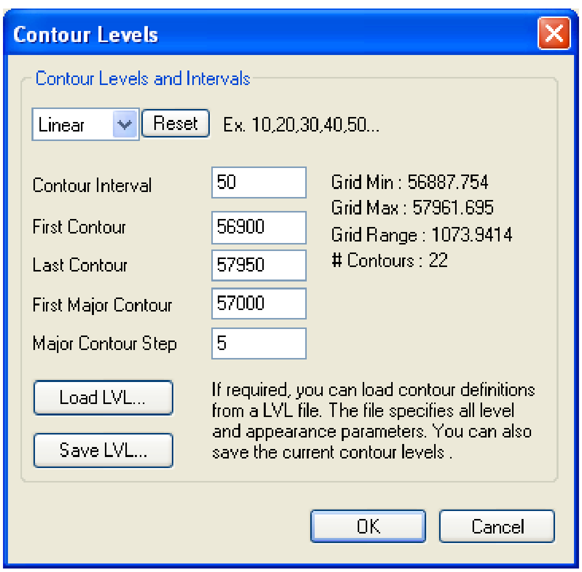


Contour control dialog and specifications for a colour look-up table.

Available controls include:

- **Data Source** -You can select and load a grid using the button. It is convenient to load a grid from the **Contour** surface property page. If the grid has multiple bands, the required band can be selected.

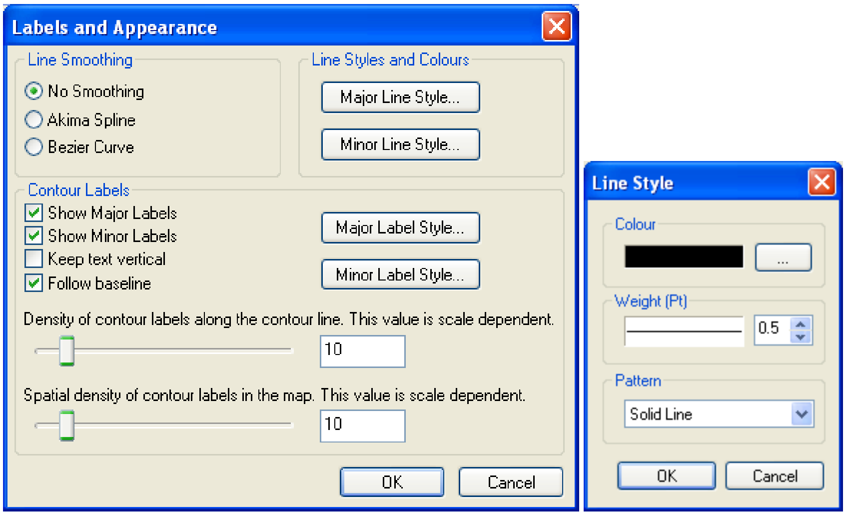
- **Contour Levels** -You can specify contour intervals, levels and their distribution from the dialog displayed when this button is selected.



Select the distribution, contour levels or save/restore from a file

Contours can be distributed linearly, logarithmically or via 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.

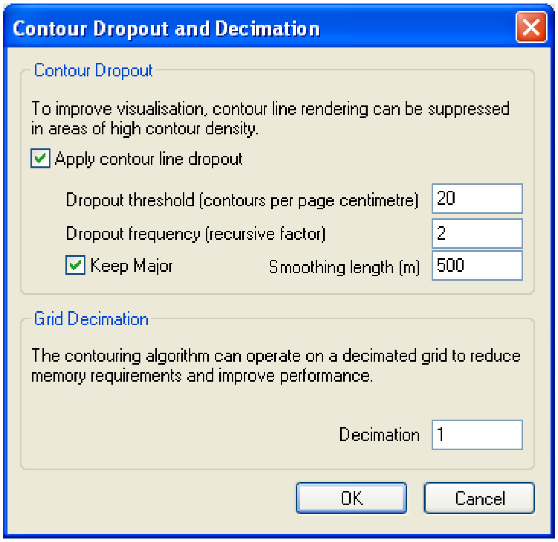


Appearance of labels of contours and line style dialog.

The line styles of major and minor contours can be defined and their smoothness determined by one of three types of contour **Line Smoothing** (No Smoothing, Akima Spline or Bezier Spline). The Major and/or Minor Labels drawn along contours can be made to have specific styles and to be either straight text or to 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 map. The dialog used to control dropout is shown here.



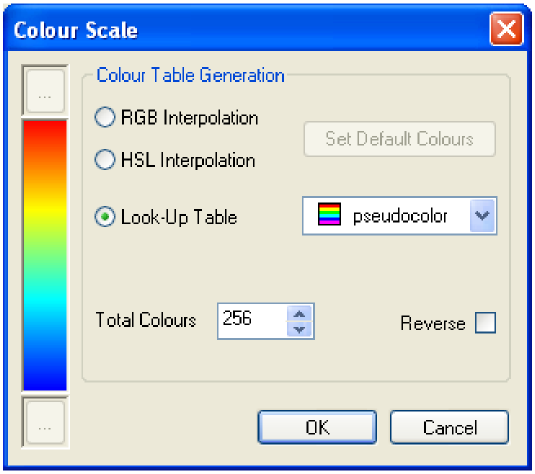
Control dialog for contour dropout and grid decimation

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 Encom Discover PA 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.
- **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.

If variable contour colour (dependent on grid values) is required, enable the **Use Colour** tick box and choose a colour scale look-up table from the pull-down list or via the **Advanced Colour Properties** dialog (shown left). A colour histogram graph is then 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 the *Controlling a Colour Layer* section.

Look-up Table – If this option is enabled, it forces the colours of the contours to be mapped by the LUT table specified rather than any histogram or band-pass settings.



The Colour Scale dialog is displayed when the **Advanced Colour Properties** button is selected.

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.

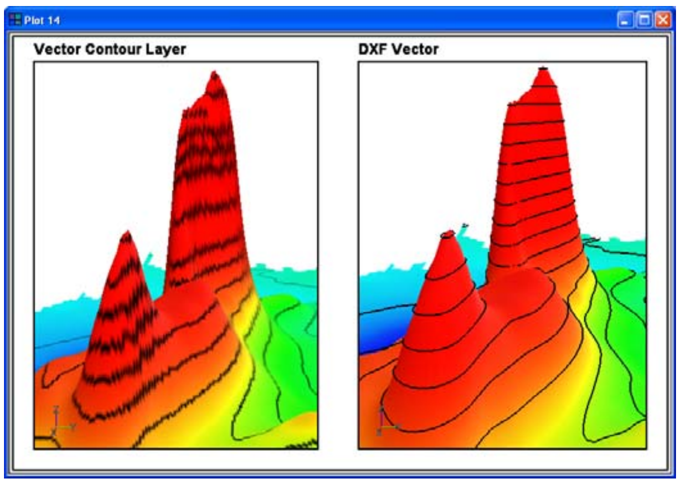
Note

In 3D imagery where contours have been added as a Grids workspace layer, steep gradients can cause poor rendering effects on contours. These effects are due to the drawn vector contour representing a single data value, passing through grid cells that may vertically span a strong gradient.

A way around this is to use the **Export Contour** button on the Surface Properties Contour dialog. Using this option can export contours created in Encom Discover PA to any of the following vector formats:

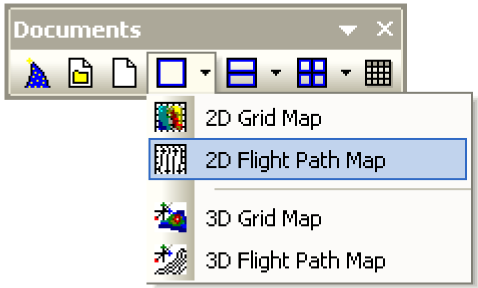
- ARCView ESRI .SHP files
- ARC ESRI .TIN files
- AutoCAD DXF format files
- MapInfo Professional .TAB or .MIF files.

Once one of these format files are saved, they can then be used (from a Vector File layer in the Map Frame) to display clean attributed contours on the 3D display. An example of this is shown below:

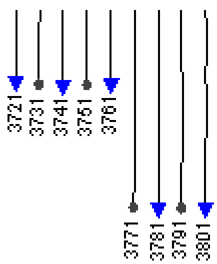


Contours added from a vector contour layer and from a saved vector file

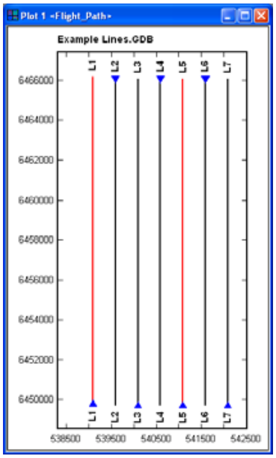
Creating a 2D Flight Path Map



To create a map (a 2 dimensional object), select either the **File>New>Map>2D Flight Path Map** or the **Flight Path** button. Maps can also be inserted into an existing document from the **Insert>Flight Map** menu item. A new window is displayed with the map and traverse lines drawnTraverse lines are initially drawn in black (by default). Line numbers and the direction of acquisition of a line can be optionally added as is indicated by a small arrow placed at the start and/or end of the line.



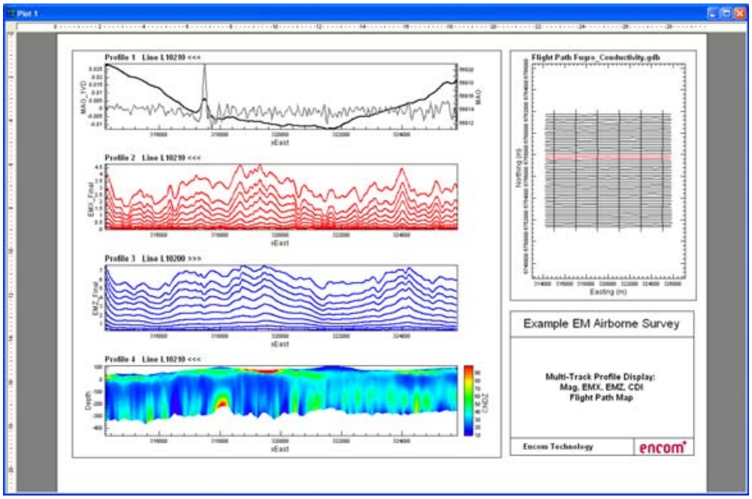
Surrounding the map are axes showing easting and northing annotations (see *Axes and Annotations*). Also note that lines selected for display in graphs or profiles from the same area covered by the map can be seen as a red, selected traverse, in the map. These selected lines are used for interactive linking of profiles and maps (refer to *Line Navigation*).



Flight map with two displayed lines

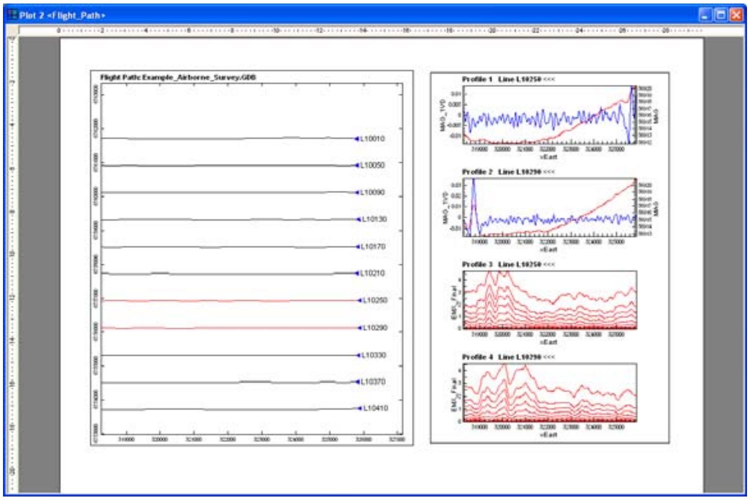
You can use the Zoom, Pan and Fit (or their data mode equivalents) to change the view of the map (see *Available Tools* for information on these functions).

Flight maps can also be inserted into other graphical objects. For example, a profile can be displayed and to indicate its location in a survey, a flight path map can be added to the display.



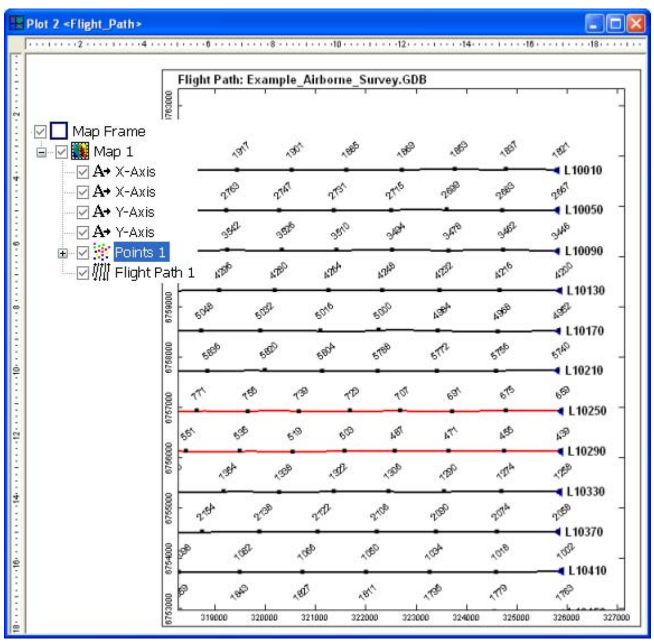
Two data lines with magnetics, elevation, EM data, inversion results, title block and flight map within a layout mode document

Alternatively, the map could be made the main focus of a display and profiles added.



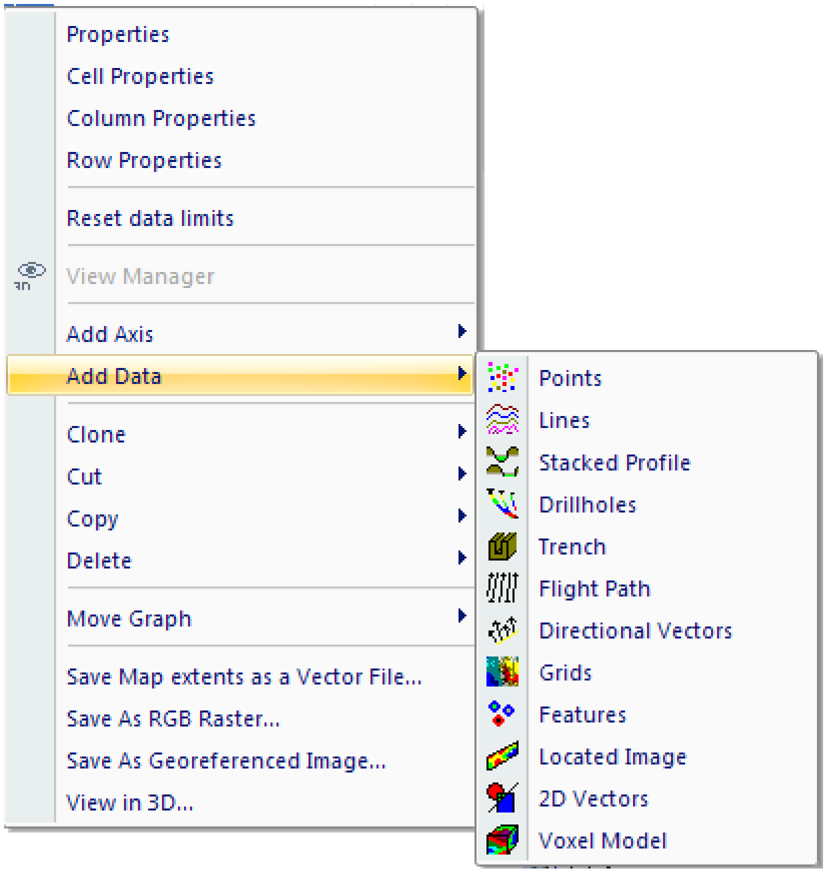
Example of Layout document with the map associated with selected profiles

If you wished to add fiducial points along the flight path, this can simply be done by adding an extra branch called Points to the Workspace tree. Use the **Add Points** button on the Data Objects toolbar or else choose **Add Data>Points** from the pop-up menu when the Map 1 branch in the workspace tree is clicked on with the right mouse button. This branch can be used to add symbols and fiducial labels with annotation at say every 50th data point (using the **Skip Factor** control).



Example of Flight Path with fiducial values using the Points data object.

The only difference between an image map and a flight path map is the data content of the map and both flight paths and images can be freely added to any map regardless of how it was originally created. Choose the **Add Data** pop-up menu to add new data to a map.



The two dimensional Flight Path and Grids branches of the Workspace tree have properties that are unique and not shared by other display documents.

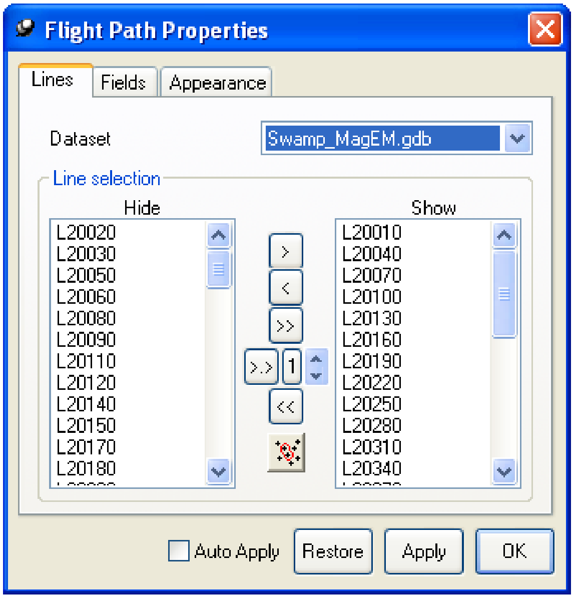
Flight Path Map Properties

To display the Flight Path Properties dialog, right-click the Flight Path in the Workspace tree, and choose **Properties**. The property dialog has three tabs:

- *Lines Tab*
- *Fields Tab*
- *Appearance Tab*

Lines Tab

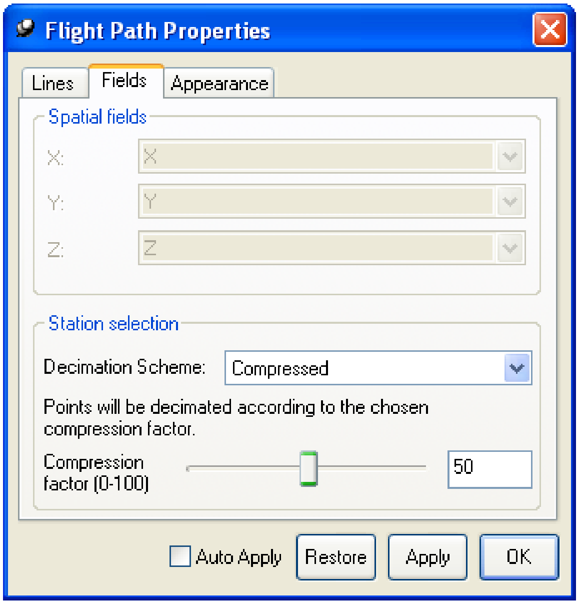
Use this property dialog to select individual lines from a nominated database currently open in Encom Discover PA. This is useful for excluding tie lines or other unnecessary flight lines from the display.



The Lines tab in the Flight Path properties dialog.

Fields Tab

Shows the fields in the database being used for X, Y and Z coordinates. Note you cannot change these in this dialog – they must be changed using the **Modify Dataset Properties**, accessed from the **File** menu.



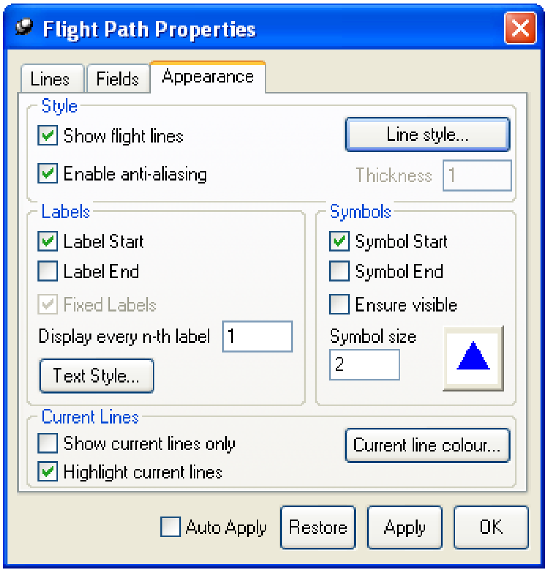
The Fields tab in the Flight Path properties dialog.

The Fields tab also allows the specification of station selection for stations along the lines. Station selection options include:

- **No Decimation** – No decimation will be applied to the station data along each line.
- **Fixed rate decimation** – A decimation of a fixed rate specified in the “Use every ‘nth’ station” box will be applied to each line.
- **Compressed** – Stations will be decimated by the specified compression factor using the slide bar or the entry in the value box.
- **First station only** – Only the first station in each line will be displayed.
- **Last station only** – Only the last station in each line will be displayed.

Appearance Tab

Use this dialog to control the appearance of the active flight lines, nominated in the Lines tab.

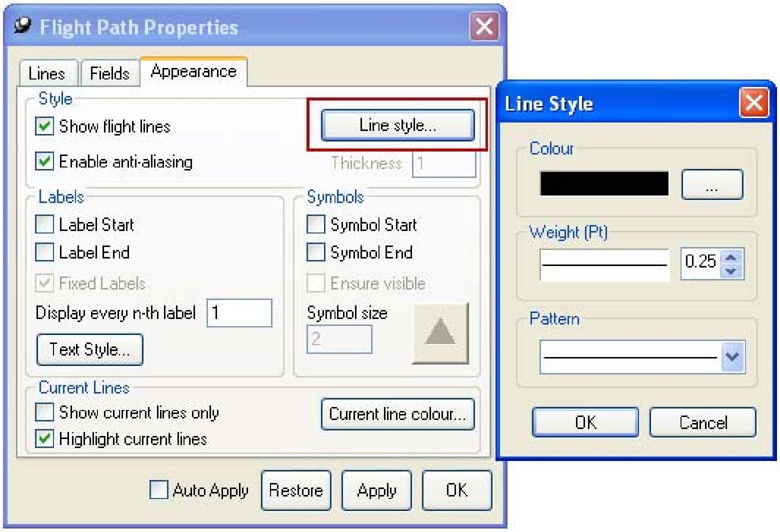


The Appearance tab in the Flight Path Properties dialog.

Appearance parameters include:

Style

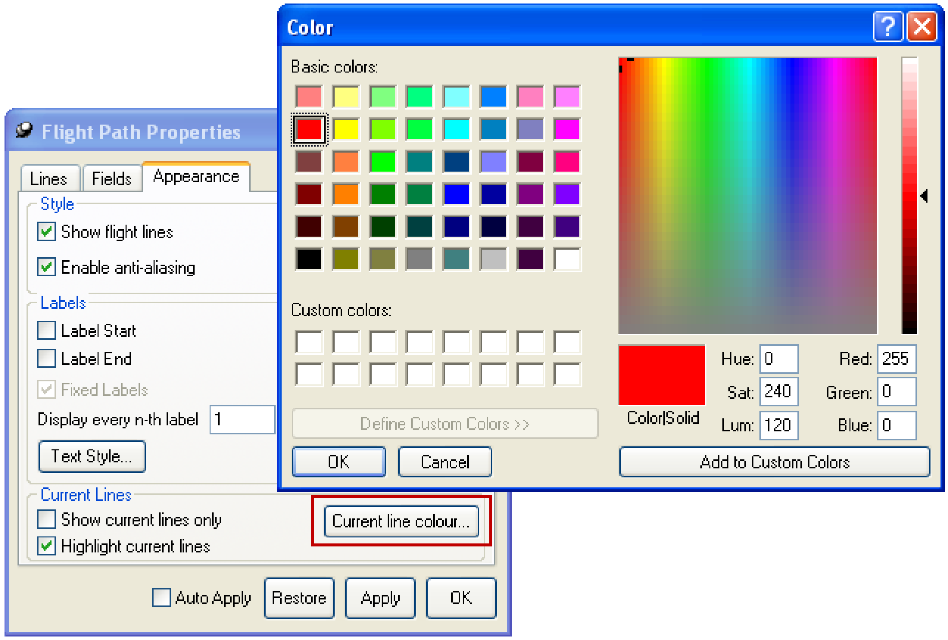
- **Show flight lines** – tick box for showing or hiding the active lines within the nominated database, chosen from the Lines tab.
- **Line style button** – click on this button to specify of the line style appearance, including colour, weight (thickness) and pattern.



The Line Style dialog accessed from the Appearance tab properties dialog.

Current Lines

- **Show current lines only** – use this option to display only the lines that are presented in a profile or graph window. Other lines specified in the Lines tab for the Flight Path Properties are not displayed until they are displayed in a profile or graph window.
- **Highlight current lines** – use this option for showing or hiding the highlight for the current lines being presented in a profile or graph window.
- **Current Line Colour** – use this button to display the Colour dialog to specify the line colour of highlighted current lines.

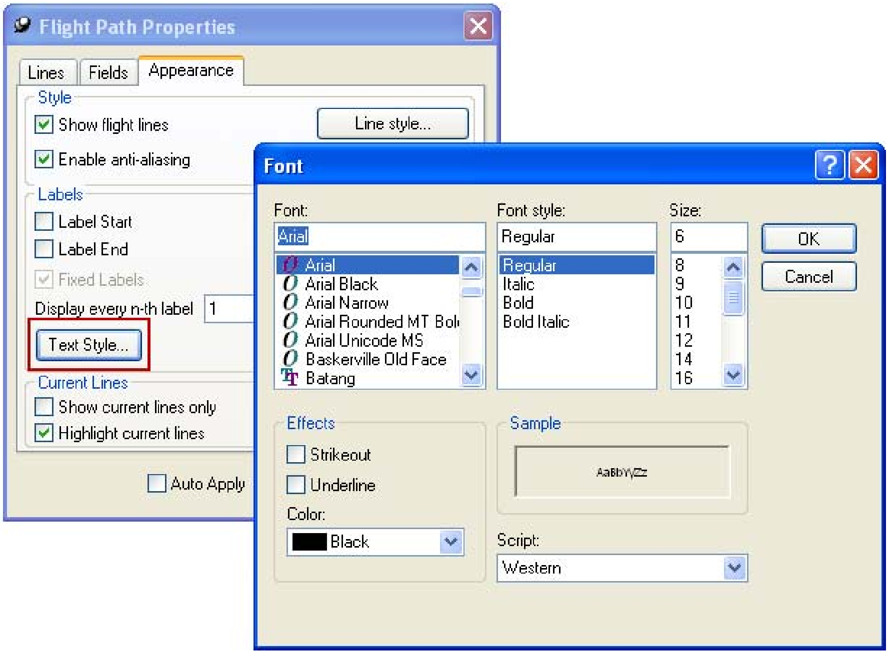


The Current line colour dialog accessed from the Appearance tab properties dialog.

Labels

- **Labels Start/Labels End** – this option places the line name that identifies the traverse at either the start and/or end of the traverse.

- **Text Style** – Use this button to display the Font dialog to control the font, colour and font style (bold, italic etc) of the line labels.



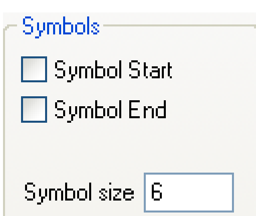
The Font dialog for line labels Text Style accessed from the Appearance tab properties dialog.

- **Display every "n-th" label** – When working with large datasets in Encom Discover PA it can be difficult to view clearly a flight path map of the survey lines with labels and flight direction symbols. The option to display every "n-th" label provides a manual drop-out of labels.

Symbols

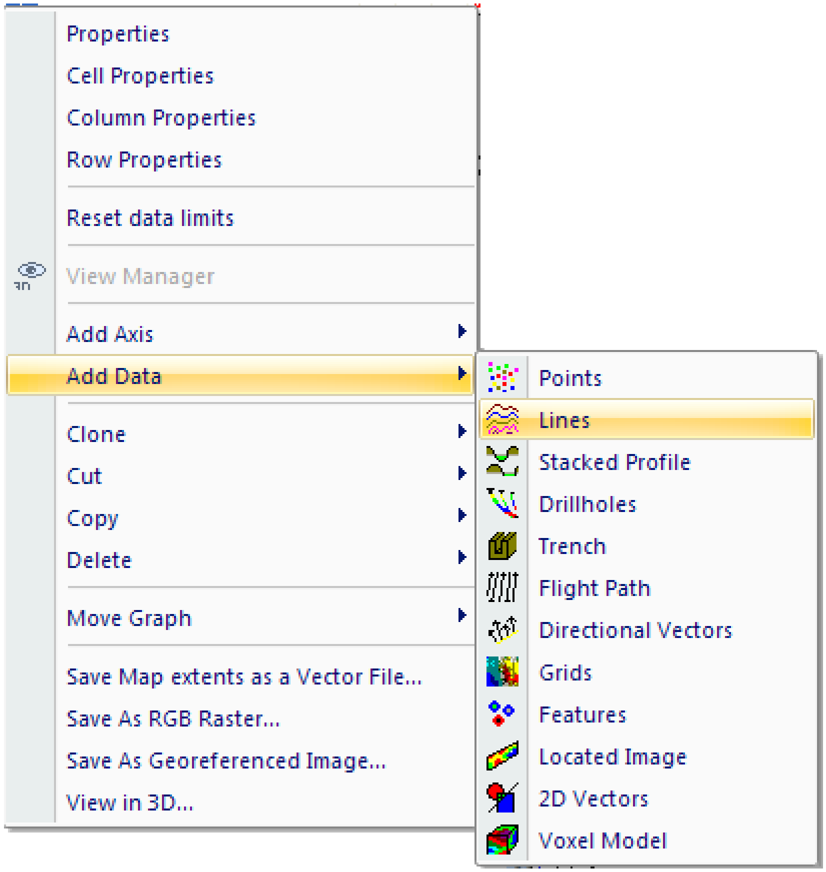
- **Symbols Start/Symbols End** - By default a filled triangle symbol is used to indicate the direction of acquisition for a traverse. Activate the **Symbols Start** and/or Symbols End tick boxes to display these.

- **Symbol size** – enter the font size (in millimetres) of the symbol used for showing the start and end of the lines.



The parameters for showing line start and end Symbols dialog in the Appearance dialog.

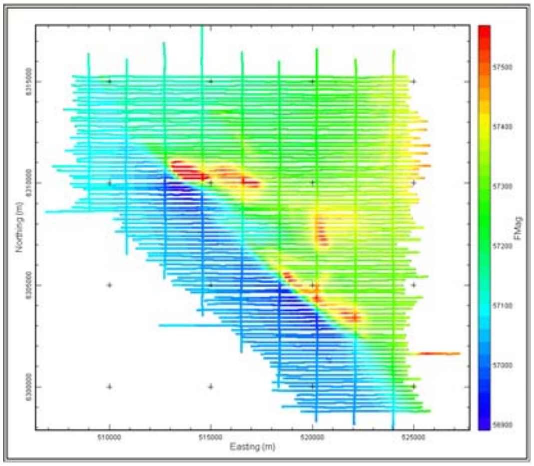
Displaying Lines



A variation of the Flight Path map is to add a Lines data object to a map. Using this display object it is possible to specify the lines required and so a subset of traverse lines can be displayed. It is useful also if, for example only tie lines or traverse lines are to be displayed.

A Lines display can also be used for presentation of data along lines. The line control has the same controls as profiles for line style, symbols, rotation and labelling. Refer to these controls for details.

An example of a Lines display is below where the Lines branch in the Workspace tree has specific line styles defined to be modulated by another data channel (for example, magnetics or altimeter).



Lines display using specific presentation applied to traverse lines

Displaying Points

Data points can be added to a map display. Points can be randomly distributed across a map area and need not be clustered into a line relationship even though they can be for ease of manipulating groups of points.

Point data can be sourced from Geosoft montaj or MapInfo Professional .TAB datasets. Once available, follow the steps below to add points to a map:

Step 1

Ensure a suitable dataset to be used for a points display is available.

Step 2

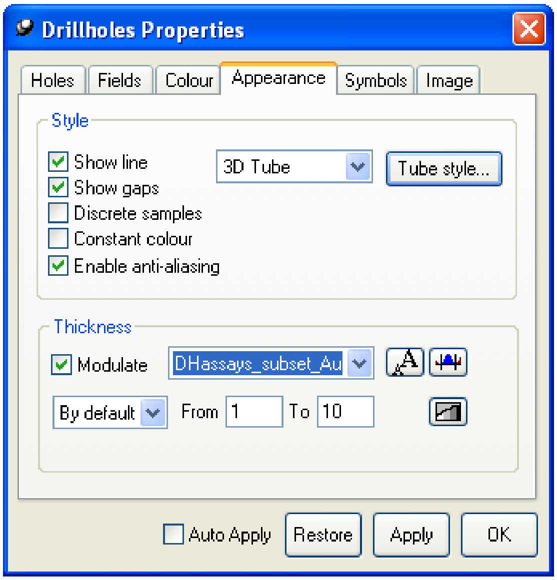
Select the map to have points displayed. Click, the **Points** button from the Data Object toolbar or use the **Insert>Data Object>Points. A Points** branch will be created in the Workspace tree.

Step 3

Display the Properties dialog of the **Points** branch. From this dialog ensure the correct data is selected from the **Fields** tab. You may need to select or deselect individual point items as required. Click the **Apply** button.

Step 4

Configure the points as necessary using the **Symbols** tab. Note that you can modulate the colour and size of displayed points according to values defined by a data channel. Different controls are available for the colour look-up, upper and lower sizes of displayed points and the units of their size.



An example of a points display with Synbols tab Points Properties dialog.

Stacked Profiles

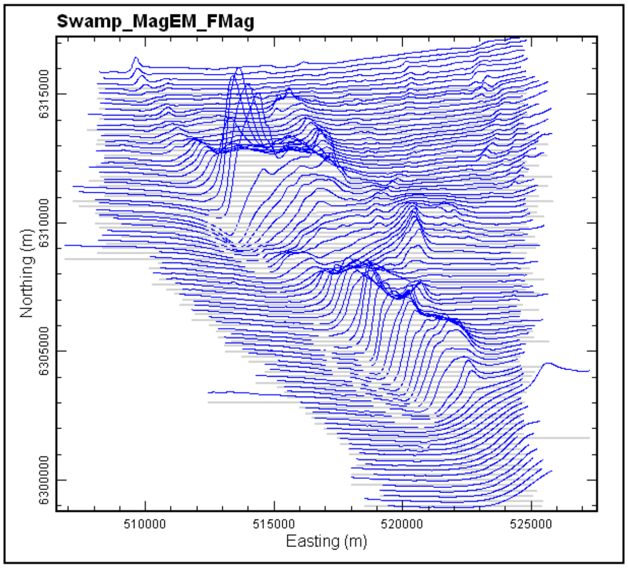
A stacked profile is a graph of a nominated channel displayed along a traverse base line. Stacked profile presentations of line oriented data are frequently used in geophysical data analysis.

Advantages offered by stacked profiles include:

- Anomaly positions can be made to lie in their correct location.
- Subtle trends and anomaly correlation from line-to-line can be extremely sensitive and often more so than as presented using contouring or imagery.
- Various filter operators can be applied to line data and optimally presented in stacked profile form.
- Stacked profiling is independent of traverse line leveling problems.

Stacked profiles created within Encom Discover PA offer the following features:

- Control over clipping (high and low) thresholds.
- Filling of profile above or below a definable baseline (a variable area presentation).
- Linear or logarithmic scaling.
- Clipping of upper and lower thresholds of data.



Example stacked profile map

Stacked Profiles can be displayed in Encom Discover PA using two different methods.

1. The **Stacked Profile tool**, accessed from the **Display Utilities** menu. This tool creates a vector file of a nominated channel from a database displayed along a traverse base line. Refer to *Stacked Profiles Tool*.

2. Within a 2D Map display, with a database open in Encom Discover PA, add a Stacked Profile data object.

Stacked Profiles created within a 2D Map Display can be sourced from Geosoft montaj or MapInfo Professional .TAB datasets. Once available, follow the steps below to add Stacked Profiles to a map:

Step 1

Ensure a suitable dataset containing line data to be used for a stacked profile display is available.

Step 2

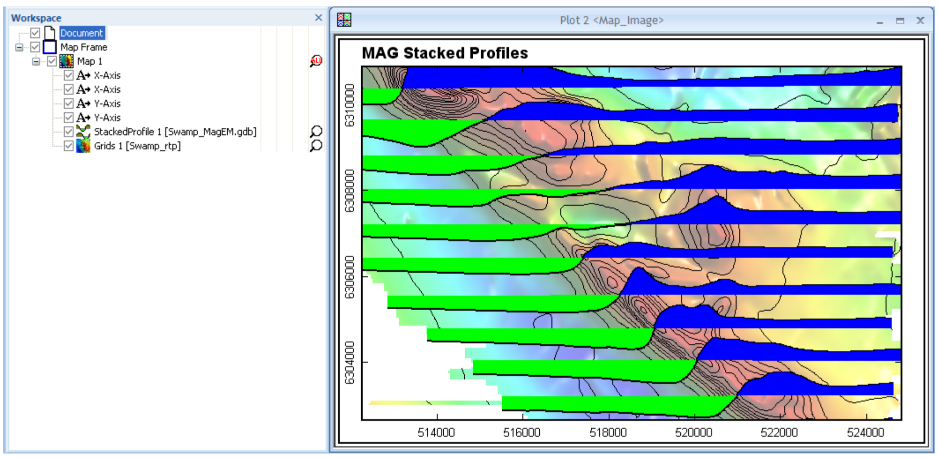
Select the 2D map to have stacked profiles displayed. Click, the **Stacked Profiles** button from the Data Object toolbar or use the **Insert>Data Object>Stacked Profile**. A Stacked Profile branch will be created in the Workspace tree for the 2D map.

Step 3

Display the **Properties** dialog of the Stacked Profiles branch. From this dialog ensure the correct data is selected from the **Fields** tab. You may need to select or deselect individual lines from the Lines tab as required. Click the **Apply** button.

Step 4

Configure the stacked profiles as necessary using the **Colour**, **Appearance** and **Fill** tabs. Note that you can modulate the colour and thickness of displayed lines according to values defined by a data channel.



An example of a stacked profiles display overlaying a grid image.

Displaying Vector Objects

Vector files can be displayed within a 2D display as a flat, planar drawing and combined with other 2D data objects, such as grids, stacked profiles and flight path lines. To display a vector file in 2D:

Step 1

Open a 2D Map display in which the Vector file is to be added.

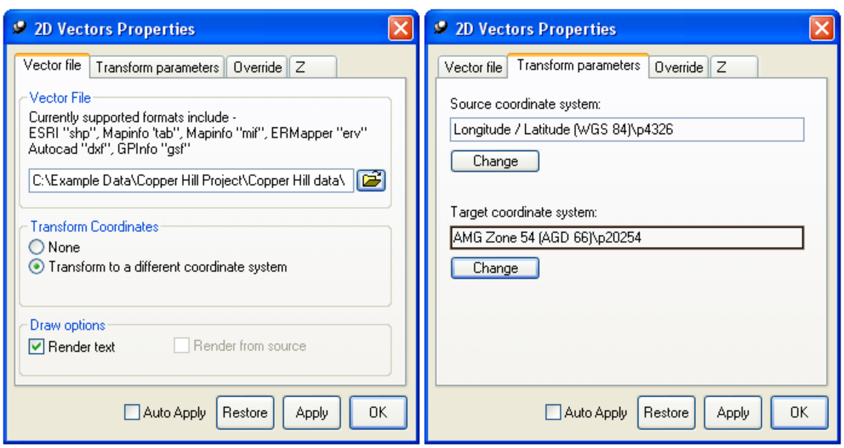
Step 2

From the **2D Map** branch of the Workspace tree, click the right mouse button to display the pop-up menu. Select the **Add Data>2D Vector File** option. Note that you cannot use the **Add Vectors** button of the Data Objects toolbar since, by default, it adds a 2D Vector branch.

Step 3

From the Properties dialog of the **2D Vector** branch, display the **Vector File** tab and navigate to the folder containing the required file. The supported 2D vector formats are listed in *Appendix E: Supported File Formats*:

If necessary, you may need to apply a transformation of the data for vector files created using a different projection from that of Encom Discover PA. Select the **Transform Parameters** tab and set the required parameters in the Transform parameters tab.

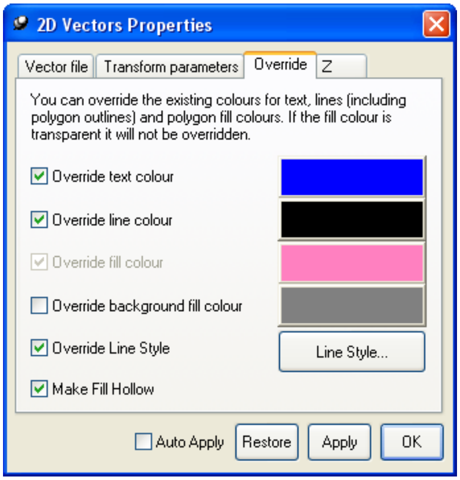


Specify the 2D Vector file and any required Transform parameters

If a coordinate system transfer is required, click the **Change** buttons on both the source and target coordinate systems.

Step 4

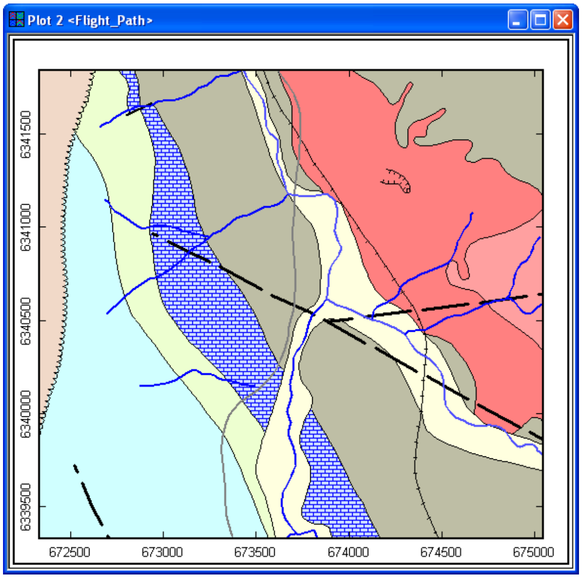
Once the 2D Vector file is specified, aspects of its appearance can be altered from the **Colour** tab. Control is provided over any displayed text, line colour, fill colour and background fill colour.



Control over the colours contained in the 2D vector file are available

Step 5

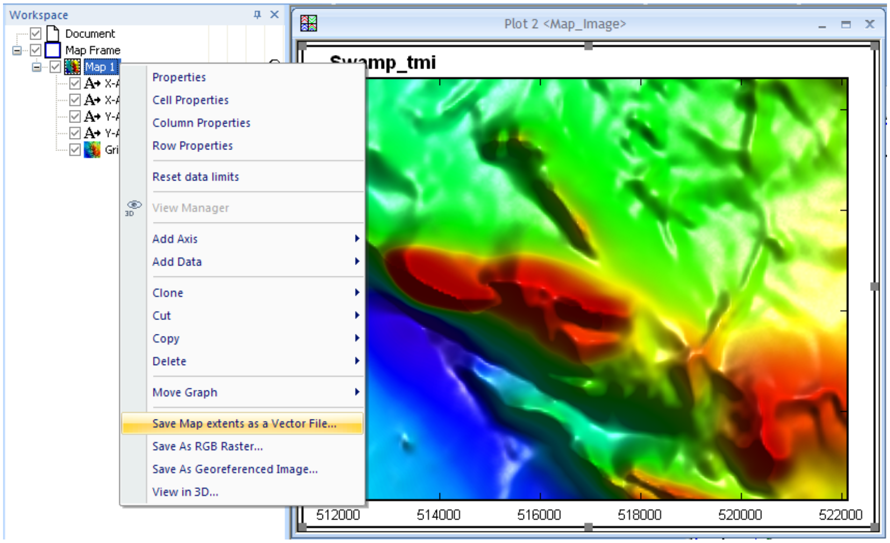
Control of the location of the 2D Vector file relative to other objects is done from the **Z** tab. This tab allows you to add or subtract a vertical offset to align the displayed 2D Vector file location as required, but in addition, the location of the 2D vectors can be made to drape over a defined grid surface. Note that the gridded surface does not have to be one already used in the display.



Multiple 2D vector files in a 2D map display consisting of surface geology, structural data and culture.

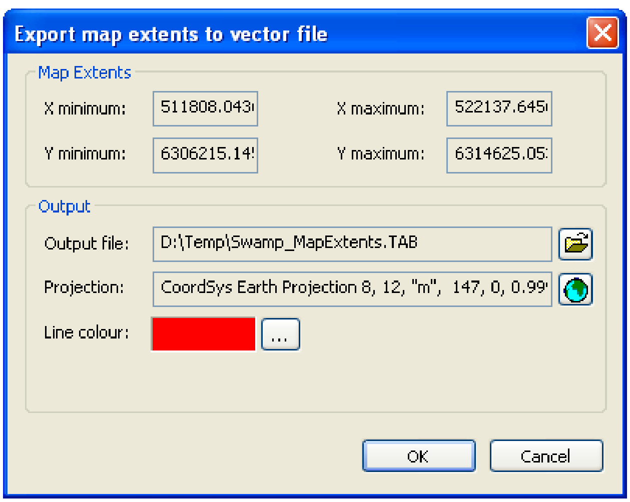
Save Map Extents as Vector File

It is often useful to be able to create a vector file consisting of a rectangular region representing a project area or an area of interest. The pop-up menu option for **Save Map Extents as a Vector File** allows a vector file of the current 2D map extents to be created. This menu option is accessed when the Map 1 branch in the Workspace Tree is clicked on with the right mouse button.



The Save Map Extents as a Vector File option is accessed from the Map 1 pop-up menu.

This will open the export dialog as shown below. The Map Extents values for minimum and maximum X and Y coordinates are automatically populated from the zoom level in the 2D map window.



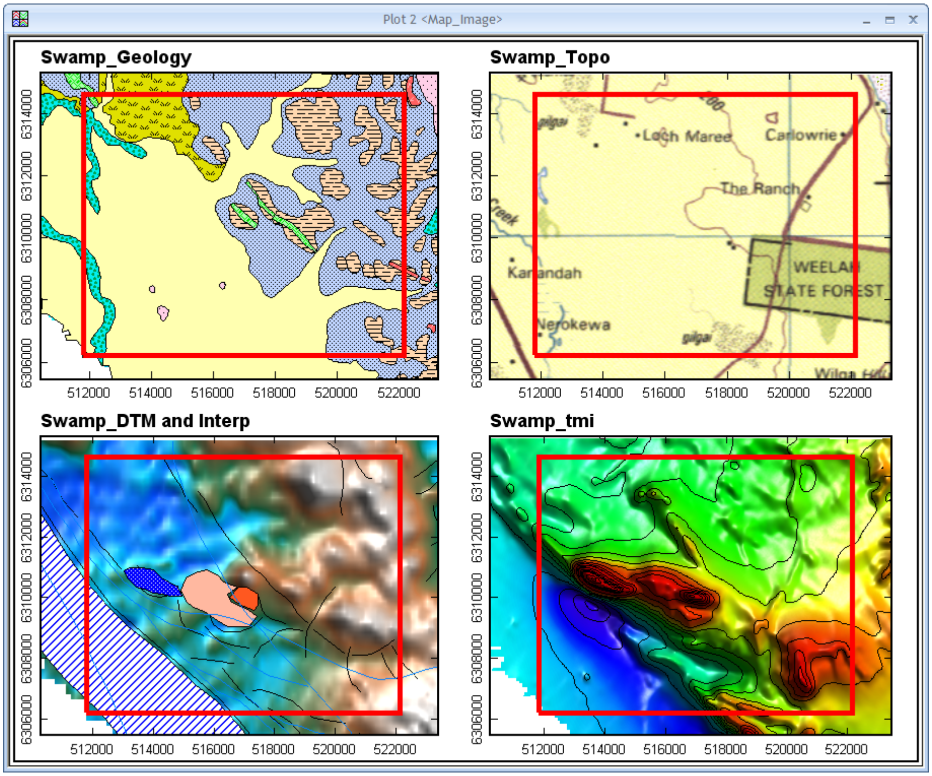
The output vector file can be created in one of the following file formats:

- ESRI Shape file (.SHP)
- AutoCAD DXF file (.DXF)
- MapInfo .TAB file (.TAB)

- MapInfo Interchange MIF file (.MIF)
- GOCAD vector file (.TS, .PL, .VS)
- Encom located image (.EGB)

Other options include the specification of the Coordinate Projection system that the output vector file is saved in and the line colour, pattern and thickness of the rectangular polygon representing the region of the map extents.

This vector file can then be loaded into one of the display window types in Encom Discover PA, namely 2D maps and 3D displays.



Example 2D map mosaic of various data objects with the map extents vector file shown in each map.

14 Three-Dimensional Displays

In this section:

About Three-Dimensional Displays

Encom Discover PA can display a wide variety of objects in 3D displays. Each 3D display is placed in a 3D Map within a Map Frame. Objects that can be displayed in a 3D Map include the following:

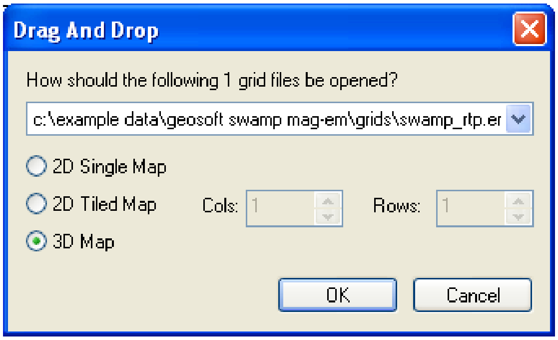
- Surfaces and multiple surfaces (grid groups).
- Flight paths
- Georeferenced located images
- Drillholes
- Voxel models
- 3D objects such as DXF vector files
- Point displays
- Multiline or single curves
- Sections and multiple sections (section groups)
- Features

Note Controlling the 3D Map display requires special commands and operation since three directions and axis are used. For details of 3D Navigation and use of the various 3D tools (e.g. 3D cursor operation), see *3D Navigation and Animation*.

Drag-and-Drop 3D Displays

A large number of file formats are available to be loaded into Encom Discover PA using the “drag and drop” facility. Wiith Windows Explorer open (or with the Explorer tab displayed in the Encom Discover PA Information Sheet) navigate to the appropriate file locations and either individually or multiply select the files. Then hold the left mouse button down and drag them into the Encom Discover PA interface window, before releasing the mouse button to “drop” the files into the application.

If a 3D document window is already displayed in Encom Discover PA the files will be added to this window. If no document window is displayed an option dialog will appear allowing you to choose how to display these files. In the example of multiple grid files being loaded into a 3D Map Window using the drag and drop facility the below Drag and Drop dialog will appear:



The Drag and Drop dialog for displaying a grid file in a 3D map window.

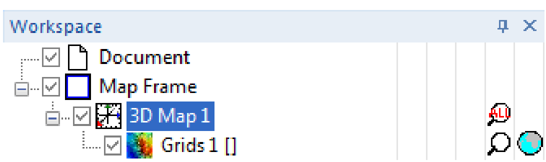
To display the selected grid(s) in one 3D Map window select the 3D Map option. This will produce a 3D map window with a Grids branch in the Workspace Tree with the grid file displayed within it. If multiple grid files are selected in the one drag and drop operation then a Grids branch for each grid file will be added to the Workspace Tree.

Note The drag and drop facility is also available for importing Geosoft .GDB databases, located images, 2D and 3D vector objects and other data files.

Chapter 14 - Three Dimensional Displays> (after Drag And Drop 3D Displays)

3D Display Workspace Tree

When a Document containing a three dimensional Map is initially created, a simple Workspace tree is displayed. Beneath the Document is a Map Frame similar to 2D Maps and Profiles.



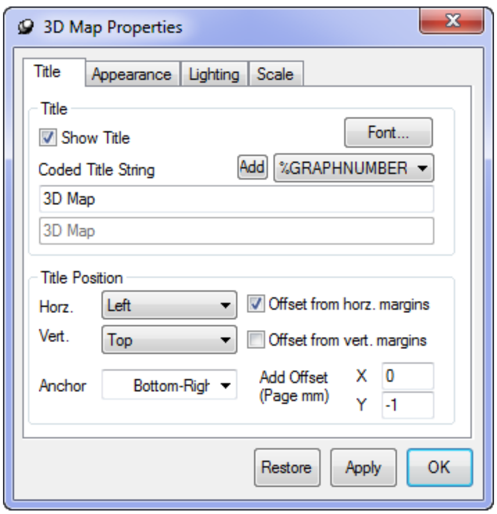
The 3D Map Frame has properties relating to Appearance (see *Appearance Tab*). The generic dialogs used to control the same features of frames in other display documents are described in *Display Properties*

3D Map Properties

Below the **3D Map Frame** branch of the Workspace is the **3D Map** branch. The options specified in the **3D Map Properties** dialog control the title, appearance, lighting, coordinate projection and scaling of the 3D Display Window. The 3D Map Properties has four tab dialogs:

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

Title Tab



3D Map Properties Title Tab dialog

The **Title** tab allows control over the positioning, style and content of a single graph title. You can show or hide the title (Show Title), select a text style and colour (**Font** button) and set the title content. The title can be located where desired. The title text can be directly typed or you can request Encom Discover PA to enter text strings as Variable Substitutes. A wide range of variable strings can be used and these are substituted for logical text (if available) at the time of drawing. A preview of the final title text is displayed beneath the coded title string as shown:

Variable Substitutes

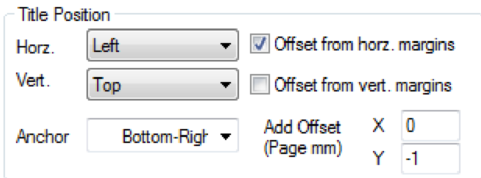
Variable Substitution assists you to label graphs by substituting text items that may be variable in nature. For example, the line name for a profile may need to be altered for each update of the profile.

Substitution strings are automatically created for you if you use the Add button to append a variable. The substitute name is enclosed in % characters. The case is not important. Many other substitution variables are also available.

Variable	Description
NAME	Data field name as reported in the name channel in the spreadsheet.
PROPNAME	Data field name derived from properties specifiers. It is the field name recorded in the properties file.
ALIASNAME	Some data channels can have aliases and these can be used to name the data presented.
DBASENAME	The database file name. the true field name stored in the database.
FULLNAME	The full field name contained in the source database. It is a constructed name combining the database name and the field.
UNITS	Units of the data channel if available—a string derived from information in the properties file.
LINE	The line name of the source profile.
BANDNAME	Data channel used as the source for the profile.
DATASET	The name of the database source.
GRAPHNUMBER	The incremental number used for each profile graph.
ANNXNAME	X axis assigned name.
ANNYNAME	Y axis assigned name.
ANNCNAME	Chord length axis assigned name.

Title Positioning

The title can be positioned anywhere in a profile area. It can be automatically positioned using the Offset switches, justification flags and text anchor. When the Offset switches are enabled, the title is positioned relative to the data plotting area. Otherwise, it is relative to the edges of the profile. For final adjustment, a manual page offset setting is available which moves the title X and Y millimetres from the specified plotting position.



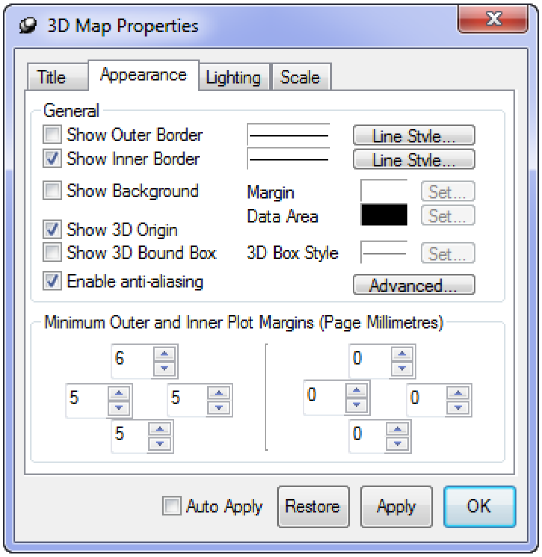
Graph title position

Appearance Tab

Each 3D Map exists within a 3D Map container (or frame). It occupies one cell of this frame (see *Working with Graphical Objects*). A frame cell has an inner margin specified in millimetres, that can be set in the **Appearance** property dialog of the 3D Map Frame. By default, this inner margin is set to zero.

Surrounding the 3D Map frame is its own outer and inner borders. The outer border is used for registration with the enclosing cell and the inner border forms the enclosing extent of the data limits. If the frame cell inner margins are zero then the plot outer border plots coincident with the frame border. For this reason, the plot outer border is not shown by default.

The appearance of the frame borders, background and margins within its frame cell are controlled by the **Appearance** property page.



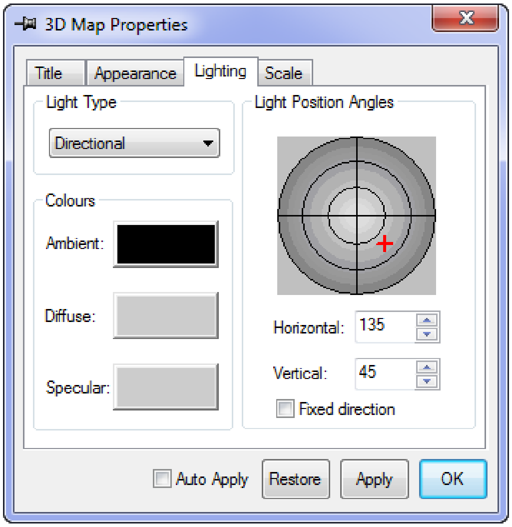
The 3D Map Appearance tab dialog

You can show or hide the inner and outer plot borders of the frame and alter the line styles of these borders. If **Show Background** is enabled, the 3D map has background colour. By definition this means the 3D Map is no longer transparent and over-plots underlying objects. You can specify the colour of the inner plot area and outer plot margins separately.

The size of the outer plot margin (specified in millimetres) is generally determined by what axis annotations are displayed in the margin. However, you can specify the minimum size of the outer margins from this property page. The inner plot margin is a redundant feature that allows you to define a margin within the plot data area. These offsets can be used in combination with the Grid Cell margins that are definable in the **Appearance** tab.

For 3 dimensional displays you can also have a **3D box** surrounding the area of the presentation.

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.

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

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.

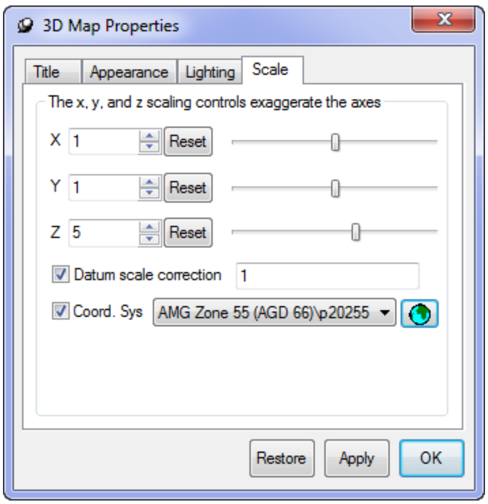
The **Light Position Angles** control contains a dynamic ellipse to define the Horizontal and Vertical positions of the directional light. The light source direction can be modified by pressing and holding down the left mouse cursor over the red cross on the ellipse and moving to an appropriate position;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



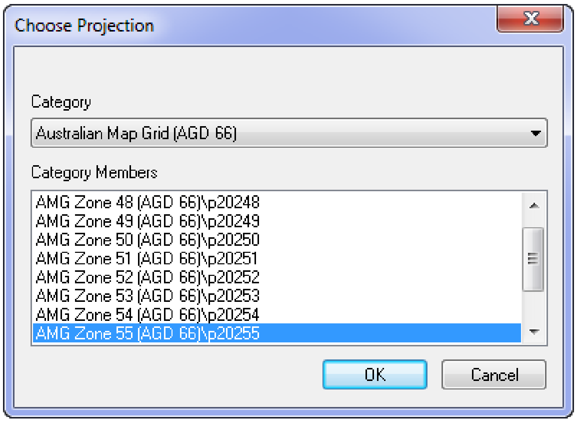
3D Map Properties Scale Tab dialog

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 **Datum scale correction** should be selected when using Latitude/Longitude datasets to equilibrate the X, Y & Z axis units. A datum scale correction is normally required with Latitude/Longitude datasets as the X and Y values are located in decimal degree units whist the Z values are located in metre units. This unit difference can make the dataset appear distorted. This can be disabled, allowing manually scaling via the Z axis control.

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.

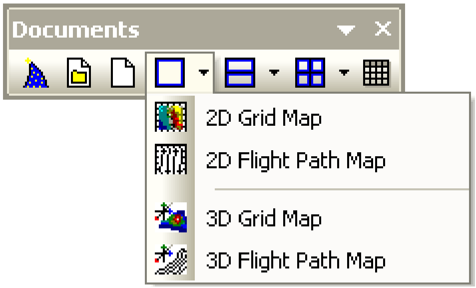
The **Scale** tab is also the location for defining the coordinate projection of the 3D display. To define the coordinate system, enable the **Coord Sys** option and then either choose a projection previously used in Encom Discover PA from the drop-down list available or else press the **Set Projection** button to select the coordinate projection system from the list of available projections as shown below.

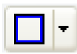


Select a coordinate projection system from the available list.

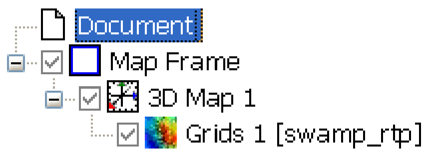
Creating 3D Displays

3D displays are created by:

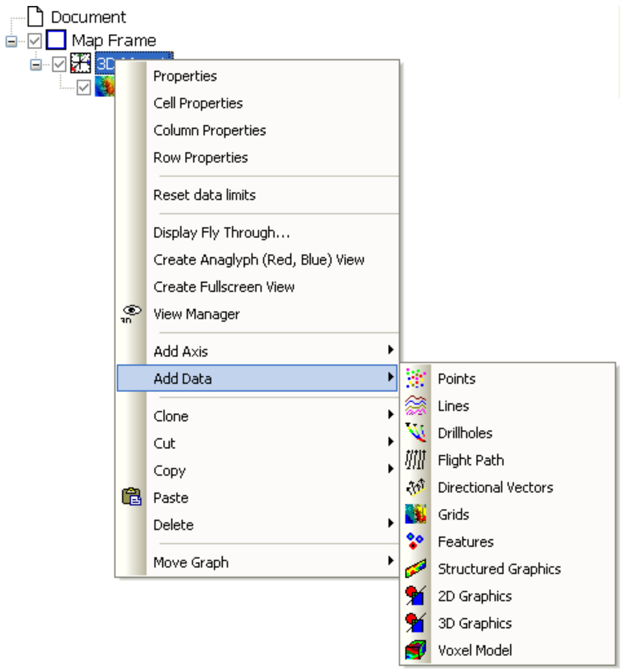




- Clicking from the Documents pull-down toolbar item for maps, or
- Using the **File>New>Map>3D Grid Map** menu item, or
- From the **New Document Wizard** and selecting a 3D Map Frame with the items required.



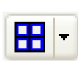
When a 3D Map display is initially created the Workspace tree appears similar to the one shown. By definition, the 3D Map has a Grid or a Flight Path branch in the tree (depending on which option you choose above).



Once the 3D Map is available in the tree, you can configure the objects within it. Alternatively you can add any of the various supported 3D display types. A list of the available types is obtained from a pull-down menu that is presented when the 3D Map branch is highlighted and the right mouse is pressed.

As you select any of the menu items, they are added to the 3D Map Workspace tree and are ready to be configured.

An alternative method of adding objects to the 3D Map is to highlight it and then click one of the object types displayed in the Data Objects toolbar.



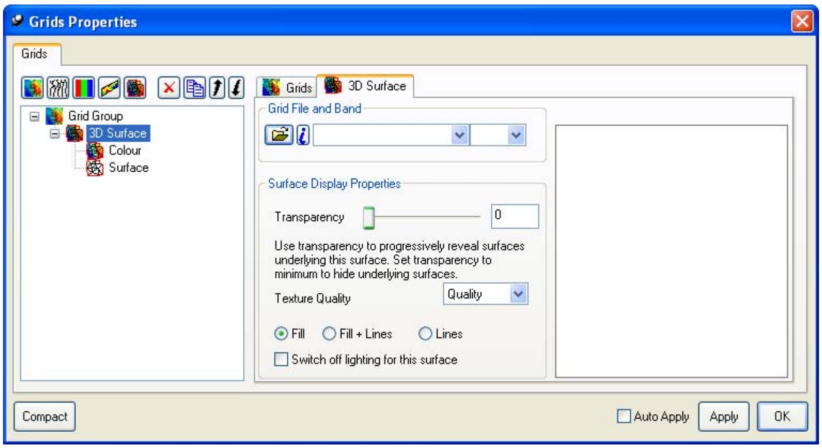
Other 3D displays operate in fundamentally the same way. For example, a 3D Graph (created from the **File>New>Map>3D Graph** option) or 3D Graph button creates a Workspace tree similar to the 3D Map, but it will be called a 3D Graph branch. A 3D Scattergram is identical and in both these cases, other objects such as Curves and Points etc can be added. The choice of objects for these object types is not as wide as for a 3D Map.

In a 3D Graph there is no restriction on the use of axes to plot 3D data within 3D space. In a 3D map there is no option to control the location axes and the defined X, Y and Z data fields are used to locate the objects refer to the Modify X, Y Z option if you need to change these fields).

Within the displayed 3D windows you can zoom in and out, roam, pan and fly-through. Intersected surfaces can be travelled through and treated as if a volume were being intersected. See 3D Navigation for information on controlling views.

View Surface Images in 3D Displays

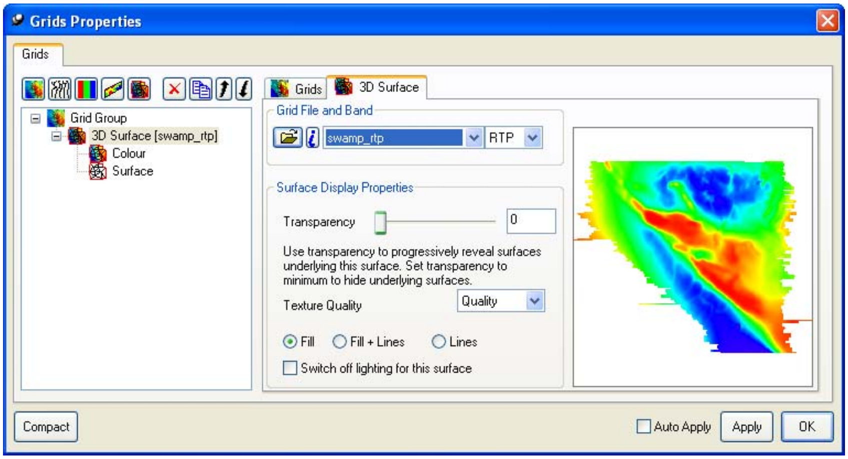
To control a surface in a 3D Map, initially a Surface branch must be present in the Workspace tree (as described above). The Surface object has the following property dialog.



Grids properties dialog for a 3D surface display

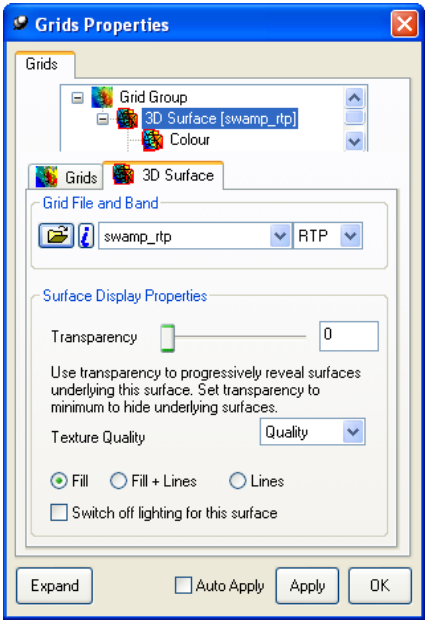
Two modes of operation are available for this dialog:

- **Expanded View** – The Expanded View dialog is divided into an image tree, control and preview area. Although providing greater control over the displayed imagery, the dialog in this format occupies a larger area on the screen and therefore may obscure underlying imagery or workspace.



Expanded view of the Surface Properties dialog

- **Compressed View** – The same dialog can be compressed to only display the controls and smaller image tree area. No preview area is displayed.



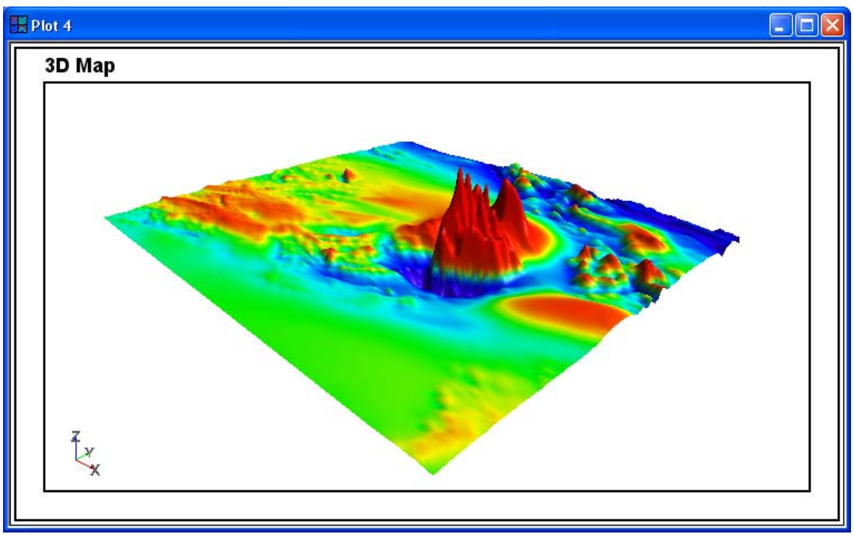
Compressed view of the 3D Surface Properties dialog

The dialog mode is controlled by the **Compact/Expand** button in the lower left corner 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.

The property dialog shows a new type of surface from a 2D dialog – the 3D Surface branch of the Grid Group tree. Specify a grid as for a 3D display into the surface using the wizard button. Click **Apply** and the surface is displayed. Note that a depth surface has both Colour and Surface sub-branches. The Colour branch is the source of colour data for the surface facets. The Surface branch is used for the surface shape and location. You can load different grids into the Colour and Surface layers. The Colour layer is controlled in exactly the same way as a pseudocolour algorithm layer.

Initially the surface is shown from one end and with low vertical exaggeration.

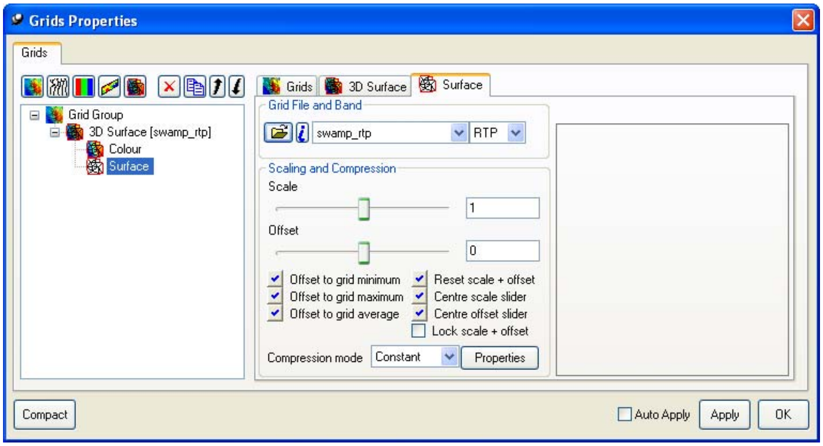


Initial view of a 3D magnetics surface

To orient the view, you may use one of two methods described in 3D Image Navigation. A look-at point crosshair in 3D views is seen when navigating. The centre look-at point and the cross-hairs are both removed as the proximity of the eye to the point reduces.

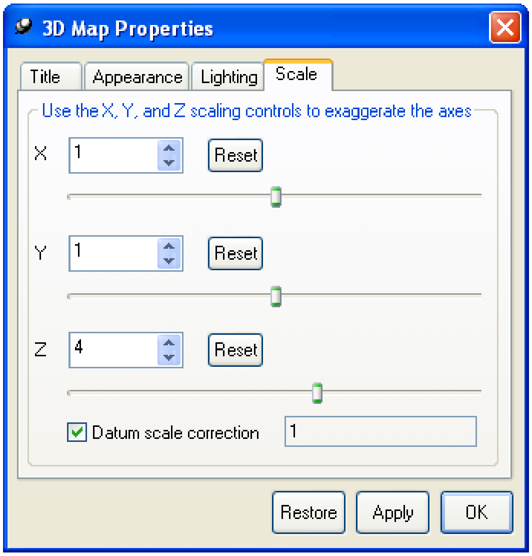
Scaling 3D Surfaces

You may wish to exaggerate the surface vertical scale by changing the scale for the Z axis. This can be done on the **Surface** tab dialog by entering a **Z Scale** and **Z Offset** to position the image vertically. Alternatively, you can use a Z Scale slider bar. Note that when adjusting a Z Scale, this applies only to the 3D Surface of that surface layer. Note too that a value entered to the right of the slider bar is the actual factor used for Scale or Offset, but it also serves as a multiplier when used by the slider bar.



Surface dialog controls for scaling, offset and compression

If you want to adjust the Z scaling of all surfaces in a view, this can be done from the **Z Scale** control box or slider bar of the 3D Map Properties **Scale** tab dialog.



Adjusting the Z Scaling control in the 3D Map Properties Scale tab dialog

Surface Compression

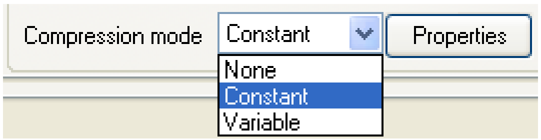
Another control on the Surface tab allows compression of grid surfaces. This feature permits Encom Discover PA to display and manipulate even large, complex surfaces quickly with little loss of surface detail. When Encom Discover PA displays surfaces, it applies a display algorithm that by default allows the presentation to be exceptionally fast but also accurate in its representation of a surface. By default, the **Variable compression** mode is used. If you wish to turn compression off, select the **None** option.

Options available for compression include:

- **Variable** - The current default which uses tomek grid compression. You can set the compression level using a slider bar. There are some options for switching off clipping if the algorithm fails.
- **Constant** - The grid is decimated. This as the default setting where the grid is decimated in three ways based on the decimation mode. .
- **None** - No compression is performed and the grid is displayed at full resolution. Care using this option is advised as this option allows no limit to memory usage.

Variable Compression

You can select the degree of Variable compression and access properties via the Properties button.

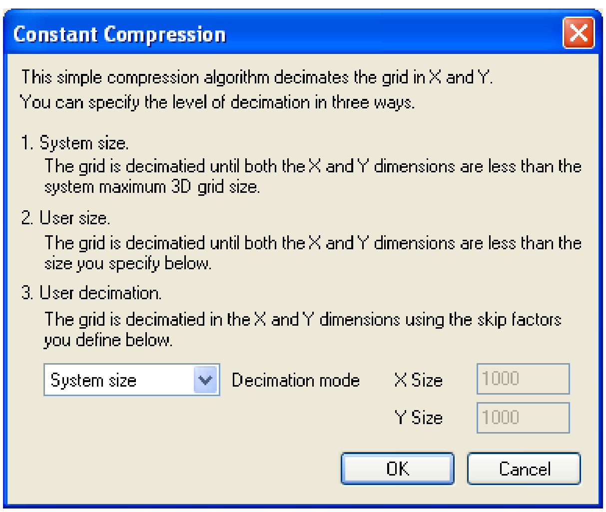


Surface compression properties controls on the Surface branch of the Surface Properties dialog.

The Clip modes are provided in an attempt to handle situations where holes or complex grid layouts can cause the variable compression to fail. Turning off the clipping entirely will resolve most circumstances.

Constant Compression

The Constant compression approach uses a simple algorithm to decimate the number of rows and columns of a grid used in a display. This approach allows fast display of the grid but may not reveal the best texturing and show all data contained.



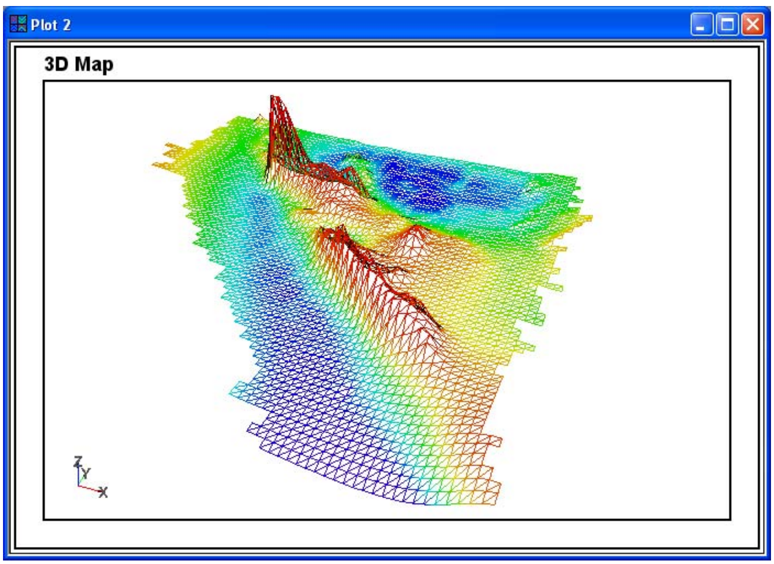
Constant Compression dialog to specify the required algorithm

The Decimation mode options available to Constant compression are:

- **System size** - The grid is decimated until it is smaller than the system settings. These settings are defined in the options property sheet.
- **User size** - The grid is decimated until it is smaller than the user size setting.
- **User decimation** – You can set the skip factor in X and Y, that is, it displays every nth point. By default this parameter is set to 1 for both the X and Y directions.

Fill and Lines of Surfaces

The 3D Surface properties dialog has three options for **Fill**, **Fill + Lines** and **Lines**. These options relate to displaying the surface as a filled surface, surface with lines or with only the triangulated lines. By displaying the Lines option only, you can make a judgement about the level of grid compression being used by the Compression mode 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 with constant compression is shown below:



Lines displayed instead of the filled surface to assess the grid compression

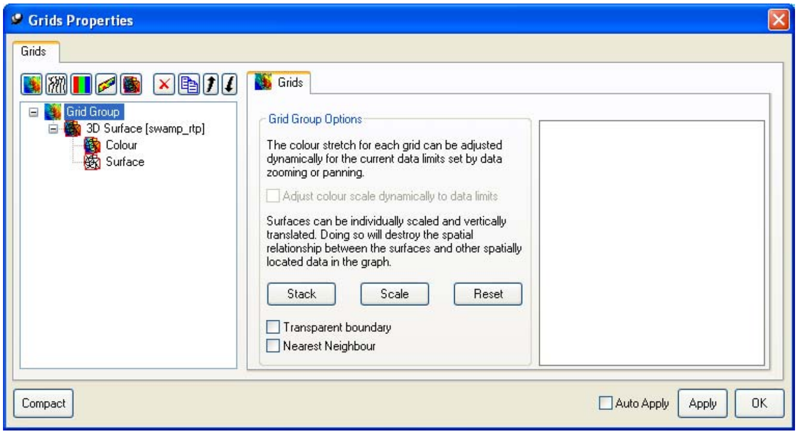
Applying Offset to 3D Surfaces

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 topographic surface may be, say 300 metres. However, it may be desirable to place a magnetics surface above the topographic surface and the amplitude or Z values of the magnetics surface may be say, 57,000 nT.

You can control the Z offset of surfaces by applying a specific offset amount or offsetting automatically by having the software adjust the amount of offset to be the grid surface minimum, maximum or average value. A slider bar is also available for interactively raising or lowering the surface.

Once an offset is determined it can be locked such that introducing other objects (like lines, other surfaces or graphical objects) do 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 scalings and offsets.



Using the Stack, Scale and Reset buttons of the Grids image dialog

You can use a 2D image map to zoom and pan around the same grid in a linked view in three dimensions. You can also use local colour stretches in the 3D display using the same technique (that is, specify the 2D data range by data zooming and panning in a linked 2D map).

The **Stack** button operation effectively adjusts the surface offsets to force them to lie within the display area. If you need to slightly move the surfaces 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).

Surface Colour

The control for surface colour is identical in operation to that of the 2D surface controls. Refer to *Controlling a Colour Layer* for details.

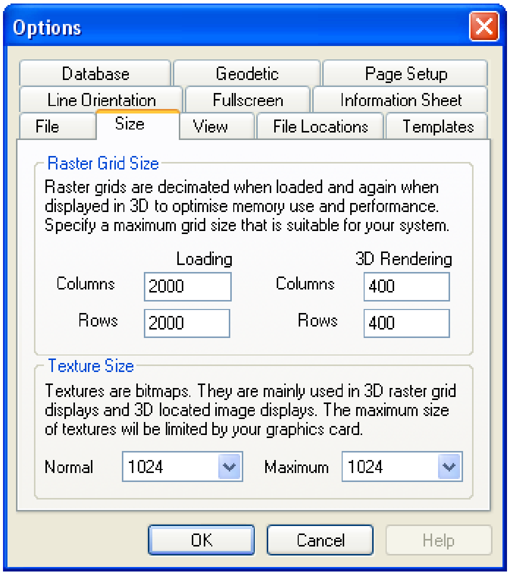
Texture Mapping

Within Encom Discover PA 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 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 Encom Discover PA 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 from **Tools>Options>Size**.



Size tab of the Options menu for Texture sizing

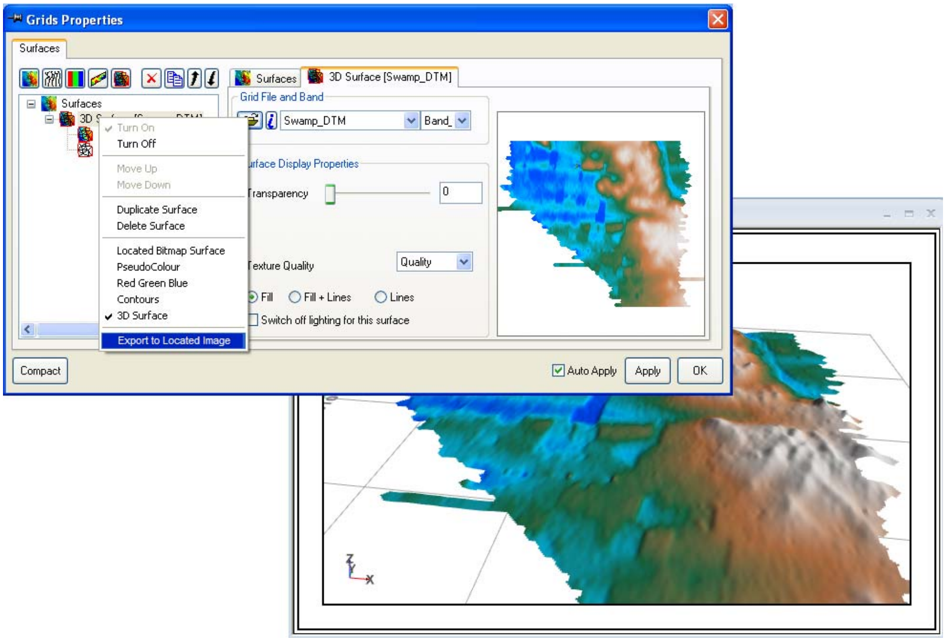
The **Texture Size** options allow entries for the maximum size of textures that are tailored to suit your graphics card.

On some graphics cards, texture sizes above 1024 are supported (up to 4096), but only if a texture interpolation method (called MIPMAP) is provided by the card. By default in Encom Discover PA, grids use this mode but controls on the Size tab allows it to be set explicitly. MIPMAP is the highest quality display mode. Located Image files generally use this mode anyway, but you should be aware that if this mode is not supported but enabled, your image may not be visible.

Specifying a maximum tile size does not guarantee that tile size is used in a Located Image file. When an image exceeds 1024x1024 pixels in size the software conserves memory and re-samples it to a power of 1024. It then determines a tile size that results in an integer number of tiles. Therefore to make best use of graphics memory, design the original image carefully. Make it a power of 2 and make it perfectly divisible by the maximum tile size (if you want an image that is that large).

Export to Google Earth (.KML)

It is possible to export a draped 3D surface to a geo-located image for use within Encom Discover PA or Discover 3D or Google Earth. This is possible using the **Export to Located Image** pop-up menu option, which is accessed when the 3D Surface branch in the Grid Group Tree is clicked with the right mouse button.



The export to located image option is accessed from the 3D Surface pop-up menu.

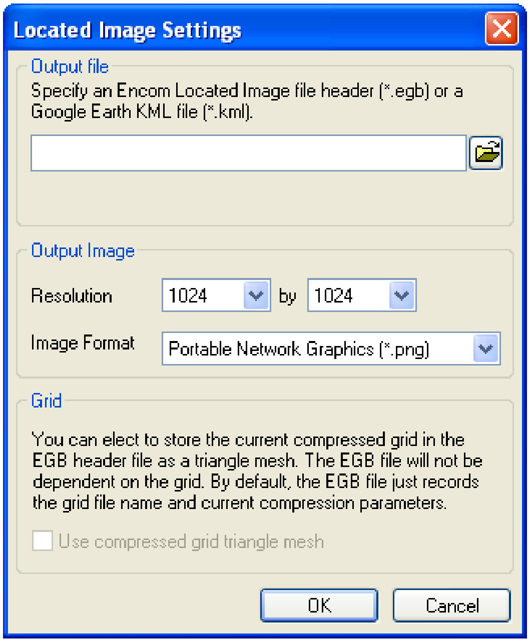
The two available output file formats are:

- Encom located image file (.EGB)
- Google Earth file (.KML)

Output Image

Use the parameters in the Output Image section to specify the image resolution and image file format. Choose from the following file formats:

- Portable Network Graphics (.PNG)
- JPEG Image file (.JPG)
- Tagged Image file (.TIF)
- Windows bitmap file (.BMP)



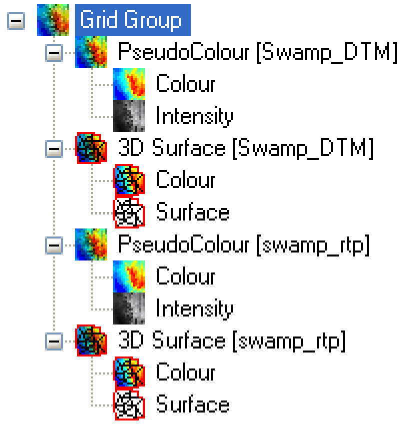
The Export to Located Image dialog.

When working with a compressed grid (see *Surface Compression*) you can elect to store the current compressed grid in the located image .EGB header file as a triangle mesh. The located image file will not be dependent on the grid. By default the located image file just records the grid file name and current compression parameters. To enable this option select the **Use compressed grid triangle mesh** option.

3D Grid Flipper

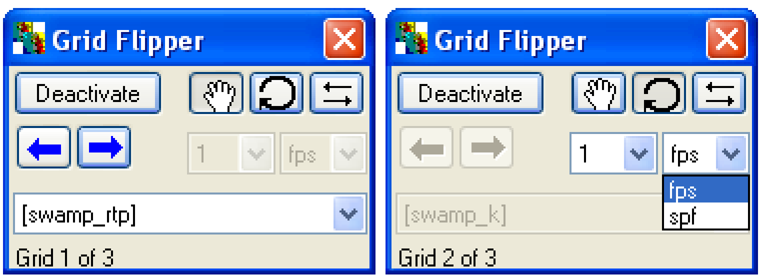
Within the 3D Map window, the Grid Flipper allows the view to be rapidly switched between any open grid surfaces (similar in operation to the Grid Flipper of a 2D Map). This tool can be accessed via right-clicking on the **Grids** branch in the Workspace Tree and choosing **Open Grid Flipper** or from the **Tools>Grid Flipper** menu command.

In order for the Grid Flipper to operate in 3D displays multiple 3D surfaces need to be available from the Grid Group tree in the grids properties dialog such as shown in the example below:



Example Grid Group tree for 3D surfaces.

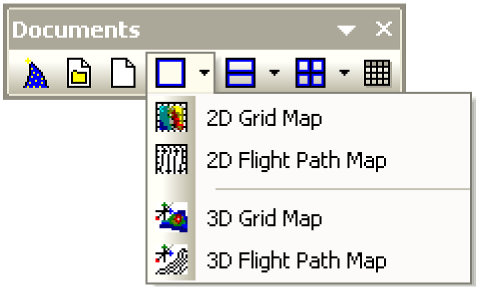
Grid selection is either via **Previous** and **Advance** buttons, or through the pull-down list. Clicking the **Deactivate** button will return grid display control to the Surface Properties dialog (in which grid display is controlled via the **Turn On/ Turn Off** options in the pop-up menu, detailed in *Creating a 2D Grid Map*).



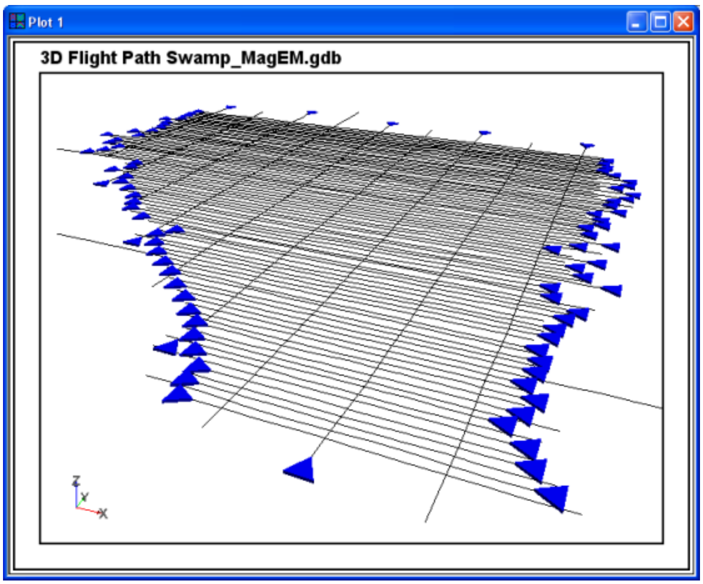
Grid Flipper dialog in manual (left) and automatic (right) mode

View 3D Flight Paths

A three-dimensional equivalent to the 2D view of a flight path can be displayed using either the menu option **File>New>Map>3D Flight Path Map** or the pull down menu item as shown.



The 3D view displays the flight path of the selected dataset and has an elevation equal to the Z value defined by the Z data field in the database.



Example of a 3D flight path map

Properties of the flight path display are identical to those of a two dimensional presentation. Note also that additional layers (images, bitmaps, models etc) can be added to the flight path display. In addition 3D internal axes can be added to assist in locating within the display.

Conventional 3D cursor navigation is available in the display. In particular also review the *Example: Flight Path over a Grid Surface*.

If you wish to have the Flight Path displays modulated by another data channels (such as Altimeter), refer to *View Line Data in 3D Displays*.

View Line Data in 3D Displays

Data derived from lines stored in an available database can be shown in 3D displays. The data lines can be enhanced in the display by:

- Offsetting vertically by a nominated data field.
- Colouring to a specific colour or colour modulated by a nominated data field.
- Colouring above or below a preset data value (average, minimum, maximum etc).
- Symbols with size and colour modulation can be added.
- Labelling of data values can be applied.
- Data displayed can be transformed.

Follow these steps to add a line display to a 3D Map:

Step 1

Create the 3D Map as described in *Creating 3D Displays*.

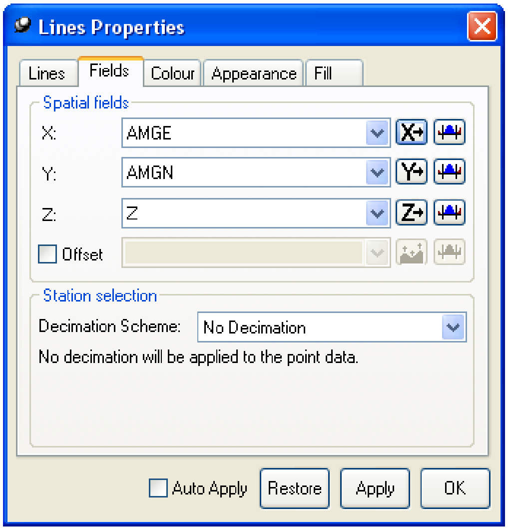
Step 2

Highlight the 3D Map branch of the Workspace tree and click the right mouse to display the pop-up menu to **Add Data**. Choose either a Curve (for a single data line) or Lines (for multiple and selectable line displays). When selected, a new **Curve** or **Lines** branch is added to the 3D Map. A faster method of adding a Curve or Lines branch to the tree is to select either the **Curve** or **Lines** button from the Data Objects toolbar after the 3D Map is selected.

Note The following steps apply to both the Curve or Lines branch items. The only difference between the two is that the Curve item controls only a single traverse whereas the Lines item can control many data traverses.

Step 3

Display the Lines Properties dialog. Select the **Fields** tab and specify the **Z** and **Offset** data fields. These fields define the elevation position (Z) and the data field to use for scaling the data (Offset).



Specify the Z (elevation) and the Offset (scale data) fields

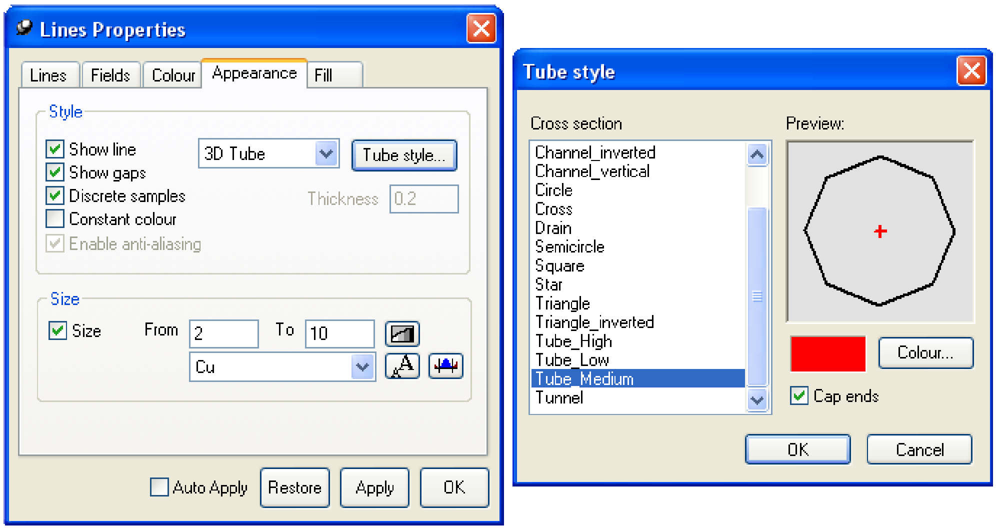
Step 4

With the data line(s) displayed, you can then modify the properties to add or control the colour, line style, symbols, labels and any additional transformation.

Controlling the Appearance of 3D Lines

The appearance of the lines is described in *Curve Properties*.

To display lines as tubes, select the **3D Line Style** tab from the Line Properties dialog, and tick the **3D Tube** option. A number of tube shapes are available. Select one from the list, and enter a **Thickness** for the tube. 3D Tube Styles can be created using the TubeShape Manager. Refer to *Tube Shape Manager*.

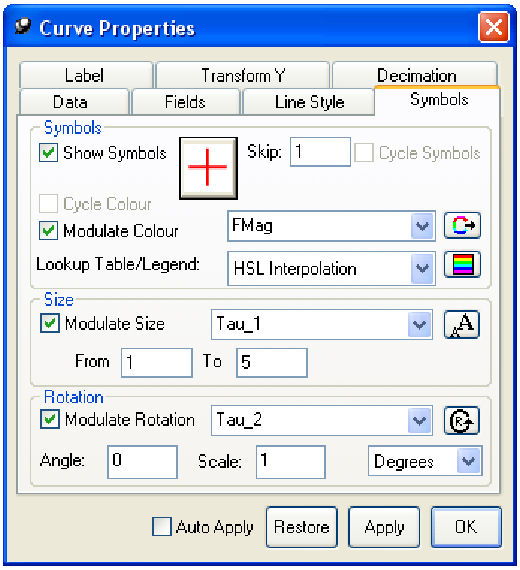


Use the Line Style tab to adjust the tube colour or modulation

Symbols

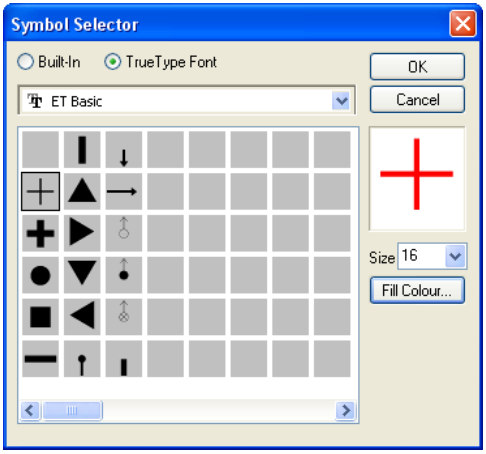
The **Symbols** tab of the **Curve Properties** dialog enables the following modifications to be made to the vertices of point data:

- Change the point symbol type
- Display points as 3D symbols based on True Type fonts, with controls over symbol orientation, extrusion and positioning
- Change the size and colour of the point symbol
- Display every point sample or a specify sample interval to display
- Modulate point data colour, symbol size or symbol rotation using values from other fields in the table and colour tables and legends.



Symbol control dialog tab

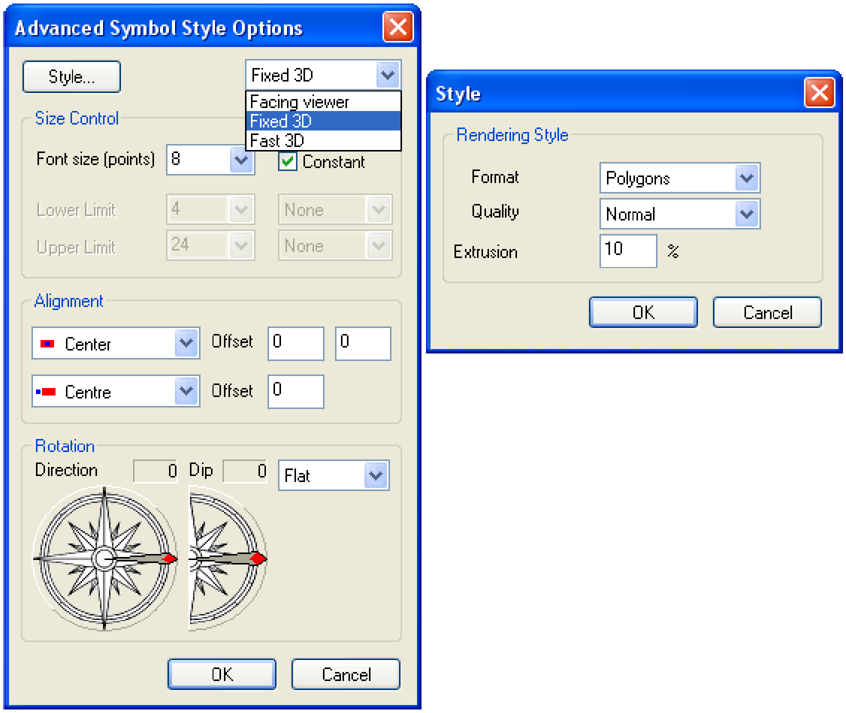
To access this dialog ensure the **Show Symbols** option is ticked. To quickly display point data using the default square symbol check the **Fast Symbol** option. This is a memory-efficient display method, and recommended when using very large datasets.



Symbol Selector dialog

To customise the symbol display, ensure the **Fast Symbol** option is deselected and click in the **Symbol** box. This opens the **Symbol Selector** dialog. A range of symbol libraries is available from the pull down list at the top of this dialog, with the selected symbol previewed to the right. The symbol colour and size can also be altered here. Three **Orientation** options are available at the bottom of this dialog:

- **None** – Symbols are displayed in the X/Y plane
- **Keep text in the view plane** – Symbols are aligned parallel to the viewing/screen plane
- **Optimum Orientation** – Symbols are placed in the principal plane (XY, YZ or XZ) closest to the viewing/screen plane.



Advanced Symbol Style Options dialog

The **Advanced** button opens the **Advanced Symbol Style Options** dialog, which provides further symbol controls. These include:

- The symbol format, including Polygons (default), Line Segments and Bitmaps. Note that Bitmaps do not have Orientation control.
- Extrusion control, expressed as a percentage of the symbol's size. The symbol is extruded perpendicular to its display plane, that is if a polygon symbol is displayed in the XY plane (Orientation None), it is extruded in the Z direction. To display a flat symbol, set the Extrusion to 0%.
- Constraint options for plane selection when using **Optimum Orientation**
- X, Y & Z **Alignment** and **Offset** Position Controls

- **Size Controls**, allowing specification of view size limits.

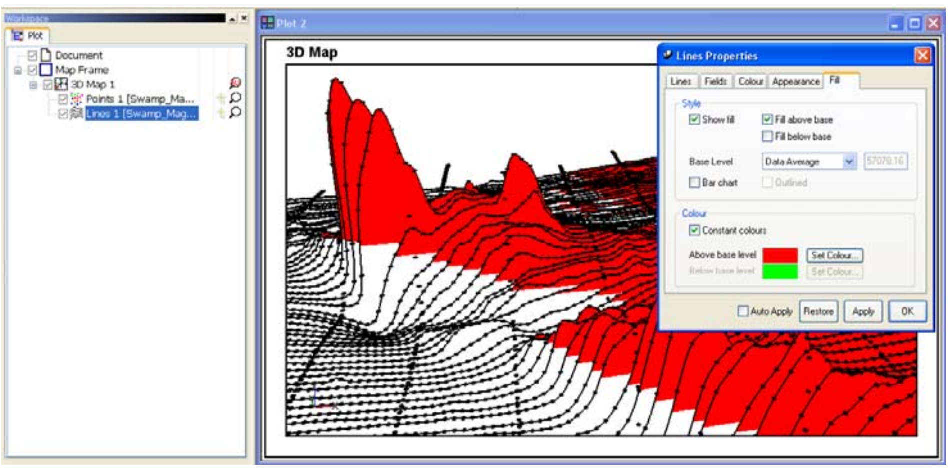
Note Displaying points as 3D Symbols requires extensive computer memory resources; it is recommended to minimise dataset size if doing so.

The symbol colour and border can also be changed using the **Line Colour** and **Fill Colour** buttons in the Symbols tab dialog. The **Skip** value determines what interval of point data samples is displayed in the 3D map window. For example, to display every 5th point data sample enter a Skip value of 5.

To **Modulate Colour** select the data field to use for the modulation and a colour look-up table or legend to use. See *Colour Table Editor* for more information.

To **Modulate Size** select the data field to use for the modulation and enter a range for the minimum and maximum modulation size. The size may be specified in millimetres, page points, screen points or percentage.

To **Modulate Rotation** select the data field to use for the modulation enter the angle and scale. The rotation of modulated symbols can use either radians or degrees anti-clockwise. An example of a 3D display with symbols and the **Fast Symbols** enabled is shown below:



Multiline display of magnetics in a three dimensional display with symbols

View Point Data in 3D Displays

Data derived from observed points stored in an available database can be shown in 3D displays. The data points represent sample locations in a survey that can be randomly placed (such as a gravity or geochemical survey). The points may not tend to be acquired along regular linear trends (or the Lines display format could be used) however, data displayed using 3D points can also be connected by lines if required. Point displays can be enhanced in a 3D display by:

- Offsetting vertically by a nominated data field
- Colouring to a specific colour or colour modulated by a nominated data field
- Symbols with size and colour modulation can be added
- Labelling of data values can be applied
- Data displayed can be transformed.

Follow these steps to add a Point display to a 3D Map:

Step 1

Create the 3D Map as described in *Creating 3D Displays*.

Step 2

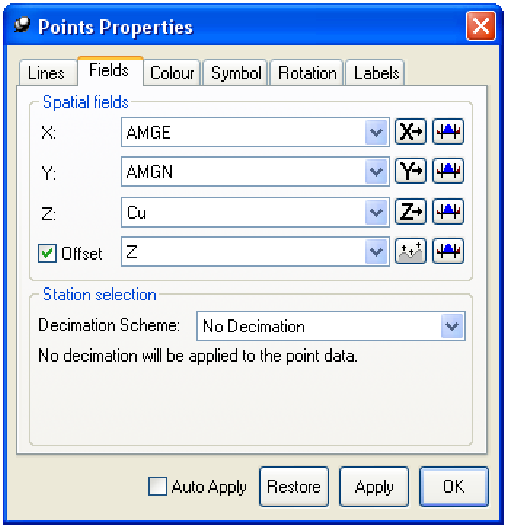
Highlight the 3D Map branch of the Workspace tree and click the right mouse to display the pop-up menu to Add Data. Choose the Points menu item. When selected a new Points branch is added to the 3D Map. A faster method of adding a Points group is to select the Points button from the Data Objects toolbar once the 3D Map is selected.

Step 3

Highlight the 3D Map branch of the Workspace tree and click the right mouse to display the pop-up menu to **Add Data**. Choose the **Points** menu item. When selected a new **Points** branch is added to the 3D Map. A faster method of adding a Points group is to select the **Points** button from the **Data Objects** toolbar once the 3D Map is selected.

Step 4

Display the **Points Properties** dialog. Select the **Fields** tab and specify the Z and Offset data fields. These fields define the elevation position (Z) and the data field to use for scaling the data (Offset).Specify the Z (elevation) and the Offset (scale data) fields.



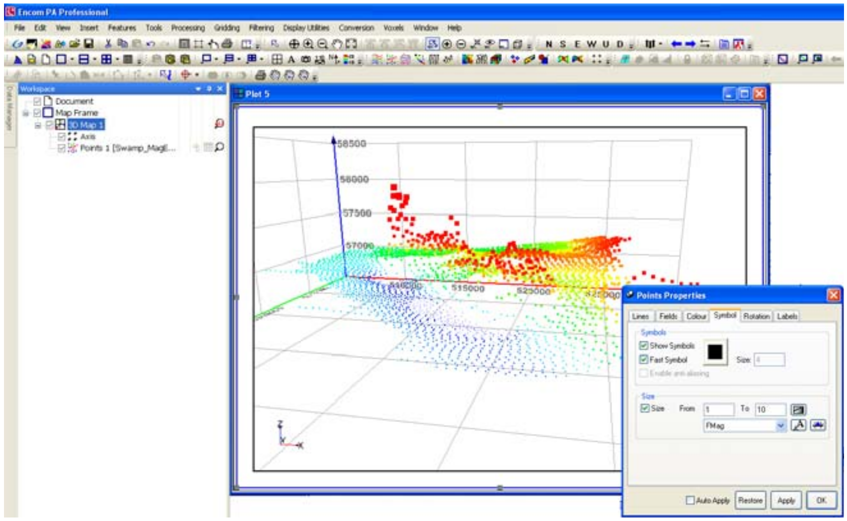
With the data points displayed, you can modify the properties to add or control the colour, symbols, labels and any additional transformation. Adjustment to data values can be applied by the Transform tab or compression and data skipping can be applied from the Decimation tab.

Controlling the Appearance of 3D Points

The appearance of the points is described in *Curve Properties*.

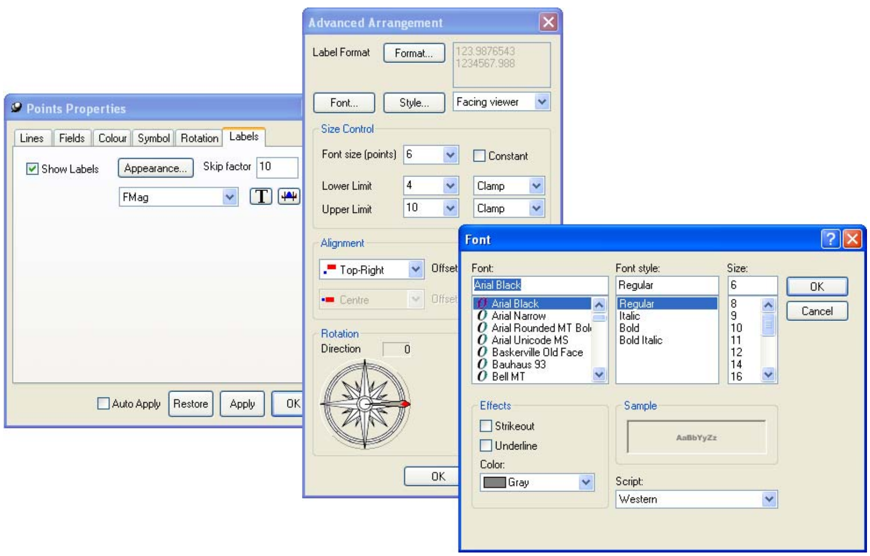
To customise the symbols displayed, ensure the **Fast Symbol** option is deselected and click in the **Symbol** box of the **Symbols** tab. This opens the Symbol Selector dialog. A range of symbol libraries is available from the pull down list at the top of this dialog, with the selected symbol previewed to the right. The symbol colour and size can also be altered here. Fast Symbols are provided as this is a memory-efficient display method, and recommended when using very large datasets. Refer to *Symbols Tab* for further information.

An example of a 3D display with symbols and Fast Symbols enabled is shown.



Points display of magnetics in a three dimensional display with symbols

Labelling of 3D points is controlled from the Label tab.



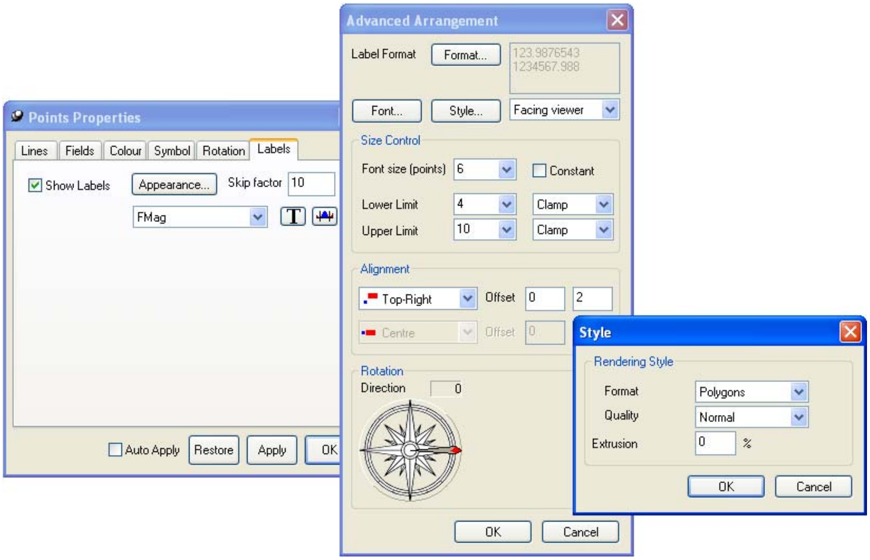
Label and font controls

The **Advanced** button provides extensive control over the text style of the labels for:

- Font
- Size
- Orientation in the 3D view
- Position (in X, Y and Z directions)
- Alignment and Offset

The dialog used to control these features is shown below:

- The **Font format**, including Polygons (default), Line Segments and Bitmaps. Note that Bitmaps do not have Orientation control.
- **Extrusion** control, expressed as a percentage of the text size. The text is extruded perpendicular to its display plane, i.e. if a polygon symbol is displayed in the XY plane (Orientation None), it is extruded in the Z direction. To display a flat symbol, set the Extrusion to 0%.
- **Constraint** options for plane selection when using **Optimum Orientation**.
- X, Y & Z **Alignment** and **Offset Position** controls allow offset of the text labels from the associated point data location.



Dialog to control the appearance of labels

Note Displaying points with 3D text requires extensive computer memory resources. It is recommended to minimise dataset size if doing so.

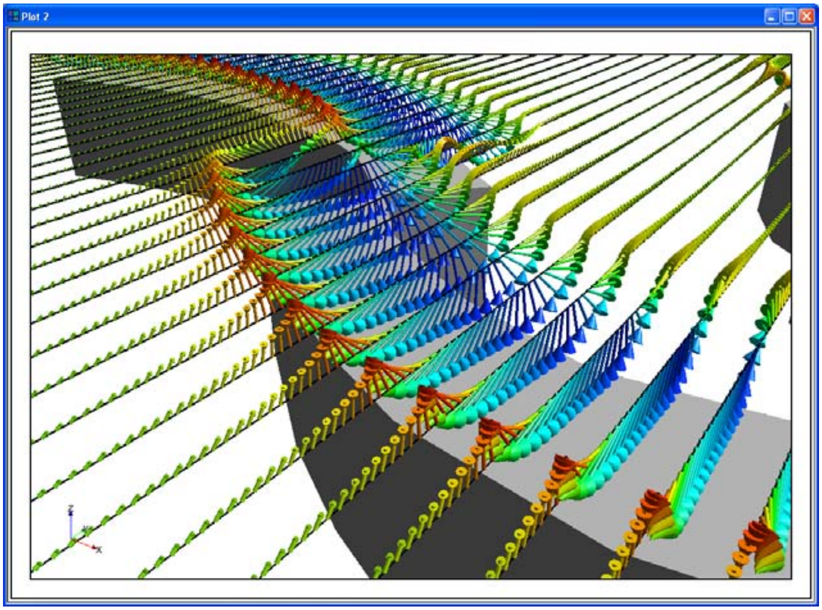
3D Vector Displays

A Vector Series data object type is available in Encom Discover PA , which is designed to display vector and tensor data in 2D and 3D graphs and maps.

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.

Encom Discover PA 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. In 2D however this just requires a single Z-axis rotation.



An example of 3D Vector series with modulated colour and size applied.

Mode

Select either vector rendering or tensor rendering. When rendering vectors you will need to define the origin coordinate and the components. The components can be sourced from different fields. When rendering tensors you will need to 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

Specify 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 just requires a single Z axis rotation.

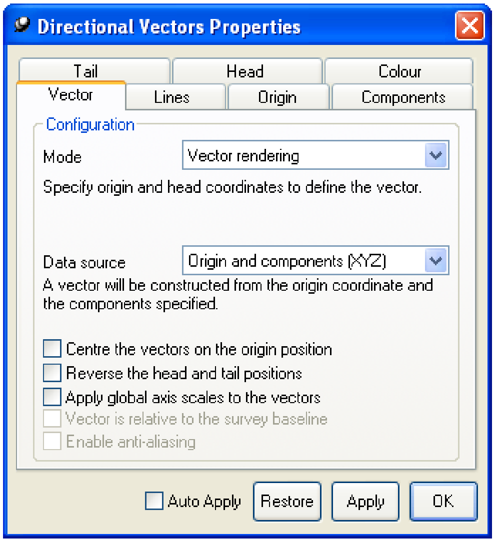
In the first two cases the length of the vector is defined by the specification of the origin and components. In the last case the length remains undefined and will initially be assigned a unit length.

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.



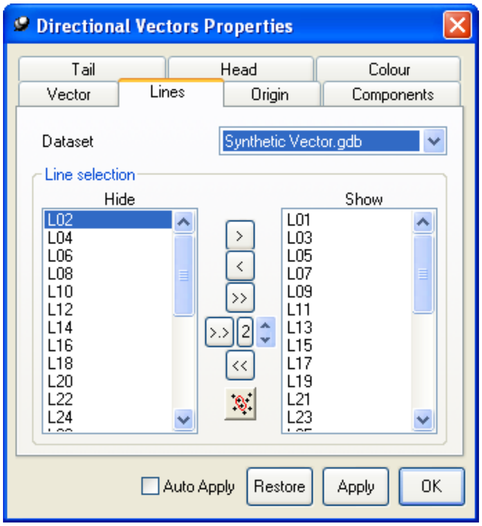
The Vector tab dialog specifying the mode and data source.

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 the space.

Line Selection

All the source data for the vectors must be derived from fields in a single database which is selected via the 'Lines' property page. You can select multiple lines as the source for the vector data. This can be done via the line lists or via the graphical selection dialog.

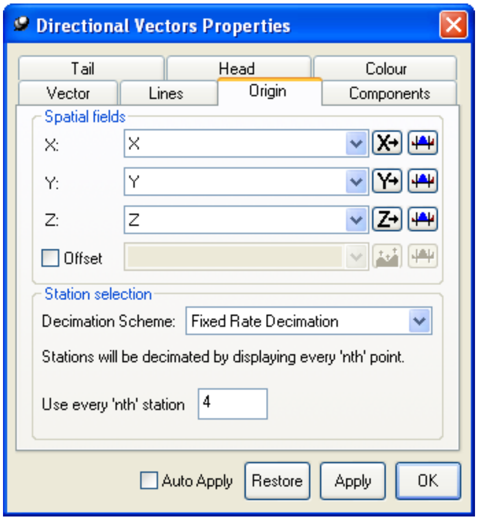


The Lines tab for specifying active lines for vector display.

Vector Origin Specification

The source data for the origin of the vector or tensor as well as the station selection are defined on the **Origin** tab dialog for the Directional vectors Properties. Specify the X, Y and Z fields if these have not been automatically selected for you. Although the fields may be multi-banded you can only select one band from each field.

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 and station decimation.

For all origin fields data conditioning is available. The data conditioning dialog allows you to apply on the fly processing to the input data to clean up data errors or otherwise improve the data prior to display. The statistics explorer is accessible for the field data via the data conditioning dialog.

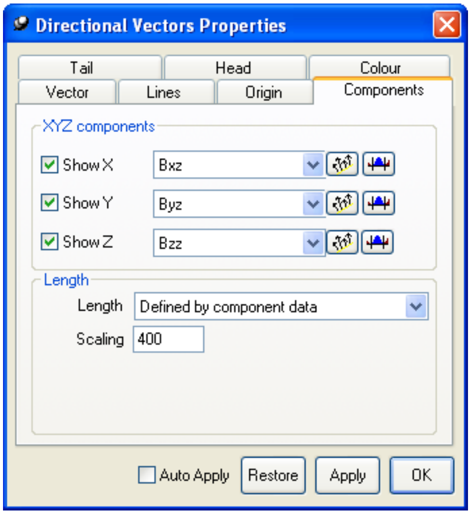
You can display every station or only a selection of stations from each line by either decimation or compression. Decimation allows you to display only the first or last station, or every ‘Nth’ station. Compression examines the X, Y and Z fields and compresses the stations so that the geometry of the line is preserved to a specified level of compression. A compression level of zero will result in no compression. The compression level can be set to between 0 and 100. Note that this only provides a hint to the compression algorithm. It does not imply that a level of compression of between 0 and 100% will be achieved.

XYZ Components

The **Components** property dialog 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. The screen shot above shows all the available options on the components property page but depending on the mode and data source chosen some of these fields and buttons will be grayed or removed or additional buttons may be available.

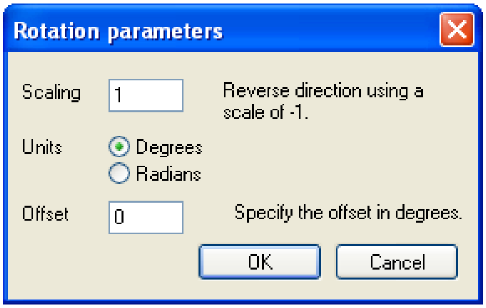
When rendering a vector select the field/band combination for each of the components. These are defined in X Y Z order. If specifying rotations the X field buttons will be grayed out. The components can be sourced from different fields or from different bands in a single field.

When rendering tensors you can specify the components for up to three vectors which comprise the tensor. In this case, the components must be sourced from multiple bands of a single field. The order of the bands cannot be specified so the bands must contain X, Y and Z data in that order.



The Components tab dialog for specifying the Z, Y and Z components of the vectors.

For all component fields data conditioning is available. The data conditioning dialog allows you to apply on the fly processing to the input data to clean up data errors or otherwise improve the data prior to display. When rendering a tensor the same data conditioning rules are applied to each component of the tensor vector whereas when rendering a vector each component has individual data conditioning rules.



When specifying rotation components (by specifying a Data Source in the Vector tab dialog of Origin and rotations) you can specify additional rules that modify the rotational data. These include:

- Scaling
- Scaling factor of a negative (e.g. -1) to be used to reverse the sense of the rotation (clockwise or anti-clockwise).
- Units of the source data (degrees or radians)
- Offset (specified in degrees).

Length

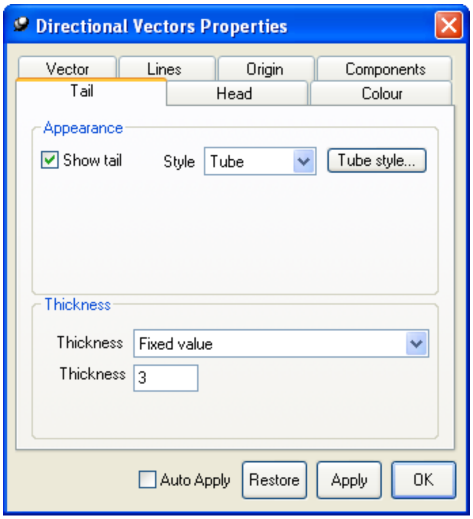
The length of the vector or tensor will be determined by the choice of components or set to a unit length if the components are defined by angles. For display purposes this is usually inappropriate and so the length of the vector can be set, scaled and modulated in a variety of ways.

- If you select **Fixed value** you can define the length as well as a scaling factor. The length of all vectors and tensors will be equal to the product of these two values.
- If you select **Defined by component data** then the length varies for each vector. A scaling factor can be specified to scale up or down the length of all vectors.
- If you select **Defined by field** then you must select a database field that describes the vector or tensor length. If you are displaying tensors then this field can be multi-banded to allow the length of each vector in the tensor to be specified. Data conditioning can be applied to this field. A scaling factor can be specified to scale up or down the length of all vectors.
- If you select **Modulated by field** you must select a database field that describes the vector or tensor length. However, the field value is not used unmodified – it is mapped to a specific length range which you also specify (‘From’ and ‘To’) via a mapping transform which you specify. A variety of pre-defined transforms are available as well as user-defined transforms. The process of defining a mapping transform is exactly the same as for colour modulation. Data conditioning can be applied to this field. A scaling factor can be specified to scale up or down the length of all vectors.

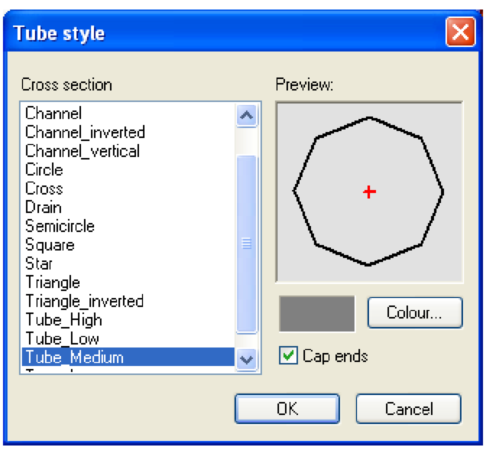
Tail

The **Tail** tab dialog allows you to define whether the vector or tensor tail ought to be rendered and, if so, what style and thickness it will have.

Tails can be rendered as either colour lines (with a number of patterns) or as coloured tubes (using a number of cross sectional tube shapes). By default the Tube_Low shape is used which gives an adequately smooth tube with low graphics rendering overhead.



The Tail tab dialog allows control over the style and thickness of a vector/tensor.



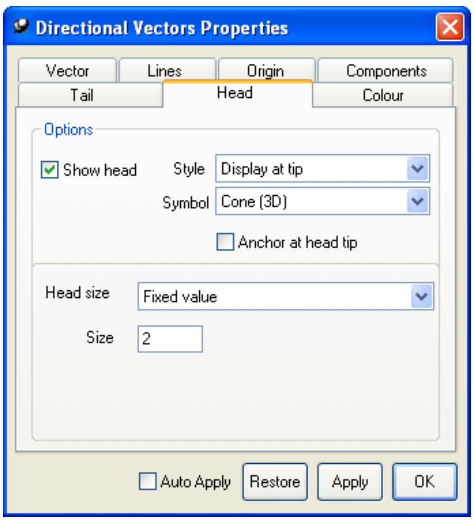
The Tube Style dialog can be accessed from the Tail tab dialog for specifying the tube shape of the vector/tensor.

The **Tube Style** button allows you to set the default colour of the vector/tensor tails. A default grey colour is specified and is also used for the default colour of the head of a vector/tensor.

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 Y 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



The Head tab dialog showing parameters for editing the appearance of the vector’s head.

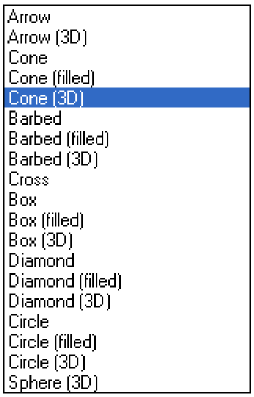
The **Head** tab dialog allows you to show or hide the heads of the vectors/tensors and to specify both the size and the rendering style of the head.

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.

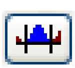
The head can be rendered using a variety of symbols listed below. Some of these symbols are simple line-work, others are flat filled polygons and others are 3D shapes. Only the 3D shapes use lighting.



The Vector head symbol style options list.

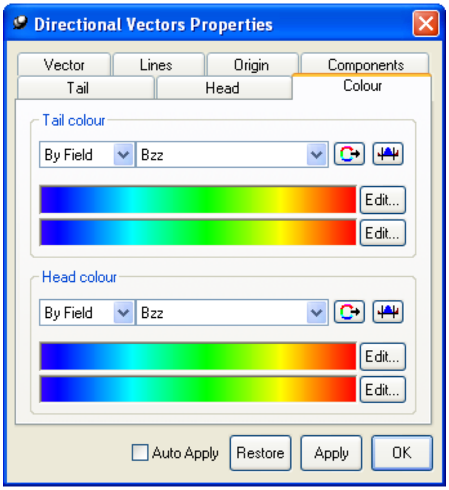
If you are displaying tensors and you choose the Tail to tip style option then only the Diamond 3D symbol option is available from the list. In this case a diamond is constructed from the three vectors of the tensor. The diamond is centred on the origin in this case.

The head size can either be a fixed size, proportional to the tail length and thus varying linearly between two sizes you specify, or modulated by a database field. In the latter case you must specify the range of the head size and a mapping transform must be defined.



Data conditioning can be applied to the field data by clicking on the **Field Data Conditioning** button available.

Colour



The Colour tab dialog of Directional Vector Properties.

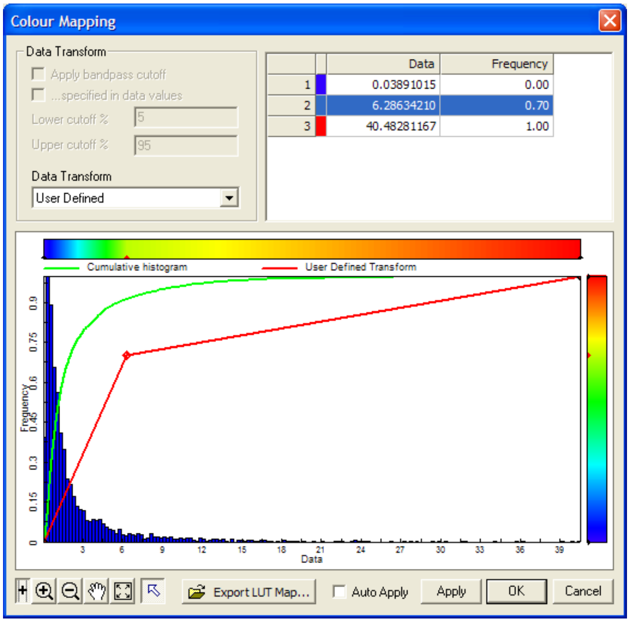
The Colour property page allows you to define the colour modulation setting for both the tail and the head individually.

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. You can construct a colour table by RGB or HSL interpolation or select any available colour table or legend. Note that only support for the foreground colour of legends is provided.

- **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.

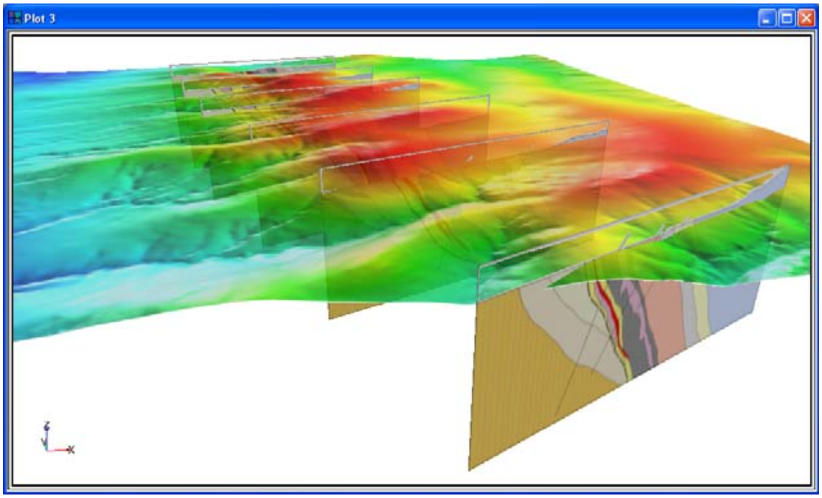


The Colour Mapping dialog allowing the specification of a data transform.

Located Images in 3D Displays

If a Located Image is to be added to a three dimensional display, the image is required to be accompanied by an Encom Georeferenced Bitmap file (.EGB - refer to *Appendix B: Data File Specifications* for additional information). This file defines the image source file by name, its type and its corner co-ordinates.

Externally created images of scanned data, seismic sections, IP sections, geological interpretations etc. can be located and placed within a 3D display. An example is shown below:



3D Map containing numerous geological section bitmaps with a DTM grid surface

In this example, a magnetic surface is viewed with a series of parallel geological interpretation sections. These sections are derived from drillhole interpretations from the GIS package Encom Discover (refer to Integration With Encom Discover). As for all 3D views, the display can be manipulated and moved to see the view from the best location. You can fly-through the surface and sections and readily obtain a perception of the relationship between the geology and geophysics.

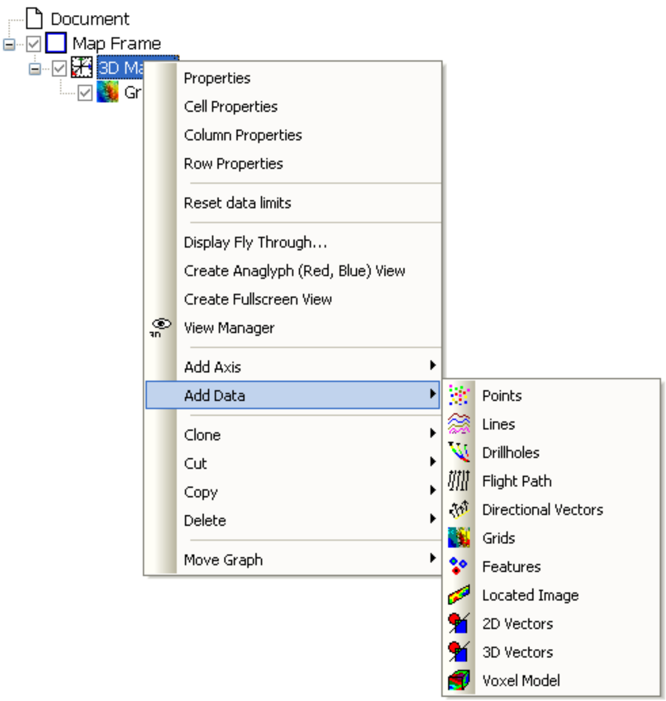
Located Images can be displayed within a 3D view by using the Images data option in the workspace tree (see below). Images are located by using an ancilliary file called an EGB file. The EGB file can be created with an ASCII editor or by using an image registration wizard (refer to *Image Registration Wizard*).

Adding a Located Image to 3D Display

3D displays of Located Images are usually added to a 3D Map although they can also be added to a 3D Graph, 2D maps, profiles and sections. The steps required are:

Step 1

Create a 3D Map or 3D Flight Path Map using the **File>New>Map>3D Grid Map** or **3D Flight Path Map** menu option (or from the appropriate buttons).



Step 2

With a 3D Map branch created in the Workspace tree, select the 3D Map item and click the right mouse button. From the **Add Data** menu item, choose **Grids**. This option creates a 3D map display as described in the *Creating 3D Displays*. Manipulate the surface as required with suitable vertical exaggeration and offset. Note that the data values of the displayed grid determine the vertical position of your display and this is important when displaying bitmaps with a different vertical position.

For example, if you wish to display a magnetic grid (with average data values of say, 55,000nT) combined with geological sections (with vertical reference of say 1-200 metres) as shown above, then an offset is required to bring the two data objects into a similar vertical reference frame. Typically therefore, you would need to reduce the vertical location of the magnetic grid down by about 55,000. Refer to *Applying Offset to 3D Surfaces*.

Step 3

You need to have prepared the necessary Located Image before display and these require associated .EGB descriptor files. Refer to *Saving as a Located Image* and *Appendix B: Data File Specifications*.

Step 4

Select the 3D Map branch of the Workspace tree and click right. Select the **Add Data>Located Image** option. An **Images** item is added to the tree associated with the 3D Map. An alternative to this is to click the **Add Image** button from the Data Objects toolbar.

Step 5

Display the properties dialog of the Located Image branch. Specify the path and .EGB file. You can use the **Browse** button to navigate to the required .EGB file.

Step 6

If the displayed map surface and image are widely separated, their vertical offset may require adjustment. This can be done either by editing the vertical location in the **Offset** tab of the **Located Image** properties dialog, or by interactively setting an offset amount in the surface display control. To do this, display the **Grid Group** properties dialog and highlight the Surface branch of the Grid Group tree. At the base of the dialog are controls to scale and offset the grid surface. Set this to an appropriate value by using the slider and increment entries. For information on editing Located Image Properties refer to *Located Image Properties*.

Drillholes in 3D

Drillhole traces, collars and recorded downhole information is ideal for display in three dimensional displays since their presentation by definition extends into the vertical direction. Encom Discover PA uses the 3D display format to present drillhole location and information in a variety of ways. Information can include assays or geological lithology.

For details of displaying drillholes and data in three dimensional displays, please refer to *Adding Drillholes to a 3D Map*.

2D Vector Objects in 3D

Two dimensional vector files can be displayed within a 3D display either as a flat, planar drawing or draped over a predetermined surface. To do this:

Step 1

Open a 3D Map display in which the 2D Vector file is to be added.

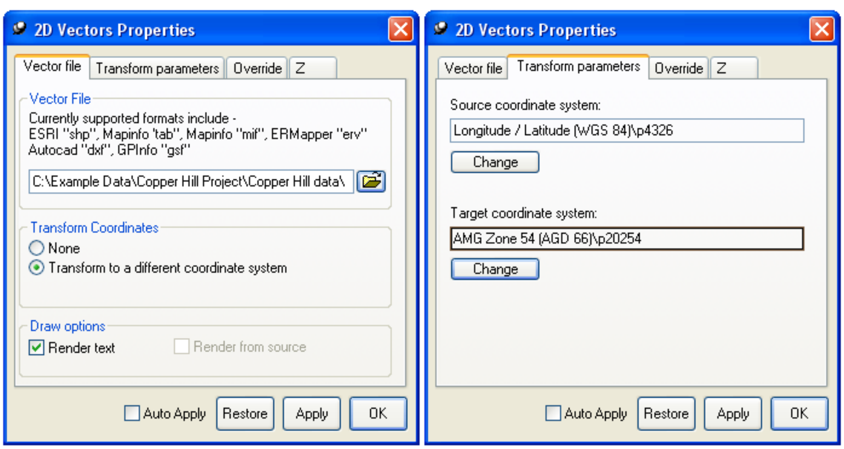
Step 2

From the 3D Map branch of the Workspace tree, click the right mouse button to display the pop-up menu. Select the **Add Data>2D Vector File** option. Note that you cannot use the **Add Vectors** button of the Data Objects toolbar since, by default, it adds a 3D Vector branch.

Step 3

From the Properties dialog of the 2D Vector branch, display the Vector File tab and navigate to the folder containing the required file. The supported 2D vector formats are listed in *Appendix E: Supported File Formats*.

If necessary, you may need to apply a transformation of the data for vector files created using a different projection from that of Encom Discover PA. Select the **Transform Parameters** tab and set the required parameters in the Transform parameters tab.

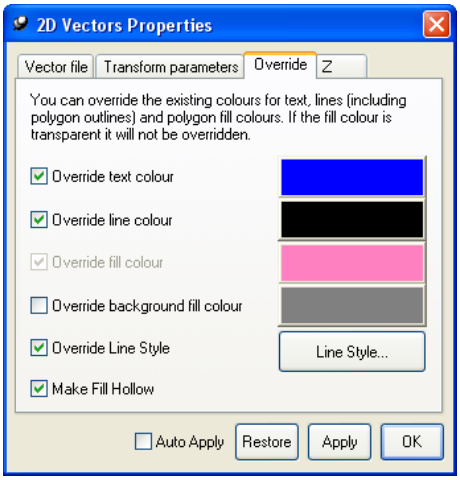


Specify the 2D Vector file and any required Transform parameters

If a coordinate system transfer is required, click the **Change** buttons on both the source and target coordinate systems.

Step 4

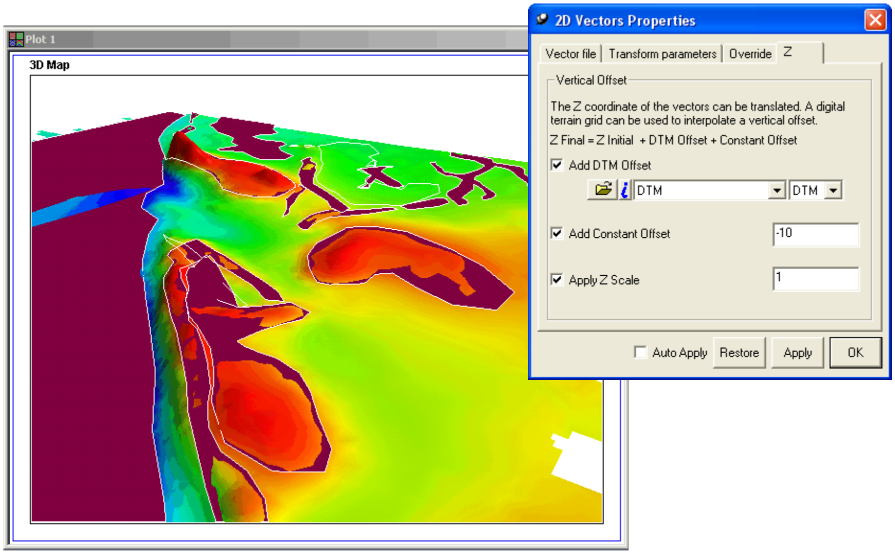
Once the 2D Vector file is specified, aspects of its appearance can be altered from the **Colour** tab. Control is provided over any displayed text, line colour, fill colour and background fill colour.



Control over the colours contained in the 2D vector file are available

Step 5

Control of the location of the 2D Vector file relative to other objects is done from the **Z** tab. This tab allows you to add or subtract a vertical offset to align the displayed 2D Vector file location as required, but in addition, the location of the 2D vectors can be made to drape over a defined grid surface. Note that the gridded surface does not have to be one already used in the display.



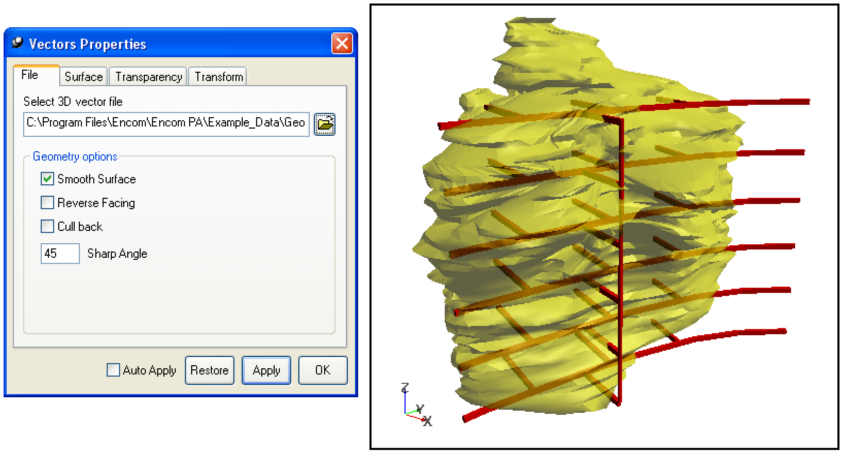
Display of a 2D Vector file draped over a surface with grid and offset definition

3D Vector Objects in 3D

Encom Discover PA can display a range of three dimensional objects. Supported 3D file format objects are listed in *Appendix E: Supported File Formats*.

Note Additional vector file formats can be imported and converted using the *Transform Vector File* utility.

An example of two 3D .DXF models imported into a 3D space is shown below and illustrates the use of 3D vector files with other data types (drillholes, located bitmaps, etc).

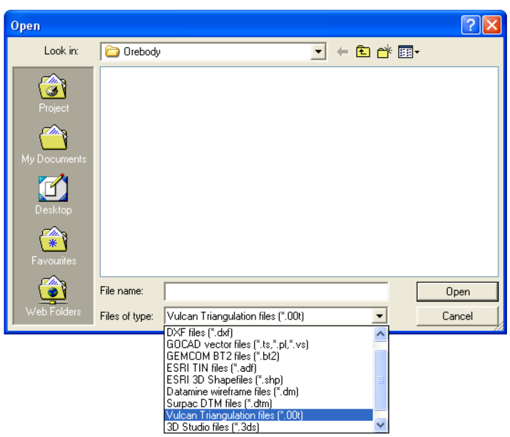


DXF and GoCad files are loaded and their appearance is controlled from the 3D Vector properties dialog

Loading 3D Objects

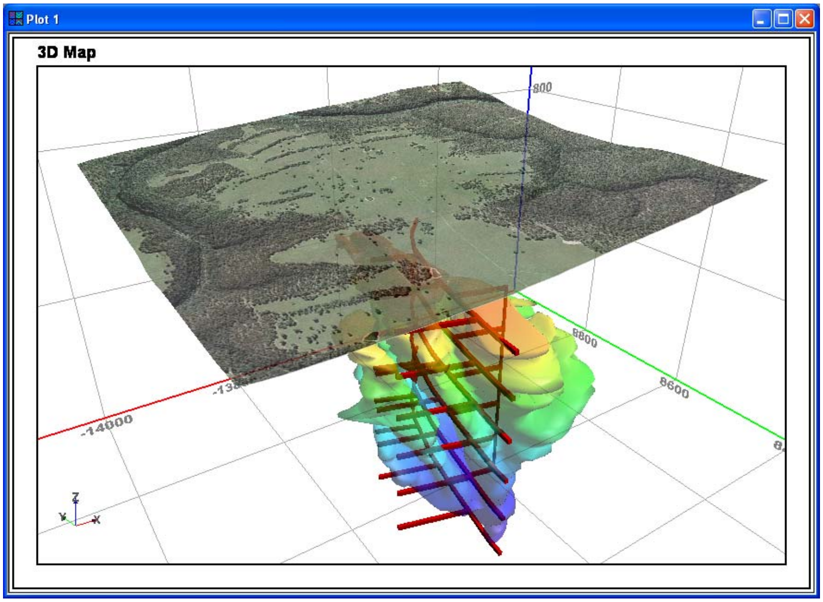
Encom Discover PA can access supported 3D format files directly by selecting the **Add Vector** button from the Data Object toolbar, or the **Insert>Data Object>Vectors** menu option.

When opening 3D files from Encom Discover PA a new, empty Vectors branch is added to the Workspace Tree. Display the **Property** dialog of this layer and use the **Browse** button to specify a 3D file after navigating to the appropriate folder. Note that the choice of file type (AutoCad .DXF, GoCAD TSurf or Gemcom .TB2) is made from the Open file dialog.



Select the 3D file format from the Files of Type pull-down option

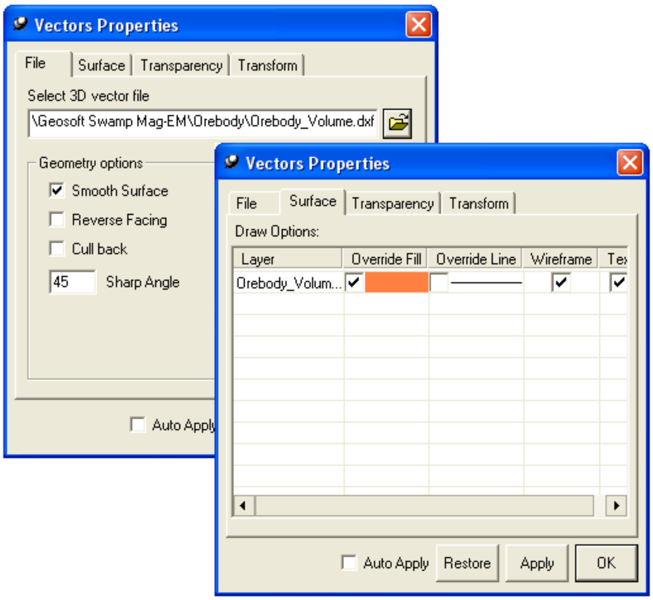
Click the **Open** and then the **Apply** buttons after file selection and the object is displayed in Encom Discover PA.



Display of an orebody and development DXF files with surface

3D Vector File Properties

Encom Discover PA provides a number of controls to improve or alter the appearance of displayed 3D vector files.



Vectors Properties tab dialogs for used for controlling and defining the appearance of a 3D vector file

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

- Apply a **Smooth Surface** of the object if the surface is faceted. The smoothing can be applied to facet angles sharper than a defined angle (sharp angle is 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:

- **Override Fill and Line** - Specification of the colour of the 3D file fill and line drawing boundaries can individually be specified. Click the colour button to display the colour selection dialog.

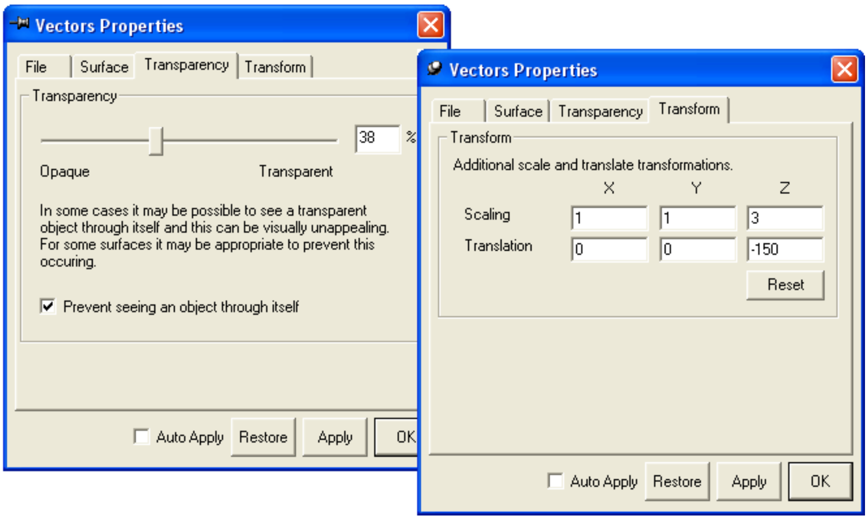
- **Show Wireframe-** - Display the wireframing of the 3D vector surface faceted shape as well as having the 3D surface displayed.

- **Show Text**

The Transparency Tab enables:

- The **transparency** of a vector object to be altered via a slider bar or manual entry of a transparency percentage

- **No Self See-Through** – In complex shapes, this option prevents the entry of one enclosing surface to see the same but further back surface.



3D Vector File tab dialogs for defining the transparency and transform.

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 (untransformed) values.

Section Profiles in 3D Displays

Sections that display individual layered sections such as Conductivity Depth Images can be presented in 3D displays. The method of display of sections and the data they present is often substantial and consequently section displays whether in tracked profiles or in 3D displays are slow to view and manipulate (refer to section chapter Introduction). Although sections can be displayed in their layered-depth format, an alternative method to present them that is much faster to draw is also available. This second method involves creating Located Image captures of the required sections and displaying these in the 3D display instead. The resultant display is extremely fast and allows real-time navigation not possible if the actual section data were displayed.

These two methods of displaying sections are discussed separately below.

- *Adding Sections to 3D Displays*
- *Section Profile Properties in 3D Displays*

Adding Sections to 3D Displays

To display section profiles in 3D a located image (.EGB file) of the 2D sections must be first created and then displayed in 3D.

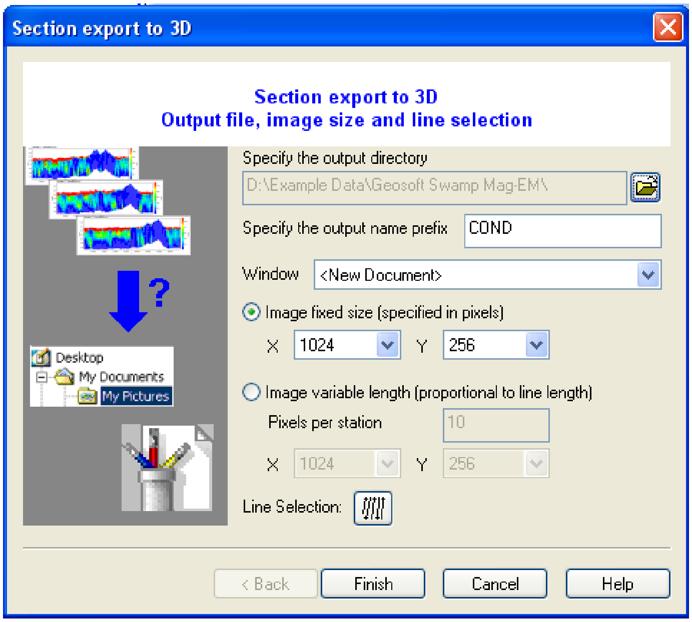
To create section(s) in a 3D display:

Step 1

Create a Section Profile as described in *Creating Sections*.

Step 2

Adjust the Colour Scale and Colour Table options if required for the section profile display and right mouse click on the Profile 1 branch of the Workspace tree to display the pop-up menu. Choose the **View in 3D** menu option and the below dialog will appear.



The Section export to 3D dialog.

Step 3

In the **Section export to 3D** dialog select the **Browse** button to enter a file location of the output .EGB (located image header) file and then specify an output name prefix for the file and associated image files. The output file format is .PNG (Portable Network Graphic) so that an example of the total output files produced by entering a prefix of “Section_CDI” for 4 survey lines (L20100-L20400) is:

`Section_CDI.egb` (located image header file)

`Section_CDI_L20100.png`

`Section_CDI_L20200.png`

`Section_CDI_L20300.png`

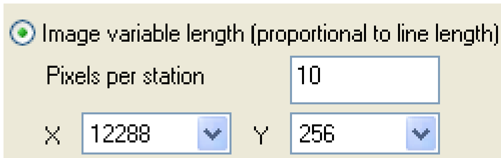
`Section_CDI_L20400.png`

Step 4

Define the **image resolution** by either:

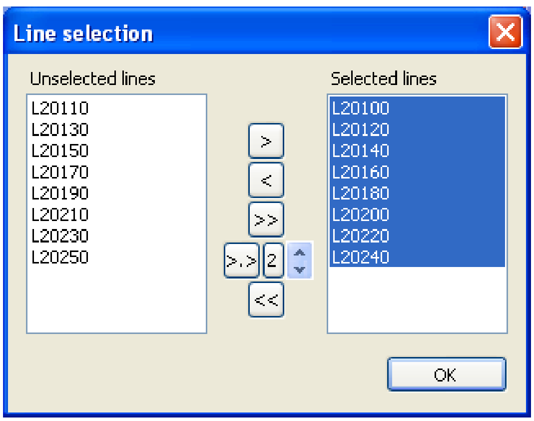
- **Image fixed file** – specifying a fixed size for X and Y dimensions (in pixels) for an image; or

- **Image variable length** – specify the number of pixels between station locations and the number of pixels for X or Y dimensions (in pixels) will vary based on this length.



Step 5

Click on the **Line Selection** button and nominate which lines in the loaded database to use in the export to 3D process. Click OK.

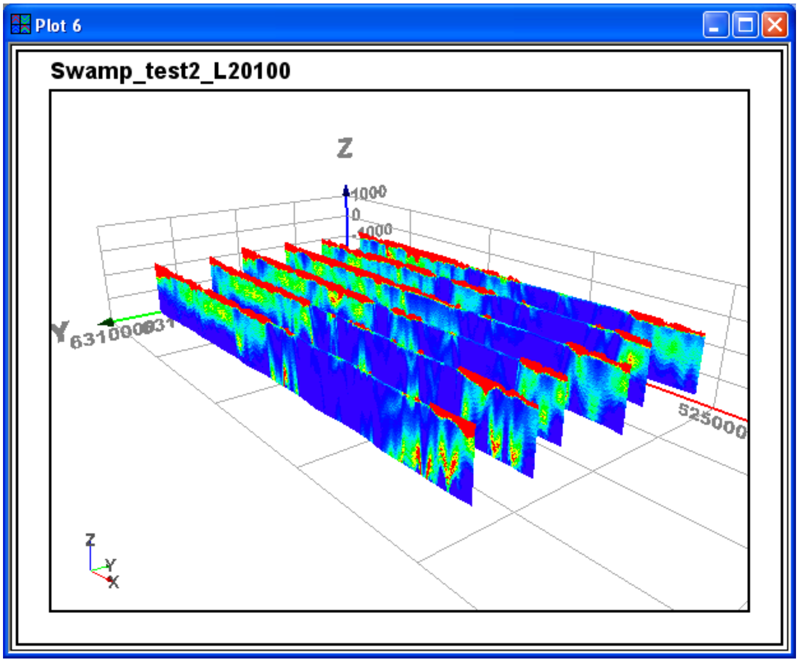


Line selection dialog for Section export to 3D process.

Step 6

Click on the **Finish** button to begin the export process. The status of the export procedure can be monitored in the bottom left hand corner of the PA interface in the status bar section.

Once this procedure is complete a 3D display window will appear with the located image data object containing the section profiles displayed. An example of a 3D display of a section profile located image is shown below:



Example of a located image of conductivity-depth image section profiles shown in 3D.

To edit the display properties refer to *Located Image Properties*.

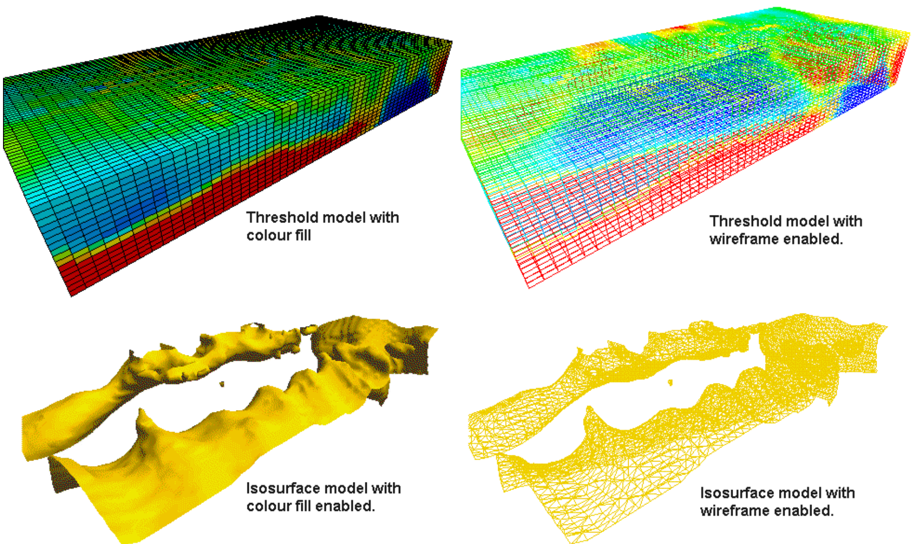
Section Profile Properties in 3D Displays

When displaying section profiles in 3D as a located image it is best to be aware of the following disadvantages:

- There is no interactive control over the colour table or scaling used.
- The images must be pre-prepared before they can be displayed. This need happen only once but it is necessary whenever a colour change or alteration of resolution is required.
- The display of an image of a section is not as accurate in its representation as the original data in the image. Image creation provides control over the resolution used, but usually a reduced quality image results.

Voxel Models in 3D

Displaying voxel models in 3D is described in detail in the *Voxel Models*. Voxel models are well suited to displaying in 3D format. An example of these displays is shown below:



Examples of voxel displays using 3D format

Feature Objects in 3D

Interpreted or created features from a feature database can be displayed in 3D displays. This can be useful when interpreted two dimensional areas are available or, where interpretations are mad on sections of data (such as CDI or geological sections) and then the interpreted features can be displayed in 3D. See *Working with Features* for a description of creating 3D features.

Feature displays in 3D can be:

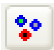
- Any of the Feature sets (point, polyline, polygon or 3-point picks). It is essential that an appropriate Z (elevation) data field is available to correctly position the features.
- Standard colour attributes can be applied to features on a Set basis.
- Multiple feature datasets can be used to layer the interpretations made.

Follow these steps to add a Feature Set to a 3D Map:

Step 1

Create the 3D Map as described in *Creating 3D Displays*.

Step 2



Open or Create a Feature Database using the **Features** menu or Data Manager window. For information on this, refer to *Features*.

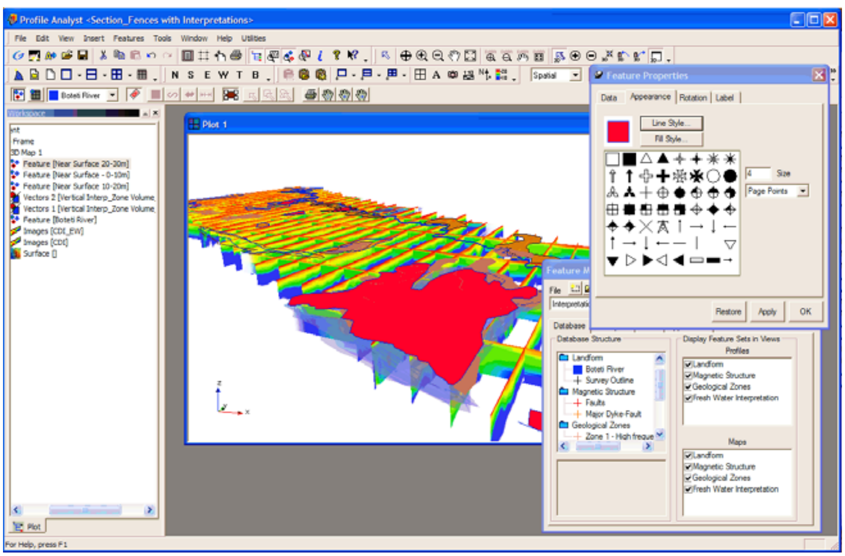
Step 3

Highlight the 3D Map branch of the Workspace tree and click the right mouse to display the pop-up menu to **Add Data**. Choose the **Feature Set** menu item. When selected a new **Feature** branch is added to the 3D Map. A faster method of adding a Features branch is to select the **Add Feature** button from the Data Objects toolbar once the 3D Map is selected. Alternatively **drag-and-drop** the Feature Set displayed in the Data Manager window into the active display window.

Step 4

To control the appearance of features within the 3D display, refer to *Feature Object Properties*.

An example of features displayed in a 3D display is shown below:



Features displayed in a multi-section 3D display to assist with interpretation

Animation of 3D Displays

Encom Discover PA allows different 3D views to be captured for replay as an animated movie. 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 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. Once a script is created, an .AVI movie can be generated using a movie codec of your choice.

At certain points, you can enter audio identification. These sound effects are then heard while a script is being replayed through the computer sound system. Sound cues such as “Note the high gold grade in drillhole DDH-121…” can be a useful adjunct to seeing the 3D display replayed.

Details of creating and operating the Fly-Through wizard are described in *3D Animation and Fly-Through*.

15 3D Navigation and Animation

In this section:

- *3D Toolbar Controls*
- *3D View Manager*
- *Navigating in 3D*
- *3D Display Modes*
- *3D Cursor Plane*
- *3D Animation and Fly-Through*

3D Toolbar Controls

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*.

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.

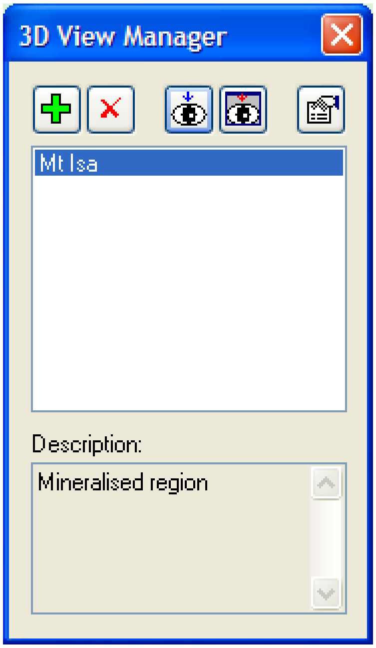
Fit to Page - Only available in **Page Layout** view; automatically resizes the page to fit to the extents of the view.

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.

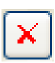
- **Orthographic** – A form of parallel projection (also known as an isometric or axonometric projection) where identically sized objects are displayed with the same size regardless of the distance they are positioned from the viewer. Orthographic view is best used when it is important to be able to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

3D View Manager



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**.

Navigating in 3D

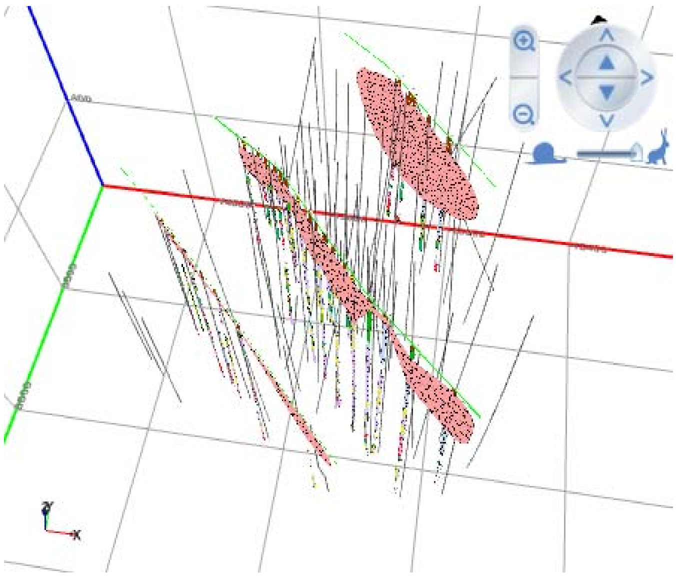
- *3D Navigation Modes*
- *3D Navigation Controls*

- *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.

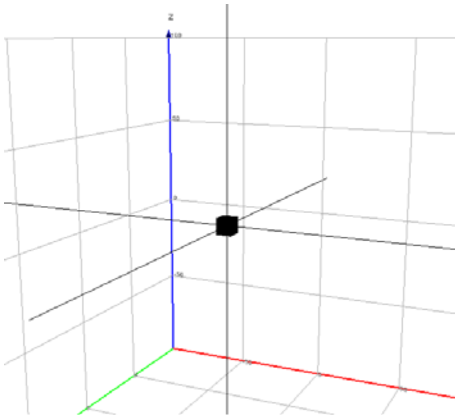


Select/Navigate—Use this mode to create, edit and interrogate objects in the 3D environment. When selected, position the cursor inside the Display Window, then hold the left mouse button down and drag the cursor to rotate pivot the view about the centre of the display. You can use the other controls 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

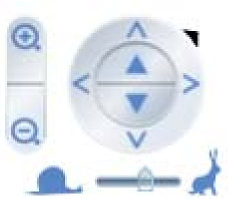
To navigate in 3D, click the  **3D Navigation** button on the Zoom Controls toolbar.

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.



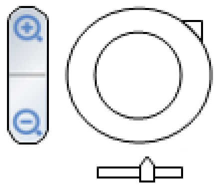
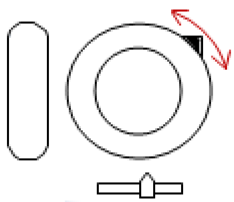
The 3D navigation view point

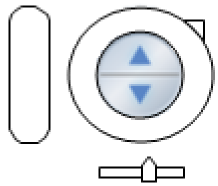
The **3D Navigation** tool is displayed in the top-right corner of the Display Window. This tool fades when there is no movement after approximately 5 seconds. To redisplay, move the cursor into the top-right corner. This tool controls the eye position and the view point (the rotation point). Similar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

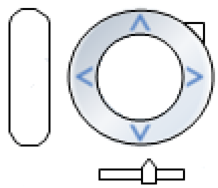
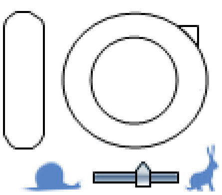


The 3D navigation tool is displayed in 3D Navigation mode.

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.

<table>
<tr><th>Movement</th><th>Using the
3D Navigation Tool</th><th>Using the
Mouse/Keyboard</th></tr>
<tr><td>Vertical pan</td><td>
Z-pan: Moves the view point vertically (in the Z-direction)</td><td>

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.</td></tr>
<tr><td>Horizontal pan</td><td>
XY-pan: Moves the view point horizontally (in the XY-plane)</td><td>
XY-pan: Hold down both the left and right buttons. The view point moves within the XY-plane in the direction of the cursor.</td></tr>
<tr><td>Speed/ sensitivity</td><td>
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.</td><td>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.</td></tr>
<tr><td></td><td></td><td>Refer to 3D Cursor Keyboard Shortcutsfor a complete list of mouse/keyboard controls.</td></tr>
</table>

The **Reset 3D View**, **View Manager** and **Change View Direction** on the Zoom Controls toolbar can also be used when in 3D Navigation mode.

Using the 3DConnexion SpaceNavigator™

Discover PA 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.

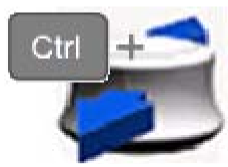
2. Connect the device via a USB port.

3. Start Discover PA 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 PA 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 Encom Discover PA can be configured to operate using several view modes. The following view modes are available:

- *Normal View Mode*

- *Page Layout Mode*
- *Full Screen Mode*

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 PA 3D can be switched into full screen mode such that the entire monitor screen will display only the content of the main 3D **Display Window**. The menus, toolbars, status bar, information windows and workspace tree will be hidden when operating in full screen mode.

Switching to full screen mode is conducted with the **Select/Navigate** tool enabled, right-click in the main 3D window and select the **Fullscreen View** menu item.

To exit full screen mode, press the ESC key on the keyboard.

In Full screen mode, the **3D Navigation** control is automatically enabled. Full screen window display parameters, such as **Resolution**, **Colour Depth** and **Display Frequency** can be modified on the **Tools>Options>Fullscreen** dialog.

3D Cursor Plane

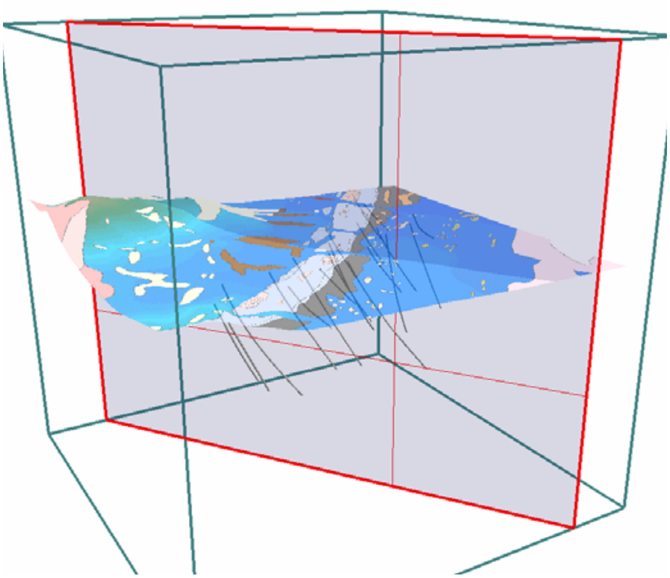
The Cursor plane is a user controllable and customizable plane in the 3D view, which has two main functions:

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

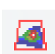
The Cursor plane toolbar contains tools required for the operation of the Cursor plane.

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.

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.

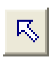
 Shrink

 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.

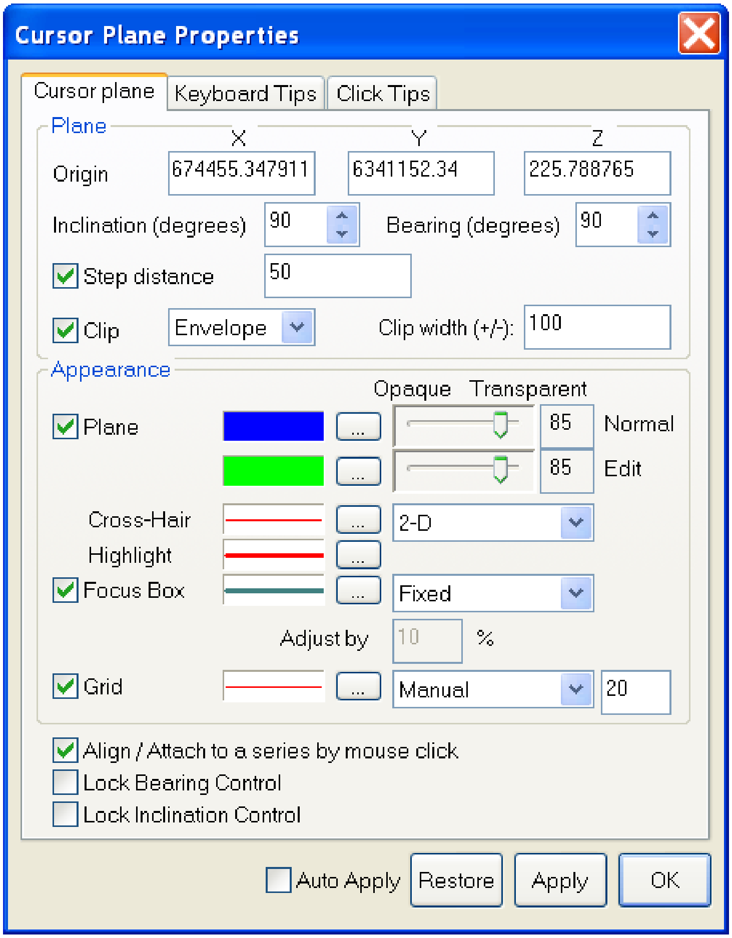
Left
Right

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

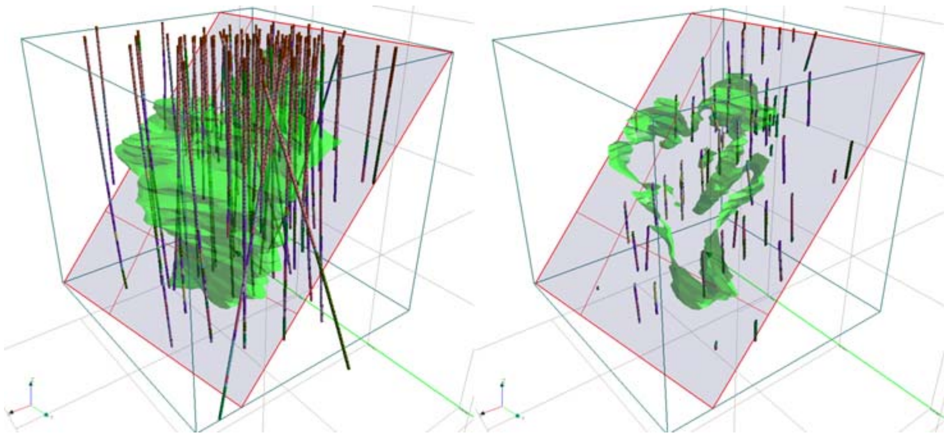
The 3D position of the Cursor plane can be set manually via the **Origin**, **Inclination** and **Bearing** options.

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).

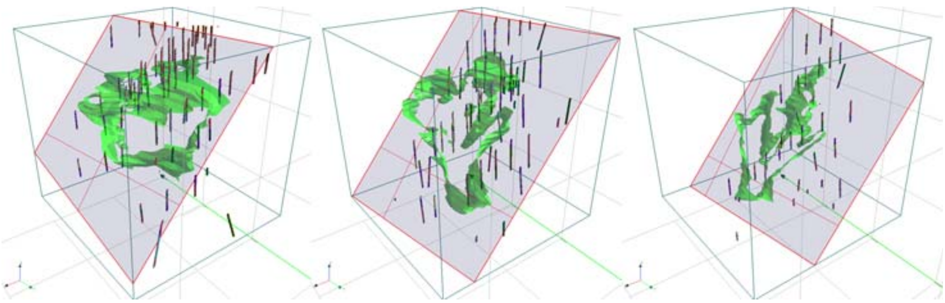
The **Clip** option refines the operation of the **Plane clipping** tool. Clipping is dynamic: if the cursor is moved (e.g. dip and azimuth, or lateral movement), the clipping region will follow.

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.



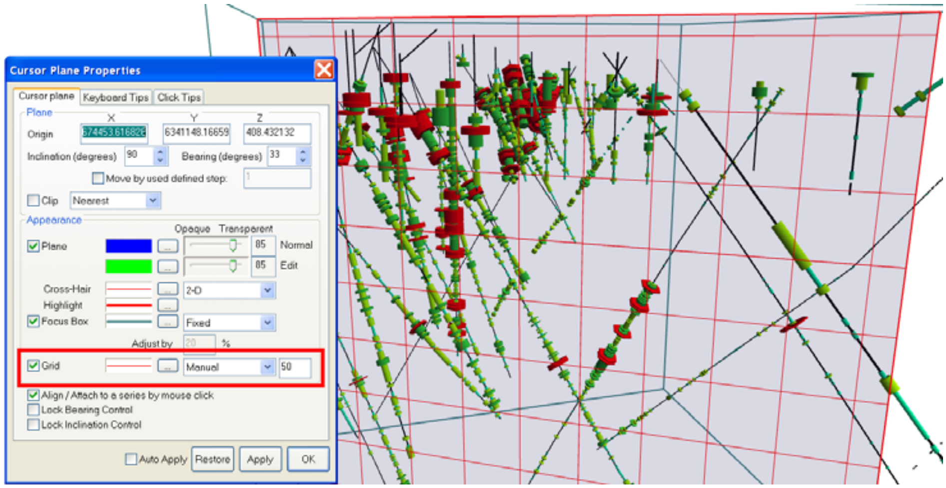
A skarn model (vector) and drillhole dataset before (left) and after application of a 40m wide clipping envelope centred on an inclined Cursor plane.



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.

Creating and Editing Features Objects Using the Cursor Plane

Another use of the Cursor Plane is to create features object in a 3D window using the **Bond To Image** button, and a displayed surface (Located Image only) as a location reference. The features available are the same as those for conventional feature objects (refer to *Features*) excluding the 3-Point-Pick feature.

The steps required to create features in a 3D Map are:

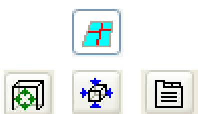
Step 1

Have a Located Image displayed in a 3D Map window. This image is used to create and edit features in 3D. The Located Image can be flat or superimposed over an undulating surface such as topography (refer to *Creating Located Images*).

Step 2

Open or create a feature database and add an associated Feature Set to the Workspace tree of the 3D Map window. Ensure the feature Set to be written to is made editable in the workspace tree.

Step 3



Making the feature set editable should also activate the 3D Cursor plane in the 3D display as well as associated toolbars.If not the select the **3D Cursor** button of the 3D Cursor toolbar.Configure the display and the Edit Plane as desired using the **Frame size** control buttons and **Cursor Properties** button.

Step 4

Orient the Cursor Plane with the same axes plane as the Located Image to be used for drawing features. Typically this will be horizontal so the XY plane is required. Use the **Plane Orientation** button to modify the orientation. Link (bond) the Edit Plane to the Located Image using the **Bond Image** button.

Step 5

To activate the 3D Cursor on the Edit Plane, click the **Attach Cursor** button (or the **Pointer** button). When this is done, positioning the cursor over the Edit Plane draws the 3D Cursor crosshairs.

Step 6

You can now start creating features on the Cursor Plane using the Feature Tools on the Features toolbar. Specify if you wish to create a Symbol, Line or Polygon. Note that to accurately draw 3D objects, it is best to orient the view from a position perpendicular to the Located Image and Cursor Plane. This can be done using the **Direction** toolbar buttons.

Repeat drawing features as necessary. You may need to alter the view or to change the Feature Set. To check the entries, display the Features Spreadsheet.

3D Animation and Fly-Through

Encom Discover PA allows different 3D views to be captured for replay using a fly-through 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. The fly-through tool is available from the **Tools>Fly Through Animator** menu option.

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 Encom Discover PA 3D.
- Manually specifying various predefined views.
- Edit an existing Fly-Through (.FLY) file.

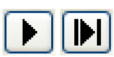
Other features of the **3D Fly-Through Animation** feature include:

- 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 **Tools>Fly Through Animator** menu option
3. In the Fly Through Creation Wizard dialog that opens (Step 1) select the **Manually from the current Encom Discover PA 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 navigation 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.

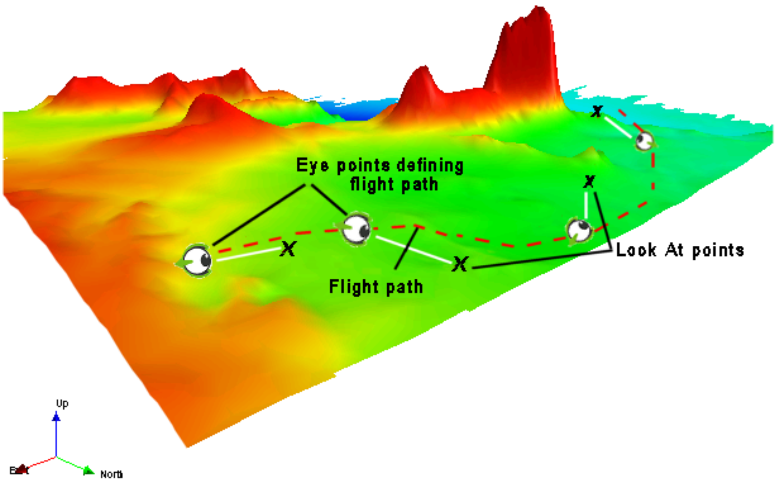


8. Play the animation in the 3D window by pressing the **Play** button. A selected number of view points can be previewed using the **Preview** button to the right of the Play button.
9. Edit the animation by adding, inserting, modifying or deleting the necessary view points. These controls and additional 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.

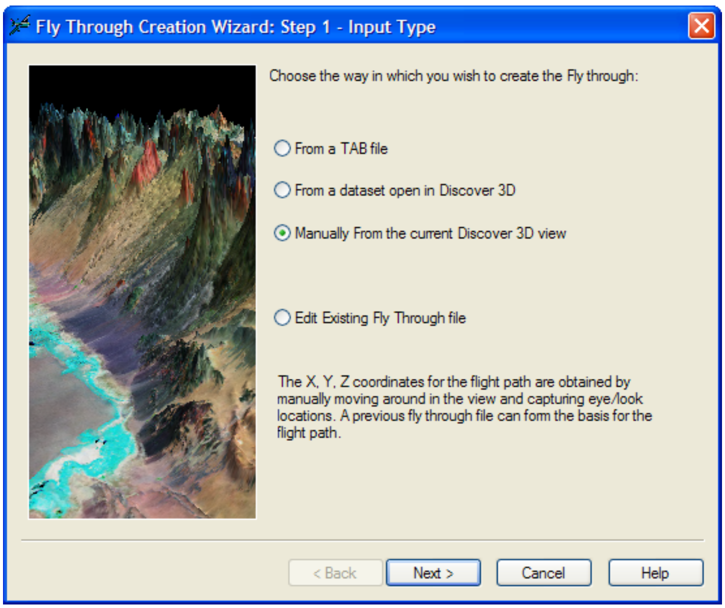
The individual **Eye Points** and their associated **Look At** point, including any associated events, are saved to a script file called a **FLY** file. These files, created for a given project area, provide the mechanism to replay the fly-through animation as desired.

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.

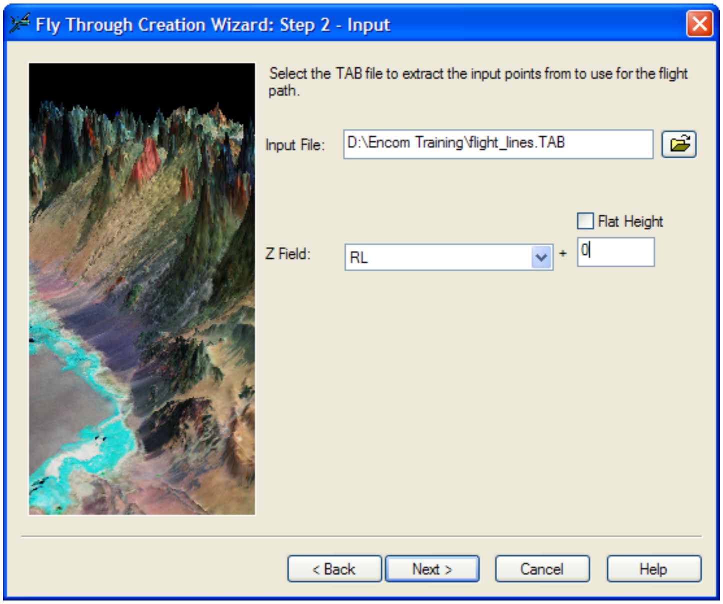
- From Encom Discover PA 3D Dataset – XYZ coordinates for flight path based on values in fields in a Encom Discover PA 3D dataset.
- Manually from current 3D view – Specify a number of 3D map window views to use as **Eye** and **Look At** points which are linked together to form the flight path.
- Edit 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.



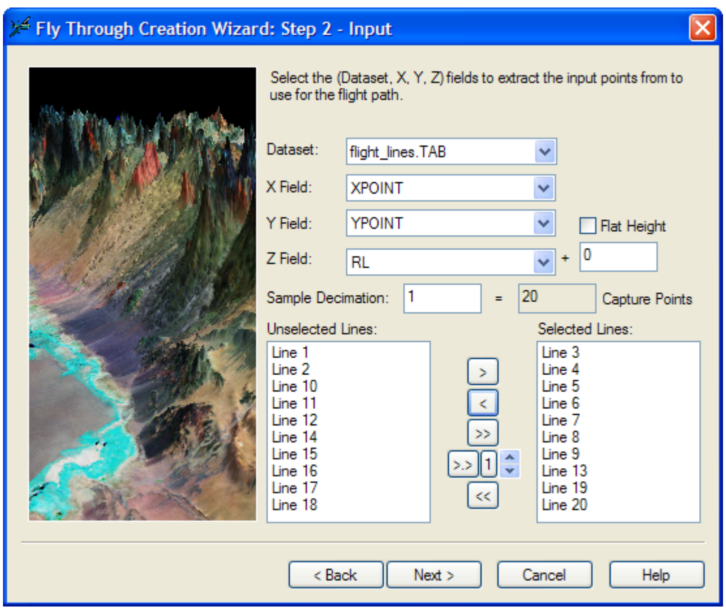
Select MapInfo table and Z values

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.

Encom Discover PA 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 Encom Discover PA 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.

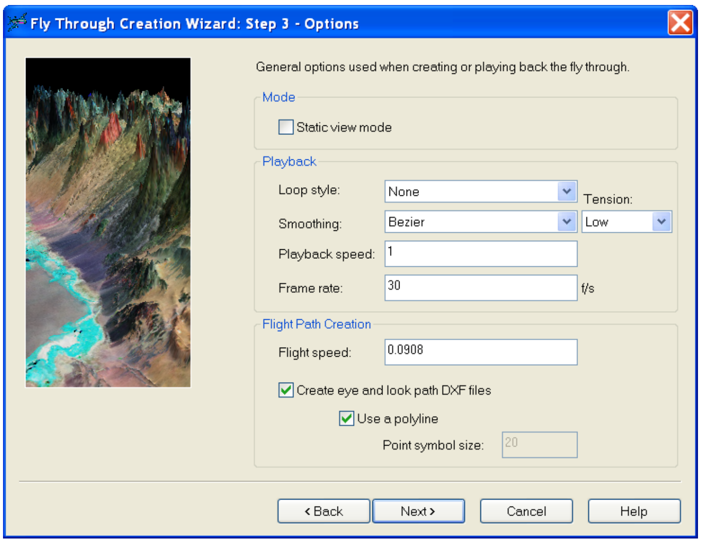
Edit Existing Fly-Through File

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.

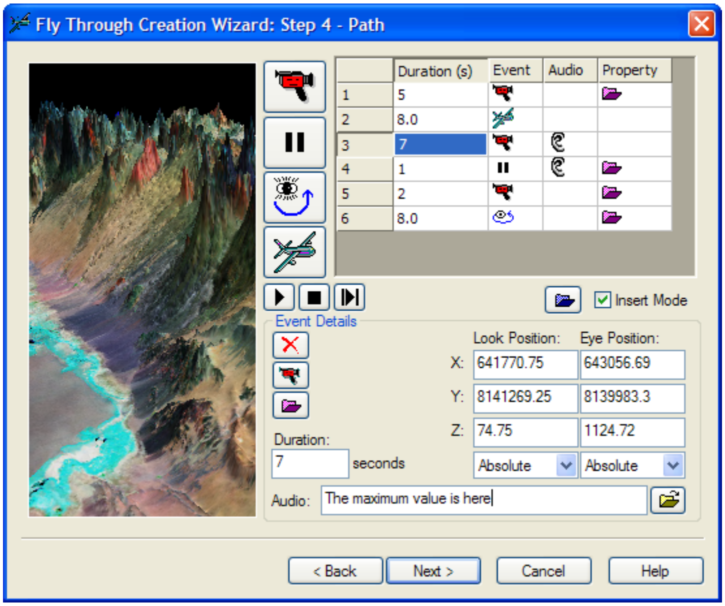
Step 4 - Capture Points and Event Details

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 Encom Discover PA 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, Encom Discover PA 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.

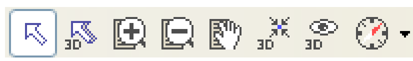
- **Event –** The type of capture point is indicated by one of four Events: **Capture Point, Pause, Rotate Eye** and **Rotate Look At.** See *Capture Point Events* for more information.

- **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.**

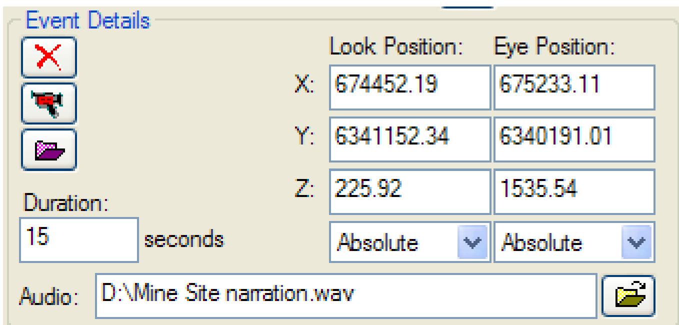
The **Properties** for the current event can be captured using this button. This will save any changes made to any data objects in the Workspace Tree since the previous event (e.g. visibility/transparency/legends/object thickness or size/etc). To refresh this, select the point event, make the necessary 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.

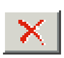
Note Properties applied to a Capture Point Event are also applied to all successive Events, unless a following Properties entry alters this.

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 Encom Discover PA 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 Encom Discover PA 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.

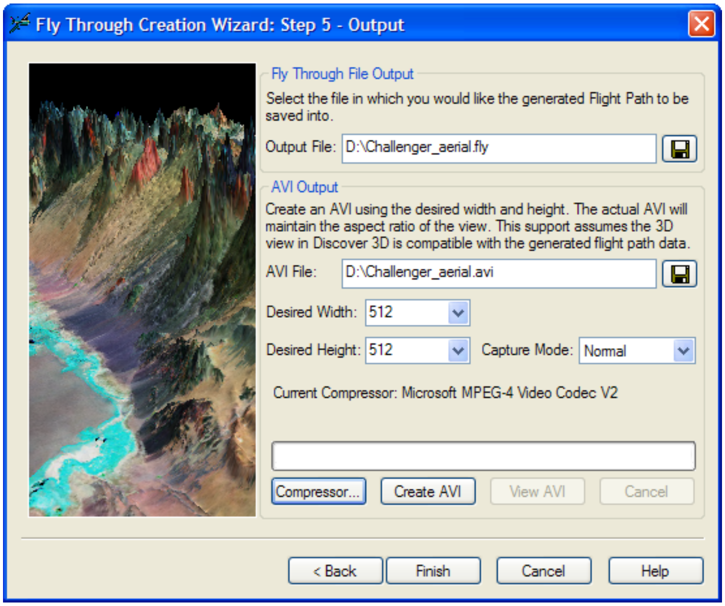
Once a Fly-Through file has been saved, it can be played back via a number of options within Encom Discover PA: see *Playing an Existing Fly-Through File* below.

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 Encom Discover PA 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 Encom Discover PA 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.

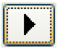
1. To activate this, enable the **Static view mode** option in the Step 3 dialog.

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, Encom Discover PA 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 Encom Discover PA 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/*).

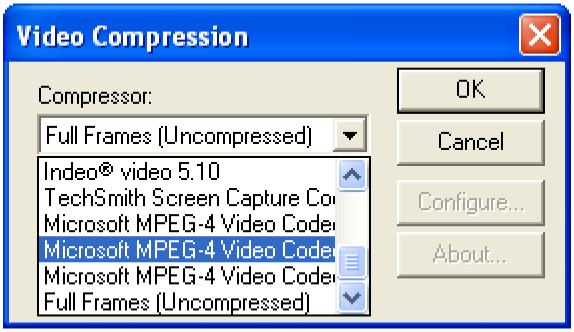
To create an video movie:

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 Encom Discover PA 3D 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 Encom Discover PA 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.

16 Located Images

In this section:

- *About Located Images*
- *Applications of Located Images*
- *Creating Located Images*
- *Using Located Images*
- *Located Image Properties*

About Located Images

Encom Discover PA makes use of a powerful feature to display a wide range of data and objects in 2D and 3D effectively, quickly and simply. Significant speed advantages, especially when using 3D displays can be obtained with this feature. In this documentation, these files are called Located Images.

The capabilities and applications of Located Images allow the display of 3D objects. In addition, a large number of file types are now directly supported and can be used as the geometry source for 3D surfaces and objects. A Located Image therefore consists of two types of source data – bitmaps (images) and 3D surfaces or objects.

The bitmaps can be simply created from sources such as:

- Maps, profiles, sections and graphs in PA
- Raster images (all common formats supported)
- Located images (eg GeoTIFF)
- Digital photography (eg aerial photos, trees, cars and everyday objects)
- Vectors (when added to a planar surface bitmap)
- Scanned data (eg sections, seismic, drawings etc).

3D surface and object geometry data can be imported from sources such as:

- Raster grids (eg ERMapper or Geosoft)

- Triangulations (eg ESRI TIN files)
- 3D Vector data (eg TKM, DXF, GoCAD or GEMCOM files)
- User designed surfaces (via EGB files)

In many cases each Located Image is comprised of data from multiple sources and may therefore require some metadata to tie the source data together. The metadata is stored in an additional, small ASCII header file with the file extension .EGB (Encom Georeferenced Bitmap).

EGB files have the following general capabilities:

- They can render one or more geo-located images in 2D or 3D. Images can be located on simple, single faceted surfaces or complex surfaces comprising many millions of facets.
- They can render one or more three dimensional objects. The objects may be arbitrarily complex.
- Maintain detailed ancillary information tables for objects at multiple levels of scope that can be queried or used to modulate the appearance of objects.

In practice this means that use of .EGB files can display almost any kind of geographical content ranging from large multi-faceted textured surfaces to thousands of geo-located objects. You can record and store information with these objects and recall that data via graphical selection of objects on-screen.

You can also display objects that do not have an associated Located Image. For example, a surface can be rendered using flat or smoothly interpolated colour. The colour can be modulated in a wide variety of ways. This ability makes it possible to directly support raster grids and TIN files with .EGBs.

Further, Located Images through .EGBs provide support for structured objects constructed from sub-objects and linked together with a series of cascading transformations. A large number of objects can be rendered efficiently and the user can graphically interact with the objects. This extension allows users to build life-like worlds and in the future will enable support for animation of 3D objects.

Applications of Located Images

There are many applications for Located Images. Often, there is more than one way to achieve a particular display and there is functionality overlap between the Located Images and other display types in Encom Discover PA. However, the Located Images can provide certain advantages.

In 2D you might use EGB series in these scenarios:

- Rendering CDI sections in profiles and section graphs. Sections require a huge amount of information to be displayed and it can take quite a long time to render a CDI section. To display multiple sections in a view may reduce interactivity due to long render and update times. An effective solution is to use the **Batch Hard Copy Wizard** to generate a located image for each section. This can then be used instead of the Section display and can improve render times by several orders of magnitude. Another similar application might be rendering seismic sections.

- Rendering Located Images (such as GeoTIFF files) in 2D maps. This can also be achieved via the Grid Group series but using Located Images is faster, more memory efficient and does not involve any reinterpolation of the image data.

In 3D you might use the EGB series in these scenarios:

- Displaying **CDI sections** in 3D. The same EGB file that was generated for viewing the sections in 2D can be displayed in 3D. One advantage of this is a simplified graph that does not require a complex 3D Section. The Located Image with its .EGB also provides better control over the transparency of sections and allows you to view sections at higher resolution.

- Rendering **3D grids**. You can display a 3D grid using the Grid Group but the Located Image also loads a grid (in any of the large number of supported formats) natively. The Located Image allows you to control the colour modulation of the grid more precisely plus it allows you to display much higher resolution images of the surfaces.

- Rendering **3D vector files** (eg DXF). Located Images of 3D vector files can also be used to display these formats but the Located Images retain ancillary data acquired from the source file that you can view after graphically selecting objects.

- Rendering **3D TIN surfaces**. These can also be viewed using the 3D vector series but the visualisation and colour modulation options of the Located Image approach is far more powerful. TIN files can be loaded natively into Located Images.

The file types supported as Located Images are listed in *Appendix E: Supported File Formats*.

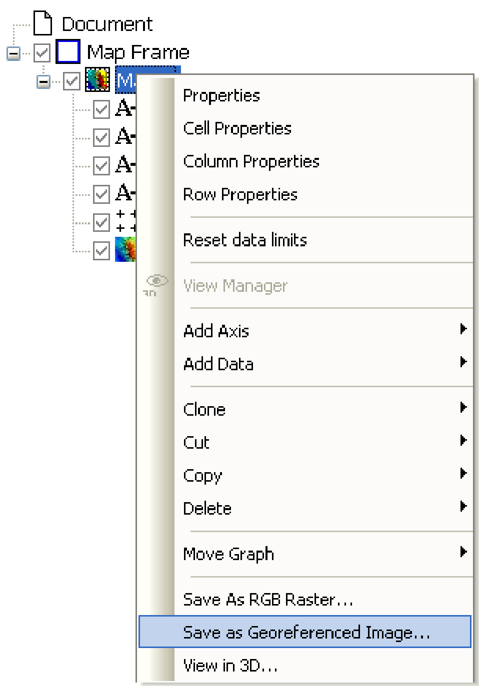
Creating Located Images

Located images can be created from any 2D map, profile, section or graph display in Encom Discover PA either by direct export or via the batch hard copy mechanism. They can also be exported from grid groups and CDI section groups displayed in 3D. You can use a supplied wizard to create Located Images. Finally, you can use a text editor to build simple .EGB files based on the examples provided with the software.

Located Images Direct From 2D Displays

Use the **Save as Georeferenced Image** option available from graphs, profiles, sections and 2D map displays.

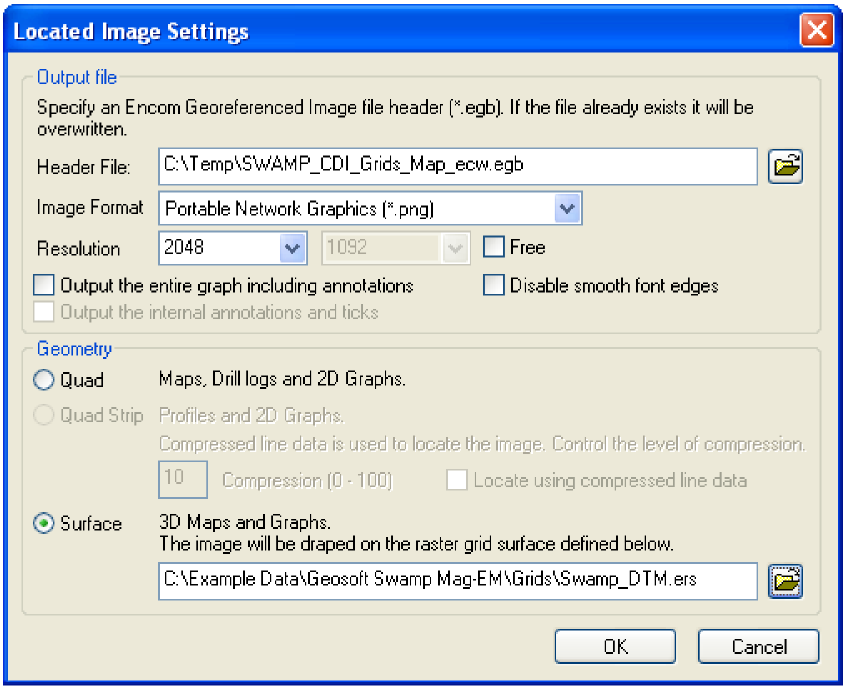
From a Grid object in a 2D Map (or profile or section etc), position the cursor within the window and click the right mouse button to display a pop-up menu (Ensure the cursor is in Pointer mode. Click the **Pointer** button if not or click the left mouse cursor while the cursor is in the display window). From the menu select the **Save As Georeferenced Image** option (as shown below).



Pop-up menu selection for saving a Located Image from a display or the Workspace tree

An alternative to this method is to select the required Map in the Workspace tree, click right and select the **Save As Georeferenced Image** option from the pop-up menu. See *Appendix E: Supported File Formats* for supported formats.

Note Located Images comprise the captured bitmap plus a locational header file (the .EGB) and so resolutions can be specified in the range from 256 to 32768 dpi. The higher the resolution, the better the quality of the original data representation, but the larger the file required to store the information.



Dialog to save a located image (.EGB) file for a display.

Located images can also have three different types of geometries depending on their intended use. These geometries are:

- **Quad** – This geometry is used for display in 2D profiles and graphs and has an entry in the associated .EGB header that references the Line number from which the Located Image is created. This means that when this type of Located Image is used with profiles and graphs, the Line Iterator can be used to update and navigate through the bitmap displays.

- **Quad Strip** – This type of Located Image also adds Line Number references in the associated .EGB header but also specifies the XYZ corner points of the bitmap such that it can be displayed in both profile/ graphs AND in 3D line displays. To assist in fast 3D display, a Compression can be applied to these bitmaps. The higher the compression applied, the faster the 3D display can be redrawn but with a reduction in resolution.

- **Surface** – This Located Image geometry places a grid surface specification in the associated .EGB header. When displayed in a 3D Map, the grid surface is then applied to the captured bitmap to simulate an undulating surface with the associated raster grids shape. An example of this is shown below where a scanned Located Image has a topography raster grid used to define its displayed shape.

Located Images can be exported directly from a 3D surface (Grid Group). The result will be a Located Image that exactly replicates the surface (including surface compression and display properties such as lighting). The advantage of using a Located Image to display this data is the high resolution of the image that can be rendered on the surface.

Located Images can be exported directly from a 3D Section Group via the **Export** option on the **Options** property page. All sections currently displayed will be exported using the current colour and transparency options. Using a Located Image removes the need to access the CDI database and improves performance.

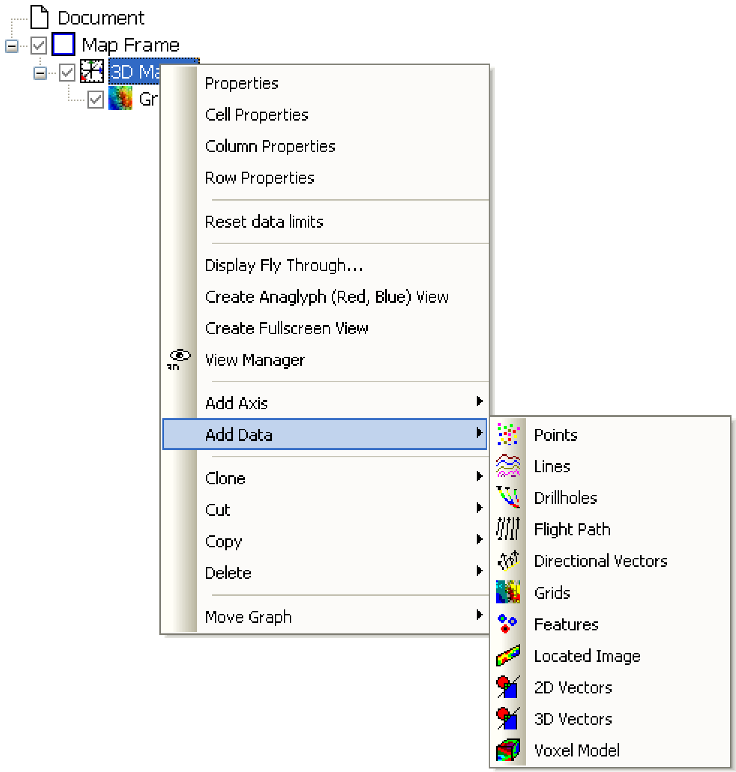
Using Located Images

Located Images can be used in profiles, sections, 2D and 3D displays. These objects are especially powerful in 3D displays as they can be grouped and many hundreds of objects can be used with optimised display performance.

3D displays of Located Images are usually added to a 3D Map although they can also be added to 3D Graphs, 2D maps, profiles and sections. In the description below, Located Images are added to a 3D display but the steps required to add Located Images to the other display types are essentially identical. The required procedure is:

Step 1

Create a 3D Map or 3D Flight Path Map using the **File>New>Map>3D Grid Map** or **3D Flight Path Map** menu option (or from the appropriate buttons).



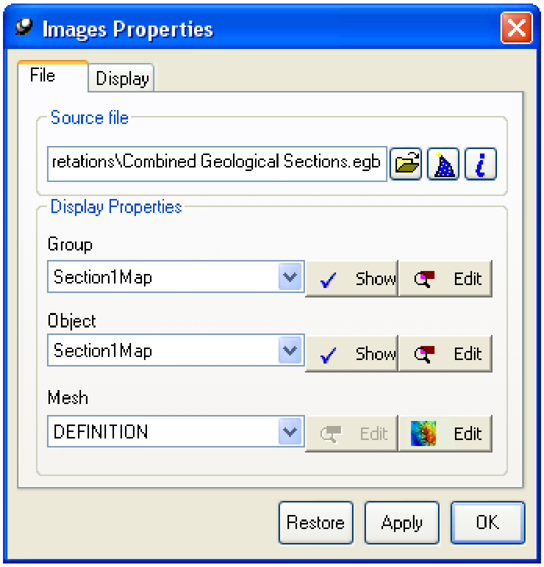
Step 2

With a 3D Map branch created in the Workspace tree, select the 3D Map item and click the right mouse button. From the **Add Data** menu item, choose **Located Image**. An alternative to this is to click the **Add Located Image** button from the Objects toolbar.

As described in *Creating Located Images*, you need to have prepared the necessary Located Image before display and these require associated .EGB descriptor files. Refer to *Image Registration Wizard* and *Appendix B: Data File Specifications* for Encom Georeferenced Bitmap Format.

Step 3

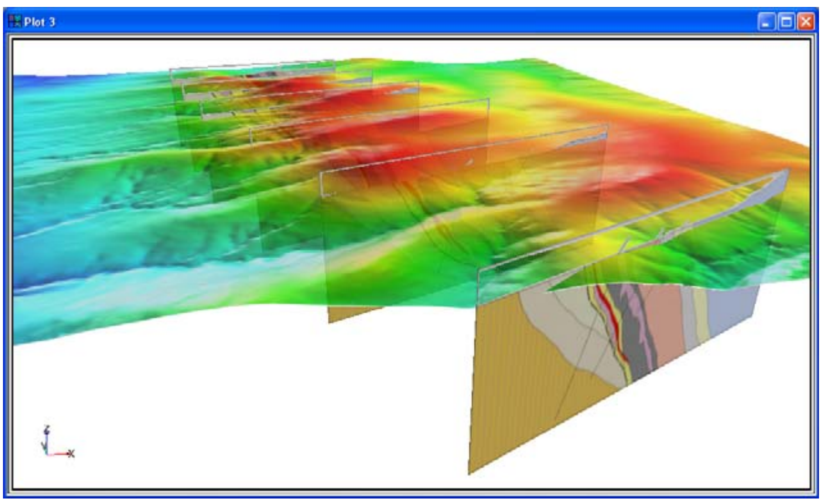

Display the properties dialog of the Located Image branch. Initially specify the path and .EGB file. You can use the **Browse** button to navigate to the required .EGB file or the **Wizard** button to access the Located Image Registration Wizard.



Located Image properties dialog to specify the EGB file and display options

Refer to *Located Image Properties* for details on the various control settings.

An example of a display using Located Images is shown below.

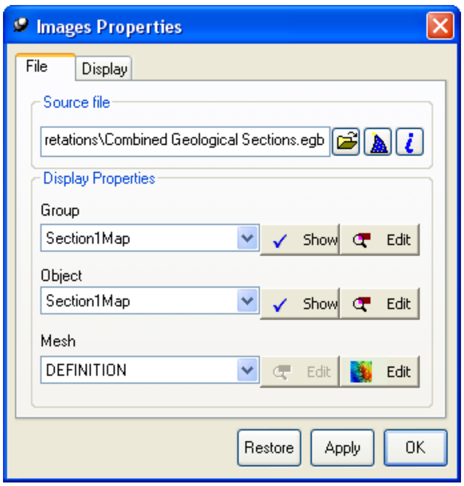


Multiple images displayed with a 3D elevation grid surface

Located Image Properties

When a Georeferenced Image is displayed in Encom Discover PA 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 Images 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 combining images of geological sections georeferenced using the *Image Registration Wizard* is selected:



Images Properties dialog to specify the EGB file and display options

The image source path and file is listed under the **Source File** section. If you need to display an alternative geo-referenced image, use the **Browse** button (shown left). The Image Registration Wizard is also available from this dialog (button shown left).

Configuring and making use of the powerful features of Located Images requires use of the Properties dialog. Note that all the properties of a Located Image can be specified within the .EGB file (if used) and these properties will be used by default. The property pages can be used to override those defaults. Controls for Located Image display is divided into:

- **Group** – To control the collection of objects that may exist in the definition of .EGBs. For example, a collection of tree objects may form a forest group.

- **Object** – Individual entity control. Individual objects may be the trees of a forest.
- **Mesh** – Each object is built from one or more meshes, each of which can be textured with an image. These form the lowest level of Located Image control. For example, a tree object will be displayed with a textured group of surfaces forming the 3D representation of a tree.

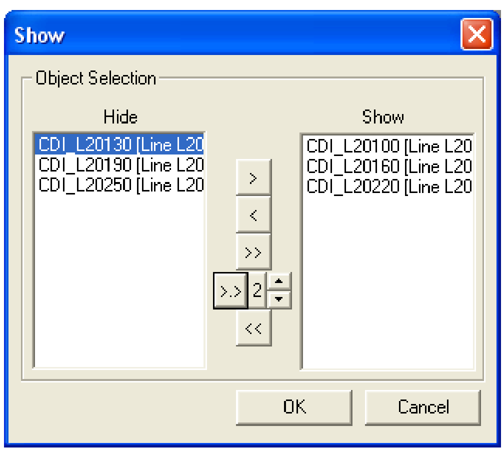
Meshes can be of three types depending on their end-use:

- **Quad** – A simple bitmap that has four corners located in space. These corners need not be planar.
- **Quad_Strip** – A group of joined Quad meshes in either a vertical or horizontal orientation form a Quad Strip.
- **Triangle Mesh** – This mesh is a collection of vertices in space joined by triangular facets. Each facet can be displayed with different shading and colour to form a surface. The mesh may be open or closed or may contain multiple distinct patches.

Further definition information on meshes and the .EGB format is described in *Appendix B: Data File Specifications*.

Group Controls

The **Show** button allows selection of the Group to be displayed:



Selection of items defined in a Group EGB with multiple groups

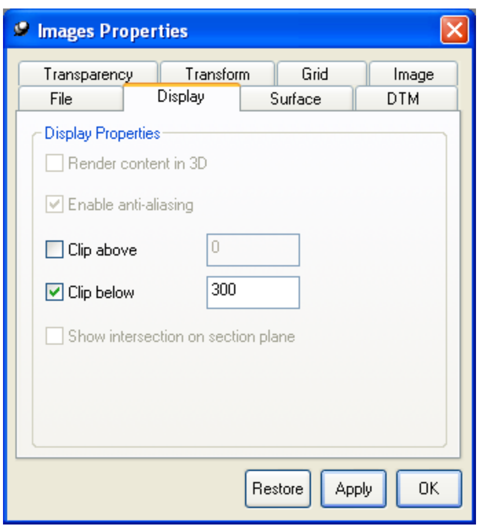
The Group **Edit** button allows the Surface and vertical position (DTM tab) of the Located Image to be configured.

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

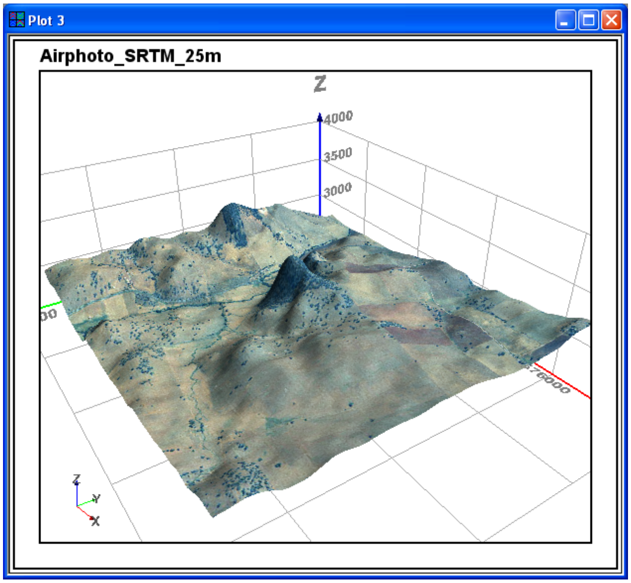
- File Tab – initially displayed, allows file selection, group controls and image registration.
- *Display Tab* – provides clipping options.
- *Surface Tab* – provides options for image colour, modulation and lighting.
- *DTM Tab* – allows the image to be offset using a digital terrain model (DTM).
-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

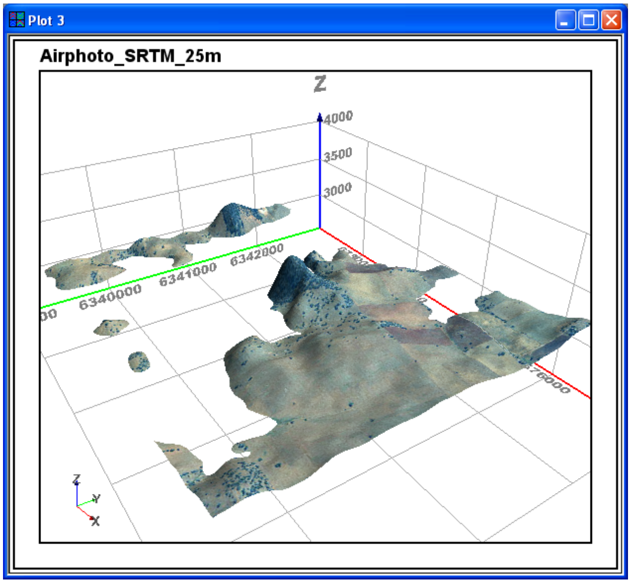
The Display tab allows the image to be clipped above and/or below two user-specified horizontal planes (i.e. Z values).



For instance, if a Clip below value of 300 was set, an airphoto draped over a DEM grid would have all portions of the image with Z value less than 300 hidden.



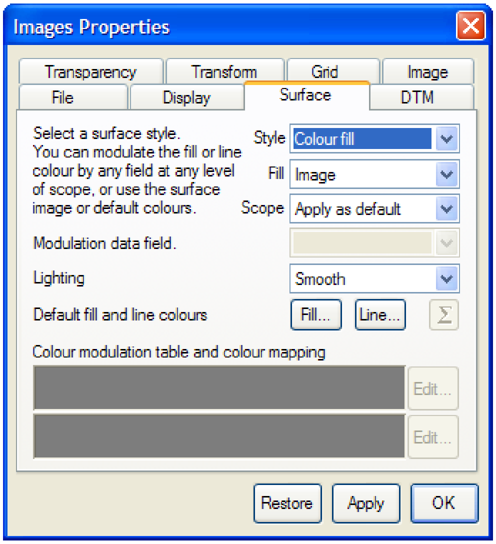
An airphoto draped over a DEM grid



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

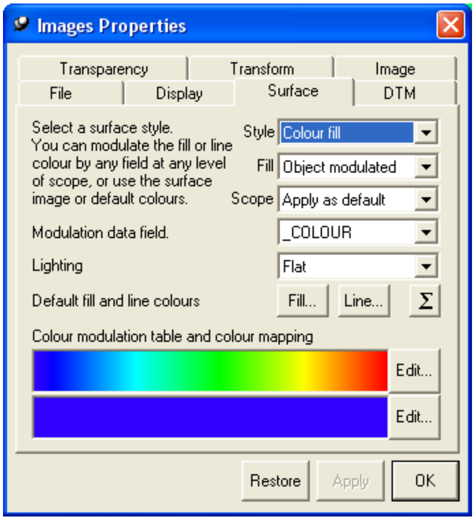
Parameters that can be controlled include:

- **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.

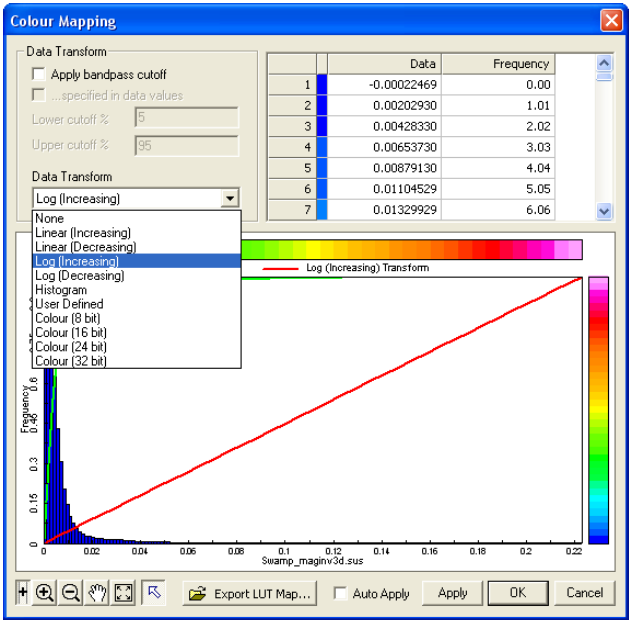
- **Edit of colour modulation table** – Control of the colour look-up table is provided.

- **Edit of 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.



Surface tab defining colour usage and appearance of Located Images

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.



Colour Mapping transform dialog

The Colour Mapping dialog 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 cutoff as required.

Available transforms include:

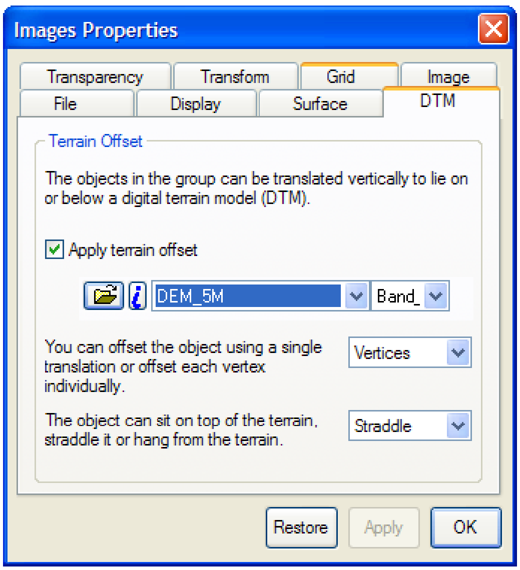
- **None** – The source data is used as an RGB colour value directly with no modification or reference to the colour table.
- **Linear (Increasing)** – Within the prescribed colour range (as determined by the Band-Pass Cutoff) the data (from the attributes of the voxel model) are linearly stretched from 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.

- **Manual** – 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 allows the objects to be vertically offset by a DTM (Digital Terrain Model) grid.

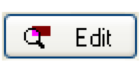
The offset can be computed for each vertex of all meshes in the object or an offset for the centre of the object can be computed and applied equally to all vertices. The object can sit on top of the DTM, hang from it or be centred on the DTM.



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

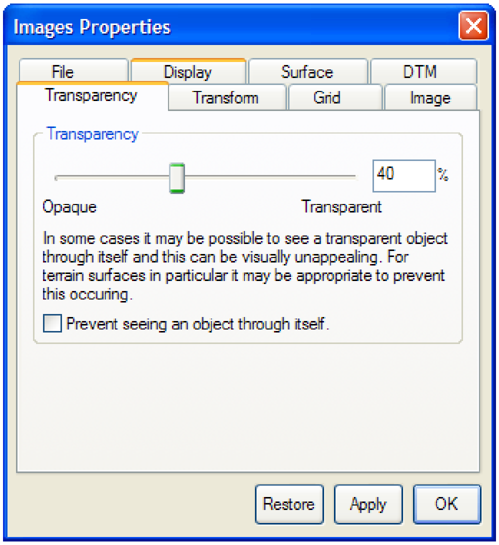
Object Controls

The Object Controls have object selection from the **Show** button similarly to Group selection but in this case, individual objects of a specified Group are being chosen.



The Object **Edit** button gains access to the Transparency and Transform tab dialogs to provide transparency control to individual objects or all in a group and additional scaling and translation transformations.

Transparency Tab



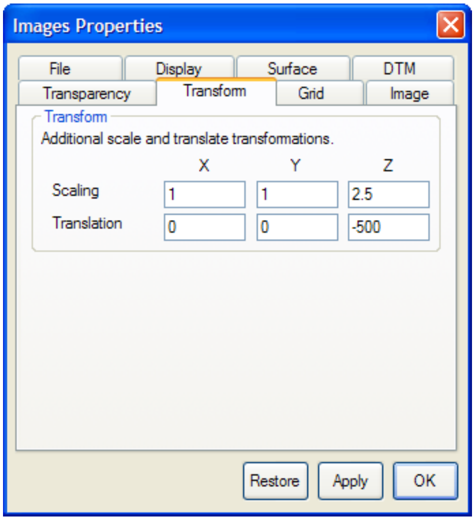
The Transparency property tab to control the transparency of located images.

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).

The transparency setting can be applied to a single object, all in a Group or All objects irrespective of the Group. 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.

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. These controls are provided for convenience but more powerful options are available via the .EGB file.

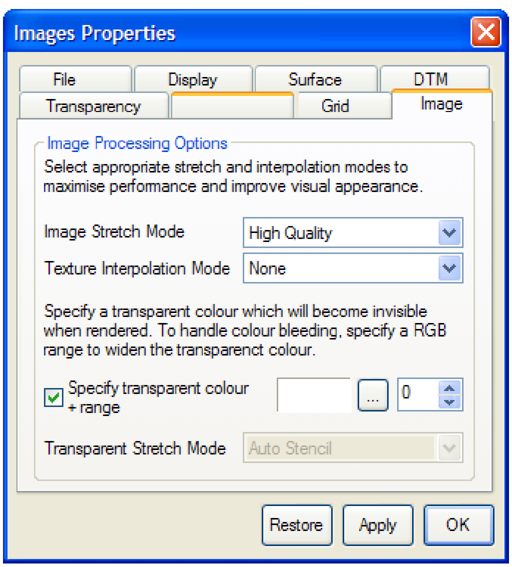


The Transform property tab to control scaling and offset of located images.

Mesh Controls

The Mesh properties has an **Edit** button to provide access to the **Image** tab which controls the image stretch, interpolation quality control plus the automatic setting of a specific colour or range of colour in which transparency is enabled.

Image Tab



Mesh controls for quality and transparency of a colour

In many cases a colour (such as white) is set to be transparent since white frequently is used as the border around a captured image. The transparency option allows the colour to be clear when displayed.

In other circumstances (for example if the image is derived from a scanned bitmap), it is helpful to have a range of colours to be made clear. A buffer of colour can therefore be determined to be clear up to the specified colour. For example, if white is selected as the default clear colour, (RGB colours of 255:255:255) and a buffer colour range of 5 is defined, then the colours made clear (in addition to white) are 250:255:255, 251:255:255….255:255:254.

Two display options are available for the Located Image for **Image Stretch** and **Texture Interpolation Modes**.

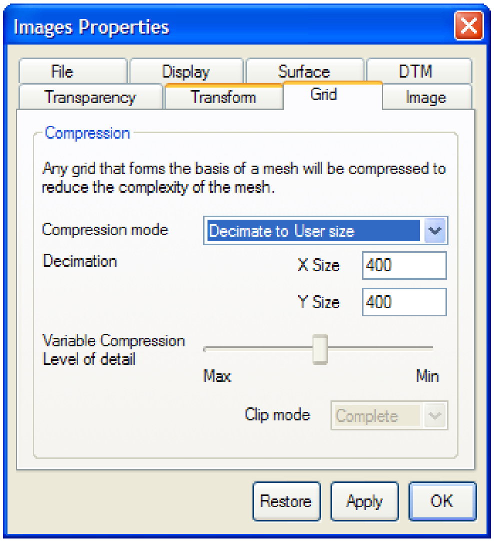
Image Stretch Mode allows for the display of specific colours plus Black and White or for these to be displayed in the highest possible quality. **Texture Interpolation Modes** of an image can be:

- **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.

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 PA 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 PA 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.

17 Features

In this section:

About Features

Note All options within the Features menu (except **Open Existing FDB**) are only accessible with a PA Professional licence.

The Encom Discover PA Features menu provides access to a powerful set of utilities for creation, storage, manipulation and display of feature objects (anomalies).

A Feature Object is a three dimensional geometric entity represent simple geometric entities such as points and lines, or more complex 3D objects such as polylines, polygons, open surfaces and closed surfaces (i.e. polyhedrons). Feature objects 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. Feature Objects can be created from directly within Encom Discover PA.

These Feature Objects are grouped together into Feature Sets and stored within these sets in a Feature Database, which is a standard Microsoft Access database format with a .FDB file extension. Encom Discover PA Professional has the ability to open and maintain many feature databases concurrently. Accompanying attribute information is stored in the Feature Database for each feature object and the database can be queried based on these attributes.

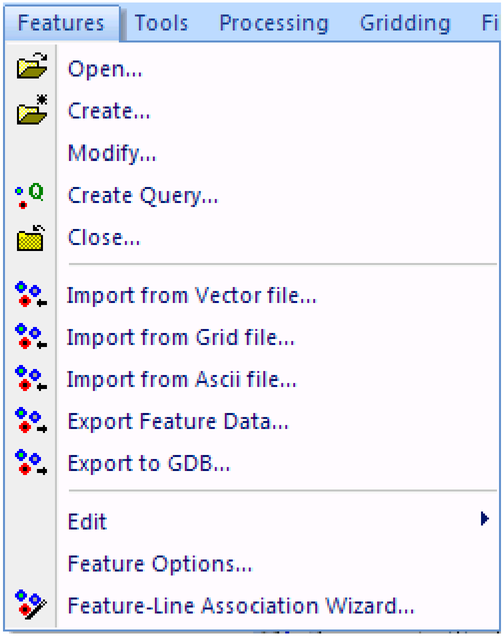
Aside from viewing in Encom Discover PA Feature Objects can be used with the *3D Solid Generator* utility. The 3D Solid Generator joins a number of Feature Objects together to produce a three-dimensional enclosed or planar surface; i.e. a solid body to represent an ore body zone or fault plane. Volumes can be calculated for enclosed 3D Solid bodies. Feature objects can also be used with the *3D Extrusion Tool* to create 3D solid bodies from digitised polygons. The Extrusion Wizard can use polygonal objects from a Feature Set as a primary surface which can be extruded between another surface, defined by a grid surface, a constant Z value or a field contained within a database. With both of these utilities volumes can be calculated for the generated enclosed 3D Solid bodies. For more information on these utilities please refer to their corresponding sections in *Display Utilities*.

Managing Features

Features are stored and maintained in a database controlled by Encom Discover PA. The feature management system:

- Can open and maintain many feature databases concurrently.
- Can have a database containing multiple Feature Sets within which feature objects can be grouped.
- Can contain a Feature Set class of features used to categorise feature objects with some aspect in common (for example, features derived from deep conductors, cultural sources, detected dykes etc). Feature objects within a Feature Set have the same display symbol attributes although these can be modulated for size, colour and rotation depending on selected data fields.
- Allows each Feature Set to be associated with data fields derived from the originally used data source. This is especially useful for feature analysis since later evaluation of the feature may depend on re-examining the initial criteria for selecting the feature methods. Such as EM decay curve analysis use this approach.

Features Menu



The Features menu options

The Features menu contains the following functionality:

- **Open**: allows an existing Feature Database to be opened into the current session (see *Opening an Existing Feature Database*)

- **Create**: creates a new empty Feature Database, including any custom attribute fields specified by the user (see *Creating a New Feature Database*).

- **Create Query**: allows attributed Feature Databases to be queried using a simple SQL syntax (see *Querying Feature Attributes*)

- **Close**: closes selected open Feature Databases.

- **Import from Vector File**: allows a range of vector file formats to be imported into either an existing Feature Database or new Feature Database (see *Importing Data from Vector or Grid Files*).

- **Import from ASCII File**: allows a range of ascii file formats to be imported into either an existing Feature Database or new Feature Database (see *Importing Features from ASCII Files*).

- **Export to Vector File**: exports feature objects from a Feature Set from an open Feature Database to a range of industry standard vector file formats (see *Exporting Feature Sets*) Some example include AutoCAD .DXF MapInfo .TAB, Gemcom .BT2 and ESRI .SHP. Feature objects can also be exported to MapInfo .TAB files which are directly compatible with a Discover Drillhole Section Boundary file.

- **Export to .GDB File**: exports feature objects from a Feature Set from an open Feature Database to a Geosoft Database (.gdb) file (see *Exporting Feature Sets*).

- **Feature Spreadsheet**: displays a spreadsheet view of feature objects listed according to their ID number and associated fields (see *Feature Spreadsheet*).

- **Edit** – allows the user to Combine, Intersect, Erase, Set Target, Clear Target, Resize, Delete, Cut, Copy and Paste feature objects (see *Editing Objects*).

- **Feature Options**: contains controls for setting the default polygon fill, selection and query highlight colours (see *Feature Options*).

- **Feature-Line Associate Wizard**: allows feature objects to be associated with line numbers from a loaded Geosoft or Intrepid survey database (see *Line Association Wizard*).

Feature Databases

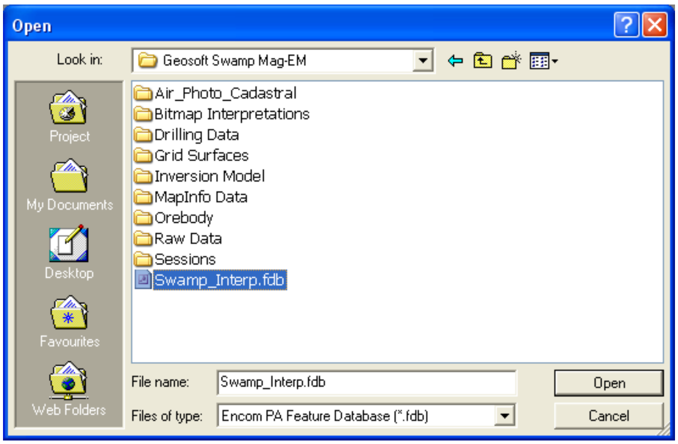
In this section:

- *Opening an Existing Feature Database*
- *Interacting with Features*
- *Creating a New Feature Database*
- *Adding a New Feature Set to a Database*
- *Feature Calculator* – provides the ability to create new fields in a feature database based upon user-defined calculator expressions.
- *Deleting a Feature Set*

Opening an Existing Feature Database

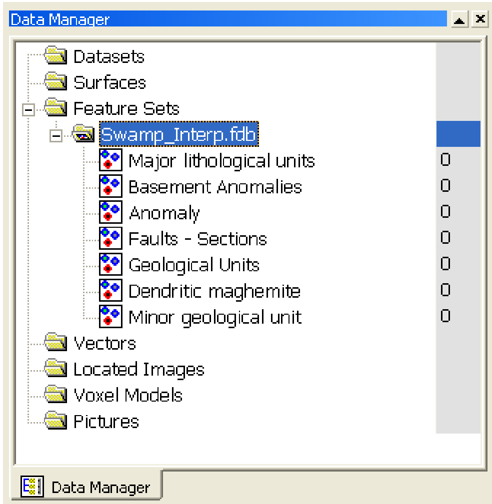
Feature Objects are stored and maintained in a Feature Database. A Feature Database is a relational database that supports complex objects and associated attributes. A Feature database can be identified by its .fdb file extension. Encom Discover PA has the ability to open and maintain multiple feature databases concurrently.

To open a pre-existing **Feature Database** select the **Features>Open…** menu option. The **Open** dialog is will be displayed.



Use the Browse window to locate an existing feature database

Browse to the location of an existing Feature database and click **Open**. A new Feature Database will be added to the Data Manager window under the **Features** folder and all associated Feature Sets belonging to the database will be listed beneath it.



A loaded Feature Database will be shown in the Data Manager window

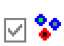
To close a **Feature Database** in an Encom Discover PA session select the **Features>Close** menu option. In the **Close Feature Databases** dialog that appears select the database(s) to close and click the **Close** button. An alternative to close a feature database is to click with the right mouse button in the Data Manager window on the Feature Database branch that you wish to close and select **Close Database** from the pop-up menu.

If you want to close multiple open feature databases click with the right mouse button on the **Feature Sets** folder of the Data Manager and select the **Close All Unused** pop-up menu option.

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:



Visability - toggles the visibility of the Feature Objects in the 3D window on and off.

Editability – enables editing for the selected Feature database. With editing mode enabled new objects can be added to the database and existing objects can be edited or deleted from the Feature database. Only one Feature database at a time can be made editable.

Selectability – enables objects 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 extents of the extents of the objects 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 options - 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*).

- *3D Solid Generator* – 3D polyline or polygon objects can be triangulated to produce open surfaces (e.g. fault planes) or closed surfaces (e.g. solid closed meshes) using the 3D Solid Generator. These triangulated objects can be output directly to an existing or new Feature database.

- *3D Extrusion Tool* - various 2D or 3D objects, including points, lines, polylines and polygons can be extruded into triangulated 3D mesh objects using the Extrusion Wizard. The output objects from the Extrusion Wizard can be directed into a new or an existing Feature database.

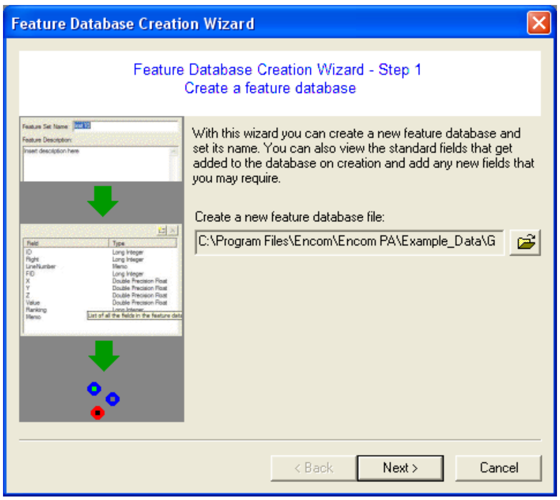
Feature Database Creation Wizard

The Feature Database Creation Wizard creates a new empty Feature Database.

1. To start the wizard, select the **Features>Create** menu option.

2. Complete the options displayed at each step of the wizard.

Step 1: Database name and location

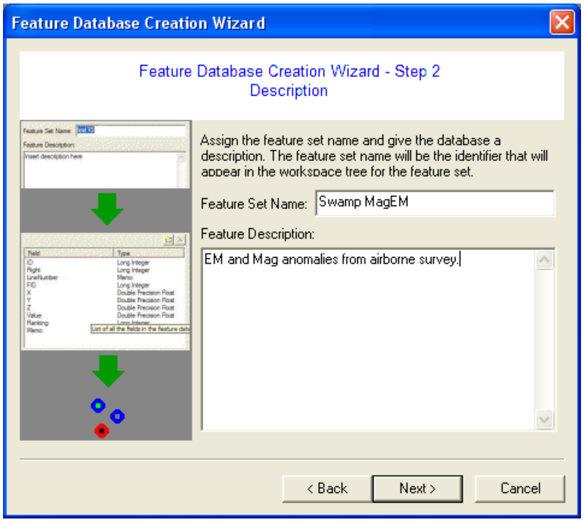


Step 1 of the Feature Database Creation Wizard – Feature Database naming and location

- 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** once the database is defined.

Step 2: Feature set name

- Enter a name for the first Feature Set belonging to the database. If required also enter a description for this Feature Set, such the characteristic that will be used to group the how feature objects together. Additional Feature Sets can be created for this database once the wizard process has been completed.



Step 2 of the Feature Database Creation Wizard - Feature Set naming

Step 3: Define Fields

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. You can nominate additional fields of your own choice, or add automatic fields that are linked to a field in the survey database. Data in these fields are automatically piped from the survey database into the feature database when you create a feature object.

1. To add new fields click on the **Add New Field** button (shown left) in this dialog. Doing so opens a dialog that allows you to name the new field and select the data type from a standard list that includes:

   - **Boolean** – can contain binary information only (i.e. Yes/No).

   - **Byte** – can contain byte values, such as

     Short Integer – can contain small integer values ranging in value from -32768 to 32767.

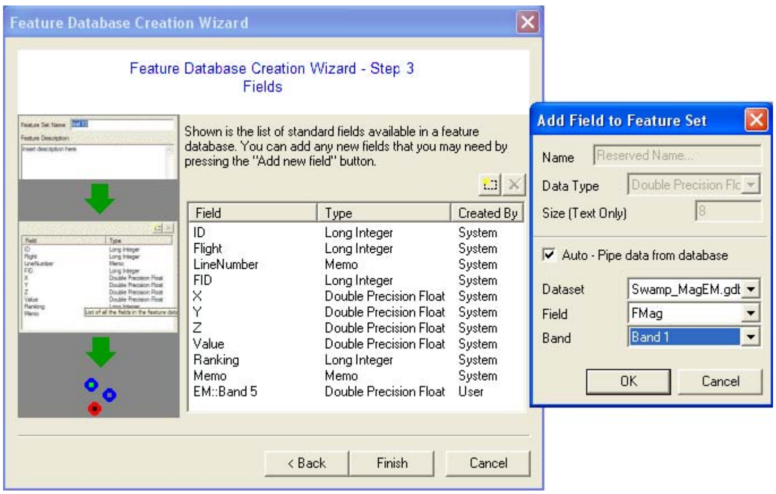
     Long Integer – can contain large integer values ranging in value from -2,147,483,648 to 2,147,438,647 .

     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.



Step 3: Associating survey database fields with Feature Sets

When specified, the data fields are displayed in the **Fields** tab. Note that these automatic fields are created by Encom Discover PA and you cannot modify the name or data type.

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.

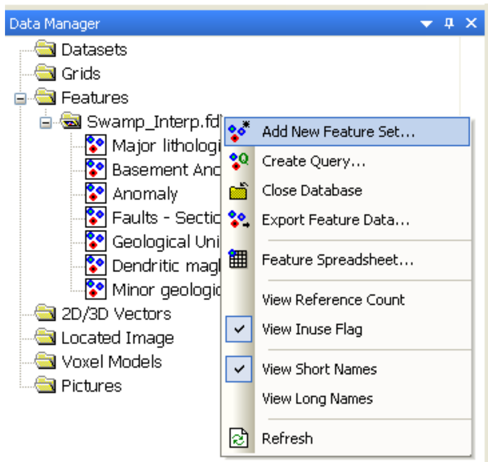
Other fields, such as Ranking and Memo are also added to the Feature Set. These are character type fields that allow you to better organise your features.

2. Click **Finish** to create the new Feature Database. A new Features item is added to the Data Manager window.

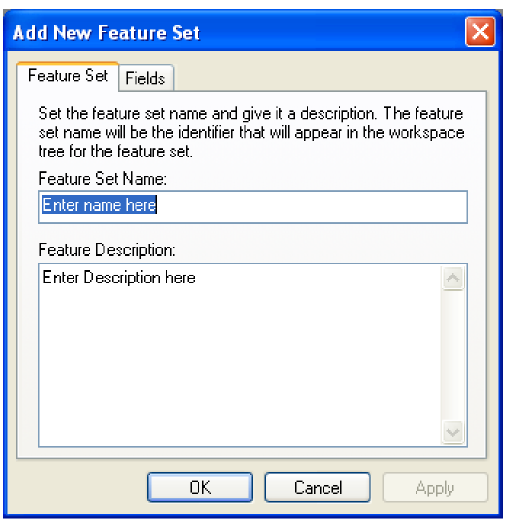
See *Creating Objects* for information on how to add features to a database.

Adding a New Feature Set to a Database

To add a new Feature Set to an existing Feature Database, highlight the database branch in the Data Manager window and click the right mouse button to display a pop-up menu. Select the **Add New Feature Set** option from the pop-up menu as illustrated below:



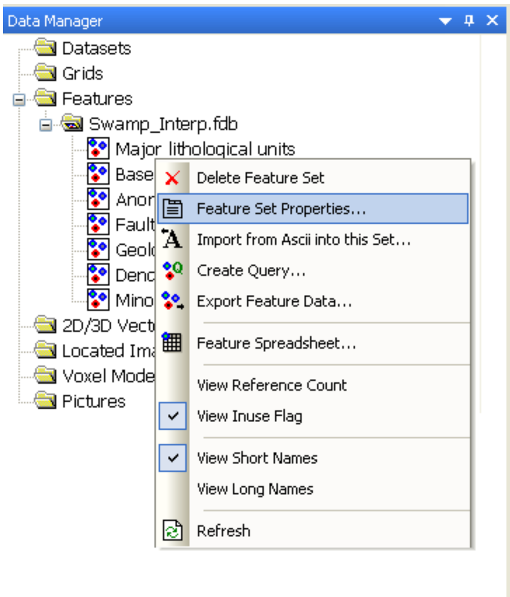
Add a new Feature Set using the Data Manager window



The Add New Feature Set dialog.

The **Add New Feature Set** dialog will appear. Enter a name for the new Feature Set and if required a description as well. To add additional data fields to the new Feature Set, select the **Fields** tab in this dialog and add additional fields as explained in the previous chapter. Click the **OK** button and a new Feature Set branch will be added to the Data Manager underneath the feature database branch.

If additional fields are required to be added to this Feature Set after the dialog has been closed, return to this dialog by highlighting the Feature Set branch in the Data Manager and clicking on the right mouse button to display a pop-up menu. Select the **Feature Set Properties** pop up menu option and the same dialog will appear again.



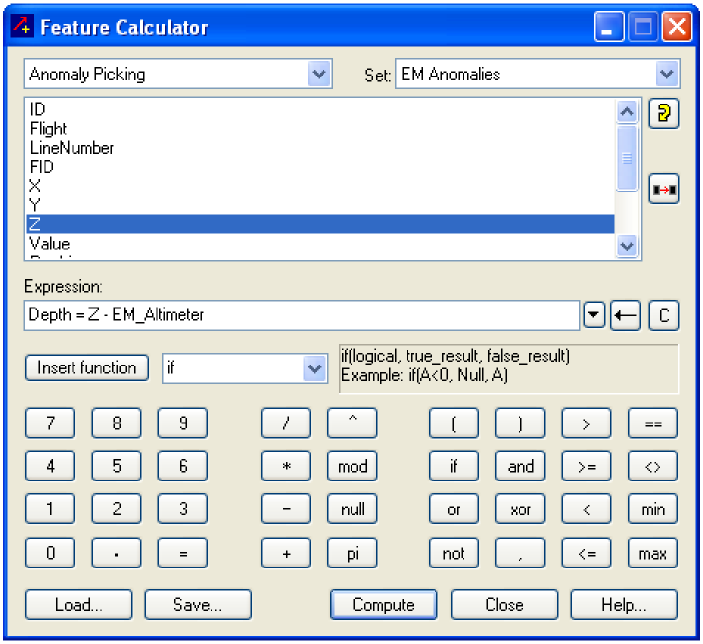
Add additional fields to a Feature Set using the Feature Set Properties pop up menu option

Feature Calculator

Note Encom Discover PA Professional is required to operate this tool. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

The Feature Calculator can apply arithmetic operations to data fields of a Feature database. You can initiate the **Feature Calculator** from the **Utilities** menu.

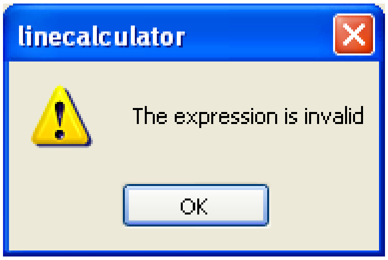
The Feature Calculator interface appears similar to the Line Calculator and its operations in general are similar. The interface for the Feature Calculator is:



Feature Calculator dialog with feature computation formula

The Feature Calculator requires a formula to be specified in the command line at the top of the dialog. Available data fields appear in the scrollable list from the nominated Feature database. Complex data field operations such Boolean, comparative, logical and decision functions can also be applied. These operations are described in *Calculator Syntax*.

Clicking individual operation buttons causes their display to be automatically placed in the command line. You can either enter formula directly into the command line by typing from the keyboard, or by selecting the appropriate fields and buttons. Note in all cases that a correct arithmetic formula must be written. If the calculator detects errors in the command entered, a message indicating an invalid expression is displayed.



Buttons along the base of the dialog allow you to **Select** a field (after highlighting), **Load** or **Save** a formula, or **Compute** the displayed formula. The formula specification is as for normal algebraic operation.

The **Insert Function** button allows you to construct complex formulae. A wide range of arithmetic, logic and comparative operations are available with the Line Calculator. A subset of the formulae are provided by the pull-down list that provides the various operators and an example of their use. Refer to *Special Line Calculator Operations* for a list of the operators available.

Calculator Syntax

For the Line, Grid and Feature Calculators, a series of advanced Boolean and algebraic operators are available to allow 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.

Below are described a series of operators that, when used with calculators, will perform various operations. Note that in the case of the Line and Feature Calculators, the output (referred to as the parameter Out) may be a newly written database channel. In the case of the Grid Calculator, the output is a supported grid.

- *Logical Operators*
- *Arithmetic Operators*
- *Anti-Log Functions*
- *Comparison Operators*
- *Boolean Operators*
- *Trignometric Operators*

Logical Operators

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 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` is greater than 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 examples 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.

Arithmetic Operators

To perform basic mathematical operations such as addition, subtraction, or multiplication; and produce numeric results, use the following arithmetic operators.

Function	Operator		Example
addition	`+`	plus sign	`A+B`
subtraction	`–`	minus sign	`A–B`
multiplication	`*`	asterisk	`A*B`
division	`/`	forward slash	`A/B`
exponentation	`^`	caret	`A^B`

Anti-Log Functions

To perform antilog 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.

Function	Operator	Example
equal to	`=`	`A=B`
greater than	`>`	`A>B`
less than	`<`	`A<B`
greater than or equal to	`>=`	`A>=B`

less than or equal to	`<=`	`A<=B`
not equal to	`<>`	`A<>B`
minimum	`MIN(value1, value2)`	`MIN(A,B)`
maximum	`MAX(value1, value2)`	`MAX(A,B)`

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.

Trignometric Operators

The following trigonometric operators are available for both Line and Grid Calculators.

`DEGTORAD(degree_value)` – Conversion of degree values to radians.

`RADTODEG(radian_value)` – Conversion of radian values to degrees

`SINH(a)` – Hyperbolic sine of an argument in radians (`Y = sinh(x)`)

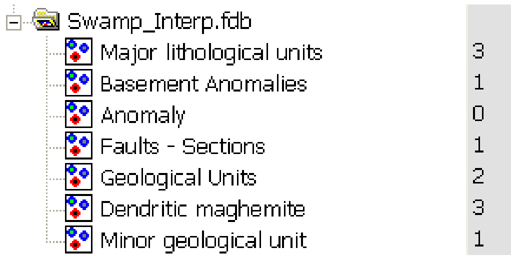
`COSH(a)` – Hyperbolic cosine of an argument in radians (`Y = cosh(x)`)

`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.

Deleting a Feature Set

A Feature Set can be deleted from a Feature Database provided that the Set being deleted is not active in any display windows of Encom Discover PA. An easy way to check this is by referring to the Data Manager window. A record of how many entries of each Feature Set is being used in display windows is recorded in a column located to the right of each Feature Set branch.



To delete a Feature Set from a database highlight the Set in the **Data Manager** window and click on the right mouse button to display a pop-up menu. Select the **Delete Feature Set** menu option and a confirmation message is displayed.

If you wish to continue deleting the Feature Set click OK. The Feature Set will be permanently removed from the Data Manager hierarchy tree.

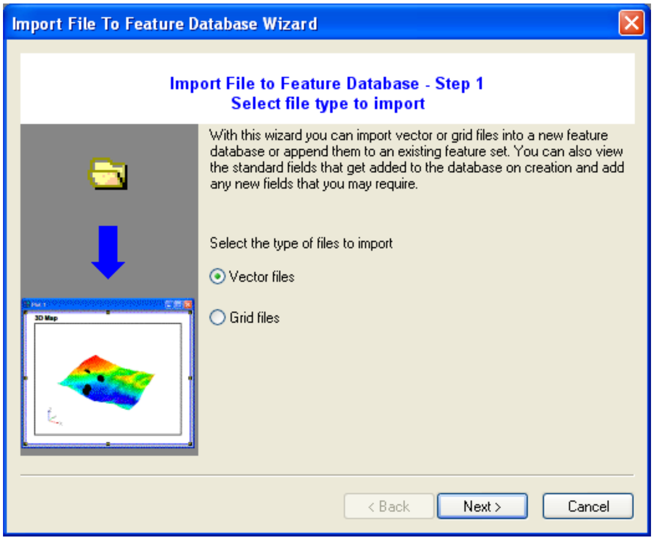
Importing Data Into a Feature Database

- *Importing Data from Vector or Grid Files*
- *Importing Features from ASCII Files*

Importing Data from Vector or Grid Files

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 From Vector file** or **Features>Import from Grid file** menu option.

Step1: Select file type



The Import File to Feature Database Wizard

Select the type of files to import. Two options are provided:

- The **Vector files** option supports common industry file formats including DXF, MapInfo .TAB, ESRI .SHP, Gemcom .BT2. You can import any contained layers into separate features.
- The **Grid files** option supports importing Grid files, such as ER Mapper, Surfer, Geosoft etc into a triangulated irregular network surface (TIN).

Press **Next** after the required format is selected.

Step 2: Select source and destination files

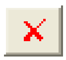
Vector files

- Click the **Browse** button to select a single source file.

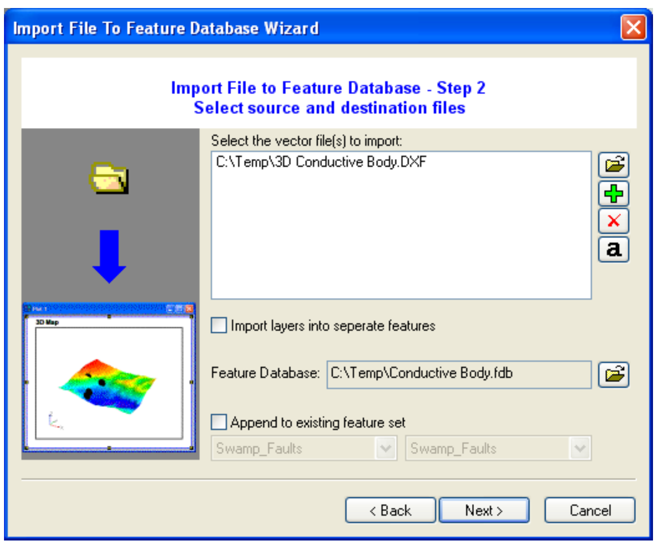
- Multiple files can be imported simultaneously by selecting them consecutively with the **Add Files** button.

- 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; 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



- Individual vector files can be removed from the Import list by highlighting them and selecting the **Delete Files** button.
- 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.



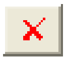
Importing a DXF file into a new feature database called 'Conductive Body'

Click **Next** to advance the wizard.

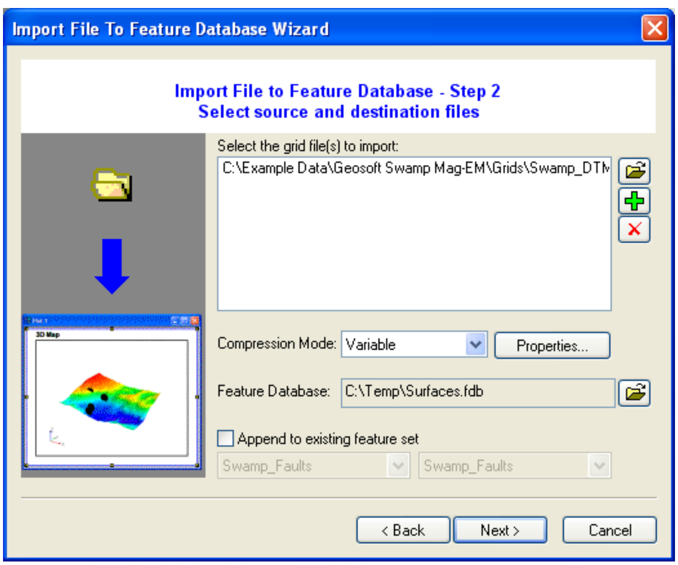
Grid files

- Click the **Browse** button to select a single source file.

- Multiple files can be imported simultaneously by selecting them consecutively with the **Add Files** button.



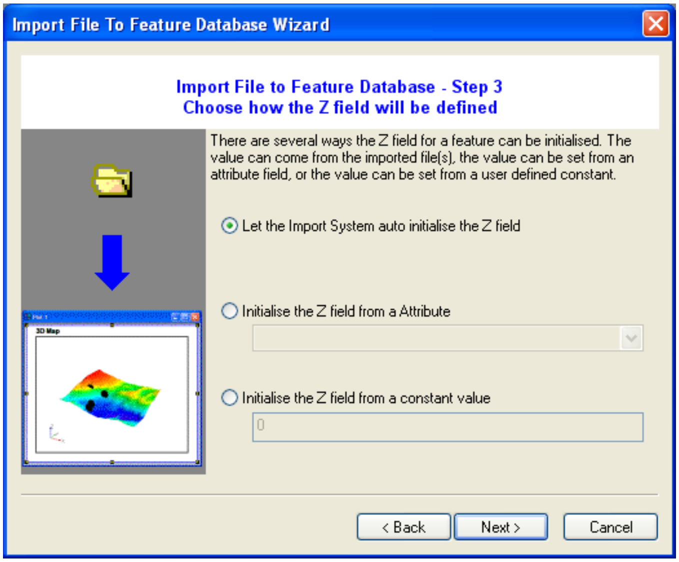
- Individual grid files can be removed from the Import list by highlighting them and selecting the **Delete Files** button.
- A number of **Compression Modes** are available to facilitate efficient importing and handling of larger grids. See the *Surface Compression* section for a full explanation of 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.



Importing a grid file into a new feature database called 'Surfaces'

Click **Next** to advance the wizard.

Step 3: Assign the Z field



The final dialog of the Import wizard, controlling the Z value assignment.

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.

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), Encom Discover PA 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. The user can either:

- **Continue the import** – all surfaces will be displayed in Encom Discover PA, but using some of the Feature Object editing tools (see *Editing Objects*), 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.

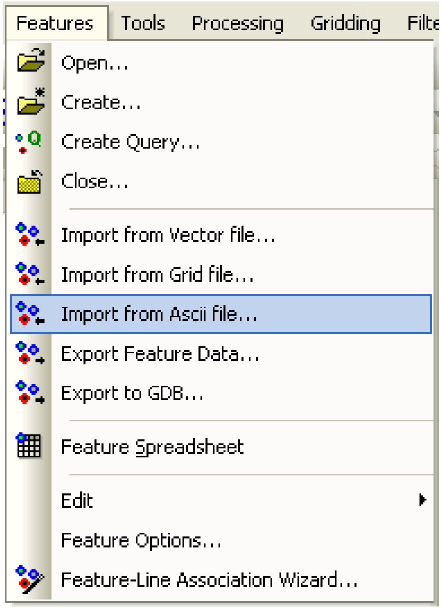
Importing Features from ASCII Files

The Feature utility can also import ASCII file data in .CSV format only. The format of these files is described in *Appendix B: Data File Specifications*.

ASCII data can only be imported into a pre-existing feature database that is loaded into Encom Discover PA. Refer to Creating a new Feature Database for more information on creating a feature database.

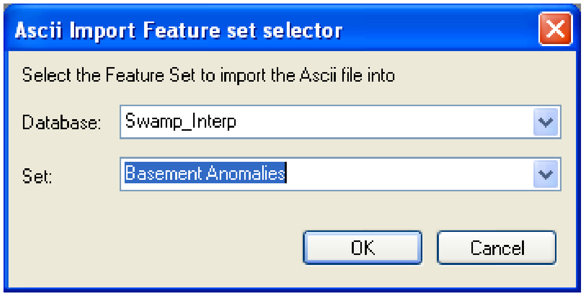
To import feature data in ASCII format into a loaded feature database:

1. Select the **Features>Import from ASCII file** menu option.

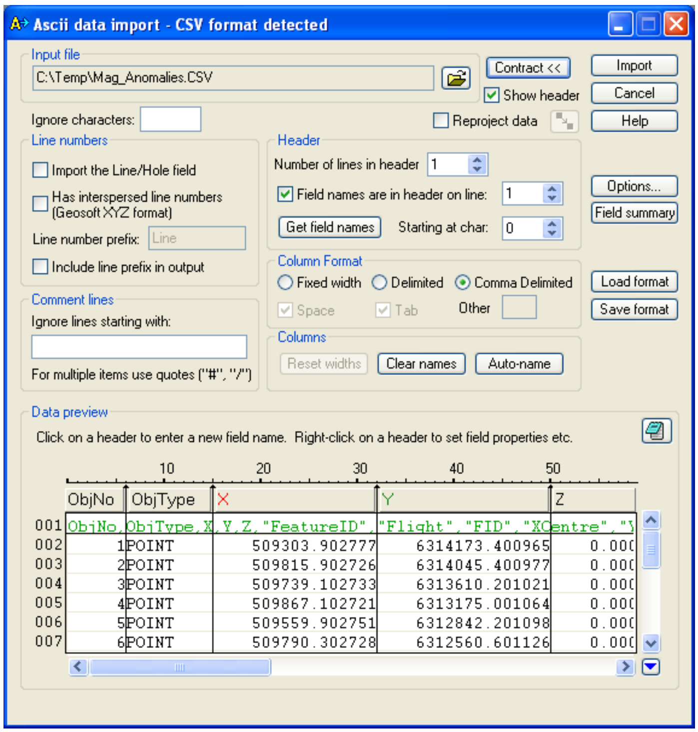


The 'Import from Ascii file' menu option.

2. An existing Feature Database and associated Feature Set must be selected from the first dialog that appears. This will be the file that the objects in the ascii file will be imported to.



3. The Ascii data import dialog will then appear. Click on the **Browse** button to open a .CSV file. When a CSV file is selected, the column names embedded are used by the import facility and automatically loaded. The import dialog appears as below after the file has been read:

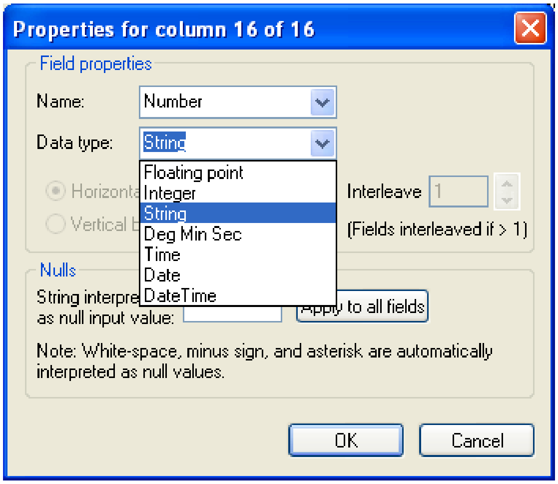


Dialog of import tool after a CSV file with column headers has been read

---

Note If the dialog appears in a condensed format click on the **Expand** button in the top right hand corner to view all parameters.

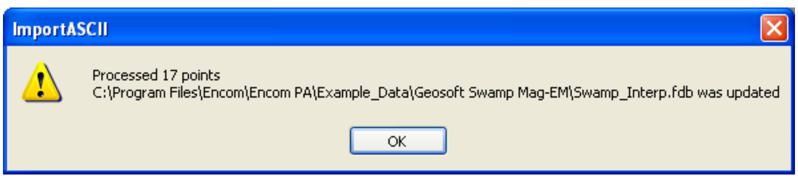
Note If a column in the ASCII file has alphanumeric fields (eg a column with values of 1A, 2A etc) this must be specified as **Strings** rather than the default **Float** data type. To change this data type to String click on the column header with the right mouse button and select the **Properties** pop-up menu option. In the Properties dialog change the data type to String.



Alter the Data type of alphanumeric fields to be String

Also note that you may have to specify null values in the various fields if applicable.

4. When the data format, null specification and columns names etc. are correct, click the **Import** button of the dialog. The contents of the selected file are then converted to a feature database with a resultant summary dialog detailing the imported data.

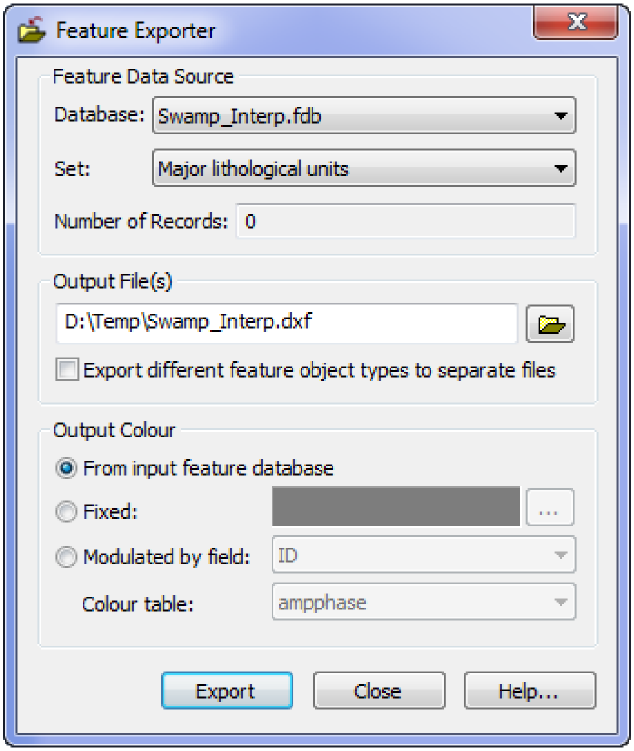


Summary dialog for importing features from an ASCII file

Exporting Feature Sets

The individual records of a Feature database can be exported to a range of output file types.

The export facilty can create output of the specified Feature database, and Feature Set. The utility reports the number of records of the selected Feature Set in the database. When you specify the output data file, the pull-down entry allows selection of the file type:



The Feature Exporter allows a selected feature Set to be exported as one of the following formats:

- Geosoft .GDB
- ArcView .SHP file
- AutoCAD .DXF file
- MapInfo Professional.TAB
- MapInfo Professional .MIF
- ASCII .CSV file

To export a feature set:

1. Choose the **Features>Export** menu option.

2. Specify the Feature **Database** and **Set** plus the output filename and format type.

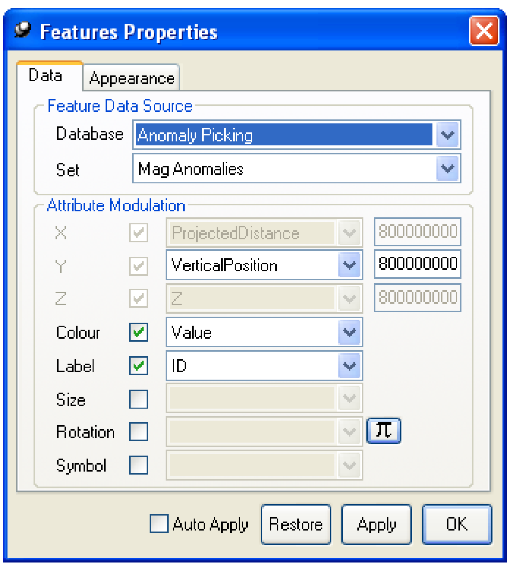
   Note at the base of the **Save As** dialog, you can specify the required export format for the objects in the **Save as Type** option.

3. Click the **Export** button when the required output is specified. The vertices for each feature are listed according to their unique ID number assigned by the software.

Feature Object Properties

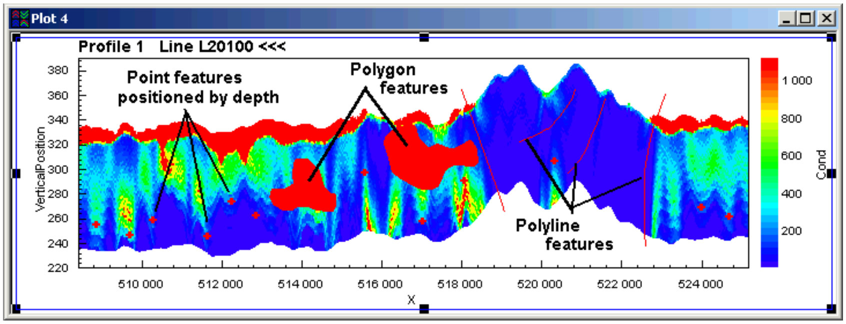
The display style and labelling options of all objects in a Feature Set can be modified in the Feature Properties dialog (double click on the appropriate Feature item in the Workspace tree to open, or right click on the feature layer and choose the **Properties** menu option).

This dialog is almost identical for all types of feature objects and regardless of what type of display they are within. However, in the case of a Profile display, a VerticalPosition field is used to specify the Y value of the features (i.e when the feature was created in profile view the value of the Y-axis is specified as the VerticalPosition).



Feature Properties dialog for a profile display

If features are to be plotted within a section and with a reference to a specified depth, you can select the data field (for the Vertical) of **Value**. This field of the database is retained when the features have been created from a depth-related section profile. An example of this is below:



Features positioned vertically and across the displayed section using the Value field

Within the Properties dialog you can also specify **Colour**, **Size**, **Rotation**, **Symbol** and **Label** modulation of the feature objects from the data page. In all cases it is necessary to select a field from the feature database. If a required data field is not available in those specified, return to the **Fields** tab of the Feature Set Properties (accessed from the Data Manager pop-up menu) and add the required field.

Feature Appearance

The **Appearance** tab of the Feature Properties dialog provides controls for point, polyline and polygon style control, including

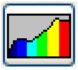
- **Symbol** type, size and colour
- **Fill** colour and transparency
- **Line** colour, weight and pattern.

Any style applied will affect all objects in the database. Polygon colour fill can be disabled by deselecting the **Fill** tick box. Once the Fill option is deselected, the Line colour will automatically become the colour of the Fill colour.

Alternatively, all objects in the database can be colour modulated:

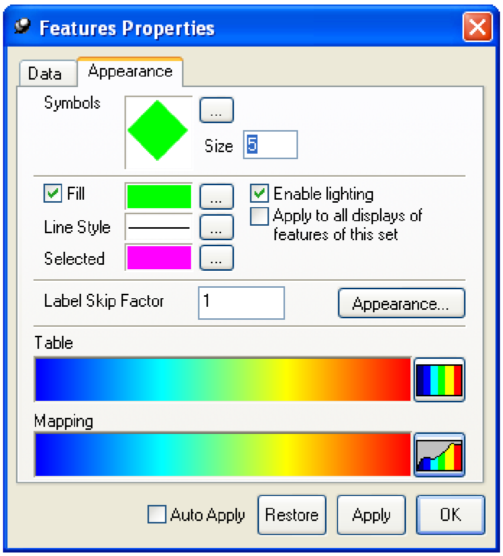
1. Specify numeric field to colour modulate the objects by using the **Colour** control under the **Data** tab.

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 a RBG or HSL interpolation. See the *Colour Table* section in Voxel Models for a detailed description of this dialog.



3. Use the **Colour Mapping Edit** button to apply a Data Transform (predefined or custom). See the *Colour Mapping* section in Voxel Models for a detailed description of this dialog.

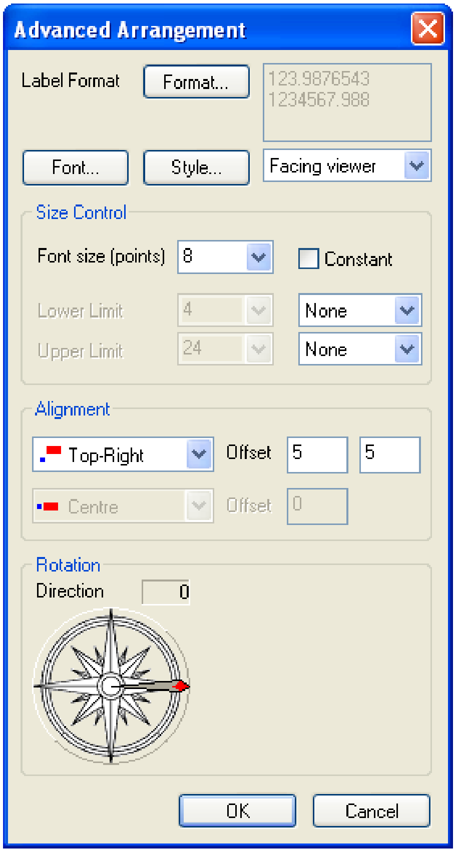
4. Click **Apply** (or enable the Auto Apply option) to display the changes.



The Appearance tab dialog for Feature Properties

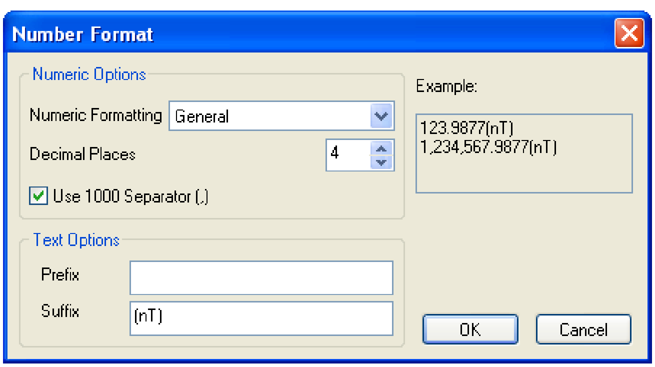
Labelling Features

Feature object 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 3D feature labels

The **Format** button provides numeric formatting options (scientific, various DMS formats, general, etc), decimal place allocation as well as suffix and prefix specification



The Number Format dialog for feature labels

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 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.

Working with Features

- *Selecting Feature Objects*
- *Querying Feature Attributes*
- *Creating Objects*
- *Editing Objects*
- *Advanced Editing Functions*

Selecting Feature Objects

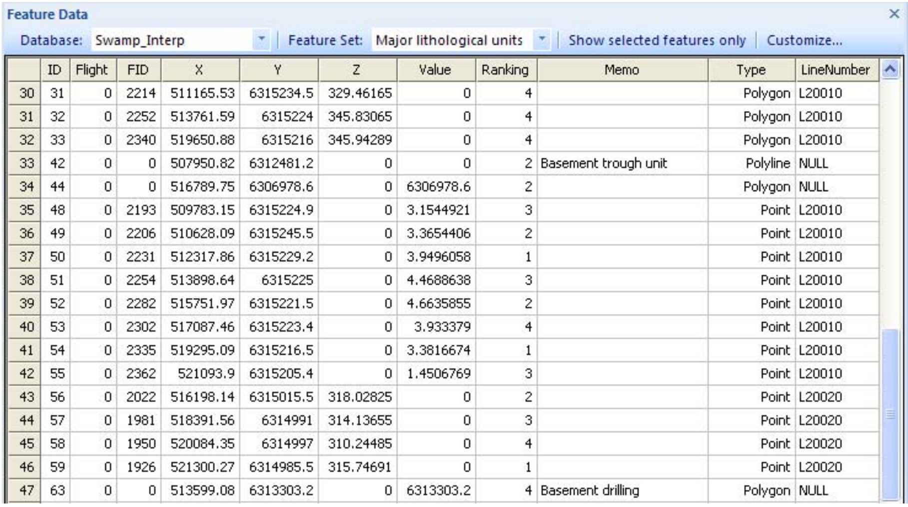
Editing Objects and Attribute Query both require one or more Feature Objects to be **selected**. A Feature Set must be **Selectable** before objects within it can be selected; this is enabled in the Workspace Tree and is typically automatically enabled with the **Editable** icon, unless multiple Feature Databases are being used. This control is useful when multiple dense databases exist in the same area; you can disable the electability of all databases except the target database.

A number of object selection methods are available:

- Enable the **Select** button on the **Zoom Controls Toolbar**, and click on the object to edit. When selected, an object will be highlighted in red (unselected objects are displayed in a default blue colour). The **Cursor Plane** will automatically be enabled and the plane will snap to the object's plane. To select multiple objects, hold down the CTRL key whilst using the **Select** button; the Cursor Plane will snap to the last selected object.

- The **Polygonal Selection** button allows you to draw a selection region in the display window; all Feature Objects 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 objects behind this plane, regardless of object orientation.

Alternatively, one or more objects can be selected from the Feature Spreadsheet display, a floating window located at the bottom of the screen when Encom Discover PA is first opened. If this display is closed at any time it can be opened again by right mouse clicking anywhere on the toolbar area of Encom Discover PA and selecting the **Feature Data** pop-up menu option.



Feature Data
Database: Swamp_Interp | Feature Set: Major lithological units | Show selected features only | Customize...

	ID	Flight	FID	X	Y	Z	Value	Ranking	Memo	Type	LineNumber
30	31	0	2214	511165.53	6315234.5	329.46165	0	4		Polygon	L20010
31	32	0	2252	513761.59	6315224	345.83065	0	4		Polygon	L20010
32	33	0	2340	519650.88	6315216	345.94289	0	4		Polygon	L20010
33	42	0	0	507950.82	6312481.2	0	0	2	Basement trough unit	Polyline	NULL
34	44	0	0	516789.75	6306978.6	0	6306978.6	2		Polygon	NULL
35	48	0	2193	509783.15	6315224.9	0	3.1544921	3		Point	L20010
36	49	0	2206	510628.09	6315245.5	0	3.3654406	2		Point	L20010
37	50	0	2231	512317.86	6315229.2	0	3.9496058	1		Point	L20010
38	51	0	2254	513898.64	6315225	0	4.4688638	3		Point	L20010
39	52	0	2282	515751.97	6315221.5	0	4.6635855	2		Point	L20010
40	53	0	2302	517087.46	6315223.4	0	3.933379	4		Point	L20010
41	54	0	2335	519295.09	6315216.5	0	3.3816674	1		Point	L20010
42	55	0	2362	521093.9	6315205.4	0	1.4506769	3		Point	L20010
43	56	0	2022	516198.14	6315015.5	318.02825	0	2		Point	L20020
44	57	0	1981	518391.56	6314991	314.13655	0	3		Point	L20020
45	58	0	1950	520084.35	6314997	310.24485	0	4		Point	L20020
46	59	0	1926	521300.27	6314985.5	315.74691	0	1		Point	L20020
47	63	0	0	513599.08	6313303.2	0	6313303.2	4	Basement drilling	Polygon	NULL

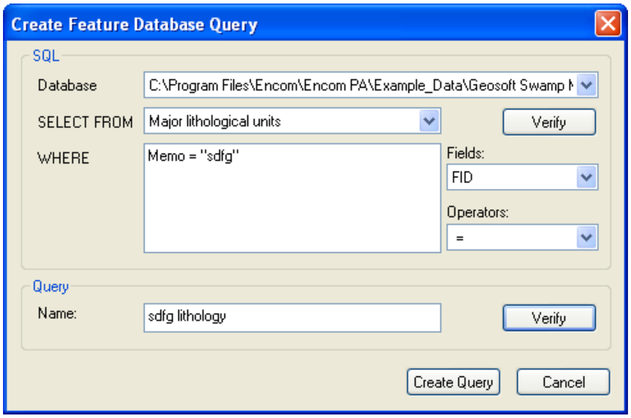
Feature Spreadsheet display with feature object selected

Querying Feature Attributes

The **Create Feature Database Query** dialog (accessed via the **Features>Create Query** menu option. Feature objects within an open Feature Database are able to be queried using basic SQL syntax.

The output of a query is a new Feature Set 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 Spreadsheet section) into a number of databases each populated with objects of a single attribute. The new Feature Set will be automatically added to the list for that specific Feature Database in the Data Manager window.

The pull-down **Field** and **Operator** selection lists to the right of the **Expression** pane allow SQL syntax creation for the feature query.

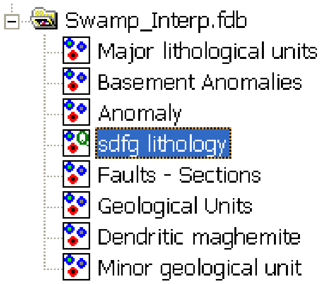


The Create Feature Set Query dialog

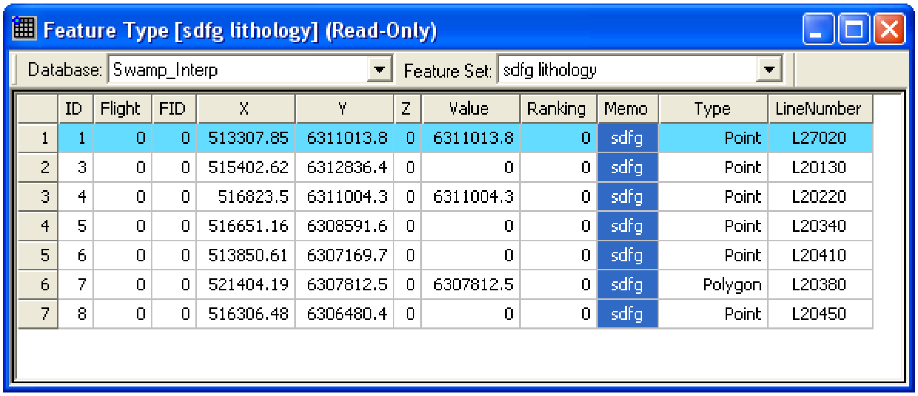
The **Verify** buttons for the query expression and query name (i.e. the output Feature Set name) allow the user to check that these entries are correctly formatted before proceeding.

The **Verify** buttons for the **Query SQL Statement** and **Query Name** will check the validity of the entries before creating the query

Once the **Create Query** button is selected the dialog is closed and the query is generated. A new Feature Set for this query will be added to the Feature Database tree in the Data Manager window. A Feature Set produced by a Feature Query has a different icon to a normal feature with a green “Q” replacing one of the blue circles in the icon.



When displayed in a Feature Spreadsheet window the success of the query can be observed with all feature objects listed having an attribute similar to that queried.


Feature Type [sdfg lithology] (Read-Only)

Database: Swamp_Interp Feature Set: sdfg lithology

	ID	Flight	FID	X	Y	Z	Value	Ranking	Memo	Type	LineNumber
1	1	0	0	513307.85	6311013.8	0	6311013.8	0	sdfg	Point	L27020
2	3	0	0	515402.62	6312836.4	0	0	0	sdfg	Point	L20130
3	4	0	0	516823.5	6311004.3	0	6311004.3	0	sdfg	Point	L20220
4	5	0	0	516651.16	6308591.6	0	0	0	sdfg	Point	L20340
5	6	0	0	513850.61	6307169.7	0	0	0	sdfg	Point	L20410
6	7	0	0	521404.19	6307812.5	0	6307812.5	0	sdfg	Polygon	L20380
7	8	0	0	516306.48	6306480.4	0	0	0	sdfg	Point	L20450

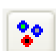
Observe the new Feature Set created by a query in a spreadsheet display

Creating Objects

Creation of **Feature Objects** in Profile, Map or 3D displays requires a user-created or imported feature database to be present in the workspace tree.

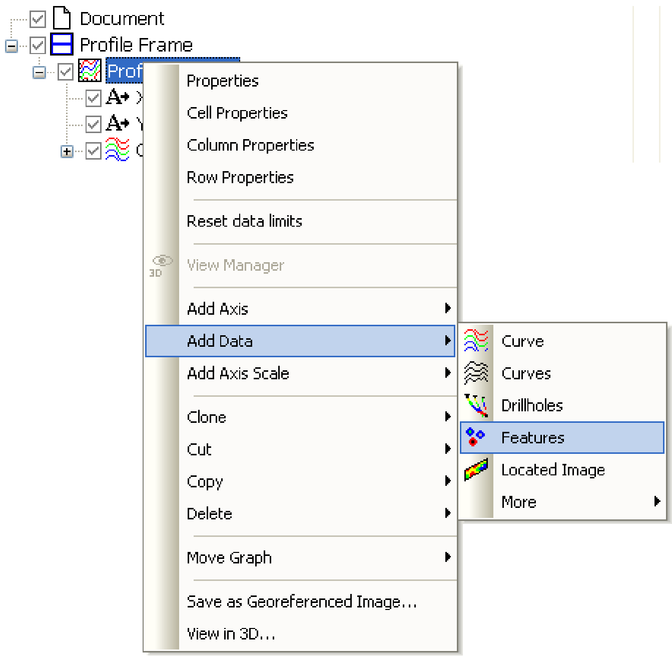
To add a Feature Set to a display (and the associated workspace tree):

1. Highlight the Profile 1, Map 1 or 3D Map 1 branch, click on the right mouse button and selecting the **Add Data>Feature Set** pop-up menu option.



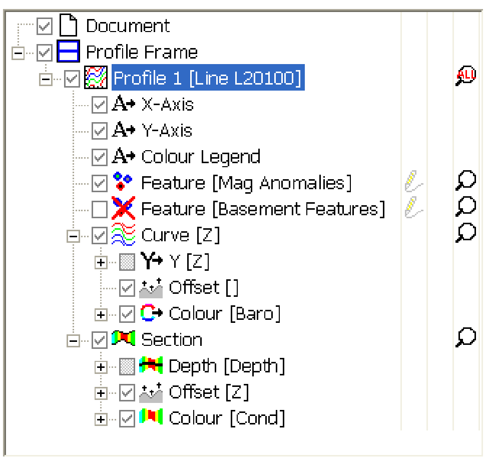
2. Highlight the Profile 1, Map 1 or 3D Map 1 branch then select the **Add Feature** button from the Data Objects toolbar.

3. Select the Feature Set from the Data Manager window and drag and dropping this object into the display window where you want it added.



Feature Set being added to a display in the Workspace Tree

Multiple Feature Set branches can be added to a tree and each entry can control the display attributes of a different Set and Type. The Feature branches that are present in a tree each have individual properties controlled by a Properties dialog (See Feature Object Properties for more information on this).



To toggle a Feature Set on or off, use the **Show/Hide** checkbox adjacent to the Feature branch in the Workspace tree.

Once the Feature Set is added to the workspace tree it must also be made **Editable** (via the pencil icon in the Workspace Tree) before any objects can be created.

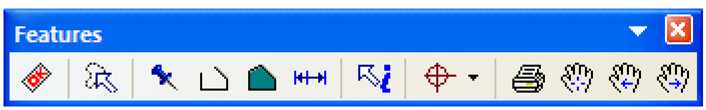
When a feature database is made editable, the Features toolbar will become active and if working in 3D the Cursor Plane will be displayed. The Cursor Plane is used as a drawing plane onto which Feature Objects are digitized: for detailed information on controlling the Cursor Plane, see the *3D Cursor Plane* section.

The icons to the right of each Feature Set branch in the Workspace Tree are as follows:

Editability – enables objects to be added or edited in the selected Feature Set. Only one Feature Set can be editable at one time. To create **Feature Objects** this icon must be active for the Feature Set that the features are to be added to.

Selectability – enables objects within a Feature Set to be selected. For instance, when multiple image branches are open, this is a useful way of limiting which branches (or images) the Cursor Plane can be bonded to. Selectability can also be used to control which Feature Set(s) can be interrogated when multiple complex databases are open.

Zoom Extents – zoom the view to the extents of the selected Feature Set feature objects.



The Features toolbar

The Features Toolbar provides the tools for the creation, selection and editing of Feature Objects, including Points, Polylines, Polygons and Three-Point-Range.

To draw objects point, polyline or polygon objects in 3D:

1. Make the target feature database editable.

2. If creating feature objects in 3D use the Cursor Plane controls to position and orientate the Cursor Plane in the required location. Otherwise position the Profile or 2D Map display to where you wish to digitise.

3. Select the appropriate object button from the Feature Toolbar (point, polyline, polygon or 3-Point-Range), and click on the point location or polyline/polygon start point.

To pick a single point feature, enable the **New Point Feature** button. A single point feature can be placed in either a profile, map or 3D window using this tool. After enabling and moving the cursor to the required window, locate the required position and left mouse click. The location is stored in the specified Feature Set in the database.

This tool is used to create a **New Line Feature** and can be used either in a profile, map or 3D window. The tool can be used to create specific vertices along a polyline (by positioning the cursor and clicking the left mouse at each vertex position). Double click to end the polyline.

Alternatively, to create a freehand polyline, position the cursor at the starting location, hold the left mouse button down and draw. The polyline and vertex locations are stored in the database.

This tool is used to create a **New Polygon Feature**. It can be used in either a profile, map or 3D window. To operate, make the required Feature Set branch editable by enabling the pencil icon in the Workspace tree. The tool can be used to create specific vertices around a polygon (by positioning the cursor and clicking the left mouse at each new vertex location). Double click to end the polyline.

Alternatively, to create a freehand polygon, position the cursor at the starting location, hold the left mouse button down and draw. The polygon vertices and location is stored in the database.

The **Three-Point-Range** tool is primarily designed for use with profiles although it can be used in map windows. The three-point pick feature marks a zone of a feature with a double-headed arrow and a centre, anomalous point.

The Three-Point-Range feature is created by selecting the New Three-Point-Range Feature button, positioning the cursor at the start location and clicking the left mouse button. Move the cursor to the centre part of the anomaly, click again and then repeat at the end, marking the width of the feature.

4. For polylines/polygons, click on consecutive points to outline the object, or hold down the left mouse key whilst moving the cursor to create a continuous series of nodes.

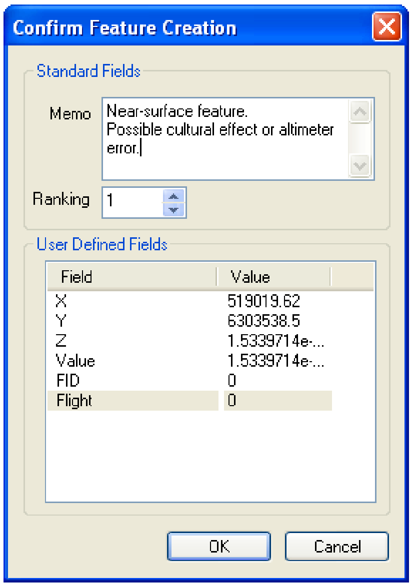
5. Press the ESC keyboard button at anytime to cancel the object creation.

6. Press the BACKSPACE keyboard button at anytime to delete the last added node.

7. Complete the polyline/polygon object by double-clicking; polylines will end at the double-click point, whilst polygons will close back to the start point

8. Enabling the **Confirm Pick** button prior to object creation will display a **Confirm Feature Description** dialog. This allows the object'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 object's geology, e.g. south_fault, supergene or zone_1, allowing a single Feature Database to hold multiple geological objects.

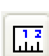
   When a feature is created in a profile or data is associated, the values stored in the Feature Database are also displayed. These can be edited in the features spreadsheet or graphically (refer to Feature Editing).



Confirm Feature Creation dialog for entering comments on created features

9. Upon completion of an object, it is highlighted in red as the current selected object (see *Selecting Feature Objects*), allowing editing (see *Editing Objects*).

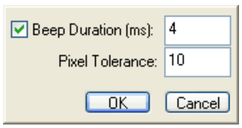
For information on using the other tools available on the Features toolbar, see *Editing Objects*.



The Ruler tool on the Main Toolbar can be a useful guide for distances, bearings and dip angles between objects on the cursor plane.

Snapping a node to existing data:

Snap to feature object node – Allows the current object's node (new object or editing an existing object) to be **snapped** to an existing object's node. 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** can also be enabled for when the cursor is within the set pixel tolerance: when the cursor moves outside the tolerance, a low pitched **Beep** will sound. The **Duration** of these beeps can be set in milliseconds.

Snap to 3D Point/Line or Drillhole data – feature object nodes can be snapped to data displayed in these formats:

1. Make the relevant item (Lines, Points or Drillholes) in the Workspace Tree **Selectable**.

2. Enable the Bond to Image button on the Cursor Plane Toolbar.

3. Move the cursor over a point in the Points or Lines dataset, or over a drillhole interval: the red crosshairs in the drawing plane will snap to that location.

4. This is particularly powerful for 3D interpretation of downhole data; for instance, digitizing a series of polylines or polygons outlining mineralization or geological features for use in the 3D Solid Generator.

Digitizing tips:

Some digitization tips which are controlled via the **Cursor Plane** tab of the Cursor Plane Properties dialog:

- 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 PageUp/Down buttons 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, 50m of the Cursor Plane.

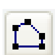
- If digitizing onto georeferenced images (e.g. geophysical profiles), use the **Bond to Image** option on the Cursor Plane Toolbar

Editing Objects

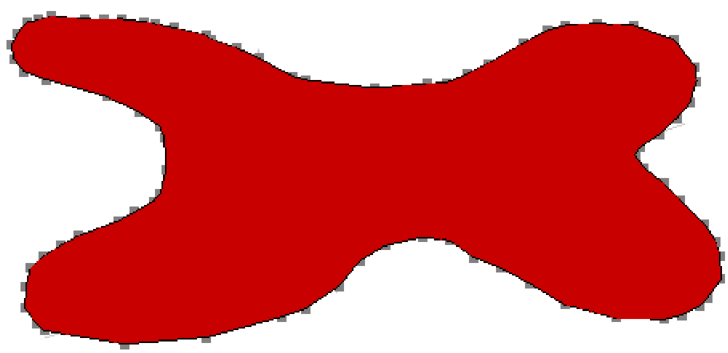
Feature Objects can be duplicated, repositioned, resized and deleted. Individual or multiple nodes within an object can be added, moved or deleted. Object attributes can also be edited.

To enable editing of an object or its nodes:

1. Object editing requires one or more objects 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 Feature Toolbar; all nodes/vertices on the selected objects will be displayed. Double clicking on a editable object will also enable the **Edit mode** for the object:



Polygon feature in edit node mode

Use the Features toolbar to perform these editing tasks:

- *Three-Point-Range Feature Editing*
- *Editing an Entire Object*
- *Editing Individual Nodes*
- *Other Feature Tools*

Three-Point-Range Feature Editing

With a 3-Point-Range feature object selected, three nodes are highlighted. If you need to widen the feature, select one of the end nodes, click the left mouse button and drag its extent to the left or right. If the central node is to be moved, position the cursor over it and select with a left mouse click and then drag within the distance of the two end nodes.

If the entire three point pick is to be moved, select the feature nodes and reposition as required. Delete a selected three point pick by pressing the DEL key or if editing, click the right mouse button and select the **Delete** item from the pop-up menu.

Editing an Entire Object

When depressed, this button allows selection of features within a polygonal frame and immediate editing (moving, deletion, vertex movement and insertion).

The following Editing functionality is available with the **Select** button enabled:

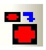
Moving a 3D Object Within the Cursor Plane

Hold down the keyboard SHIFT key whilst dragging the object to its new location. The object will be moved within the Cursor Plane. If more than one object is selected, all objects will be moved parallel to the plane of the last selected object (i.e. the object to which the Cursor Plane has snapped).

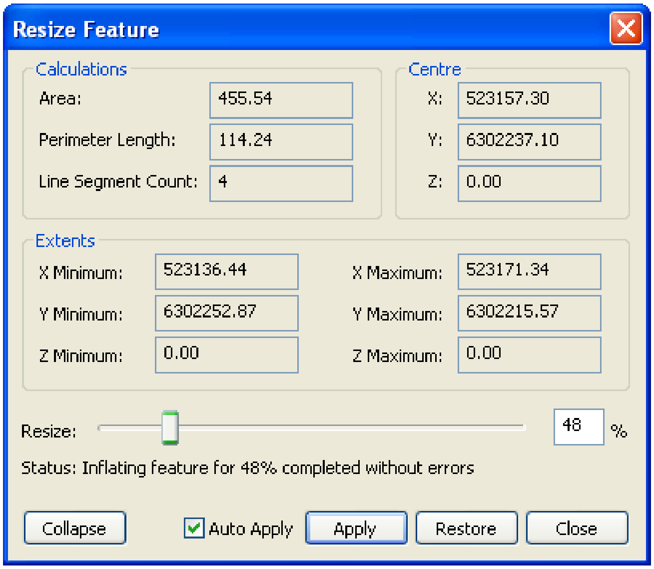
Offsetting a 3D Object from the Cursor Plane

Hold down the keyboard CTRL and SHIFT keys whilst dragging the object to its new location. The object will be positioned and remain parallel to its initial orientation, i.e. it will be offset. If multiple objects are selected, they will be offset to the plane of the last selected object.

Resizing an Object



With a single object selected and editable, select the **Resize** command from the **Features>Edit** menu.



The Resize Feature dialog.

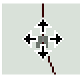
The Resize Feature dialog provides both a slider bar and manual entry to specify the new object size, as a percentage of its current size. Press **Apply** to visualise the new size (this can be dynamic if the Auto Apply option is enabled). The original object size can be restored (if the dialog has not been closed) by pressing the Restore button. A number of statistics and parameters can be examined by selecting the Expand button.

Deleting an Object

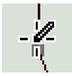
With or without the Edit mode enabled, press the keyboard DELETE key to remove an object. Alternatively, right click on the object (not a node) and select the **Delete** option from the shortcut menu.

Object Cut, **Copy**, **Paste** and **Delete** commands are also available from the Features Menu.

Editing Individual Nodes



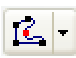
Repositioning a node – placing the cursor over a node will change the cursor to the **Move** cursor (shown left). Click and drag the node to its new location within the Cursor Plane.



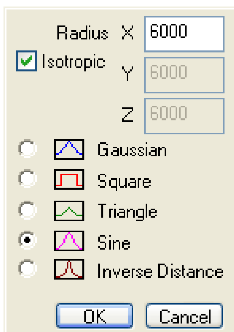
Add a node – placing the cursor over a polyline/polygon segment will change to cursor to an **Add Node** cursor. Click to add a node at this location; this node can then be repositioned as above. Alternatively, right-click when the cursor is over the segment, and choose **Insert Node** from the shortcut menu.

Delete a node – select a node by clicking on it (the Cursor Plane cross-hairs will converge on the selected node) and press the keyboard DELETE key.

Alternatively, right click on the node and choose the **Delete** option from the shortcut menu.

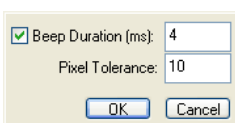


Elastic movement of nodes – Moving a node whilst the Activate Elasticity button (shown left) is enabled will move its neighbouring nodes.

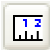


The **Radius** of this option's effect is set via the arrow button adjacent to the Elasticity button. This dialog also contains options controlling how the affected nodes are moved: this effect is represented graphically next to each option (Gaussian, Square, etc).

Snap to feature object node – Allows the current object's node (new object or editing an existing object) to be **snapped** to an existing object's node. 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 can also be enabled for when the cursor is within the set pixel tolerance: when the cursor moves outside the tolerance, a low-pitched Beep will sound. The **Duration** of these beeps can be set in milliseconds.



The **Ruler tool** on the Main Toolbar can be a useful guide for distances, bearings and dip angles between objects on the cursor plane.

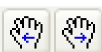
Editing node XYZ coordinates manually - The node coordinates can be edited within the Information Sheet>Data tab. Select the object to edit, and in the spreadsheet, right click on it and select Start Edit from the pop-up menu. Select the node to edit either from the spreadsheet list or graphically, and 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: Encom Discover PA will perform a check to ensure that the new node position retains the objects planar state. If it does not, the newly entered value will be ignored

Other Feature Tools

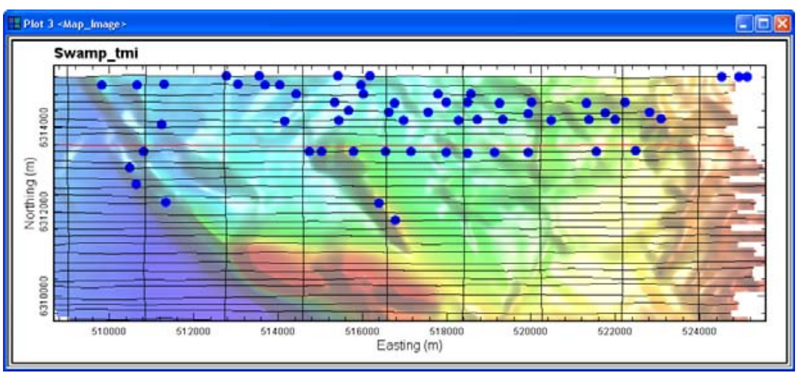
Object creation tools are described in *Creating Objects*. The remaining buttons of the Feature toolbar have the following functionality:

The **Print Feature(s)** button allows batch printing of selected feature objects in a display. This item is useful for producing report output for group selected anomalies or features.

Using the **Centre View** button moves the displayed coverage to centre on any selected feature. This is a useful option when examining features such as sampling locations or geochemical locations on a map with many features (anomalies) displayed.



Within a database Set, you can move sequentially forward or backward through the record entries, selecting individual features each time the **Next Feature** or **Previous Feature** button is clicked.



Point Feature Objects over a grid surface and flight line map

Advanced Editing Functions

A range of powerful multiple feature processing tools are available on the Features Editing toolbar, allowing intersecting feature objects to be manipulated.

By combining these advanced feature manipulations with Encom Discover PA's fast automated triangulation (using the *3D Solid Generator*) and extrusion (using the *3D Extrusion Tool*) 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* — internally triangulates selected polygons 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 features that have been broken into multiple parts with the Break tool.

Triangulate

Triangulate is primarily used to convert selected polygons into triangulated surfaces. This allows converted polygons to:

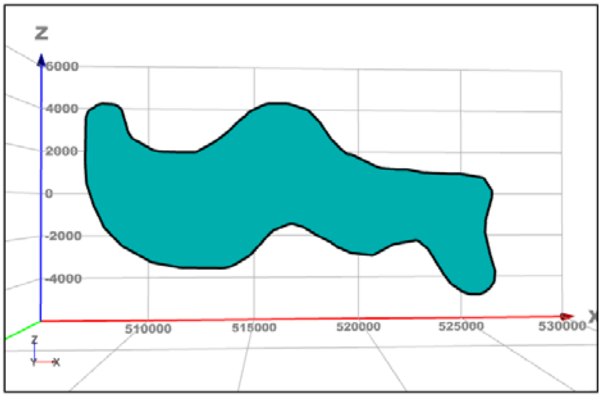
- Insert internal nodes.
- Move internal and external nodes perpendicular to the polygon plane.

And so, the converted polygon can be completely reshaped into an undulating/ non-planar feature surface (e.g. using the elasticity tool to mould regions of internal nodes).

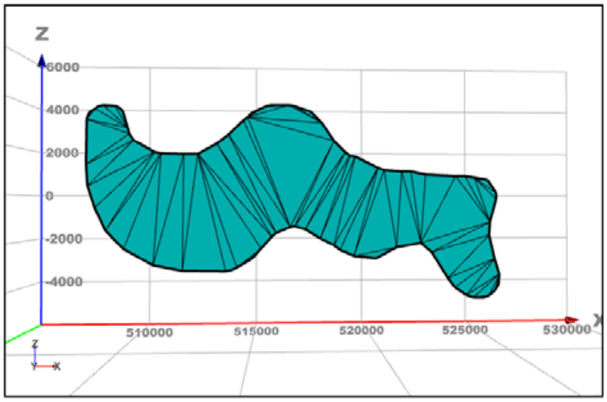
One prime example of the use of this function is the 3D modification of imported drillhole section boundary polygons from a 2D drillhole project. These polygon features cannot be modified outside of the polygon plane, but the geoscientist may wish to snap these to the actual drillhole locations in 3D.

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 position.

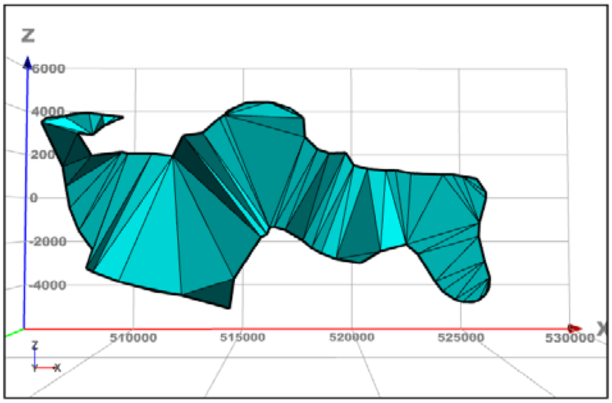
a)



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 vertex/node editing tools and/or the elasticity controls.

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.

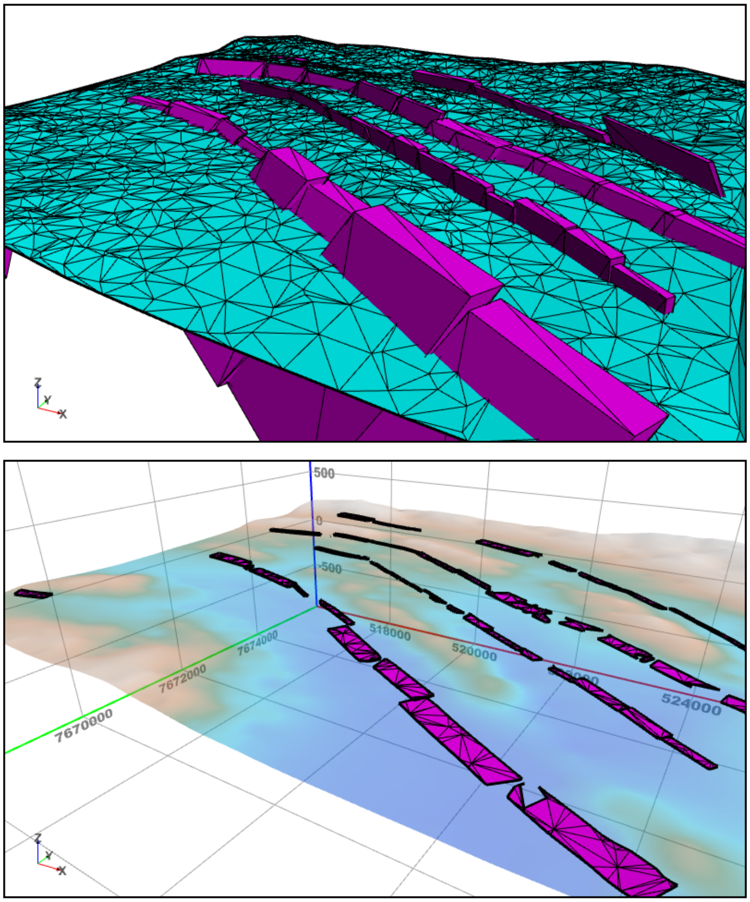
4. Press the **Intersect** button.

5. 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.

a)

b)



An example of an Intersection operation. The resulting intersection surface (b) represents the tabular bodies where they intersected the topographic surface shown in (a).

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:

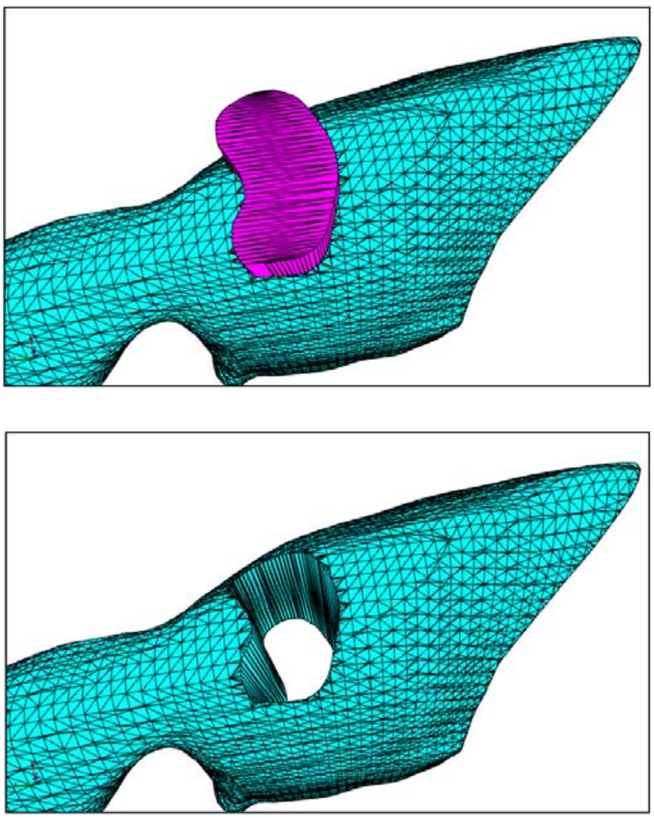
- Cut (or remove) part of a polygon feature where it overlaps with other polygon feature(s)
- Cut a polygon feature along the intersection with other polyline feature(s)

Selected features in different planes:

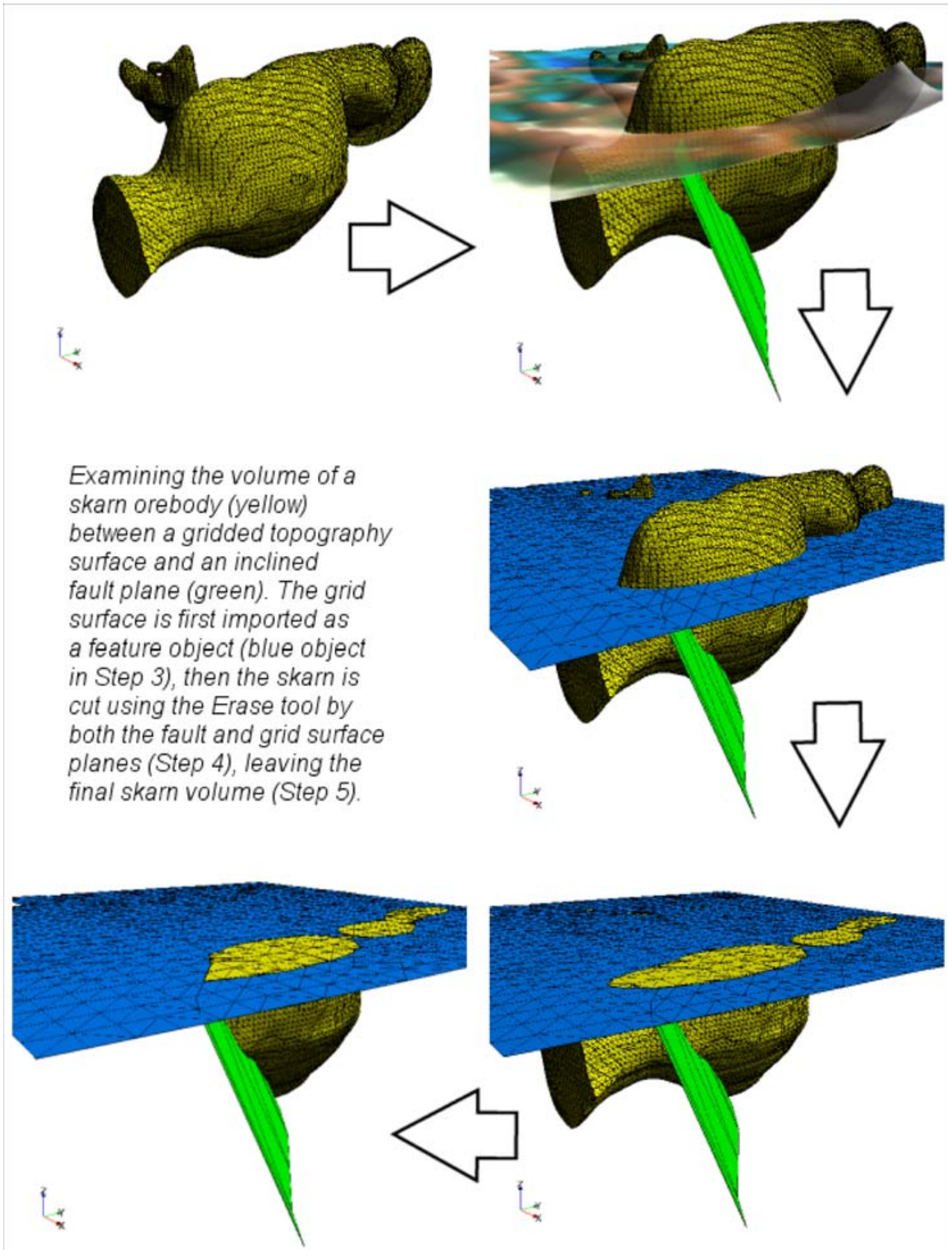
- Cut a polygon feature along intersections with other polygon feature(s)
- Cut a TIN surface along intersections with other TIN surface(s)

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).



Result of cutting the blue solid with the pink solid.



Examining the volume of a skarn orebody (yellow) between a gridded topography surface and an inclined fault plane (green). The grid surface is first imported as a feature object (blue object in Step 3), then the skarn is cut using the Erase tool by both the fault and grid surface planes (Step 4), leaving the final skarn volume (Step 5).

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. Press the **Cut** button.

5. In the **Operation Options** dialog, choose the Feature Set that the resultant feature object is to be saved into and ensure that the **Delete original features** option is disabled if you want to preserve the input features.

6. Press **OK**.

The output feature database will likely have two or more features (the target feature cut into sub-portions). Select and Delete the unwanted features.

It is recommended that any bodies being cut are enclosed bodies (i.e. if created with the *3D Solid Generator*, 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.

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 K.

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 multiple 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 set 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**.

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.

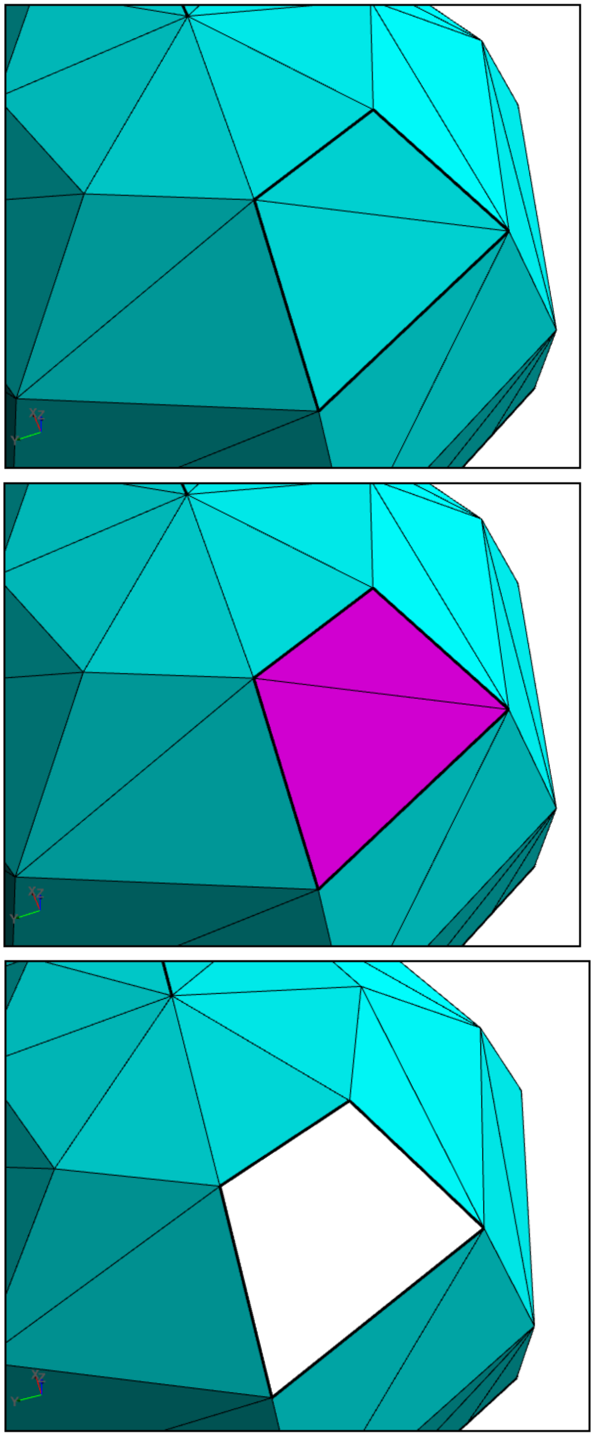
5. Repeat step for each additional interior edge until a continuous line from one side of the surface to another has been traced, or an internal region outlined (see example below).

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

It is recommended that any bodies being intersected are enclosed bodies (i.e. if created with the 3D Solid Generator, 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.

Feature Object Attributes

Feature object attributes can be examined via either the Feature or Data tabs within the *Explorer and Data Windows*; attributes can also be edited in the Feature tab. Open the Information Sheet window using the button at the right end of the Main Toolbar or via the **Insert** menu option.

- *Data Tab*
- *Feature Spreadsheet*
- *Attribute Editing*
- *Node Editing*

Data Tab

The Data tab displays the attributes of only the selected feature object (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 objects regardless of whether their database is selectable or not.

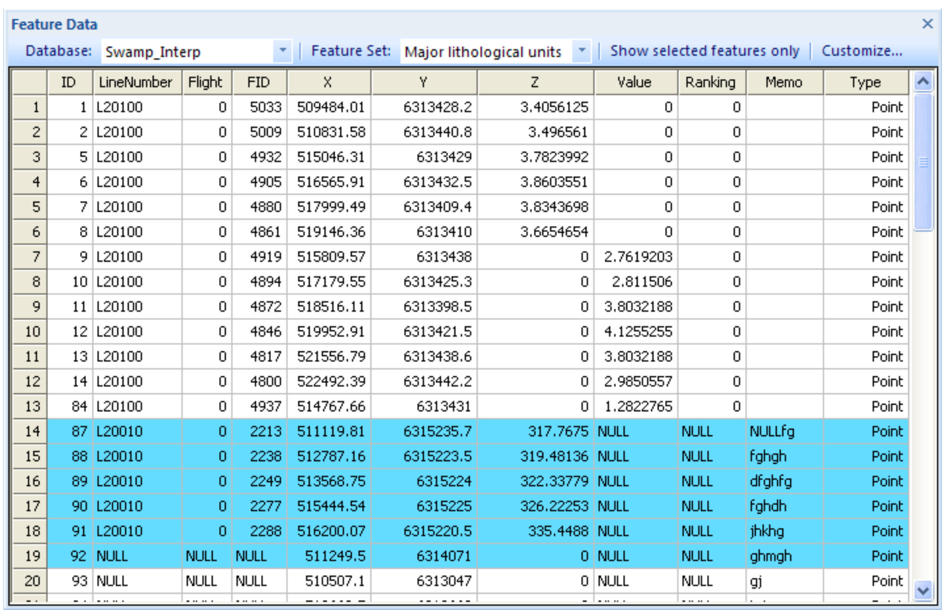
Feature Spreadsheet

The Feature Spreadsheet displays all the retained information about the feature objects contained in the specified Feature Set and is used to review and edit the contents of the opened Feature Database Sets.

When Encom Discover PA is first started, the Feature Database floating window appears towards the bottom of the screen. If this window is closed it can be opened again by right mouse clicking on the toolbar area of Encom Discover PA and selecting **Feature Data** from the pop-up menu that appears. While the Feature Spreadsheet display is floating it can be positioned anywhere including outside of the Encom Discover PA application.

To dock the feature spreadsheet display hold the left mouse down on the titlebar area of the display and drag it to one of the guides that appear on the screen.

The Feature Spreadsheet displays a table view of the attributes of every object in a Feature Database. By default these features are ordered by their ID number. This allows multiple object attributes to be examined and edited after creation; alternatively, individual objects can be attributed during creation by enabling the **Confirm Pick** button on the Features Toolbar.



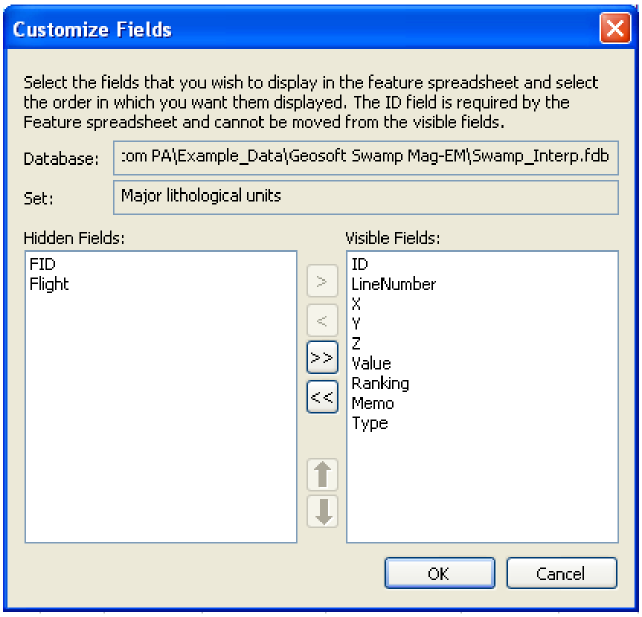
	ID	LineNumber	Flight	FID	X	Y	Z	Value	Ranking	Memo	Type
1	1	L20100	0	5033	509484.01	6313428.2	3.4056125	0	0		Point
2	2	L20100	0	5009	510831.58	6313440.8	3.496561	0	0		Point
3	5	L20100	0	4932	515046.31	6313429	3.7823992	0	0		Point
4	6	L20100	0	4905	516565.91	6313432.5	3.8603551	0	0		Point
5	7	L20100	0	4880	517999.49	6313409.4	3.8343698	0	0		Point
6	8	L20100	0	4861	519146.36	6313410	3.6654654	0	0		Point
7	9	L20100	0	4919	515809.57	6313438	0	2.7619203	0		Point
8	10	L20100	0	4894	517179.55	6313425.3	0	2.811506	0		Point
9	11	L20100	0	4872	518516.11	6313398.5	0	3.8032188	0		Point
10	12	L20100	0	4846	519952.91	6313421.5	0	4.1255255	0		Point
11	13	L20100	0	4817	521556.79	6313438.6	0	3.8032188	0		Point
12	14	L20100	0	4800	522492.39	6313442.2	0	2.9850557	0		Point
13	84	L20100	0	4937	514767.66	6313431	0	1.2822765	0		Point
14	87	L20010	0	2213	511119.81	6315235.7	317.7675	NULL	NULL	NULLfg	Point
15	88	L20010	0	2238	512787.16	6315223.5	319.48136	NULL	NULL	fghgh	Point
16	89	L20010	0	2249	513568.75	6315224	322.33779	NULL	NULL	dfghfg	Point
17	90	L20010	0	2277	515444.54	6315225	326.22253	NULL	NULL	fghdh	Point
18	91	L20010	0	2288	516200.07	6315220.5	335.4488	NULL	NULL	jhkhg	Point
19	92	NULL	NULL	NULL	511249.5	6314071	0	NULL	NULL	ghmgh	Point
20	93	NULL	NULL	NULL	510507.1	6313047	0	NULL	NULL	gj	Point

The Feature Spreadsheet display allows selection and editing of feature attributes

The **Show selected features only** button on the Feature Data Spreadsheet display will condense the list of feature objects in the currently "editable" feature set down to show only the feature objects that are highlighted, i.e. selected.

To change the order of the field columns in the feature spreadsheet click the **Customise** button on the Feature data Spreadsheet display. The **Customise Fields** dialog will appear allowing the fields columns to be rearranged or to be hidden from the spreadsheet.

Attribute Editing



The Customise Fields dialog from the Feature Data Spreadsheet display allows fields in a Feature Set to be rearranged or hidden.

Individual attributes of an Editable Feature Database can be edited directly within *Feature Spreadsheet* cells (including the X, Y and Z values). Right clicking within the attribute spreadsheet displays a pop-up 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 DELETE key, or by right-clicking on the row header and choosing the **Delete Selection** option. This pop-up menu also contains options to **Cut**, **Copy** and **Paste** the current selection, allowing object duplication (duplicated objects could then be offset using the controls discussed in *Editing Objects*).

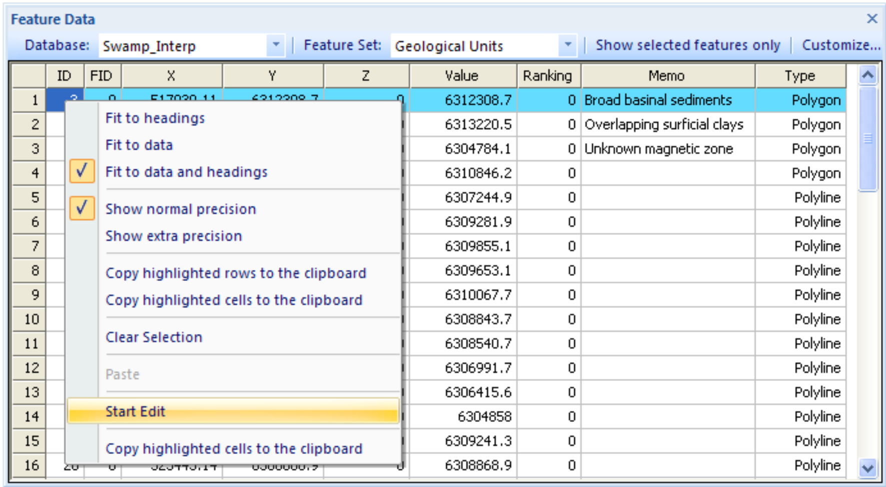
Node Editing

The X,Y & Z values in the attribute spreadsheet only represent the centroid locations for polyline and polygon objects: to edit individual node co-ordinates, right click within a selected row, and select the **Start Edit** option from the pop-up menu. An editable list of the coordinates for every node comprising the Feature Object 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 pop-up menu with three mode-specific options (see image below):

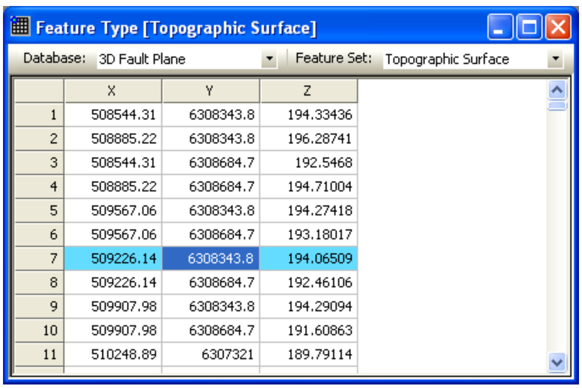
- **Insert Node**: inserts a new node below the currently selected node, populated with the currently selected node’s coordinates.
- **Delete Point**: deletes the selected node.
- **Stop Edit**: finishes node editing and returns the view to the attribute spreadsheet.

Features in the spreadsheet can be selected by clicking on the spreadsheet row header (at the left of the sheet). Multiple features can be chosen using the SHIFT and CTRL keys. Note that selected features are highlighted (whether selected graphically from a map or profile or from the spreadsheet). Highlighting of selected features both in displays and in the spreadsheet is done immediately the selection has occurred.

Right-clicking on the row header of a selected feature displays a pop-up menu. Options in the menu allow automatic heading and/or data spacing operations, plus ascending or descending sort operations for the selected column. Note these spacings can also be made manually by selecting and moving the header column widths. Other options available from the pop-up menu include deleting the selected feature row or all selected features.



Select Start Edit from the pop-up menu for a highlighted polygon feature object to edit node locations

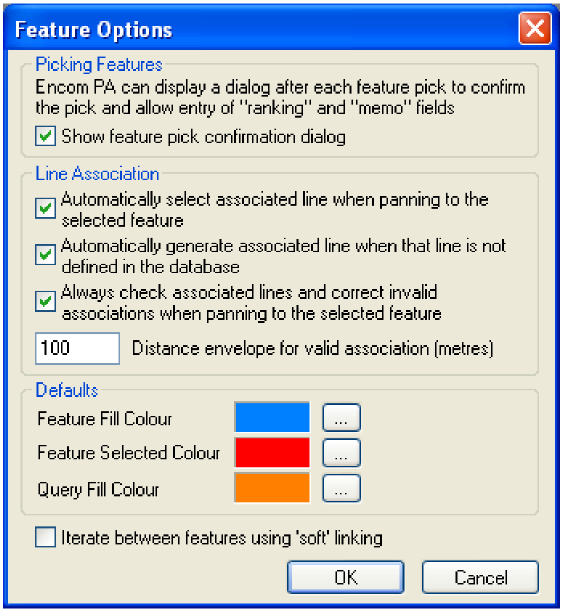


The node coordinates will then be displayed in the spreadsheet.

Certain fields of the open spreadsheet can be edited (such as the Ranking or Memo fields). User-defined fields can also be edited. To perform an edit, the feature set must be made editable in a display window and then you can position the cursor in the cell required in the spreadsheet display. When the cell is double clicked on with the left mouse button the cursor alters shape to a "|" and permits keyboard text entry.

You can select a row, column or group of cells by positioning the cursor over a cell, depressing the left button and dragging the mouse (as you would in other applications, such as Microsoft Excel). Once a block of cells has been selected, you can copy (CTRL+C keyboard sequence) and paste the cell contents into other spreadsheet or text-based programs (eg Notepad). It is not possible to cut a block of cells from the database.

Feature Options



Feature Options dialog

The **Features>Features Options** menu item enables the following default options to be set:

- *Picking Features*
- *Line Association*
- *Defaults*

Picking Features

Turn on/off the feature pick confirmation dialog which is displayed at the end of the creation of each feature object. 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.

Line Association

The line association option allows feature objects to be more intuitively integrated with other line data by selecting the following options when using the **Line Association Wizard**. Refer to *Line Association Wizard* for more information on this utility.

The following options are available to set for line association:

- Automatically select associated line when panning to the selected line in a section view to a particular feature plus populating the feature database with the associated line entry when a feature is selected.
- Automatically generate associated line when that line is not defined in the database.
- Always check associated lines and correct invalid associations when panning to the selected feature.
- The distance (in metres) envelope for applying valid associations.

Defaults

Set the default colour for:

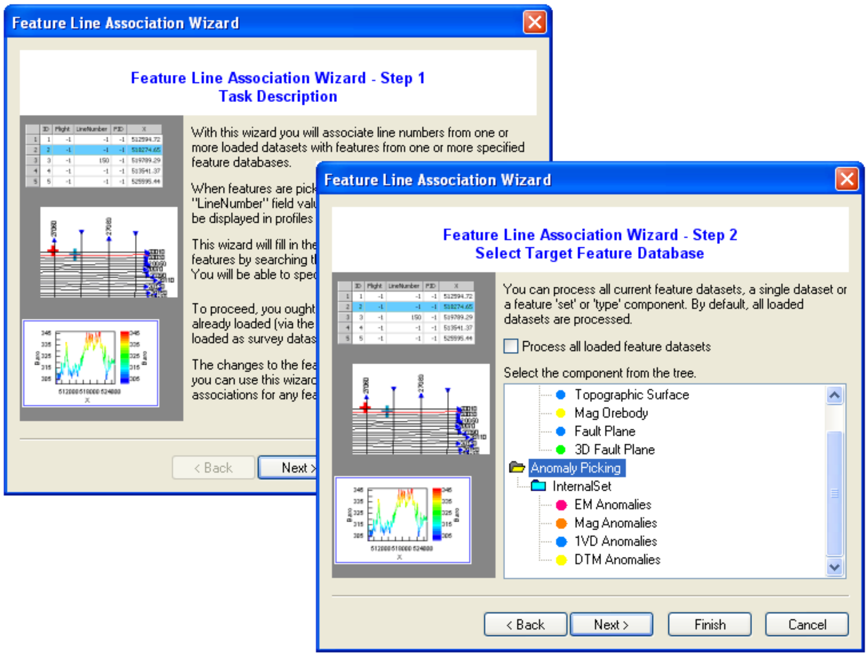
- **Feature Fill Colour** - Set default colour for the Fill of a feature polygon.
- **Feature Selected Colour** – Set default colour for highlighting a selected feature object. This colour must be set before creating feature objects.
- **Query Fill Colour** - Set default colour for Queried feature objects.

Line Association Wizard

Features can be created graphically by picking in a profile or CDI display etc. In these cases, the feature location is directly associated with the line on which it was picked from the related line database. If features are imported (for example from a geochemistry dataset), or are picked graphically from a map, then no automatic line association occurs.

It is often important to combine feature location with profile information (even though the feature may not lie directly under a traverse line). For example, you cannot easily display features, maps and profiles in a related manner where the method of feature detection is not related to a line.

As a consequence, Encom Discover PA has a method that uses a wizard to allow features to be related to traverse lines. It operates by choosing features within a buffer zone surrounding the path of the line. Access to the wizard is from the **Features>Feature-Line Association Wizard** menu item.



The first two wizard screens for the Line Association Wizard of feature databases

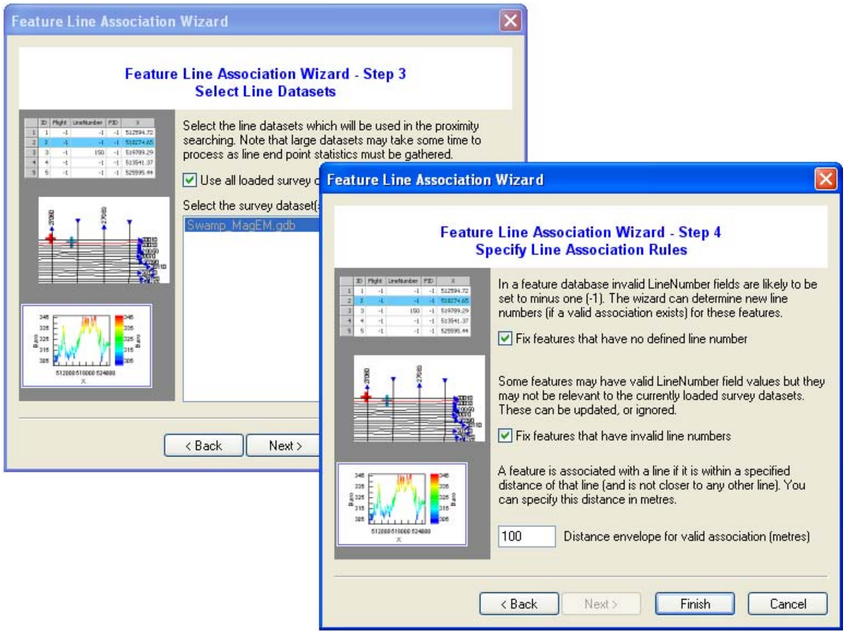
Screens presented with the Line Association Wizard are:

Step 1

Introductory screen only, see above.

Step 2

Specify the Feature Database, Feature Set. You can process all the loaded feature datasets if required or you can specifically select the Feature Set to be line associated.



Steps 3 and 4 of the Line Associate Wizard

Step 3

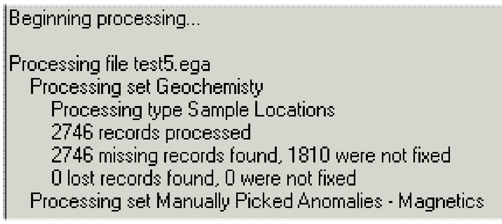
Select which line databases to associate the Feature Set as defined in Step 2.

Step 4

Options to control the line association include:

- For features with no line association, a data value flag of –1 is usually used. If a valid line association is created for features, the –1 value is substituted by the associated line number. The first option enables the substitution of the flag value with the new line value.
- If a feature database has already existing line relationships, but the line numbers do not relate to the specified line database, these line values can be overwritten via the second option.
- A buffer zone is required around a line to locate and build a line association with a feature. If more than one line is associated with a feature within the defined buffer distance, the closest is used.

After clicking the **Finish** button processing of the feature object is performed, and a summary dialog will be shown to indicate the number of features processed and associated with the lines of the line database. If you wish to check the results more exactly, you can display the Feature Spreadsheet and examine the LineNumber values related to specific features.

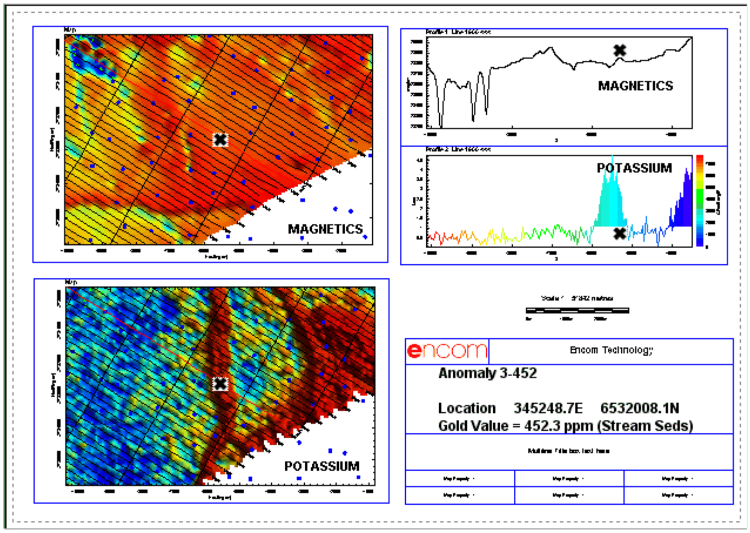


Summary information describing the number of features with successful and unsuccessful line associations

Batch Printing Features

Often large numbers of features can be created within databases. To individually present these using a manually specified display format for a report would be very time consuming. Encom Discover PA provides an alternative batch printing facility to accommodate this requirement.

You may wish to create a display that contains a map, profile and description of features and use this as a template for successive features. Such templates can be used to produce printed reports of selected features. An example of such a report containing displays of the feature location associated with map displays of magnetics, geochemistry and profiles is shown below. A title block defines the feature name, location and specific characteristics.

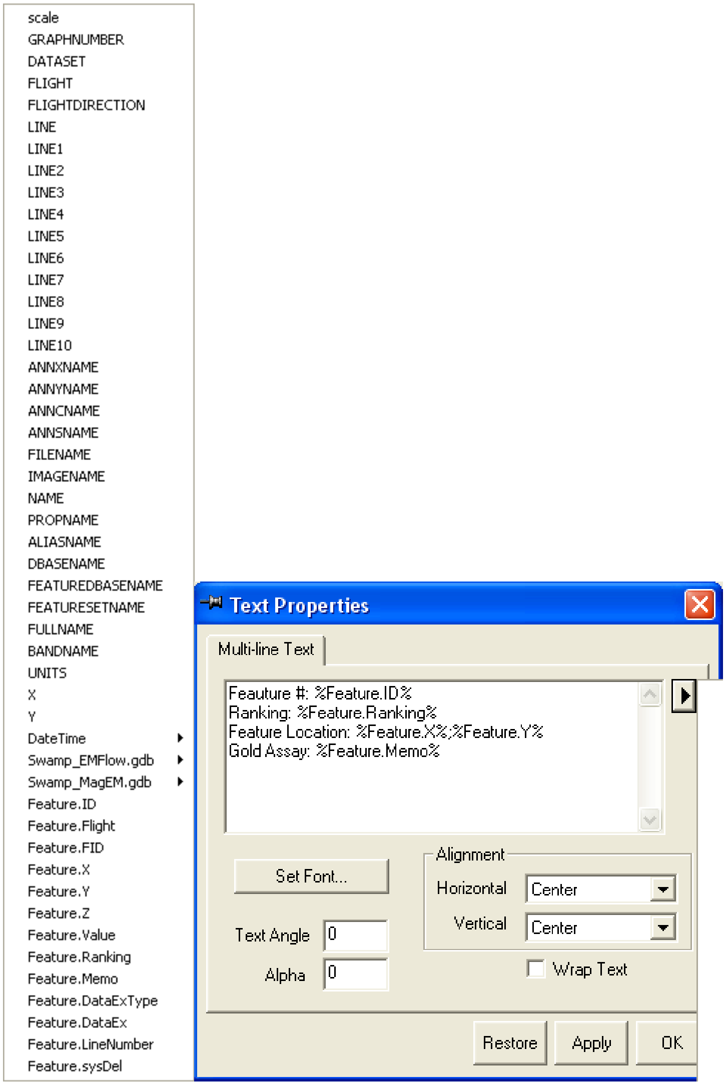


Example feature template with magnetics and radiometrics maps, profiles plus title block and text information

This template (or any other you prefer) can be used for batch printing of multiple feature hard copies. For example, using the template above, if you have selected only those features of a particular Feature Set, select the **Feature Centre** button to position the feature in the middle of the map and to produce hard copy output of all the selected features, then choose the **Feature Batch Printing** button on the feature toolbar. Note that for each print produced, the text in the title block with Feature ID, location and geochemistry results would be updated automatically.

Specifying Variable Text for Printing

For text to be automatically updated by the value contained in the Feature Database, you must define a Variable Text substitution string. Initially, create your title block or legend (as described in Title Box). For individual text items (such as Feature ID or location), insert the variable by choosing the arrow on the right side of the text box properties. A list of available feature database items is shown.



Text Properties dialog with pull-down list of available database items for variable substitution

When inserted into text, the variables have the syntax of %string%, such as %Feature.X%.

Features Database Structure and Links

Encom Discover PA uses Microsoft Access to store all feature databases. This has a number of advantages since full ODBC (Object Data Base Compression) and DBMS (DataBase Management Systems) can be used to provide direct access and updateable links to external software packages such as MapInfo, Access, Excel and ArcInfo.

The database format provides live access. A live access table is a table that can be accessed from a remote application. It is called live because there is no local copy of the data in the third party software. All operations against the data go directly to the server. This differs from linked tables that download a snapshot from the remote database into a native third party table. A linked table is a table that is downloaded from a remote database and retains links to its remote database table. The remote database table is known as a DBMS table.

Using a linked database it is possible to create and maintain a feature database and have the features added and data transferred to other software systems such as MapInfo Professional (see *Linking a Feature Database to MapInfo Professional*).

Linking a Feature Database to MapInfo Professional

Features derived from a geophysical data interpretation in Encom Discover PA can be displayed and updated in real-time with other software packages such as MapInfo Professional. To create a directly linked database accessible from MapInfo Pro, follow the following steps:

Step 1

Ensure both Encom Discover PA and MapInfo Professional are available on the one machine and operating. Also ensure that within Encom Discover PA a feature database has been created and contains at least one Feature Sets.

Note When MapInfo Professional was initially installed, ODBC and DBMS support was also installed. Also note that ODBC and DBMS support libraries were provided only with MapInfo Professional releases later than Version 5.0. The example uses MapInfo Professional Version 9.0.

Warning

A problem to be aware of within the MapInfo Professional link to Encom Discover PA is a limitation on the path length supported in creating .TAB table files. An upper limit of 60 characters is provided by MapInfo Professional. If this number of path characters is exceeded while the linked file is creating the .TAB file, MapInfo Professional will fail. Encom Discover PA remains unaffected.

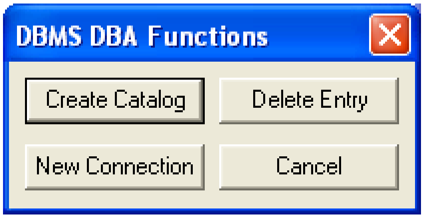
Step 2

Before you can access a DBMS table, you must initially create a link to the specific Encom Discover PA data source where the data to be linked resides. To manage this, MapInfo Professional requires a Map Catalog file be present. This file is necessary to maintain the data link but also to allow the incoming data table to be mappable in MapInfo Professional.

A MapBasic program (called MIODBCAT.MBX) is executed to create the Map catalog. Within MapInfo Professional, access the **File>Run MapBasic Application** option and path to the MAPINFO\TOOLS folder to specify the MIODBCAT.MBX program.

Step 3

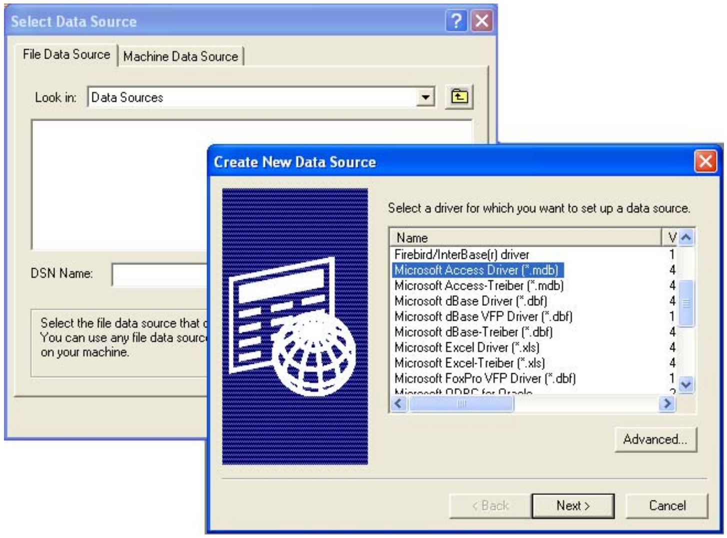
From the MapInfo Professional Tools menu option, select the **DBMS Catalog >DBMS Catalog** option. A four button toolbar is displayed. Select the **Create Catalog** button.



The DBMS Catalog dialog.

Step 4

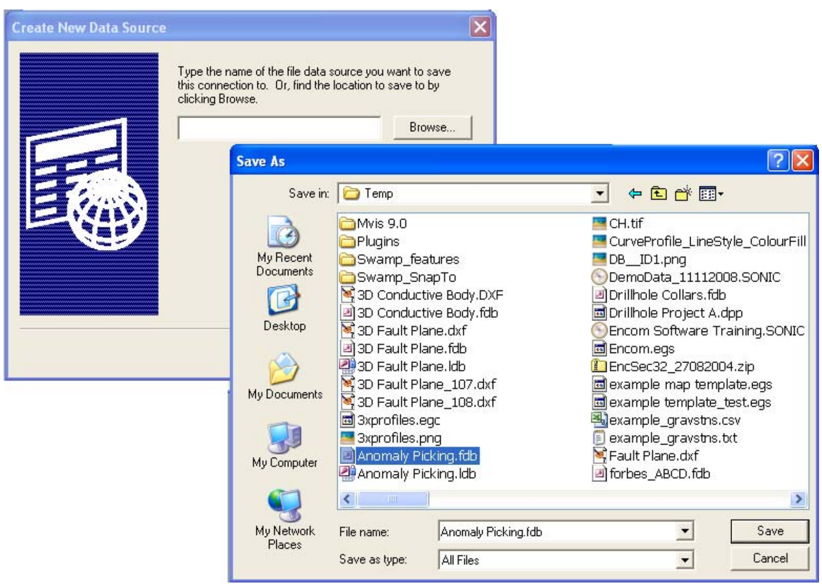
From the Field Data Source tab select the **New** button of the **Data Source Name** (DSN) entry field. This option defines the control to be used in building the link to the Encom Discover PA feature database. In this case the DSN name to select is Microsoft Access (see figure below).



Specify the DSN name (Microsoft Access) for the database link

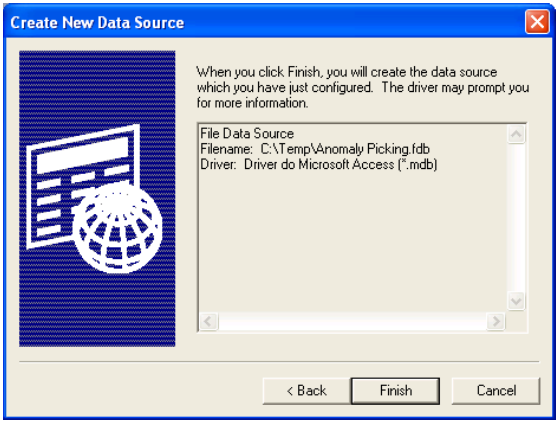
Step 5

Select the **Next** button from the MapInfo Professional wizard and **Browse** for the location of the Encom Discover PA feature database. Create a new .DSN file to associate with the feature database that will be associated with it.



Specification of the data source linked database

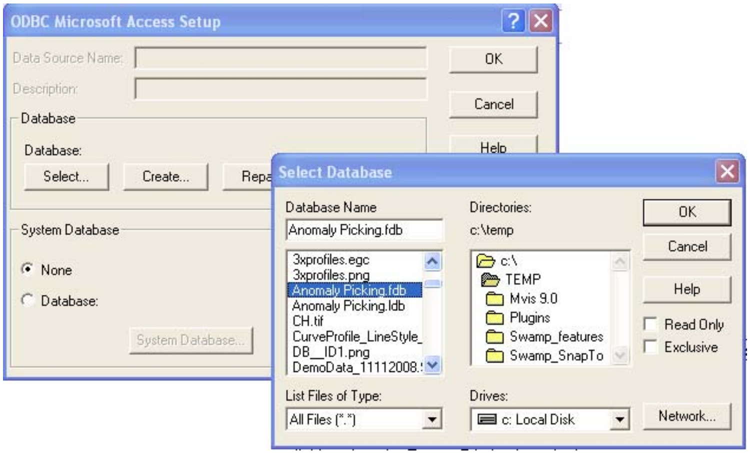
From the final MapInfo Professional wizard dialog entitled Create New Data Source, select the **Finish** button.



The Create New Data Source dialog.

Step 6

Within the ODBC Microsoft Access Setup dialog, click the **Select** button. Navigate and explicitly select the Encom Discover PA database in the displayed dialog. In this case, because the default data source type is Microsoft Access, a file extension of MDB is used as the default. The database from Encom Discover PA has the extension FDB.



Selection of the specific database table assuming the Access link is established

Step 7

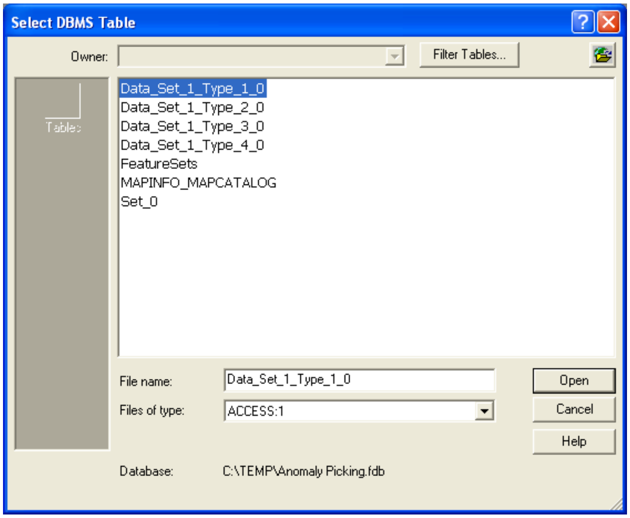
Once the database has been specified, click OK (from Select Database), click OK (from ODBC Access Setup), and OK from Select Data Source dialog. MapInfo Professional should then report that the MAPINFO_CATALOG has been created. This phase of the link has instructed MapInfo Professional that the Microsoft Access protocol is to be used for the link, the link has been established and the specific database has been defined.

Select **Cancel** from the DBMS DBA Functions toolbar.

Step 8

The next phase is to make the Encom Discover PA table mappable within MapInfo Professional. Select the **Table>Maintenance>Make DBMS Table Mappable** menu option. (from the displayed dialog can be opened). All the individual feature tables contained within the accessed feature database.

Note that only one feature table can be open at a time. Multiple MapInfo Professional TAB files can however be accessed and for each database table, a separate TAB file will be created. Consequently, although this step must be repeated for each individual Feature Set, each set can be displayed in MapInfo Professional concurrently.

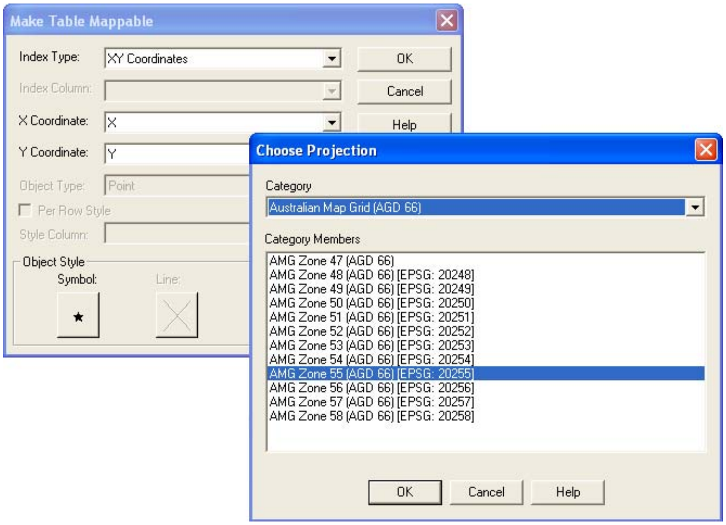


Selection of the feature within the opened Encom Discover PA database

After selecting the appropriate table, select OK.

Step 9

To complete the phase of making the DBMS table mappable, you must specify the **Symbol** to be used in MapInfo Professional for the selected table, plus the **Projection**. It is wise to use the same symbol for a Feature Set in Encom Discover PA as for MapInfo Professional to avoid confusion.



Select the symbol type and Projection for the features

While selecting the options with this dialog, ensure the **Index Type** is **XY Coordinates** (and not MapInfo Professional), and that the **X Coordinates** are X (Easting) and **Y Coordinates** are Y (Northing). When the **OK** button is clicked, MapInfo Professional completes the Table Now Mappable operation.

Step 10

The only step remaining to display the data in MapInfo Professional linked to Encom Discover PA is to open the mappable table. With the link in place, select **File>Open DBMS Table**. You can again select the required Feature Set table and any data fields that are required to be transferred during the table access.

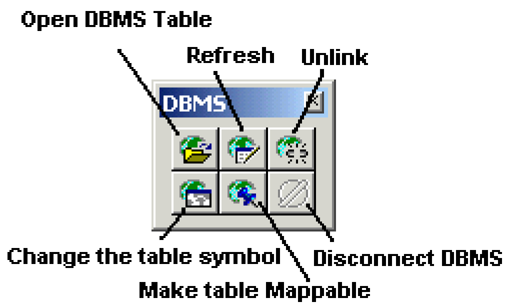
You can specifically nominate required data channels or alternatively, by default, MapInfo Professional transfers all data fields with the asterisk (*) character.

Step 11

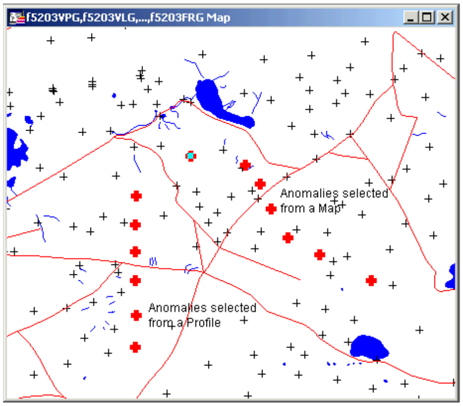
The final step is to verify the feature database to be accessed and the presentation display required. You can select a Browser window, a Map or No view. Ensure too that a suitable name is assigned to the TAB file that is to be created.

Once the new TAB file is written, it can be opened and re-opened without having to set up the data links between MapInfo Professional and Encom Discover PA.

Displaying a linked table in a MapInfo Professional map uses the selected symbols for the available features. If other tables (such as geology, topography or cadastral maps) are also displayed, the features with this backdrop information can be viewed as well.



Note that when new features, or changes to the existing features are made in Encom Discover PA, you need to select the DBMS **Refresh** button to update the database link and redisplay the new features in the MapInfo Professional map.



Feature display within a MapInfo Professional map but updated and derived from Encom Discover PA.

18 Scattergrams

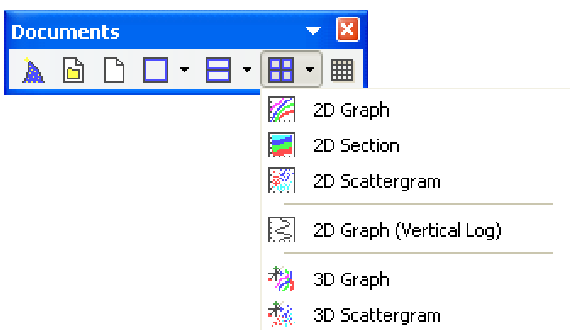
In this section:

- *About Scattergrams*
- *Scattergram Properties*
- *Using Scattergrams*
- *Three-Dimensional Scattergrams*

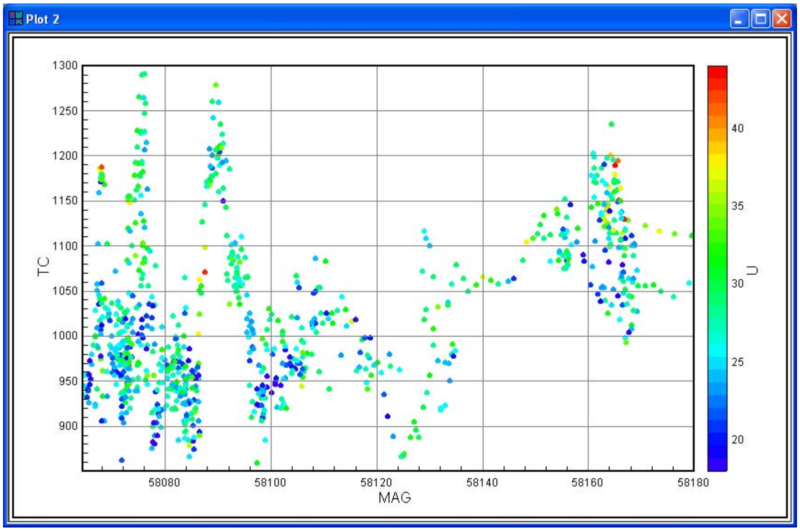
About Scattergrams

The Scattergram display is used to compare and determine if any relationships exist between data fields in a database. Scattergrams can represent two data fields (in a 2D Scattergram) or relationships between three data fields if using 3D Scattergrams.

Any data fields can be selected for display with one being chosen for the horizontal (X) axis and the second the vertical (Y) axis in a 2D scattergram. A third data field for the vertical (Z) axis is required for a 3D scattergram. Each data point is represented in the Scattergram as a symbol. In addition to the two selected data fields, the symbol representation for each data point can be modulated by colour, size or rotation derived from other data fields. No distance criteria is used in the drawing of the graph.



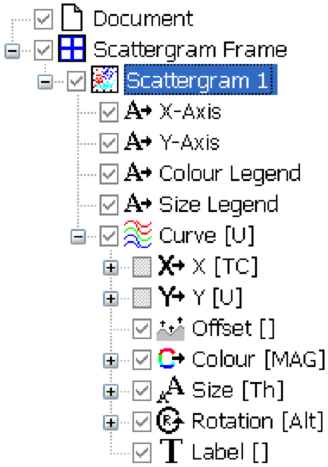
The Scattergram is initially created by selecting the button or the **File>New>Scattergram** menu item. A Scattergram can be inserted into other layout or frames using the **Insert>Scattergram** item. An example of a scattergram plot is shown below:



Scattergram of two data fields derived from radiometric and magnetic channels and modulated by colour, angle and size

Scattergram Properties

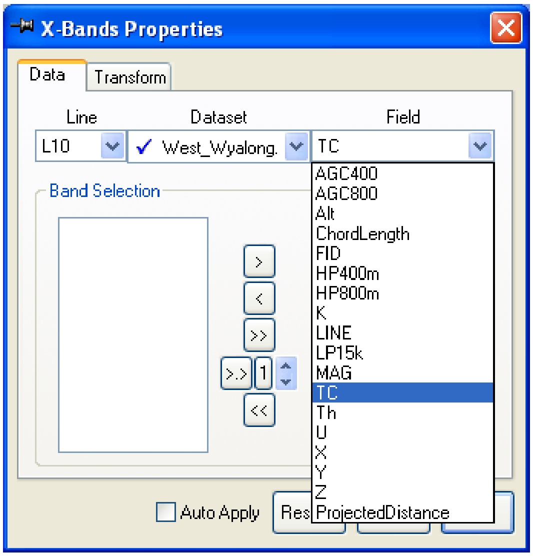
As for all displays in Encom Discover PA the Workspace tree shows the relationship of the various branches of the Scattergram display. The properties of the Scattergram Frame, Scattergram and Axis branches are identical to those of the Profile and Map Documents Displays. See *Display Properties* for descriptions of the various property tab dialogs.



In particular, the Symbols tab controls the appearance of the Scattergam points and a high degree of control for these is available, especially for a 3D Scattergram (refer to *Symbols Tab*).

An example of the Workspace tree is shown below. The data sources (X) and (Y) are the layers that provide definition of the dataset fields. Colour modulation is enabled by default when a Scattergram is initially created and its data field can be nominated as for X and Y layers.

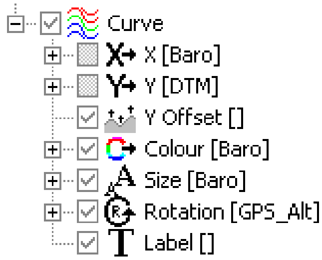
Display the property dialog to nominate a data field for each of the layers required.



Field selection dialog using the pull-down field list to nominate the data source

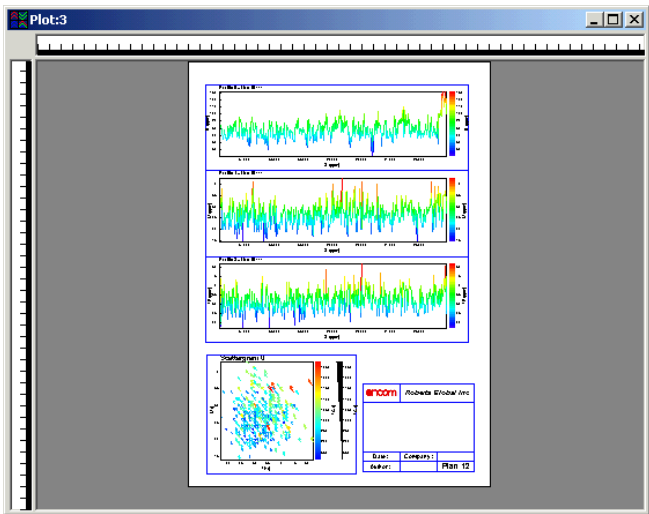
Symbol Modulation

By default, each data band displayed uses a new colour and symbol designated by the system. If you wish to enable colour modulation, switch off **Cycle Colour** on the **Symbols** property page and switch **Mod Colour** on. Symbol size, rotation and label modulation can also be enabled in the same fashion as for a multi-curve.



Using Scattergrams

Scattergrams can be incorporated in other plot documents by inserting or copying and pasting.



A Scattergram incorporated into a Layout with profiles and title block

They provide a fast and powerful method of determining and illustrating the relationship of one or more data fields to one another.

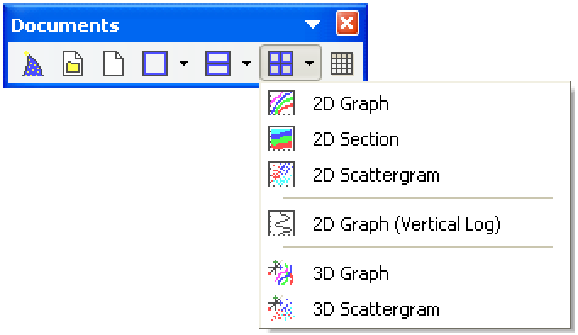
Three-Dimensional Scattergrams

Scattergrams are not restricted to being only 2 dimensional in Encom Discover PA. They can be created in 3D displays and as such are powerful tools for display of such things as:

- Geochemical assay results
- Quality control for comparison of flight data

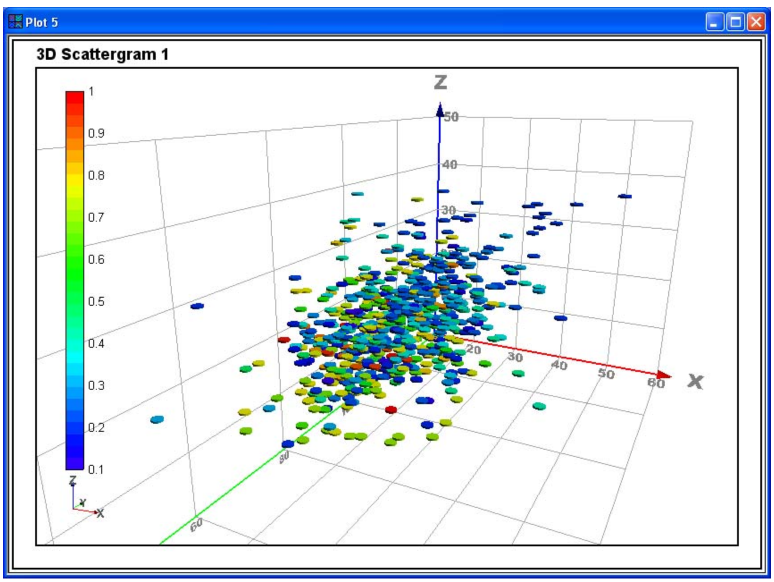
- Analysis of radiometric data channels.

To create a 3D scattergram you can: use the menu option **File>New>Graph>3D Scattergram** or the pull down menu item as shown.



Once a 3D Scattergram window is displayed, highlight the X, Y and Z data fields in the Workspace tree and assign these to the required data channels.

The size, colour and symbol types can be selected and modulated as required using the various standard tree layer controls (Colour, Size etc). An example of a 3D Scattergram is shown below:



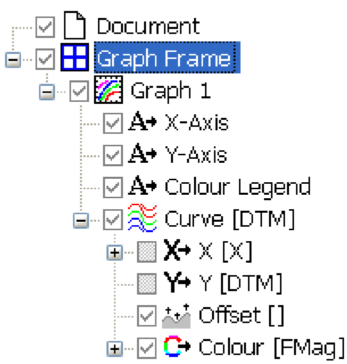
Example of a 3D scattergram with radiometrics geochemical data (K, U, and Th) and colour modulated by magnetics.

19 Graphs

In this section:

About Graphs

The Graph option creates a Graph display containing a single Graph Frame. This is the most basic type of graph, graph frame and graph document in Encom Discover PA. A Graph can have any two data fields chosen for scaling the horizontal and vertical axes in a 2 dimensional graph. For a 3D Graph, a third, vertically oriented data field is also required.



As for all other documents, the graph document structure is hierarchical. It contains objects like graph frames that can contain child frames and objects. The document structure is revealed in the Workspace Tree. A Graph Frame can have multiple rows and columns. Each cell can contain a single graph or other object. Each graph is shown in the tree as a child of the graph frame and they can be moved around the cells of the frame using options from the pop-up menus.

At the top of the Workspace tree for a graph object you can control the appearance of the plot area, plot title etc. A graph can contain one or more X axes. An X axis can be shown graphically by adding one or more X axis. A few important points concerning graphs are:

- An **X axis** can own one or more Y axes. A Y axis can be shown graphically by adding one or more Y axes. A Y axis can also own XY Internal Annotations (refer to XY Internal Axis).

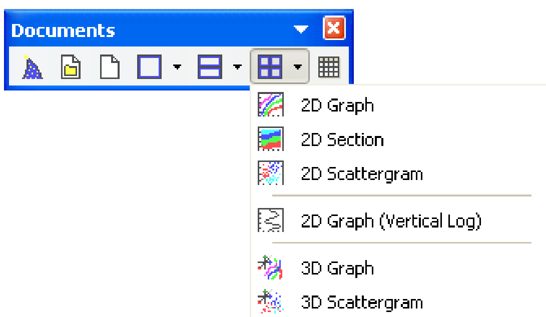
- A **Y axis** can own one or more Colour Axes. A Colour axis can be shown graphically by adding one or more Colour axes.

- A **Colour Axis** can own one or more Size Axes. A Size axis can be shown graphically by adding one or more Size Axes.

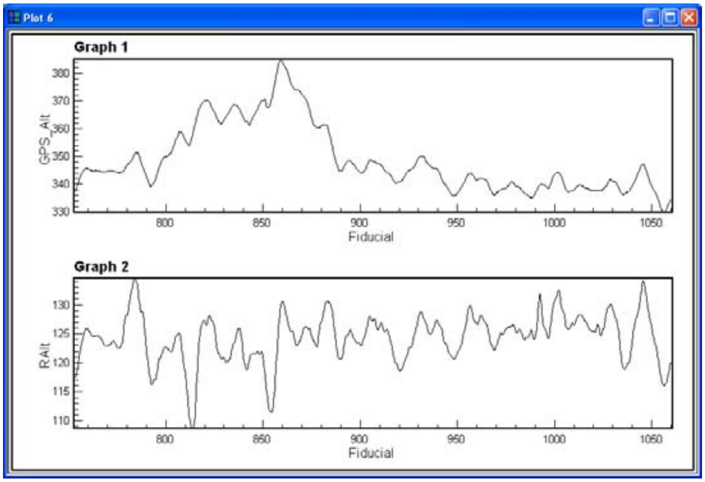
- A **Size axis** can own one or more Data Series. A data series may be, for example, an image or flight path in a map, a multi-curve or multi-layer or a feature curve.

So, in principle, to add a new Curve to a graph, you need to find the Size axis and add the curve to this object. or, to add a new Y axis to a graph you have to find an X axis and add the Y axis to this object. In practice, to simplify the appearance of the Workspace tree the axes are not displayed unless they are non-unique. Consequently, options applicable to axes are wound back up the tree as far as possible to a parent object. In a simple graph this might often be the graph object itself. To familiarise yourself with the structure of a graph, you can show all object levels by setting the Workspace Tree content to **Verbose** (refer to *The Workspace Tree*).

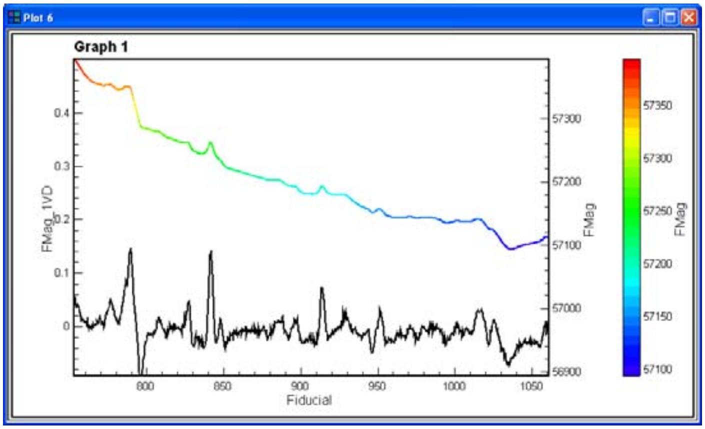
Creating a Graph Document



A Graph is initially created by selecting the button on the toolbar or the **File>New>Graph>2D Graph** menu item. A Graph can also be inserted into another layout or frame using the **Insert>Graph** menu item. Either 2 or 3 D graphs can be created. Two examples of Graph plots are shown below:



Comparative plots of Radar Altimeter and GPS within a graph



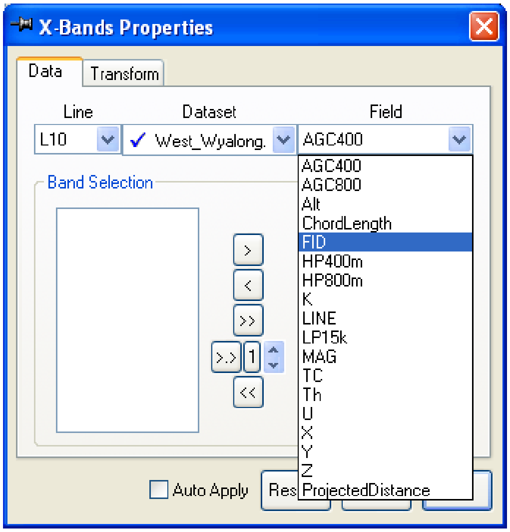
Graph of data derived from two fields with extremely different data ranges

Note in the example here that the data ranges of the two nominated fields have different data ranges. With the Graph properties you can display the axes such that different vertical scales and data ranges are accommodated.

Graph Properties

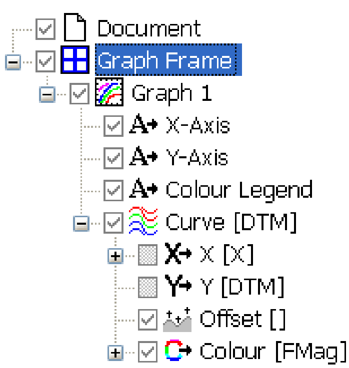
As for all displays in Encom Discover PA the Workspace tree shows the relationship of the various branchs of the document. The properties of the Graph Document, Frame, Graph and Axis branches are identical to those of the Profile and Map Documents. Refer to *Display Properties* for descriptions of the various Property tab dialogs.

Generally, a graph is used to display Curves. Unlike the Profile, a graph places no restriction on the number of X axes created or the data field used as X in a Curve. Consequently, you need to select data for the X and Y channels, and can optionally also specify colour, size, rotation and label modulation channels. As usual, Y values of the curve can also be offset by an Offset (usually topography) branch.



Field selection dialog using the pull-down field list to nominate the data source

An example of the Graph Workspace tree is shown here. The data sources (X) and (Y) are the branches that provide definition of the dataset fields. Colour modulation is enabled by default when a Graph is initially created and its data field can be nominated as for X and Y layers.

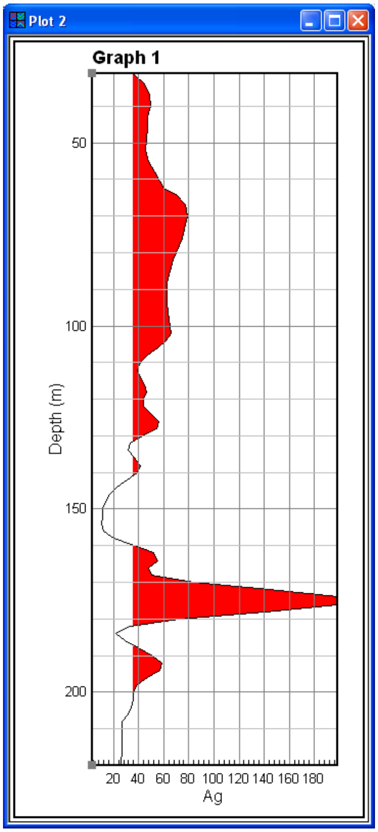


Note that a transformation can be applied to the data channels if required. Refer to Data Transformation for additional information.

Vertical Graphs

Vertical Graphs are useful in many instances (such as for wireline downhole logs). This form of graph can rotate the standard, horizontally oriented graph document such that axes and cursor movement are oriented vertically. The Vertical Graph document operates exactly like its horizontal equivalent where individual data field control of both the vertical (Y) and horizontal (X) axes is available.

When created, these documents have the standrad zoom, pan and cursor movement vertically oriented to allow data analysis up and down. An example of this format is shown below with drillhole EM data and a density log:



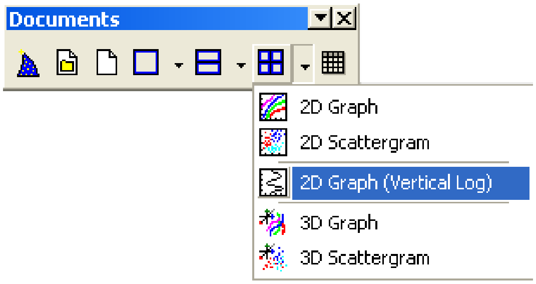
An example of a Vertical Graph display showing Ag assay concentrations at drillhole depth and colour filled above data average.

Creating a Vertical Graph

One method to display a vertically oriented graph is:

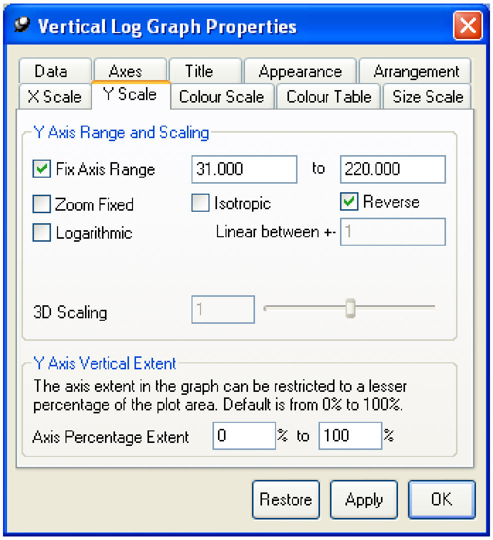
1. Open the required database or import the necessary data

2. From the document toolbar pull down the Graphs and Scattergrams item. Select the **2D Graph (Vertical Log)** menu item.



3. Assign data channels to the X (horizontal) and Y (vertical) axes. Note that typically (as for example in a drillhole, the vertical axis (Y) would be assigned to a distance field such as depth. Apply the channels.

4. You may need to reverse the vertical axis (and consequently, the data) to ensure the top of the graph truly represents the correct, higher portion of data. This is done from the Y axis control tab of the Axis branch of the Workspace tree.



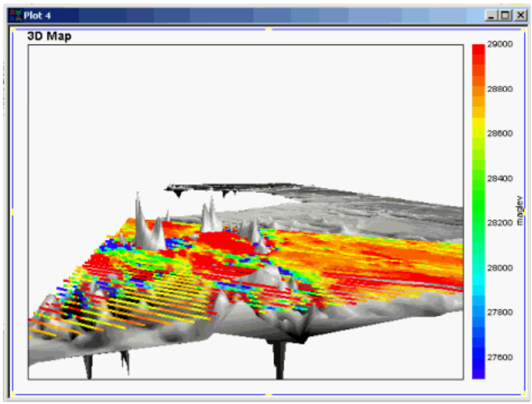
Y Axis control tab of the axis controls and note Reverse is enabled for Depth positive downwards.

From the Y Axis control tab of the axis controls ensure the **Reverse** option is enabled.

Alternative methods of creating a Vertical Graph are also available by using the **Add** option of a pop-up menu (from a Graph Frame) and select the **2D Graph (Vertical Log)** or from the **Insert>Graph** menu item.

Creating 3D Graphs

Three-dimensional graphs can be displayed in Encom Discover PA where any three data fields are plotted against one another. For example, you may wish to plot the relationship between easting and northing and elevation. This form of plot could be used for quality control when draped across a terrain surface. An example is shown below:



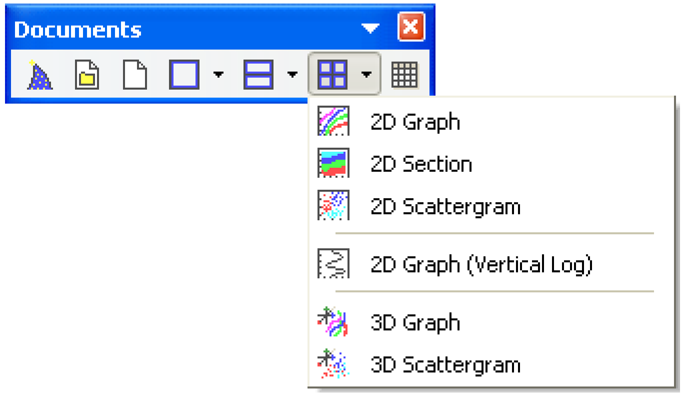
Display of 3D graph representing flight lines modulated by magnetic response

Other uses of the 3D graph application include analysis of EM decays and radiometric response modulating flight paths.

Note that once in a 3D display, other surfaces, models, bitmaps, drillholes etc can be added. The control of a 3D Graph is identical to that of a 3D Map display. For further information refer to Three Dimensional Displays. The conventional 3D cursor navigation is available as is the Internal Axes for positioning while roaming and zooming the view.

To create a 3D graph you can:

- Use the menu option **File>New>Graph>3D Graph** or
- Use the pull-down menu item as shown.



Use the Documents toolbar to select the 3D Graph display option.

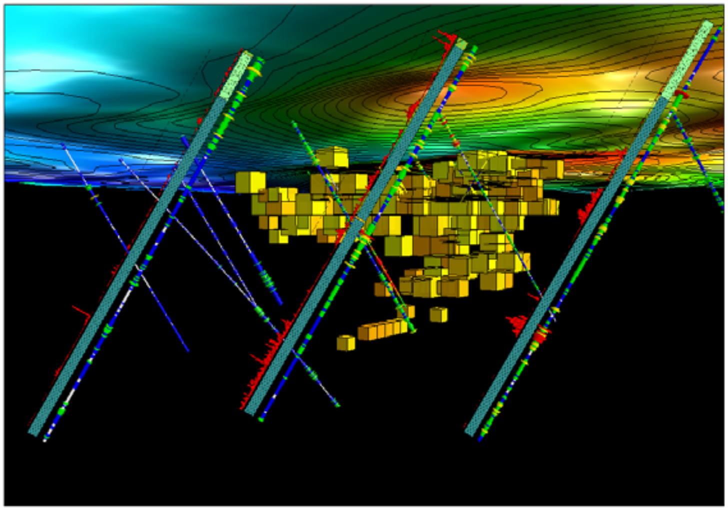
20 Drillholes

In this section:

Displaying Drillholes in Encom Discover PA

Encom Discover PA allows drillholes 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 user to view their exploration projects in a complete and coherent 3D environment. This allows the user to start assessing and interpreting their drillhole project’s downhole information in an integrated world only previously accessible in high-end mining packages.



Example 3D drillholes display showing drillholes as tubes, colour and thickness modulated by Cu assay concentrations and droped vertical log plots.

Once drillhole data is displayed in Encom Discover PA, there is an extensive range of display controls available for both the drillholes and the data that may be associated with them. For example:

- Trace Style - trace paths can be displayed to show the accurate location of the drillholes.
- Labeling - 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 - 2D cross-sections and vertical logs can be quickly displayed as georeferenced images in 3D.
- Drillholes with trace thickness control and assay modulation

Drillhole Data Formats

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 but it may also include dip and azimuth.

- **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).

Note Encom Discover PA handles dip direction with the convention that a downward hole dip has a positive (0-90) angle. If negative dips are encountered, a message is displayed to reflect this negative dip.

Importing Drillhole Data

The drillhole data import capabilities of Encom Discover PA include:

With Encom Discover PA running, you can access drillhole data from the **File>Open>Downhole** menu option. Drillhole data can be imported from these sources:

- *ASCII Drillhole Data* including collar, survey, downhole and desurveyed data in a wide range of tabulated and delimited formats.

- *Encom Discover Drillhole Data* - Encom Discover drillhole data can be read into Encom Discover PA directly using the Discover Drillhole Project. Collar, survey and data .TAB files are used and can be displayed with assay and lithology data.

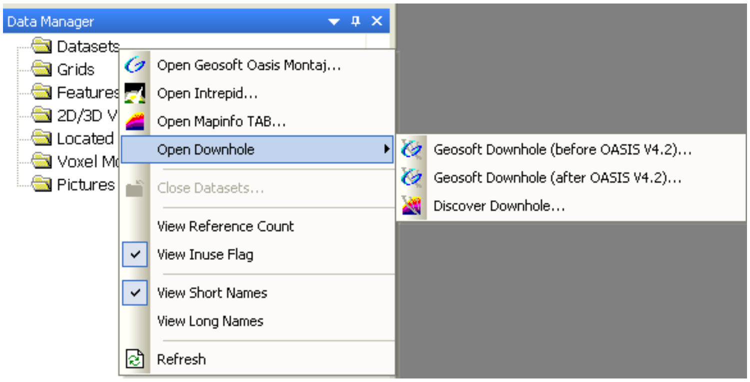
- *Geosoft WHOLEPLOT Data* - Geosoft WHOLEPLOT databases can be combined or read as separate database files containing Collar, Survey and downhole recorded data. Two types of Wholeplot databases are supported directly from the **File>Open>Downhole** menu option (as determined by the version of Geosoft WHOLEPLOT being used). Both point and from-to data can be accessed and displayed.

- *acQuire Drillhole Data* - Support for acQuire databases is provided. Using this support, acQuire data can be read from Microsoft Excel, FoxPro dBase, Microsoft Access and VisualFoxPro databases.

Encom Discover PA allows subsurface information derived from drilling to be displayed in 3D. These displays can be shown with additional information such as surface data, raster images, survey results and other subsurface objects such as structural models, drillhole logs or interpreted sections.

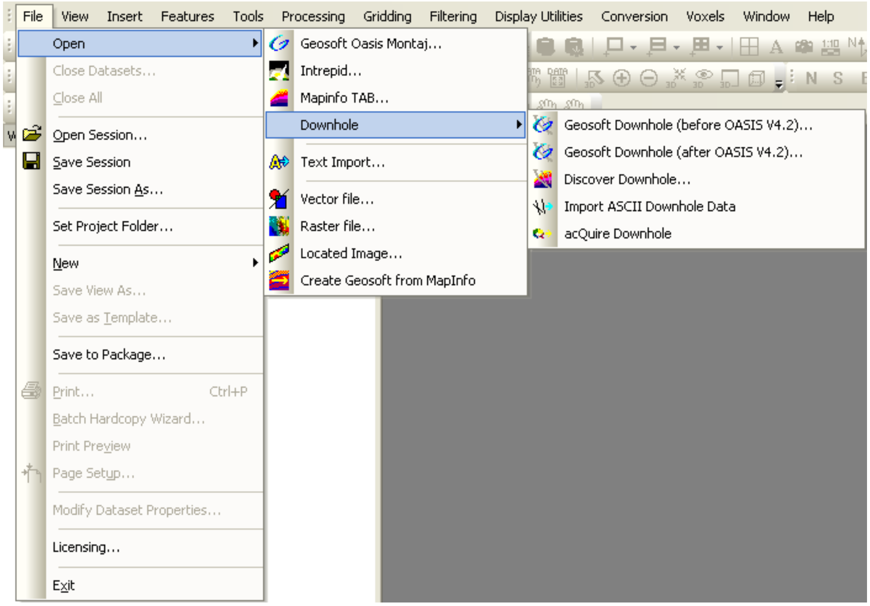
The drillhole data can be opened from two sources:

- By clicking with the right mouse button on the Datasets folder in the Data Manager and selecting **Open Downhole** from the displayed pop-up menu, or



Choose the drillhole database format from the datasets pop-up menu in the Data Manager window.

- By selecting the **File>Open>Downhole** menu option and selecting the data format type. The data format options are Discover Downhole, Geosoft WHOLEPLOT, acQuire Downhole or the Import ASCII Downhole data option.



Choose the drillhole database format from the File>Open>Downhole menu option

ASCII Drillhole Data

A facility to import drillhole survey data is provided which converts the recorded distance, dip and azimuth data of drillholes into de-surveyed XYZ information as it is imported. You can import either:

- *Files with Collar and Downhole Survey Data*
- *Files with XYZ Drillhole Data*

A import wizard (*Import Drillholes Wizard*) is provided to assist you with selecting the formats.

Files with Collar and Downhole Survey Data

This import facility can operate on either a collar file containing hole orientation data (straight holes only), or a collar file with a separate downhole survey file.

- *Collar file only*
- *Collar and survey file*
- *Downhole samples and log files*

Collar file only

A collar file only which defines the collar XYZ location, distance to the end-of-hole (EOH) plus a dip and azimuth. An example of a Collar file of this type is:

```
HOLE_ID RL X Y EOH DIP AZIMUTH
ASN213  317.9  3867256.2  6542983.2  201.8  -78.2  275.6
ASN216  320.7  3867372.4  6543961.9  186.9  -83.2  263.9
ASN217  321.3  2867386.4  6543978.3  196.4  -83.4  269.3
```

Sample collars file with hole depth, dip and azimuth fields

Note that this drillhole data definition can only define straight holes.

Collar and survey file

A Collar file accompanied by a Survey file that accurately defines the hole shape by recording downhole survey information using a drillhole sensor or camera. Examples of Collar and Survey files are:

```
HOLE_ID  RL  X  Y
AN3410  647.45  516166.2  6310431
AN3409  622.7   516025.6  6310278
AN3408  621.51  516132.1  6310196
AN3407  621.4   516113    6310208
AN3406  622.28  516090.2  6310224
AN3405  622.57  516074.4  6310236

HOLE_ID  DISTANCE  DIP  AZIMUTH
AN3410  5      -88.5   215
AN3410  22.5   -88.5   215
AN3410  51     -87.75  215
AN3410  70.5   -87.5   200
AN3410  100.4  -87.25  203
AN3410  122.8  -87.25  213
AN3410  153.5  -87.5   253
AN3410  178    -85.5   254
AN3410  202    -85.25  253
AN3409  0      -88.5   88.5
AN3409  25.6   -88     88.5
AN3409  51.5   -88     88.5
AN4209  125.3  -84     80.7
```

Sample collars file with separate survey file

Downhole samples and log files

Additional data files that can store geochemistry or lithology (in From-To measurements) or geophysical logging (measured at distance points) can also be read in.

Files with XYZ Drillhole Data

A line of readings that can be specified at positions in a drillhole can be defined in an ASCII XYZ file. In this format, each reading has its own record corresponding to a location in the horizontal plane (X and Y) and in the vertical plane (depth or Z). For inclined holes, the X and Y positions both change. Sensor readings such as downhole magnetometry, susceptibility readings or other logging measurement methods can have readings associated as extra data fields in each record. From-To type data (eg lithology interval samples) cannot be specified using the data type.

This type of data can only be created by a post-surveying step called de-surveying. In de-surveying, a software program uses the relative recorded positions of distance down the hole axis, dip and azimuth to convert to absolute locations of Easting, Northing and Vertical depth. Survey data in this form can be imported into Encom Discover PA as described in *Importing ASCII Data Files*.

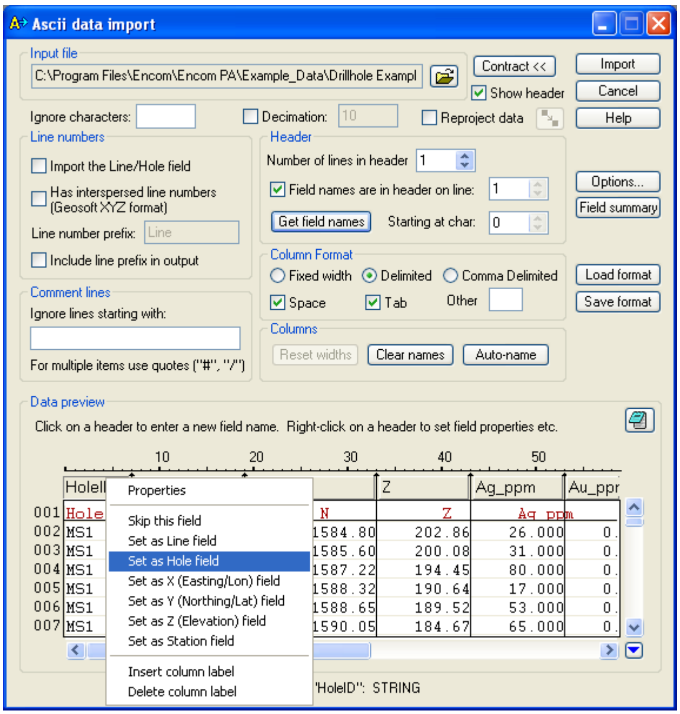
An example of a downhole XYZ file with some geochemical assay data is shown below:

```
ID  X  Y  Z  Ag  Au  Cu  Pb  Zn
MS1  641906.7  8141585  202.855  26  0.005  656  100  800
MS1  641905.4  8141586  200.083  31  0.005  206  800  1100
MS1  641902.5  8141587  194.454  80  0.005  150  700  1100
MS1  641900.6  8141588  190.644  17  0.005  624  1500 2300
MS1  641900.1  8141589  189.518  53  0.005  743  500  800
MS1  641897.6  8141590  184.668  65  0.005  799  400  1600
MS1  641895.5  8141591  180.338  14  0.005  580  300  1000
MS2  641604.6  8141592  178.642  70  0.005  891  200  500
MS2  641603.7  8141593  176.946  68  0.005  681  200  300
MS2  641601.8  8141594  175.253  80  0.005  693  300  300
MS2  641599.0  8141594  173.554  30  0.005  665  100  100
```

Sample XYZ data file

Note The above data file is supplied with Encom Discover PA and is located in the \DRILL EXAMPLE folder as Assays_NT_3D.XYZ. The data example above specifies a Hole ID column. In fact, when imported into Encom Discover PA, each drillhole is specified as a Line rather than a drillhole. This is related to the fact that the import facility is generic and can load both XYZ and line data. To display as drillhole data however this makes no difference.

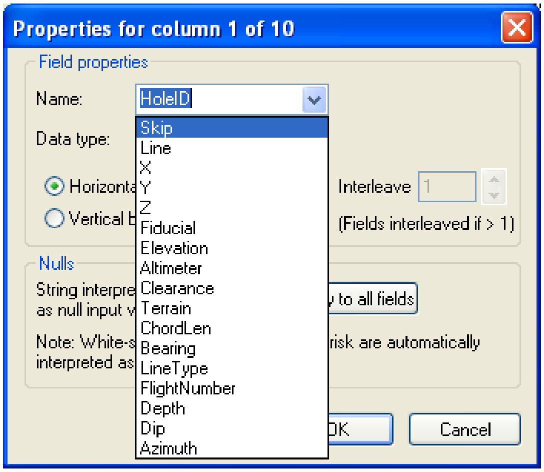
To import an XYZ data file into Encom Discover PA, use the ASCII Import tool (refer to *Importing ASCII Data Files*) from the **File>Text Import** or from the **ASCII Import** button. Navigate to the file to import and select it.



ASCII Import tool dialog with an XYZ file specified

Note in the above dialog that the XYZ (from the example file supplied) is defined. The field names for each column have been determined from the column headings in record 1 (using the **Get field names** button).

If the column headings need to be reset, you can position the cursor over the column heading, click the right mouse button and select from the available headings in the pull-down list, or choose **Properties** and manually configure the heading (see dialog below).

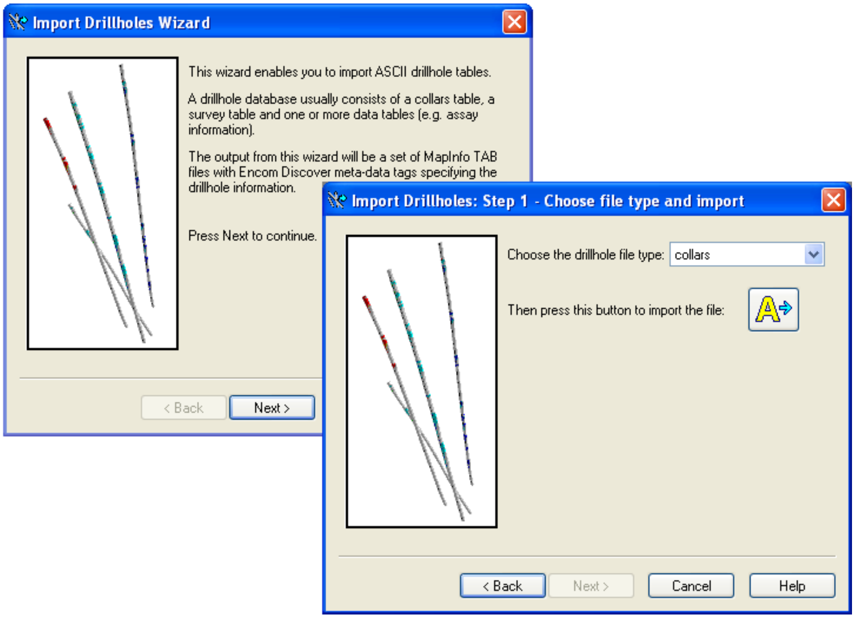


Modify the HoleID column to be recognised as a Line by Encom Discover PA

Click the **Import** button to create a database (you need to specify a database name) and have the data automatically accessed by Encom Discover PA. The import process displays a message to advise you that the Hole_ID column may have non-numeric data (that is MS1, MS2 etc). Select the OK button and the import of the drillholes are numbered sequentially as they are read.

Import Drillholes Wizard

Select **File>Open>Downhole>Import ASCII Downhole Data** to display a wizard for loading either Collar, Survey, From-To Data or Point Data files as shown below:



Wizard dialog to select the type of data to be loaded

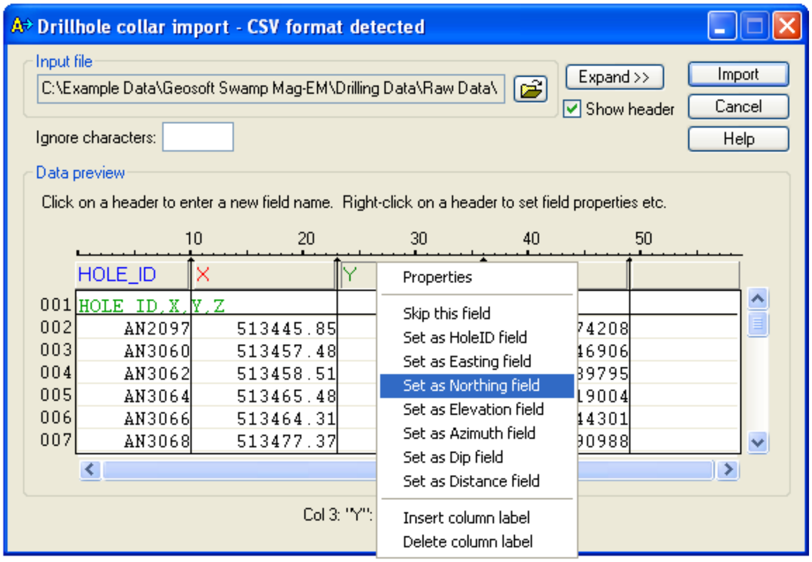
Follow these steps to import ASCII drillhole data:

Step 1

To load a drillhole file, select the Collar drillhole data file type from the drop-down list and then the ASCII Import button. A File Open dialog displays allowing the input data file to be specified. The file can be of any ASCII format suitable for import as specified in the *Importing ASCII Data Files* and includes ASCII column-based or CSV formats as shown in the examples above.

Step 2

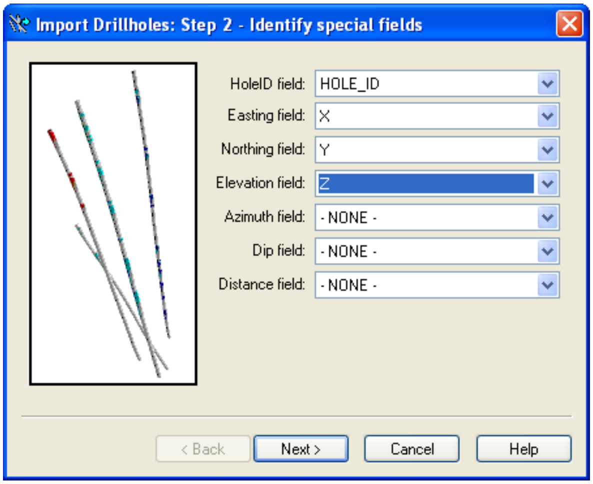
The import tool allows you to name the columns of data required. By positioning the cursor over the column headings and clicking the right mouse button, a pop-up list of relevant channels are displayed as shown below for a *Collars* file. The drillholes HoleID field should be highlighted in blue. The Easting (X) and Northing (Y) fields should be highlighted in red and green. If these are not automatically detected and highlighted you can specify these from the pop-up list. For additional information on using the ASCII Import tool, please refer to *Importing ASCII Data Files*.



Pull-down menu list for a Collars file specification

Once all columns in the preview have been defined, click the **Import** button. You are requested to specify a path and name the output .TAB file to save the output of the Collars data. When this is done, the Drillhole Importer wizard asks if you would like to specify an output coordinate projection for the collars. If No is selected then the collars XY coordinates are placed in Non-Earth projection. If Yes is selected then the Choose Projection dialog will appear and you will need to specify a Category and a Category Member from the two drop-down lists.

Once **OK** is selected the Import Drillholes wizard dialog re-displays, reporting that the Collars .TAB file has been created and the next dialog in the wizard asks you to identify the drillhole fields. The fields in the drop-down boxes are those available in the imported Collars ASCII file and these need to be matched with those specified to the left of each drop-down list. If no Azimuth, Dip or Distance fields exist in this file leave the selection as the default -NONE-.



Assign the data channels as required to their appropriate matching data

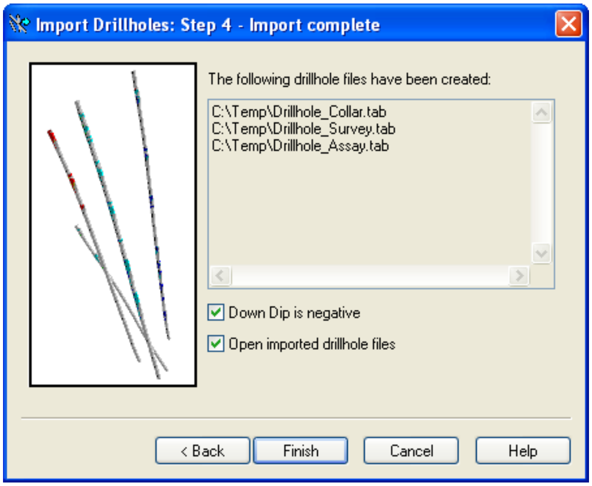
Step 3

The final process of the Import Drillholes wizard asks if you have any additional drillhole files to import. If you have a Survey or Downhole Data (lithology or assays) file to add click on Yes and you will be redirected back to Step 1 to choose this drillhole file type.

If you have Survey and Downhole Data files available you will need to repeat the above procedure for each of these files. Click the Yes button to load additional data and then go back to Step 1 above. Repeat this process until all files are imported. Mixing From-To data with Point data files for import is provided so drillhole information such as assays can be displayed with logging point data (eg specific density).

Step 4

When you have imported all the drillhole files required, select No when prompted about additional files for import. When this is done the following dialog is displayed:



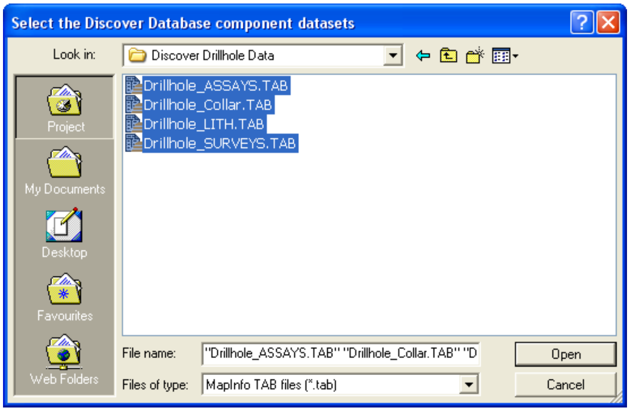
Final wizard drillhole import completion dialog

The paths and filename for all individual drillhole files are displayed. Two additional options available include:

- The direction of the drillholes relative to the horizontal plane is important for correctly displaying the drillhole axes. If the dips are negative down from the horizontal plane ensure the **Down Dip is negative** option is checked on.

- The data can be automatically loaded into Encom Discover PA if you enable the **Open imported drillhole files** option. This loads the imported and listed files and displays a spreadsheet of the data.

If not loaded automatically (see option above), to import the drillhole data into Encom Discover PA, select the **File>Open>Downhole>Discover Downhole** menu item and navigate to the folder containing the imported files. Select the Collar, Survey (if it exists) and any required data tables all together as shown below.



Multi-select the required .TAB files of the drillhole project

When imported into Encom Discover PA, the project drillholes can be seen in the data spreadsheet and can be displayed in two or three dimensional maps or sections. Refer to the relevant display section of this guide for more information.

Encom Discover Drillhole Data

You can access drillhole data for display in Encom Discover PA created in Encom Discover. For additional information on MapInfo Professional .TAB files the data files and Discover drillhole projects, refer to the Encom Discover User Guide. Also refer to Integration with Encom Discover in this manual.

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.

Once a drillhole project has been established in Encom Discover, select the **File>Open>Downhole>Discover Downhole** menu option or the **Open Discover Drillholes** toolbar button. Navigate to a valid downhole dataset. To import all drillhole information from the collars, survey and downhole data files the four .TAB files must be selected and imported in one operation

When imported into Encom Discover PA, the project drillholes can be seen in the spreadsheet window and can be displayed in two, three dimensional maps or sections. Refer to the relevant display section of the User Guide for information on this.

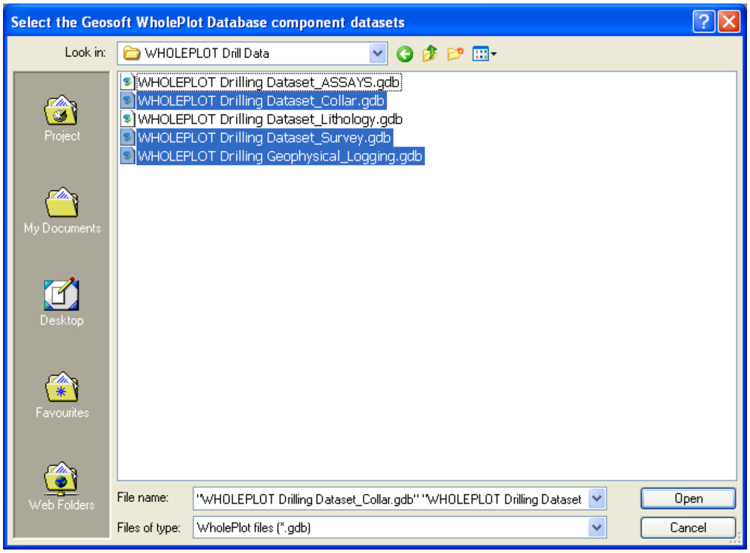
Note When using a Discover Downhole dataset Encom Discover PA provides the facility of converting downhole data from one depth unit to another. A **Define Dataset Properties** dialog allowing this conversion is displayed when the data files are specified.

Geosoft WHOLEPLOT Data

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 a single .GDB file or multiple .GDB files.

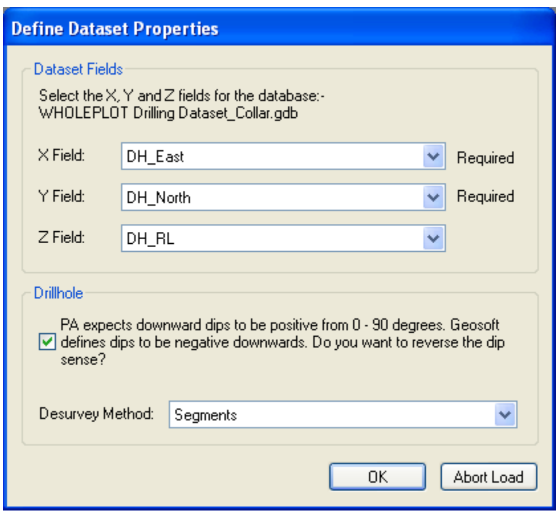
Two Geosoft WHOLEPLOT formats have evolved with differing versions of the Geosoft montaj WHOLEPLOT software. Prior to version 4.2, WHOLEPLOT databases stored collar, survey and acquired data within a single .GDB database file. After the release of version 4.2, separate databases containing the collar, survey and reading data were provided. Both single and multiple .GDB datasets are supported by different menu options in Encom Discover PA.

Select **File>Open>Downhole>Geosoft Downhole** menu option to access an Open File dialog. Navigate to the drillhole folder, choose the geosoft database and click **Open**. The Geosoft .GDB database files will be either suited for Geosoft OASIS version pre 4.2 (in which a single database file is input) or after OASIS version 4.2 in which case, multiple database files need to be selected as shown below):



Multiple GDB Geosoft databases for input of a WHOLEPLOT dataset

Geosoft drillhole databases generally define downward dipping drillholes as negative values so Encom Discover PA assumes the Geosoft database adheres to this format. If this needs to be reversed, the **File>Modify Database Properties** dialog provides an option to do so for any Geosoft drillhole database loaded.



The Define Dataset Properties dialog

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.

When imported into Encom Discover PA, the project drillholes can be seen in the spreadsheet window and can be displayed in two, three dimensional maps or sections. Refer to the relevant display section of the User Guide for information on this.

For additional information on this data format, please refer to the Geosoft WHOLEPLOT documentation or:

http://www.geosoft.com/pinfo/oasismontaj/extensions/drillholeplotting.asp

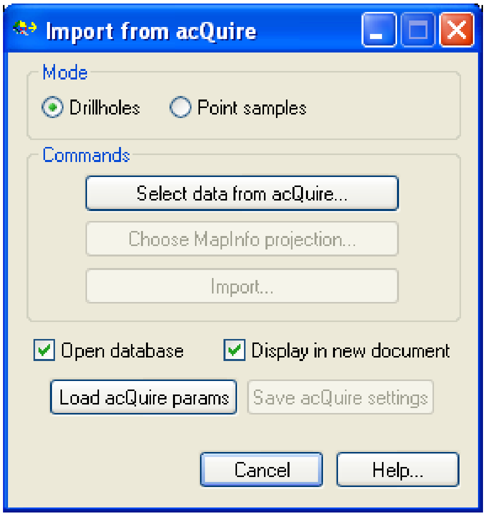
acQuire Drillhole Data

The acQuire software system is widely used within the geoscience community for the storage and retrieval of drillhole data. The acQuire software system allows a wide range of data formats to be supported including:

- FoxPro .DBF
- SQL Server
- Microsoft Excel
- Microsoft Access
- text files
- VisualFoxPro Microsoft

Note To use the acQuire drillhole database software you are required to have an operational licence of acQuire. The necessary software to operate acQuire with Encom Discover PA is automatically installed with the software..

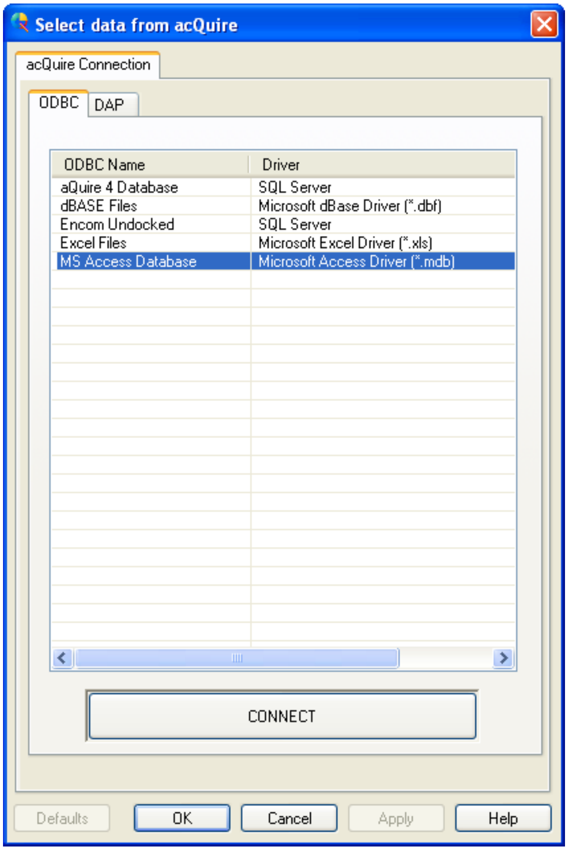
Select the **File>Open>Downhole>acQuire Downhole** menu option. When selected a dialog as shown is displayed.



Import from acquire dialog.

If you click the **Select data from acQuire** button an acQuire format selection dialog is presented. From the list you can specify the format of the drillhole data and then the program requests you navigate to the correct folder and select the input data file.

You can also ask that the data be automatically displayed once accessed through acQuire (using the **Display in a New Document** option).



Selection of drillhole data to import using acQuire

Once the acQuire data has been opened and read, it creates a set of MapInfo Professional .TAB files with the requested data and data channels (geology, assays, survey information etc). When you need to re-access the same data, you can either use the acQuire import again or the **File>Open>Downhole>Discover Downhole** menu item.

Desurvey Methods

Upon loading a drillhole project into Encom Discover PA, the drillholes are desurveyed using the **Back Calculation** method (detailed below) by default, as this is the method used by the Drillhole module in Encom Discover.

You can however choose a different method by opening the **File>Modify Database 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 Encom Discover PA, until a further change is made in this dialog.

The methods available are:

Minimum Curvature (recommended)

This is the most robust method and is highly recommended for use in Encom Discover PA. 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.

Back Calculation

This is the only method offered in the Drillhole module for Encom Discover and is the default method in Encom Discover 3D. Although it is a simple method it does produce as reasonable results as Minimum Curvature.

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.

Note that in Encom Discover PA the Back Calculation method ought to be used in preference to segments method.

Segments

This method was used in previous versions of Encom Discover PA and Encom Discover 3D. It is very similar to back calculation but produces poorer 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.

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. Akima ought to be used in preference to Bezier.

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.

Drillholes in Profile Sections

Once drillhole data has been imported into Encom Discover PA using any of the methods above, the data can be displayed in 2D vertical graphs for drillhole logs, 2D maps for collar and trace illustration and 3D tubes or rose presentation for advanced presentation.

Drillhole locations, collars and modulated downhole traces can be displayed in Profile Sections. Initially a Section branch must be present in the Profile and its vertical scaling must be in suitable units to position the drillhole traces. Similarly, only drillholes that lie within a defined buffer zone on either side of the profile line can be extrapolated onto the profile section. It is rare that the exact collar location and trace of a drillhole will be aligned with a survey line position and so the drillhole trace must be extrapolated onto the section.

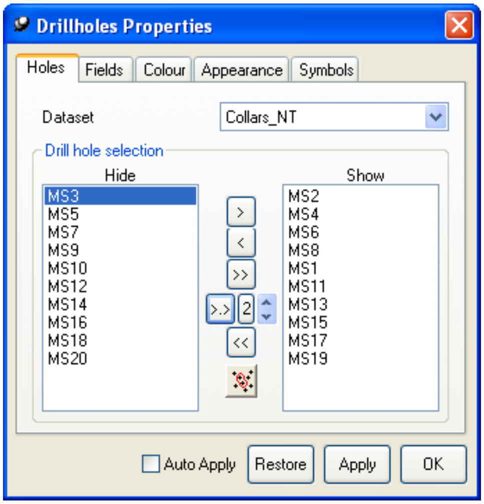
After importing the drillhole dataset displays a Section Profile in the usual manner. Add the drillholes to the Profile 1 branch by using the **Add Data>Drillholes** pop-up menu option or the **Add Drillholes** button.

For more informatio, see:

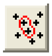
- *Drillhole Selection*
- *Data Compression*
- *Colour Modulation*
- *Trace Style*
- *Labelling*
- *Symbols*
- *Off-Section Clipping*

Drillhole Selection

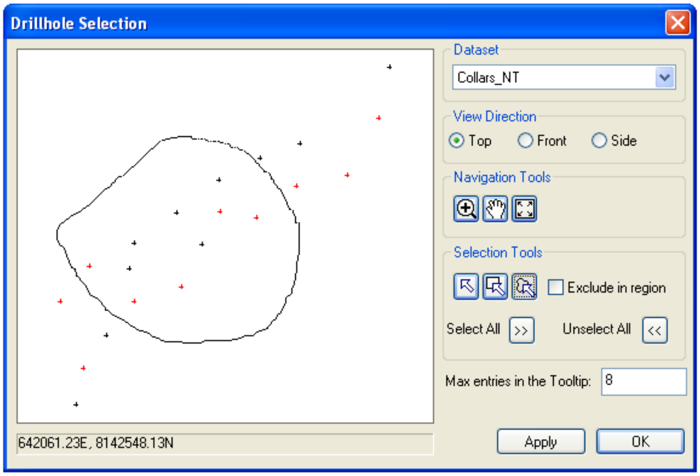
The **Holes** tab of the Drillholes Properties dialog allows the user to control which holes are displayed



Drillhole selection using the Holes tab

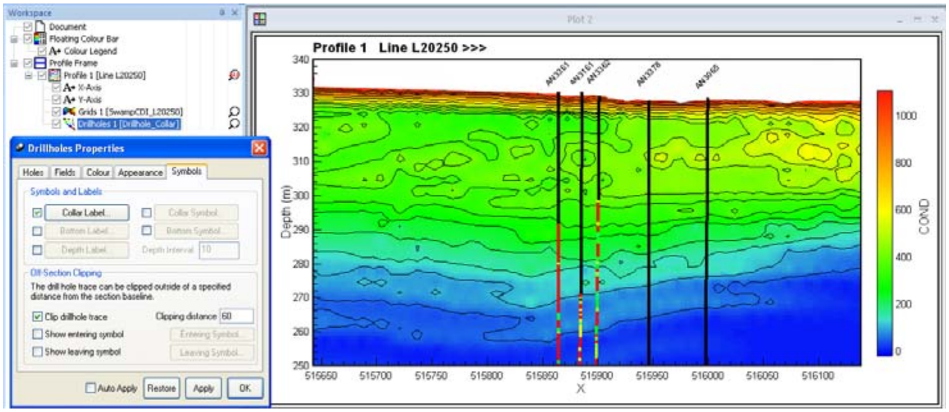


Specify the holes to be displayed by moving the drillhole name(s) from the Hide to the Show list. Alternatively, you can graphically select the holes to be displayed from a collar map using the **Graphical Selection** button.



The Graphical Selection dialog for Drillholes.

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.



Drillholes displayed in a CDI section of a Profile window

Data Compression

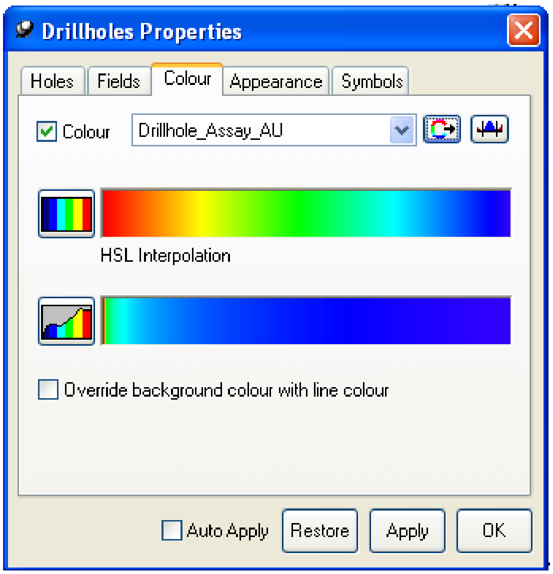
The Decimation Scheme option in the **Fields** tab enables the display rendering performance to be improved by reducing the number of samples displayed down the hole. This option is generally only appropriate for larger datasets where geophysical data has been collected at a small scale (e.g. centimetre 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 userdefined value.
- **Compressed** – a user-specified percentage of the downhole dataset is compressed.

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 gold (Au_ppm) assay field and a logarithmic colour mapping transform.

Enable the **Colour** tick box and set the desired data field from the adjacent pulldown 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 Look-up Colour Table or Legend Table if one has already been created using the *Colour Table Editor* or *Legend Editor*. Alternatively a range of standard colour interpolations (RGB & HSL) and Look-Up Tables (created using the Colour 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).

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 and 15 g/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 *Appearance Tab* will replace any background colours applied via a **Legend** in the **Fields tab** with the colour specified in the Appearance tab.

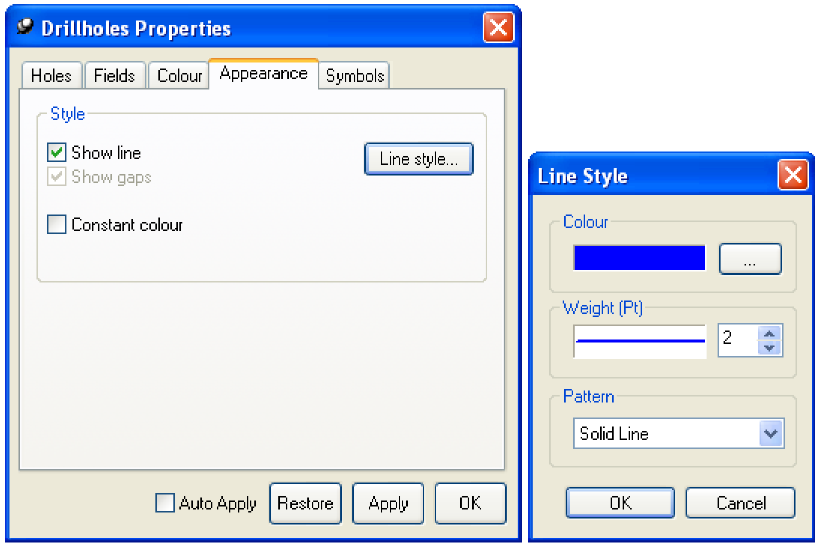
Drillhole Colour Legends

An on-screen colour legend Floating Colour Bar can be added to the 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 for further information.

Trace Style

The **Appearance** tab allows the trace style of drillholes to be altered.

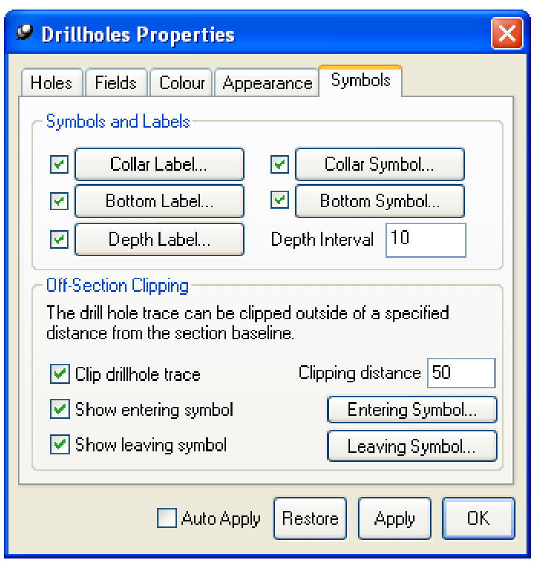
To alter the **Trace Style**, ensure the **Show Line** option is enabled, and select the **Line style** button. This will display the Line Style dialog allowing the selection of a new colour, weight and pattern for the drillhole trace.



The Drillhole Properties Appearance tab dialog.

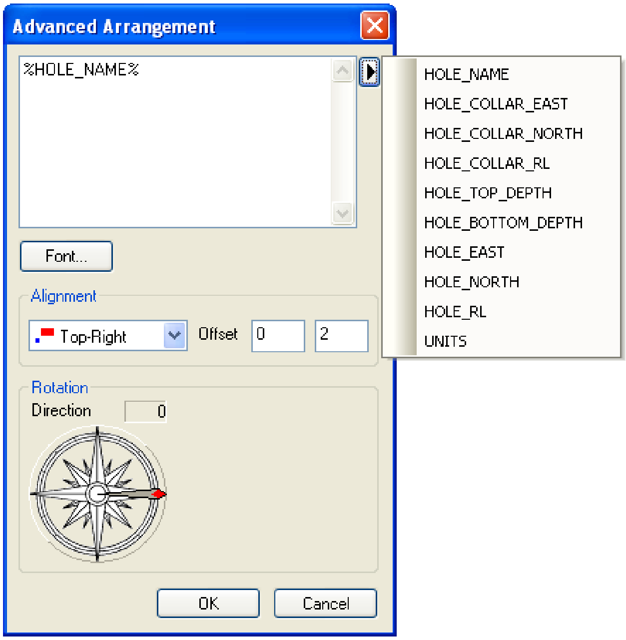
Labelling

Select the **Symbols** tab to activate **Symbols and Labels** for the drillholes and to apply **Off-Section Clipping**.



The Symbols tab for Drillholes Properties.

Drillhole collars and bottoms can be annotated with labels by enabling the appropriate tick box in the **Symbols and Labels** section. To change or customise the label display, click the adjacent **Collar Label**, **Bottom Label** or **Depth Label** button to open the Advanced Arrangement dialog.



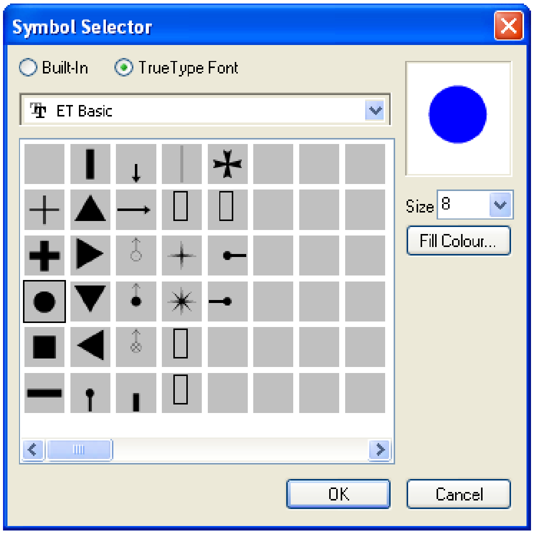
The Advanced Arrangement dialog for applying drillhole collar and bottom labels

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).

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

Off-Section Clipping

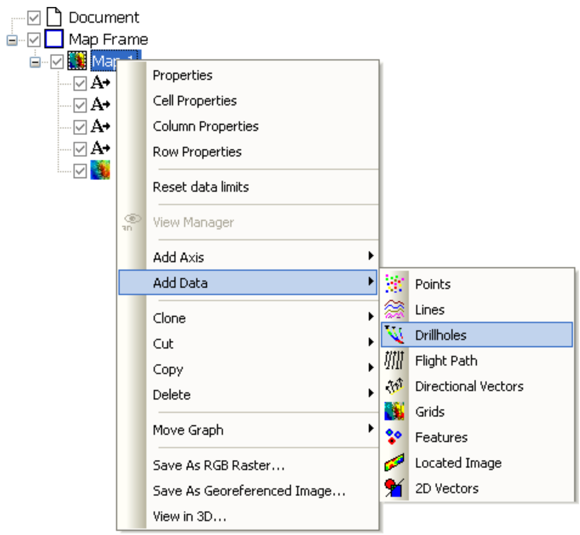
The **Off-Section Clipping** section controls the visibility of drillhole traces in a section based upon their distance from the section baseline. To apply a drillhole clipping to each section displayed select the **Clip drillhole trace** tick box and enter a **Clipping distance** value (in metres). For example a value of 50 will display all drillholes located within 50 metres of a section profile.

Some drillhole traces will appear truncated in section profile displays. This is because a drillhole may intersect the section profile and disappear out the other side of the section. In this case it is recommended to enable the **Show entering symbol** or **Show leaving symbol** options which will highlight the position and behaviour of the drillhole trace.

Adding Drillholes to a 2D Map

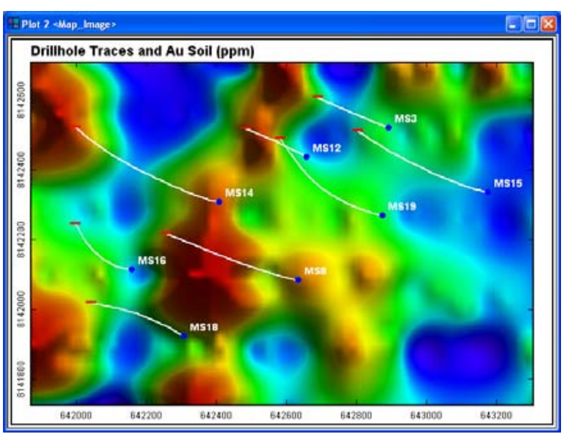
Drillhole locations, collars and modulated downhole data can be displayed in a two dimensional map. Rose diagram displays cannot be shown.

After importing the drillhole dataset display a 2D Grid Map in the usual manner. Add the drillhole locations to the Map 1 branch of the Workspace tree by using the **Add Data>Drillhole Group** pop-up menu option or the **Add Drillholes** button.



Select the Add Data>Drillhole Group item from the pop-up menu

With the holes displayed in the map, configure them using the drillhole properties as explained in the section for 3D Displays.



Example of drillholes within a 2 dimensional map showing Au soil grid.

Adding Drillholes to a 3D Map

Once drillhole data has been imported into Encom Discover PA, the data can be displayed in a 3D Map. Create a 3D Map display as described in *Three-Dimensional Displays* or select the **File>New>Map>3D Grid Map** menu option or the toolbar option.

With a 3D Map active, click the **Add Drillhole** button from the Data Object toolbar or choose the **Add Data>Drillhole Group** menu item from the pop-up menu of the 3D Map branch in the Workspace tree.

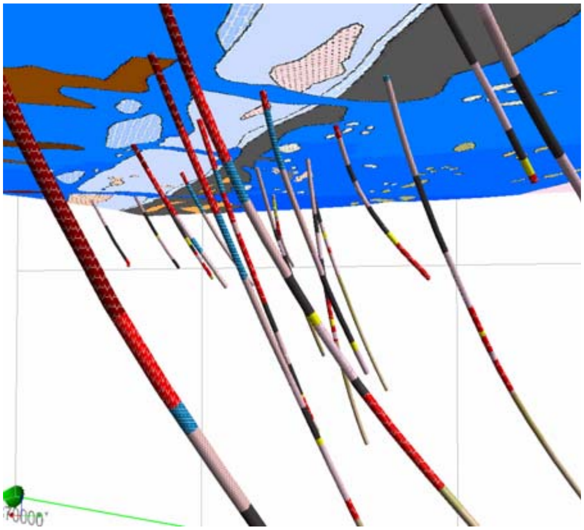
A **Drillhole** branch is created in the Workspace tree and the first of the available holes is displayed as a 3D hole. See *Drillhole Properties* for information on how to change the display properties.

Drillhole 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
- Select the **Drillholes** branch in The Workspace Tree and click the right mouse button. From the resulting pop-up menu, select the **Properties** item.

Various display controls can be applied to the drillholes depending on the type of data available. An example of a downhole display with lithology is shown below.



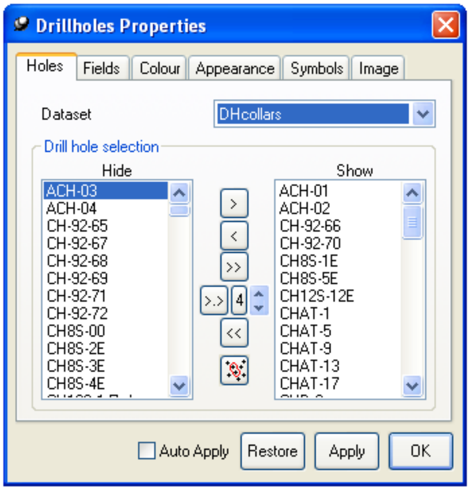
Example drillholes with downhole lithology displayed

For information on using the Drillhole Properties dialog, see:

- *Holes Tab*
- *Fields Tab*
- *Colour Tab*
- *Appearance Tab*
- *Symbols Tab*

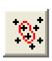
Holes Tab

The **Holes** tab of the Drillholes Properties dialog allows you to control which holes are displayed.

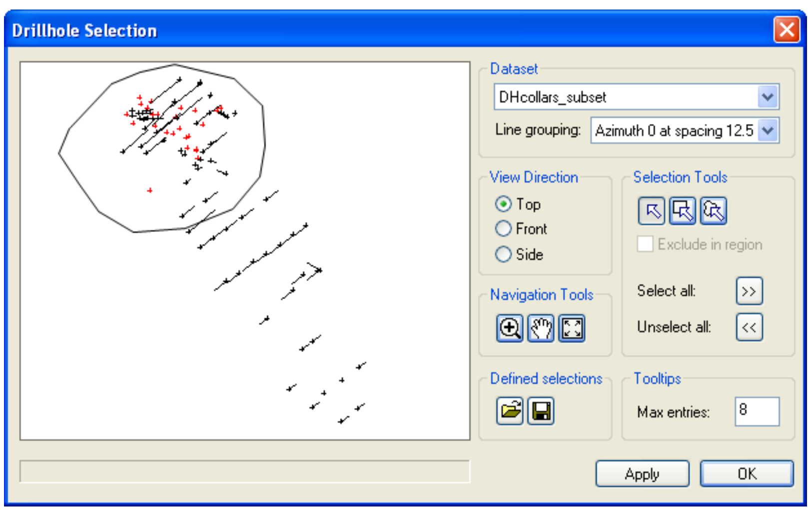


Drillhole selection using the Holes tab

If more than one dataset is open in Encom Discover PA make sure that the correct one is selected from the **Dataset** drop-down list.



Specify the holes to be displayed by moving the drillhole name(s) from the **Hide** to the **Display** list. Alternatively, you can graphically select the holes to be displayed from a collar map using the **Graphical Selection** button.



Drillhole graphical collar selection

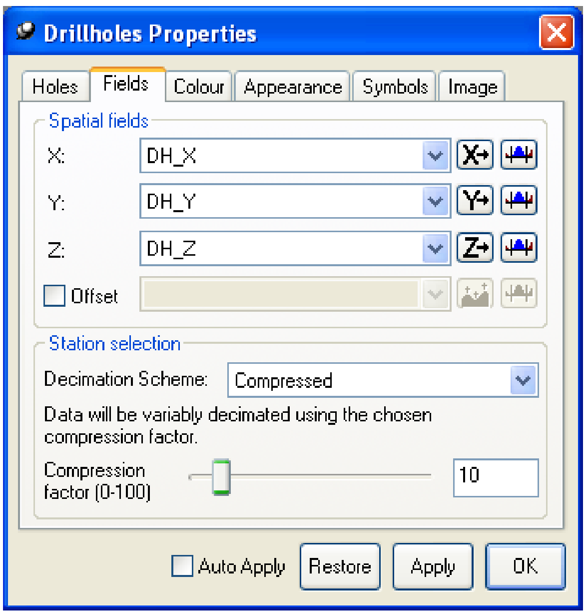
Note Drillhole collar name can also be displayed by placing the cursor over a drill hole and viewing the name in the left corner of the status bar at the bottom of the 3D screen.

Fields Tab

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:

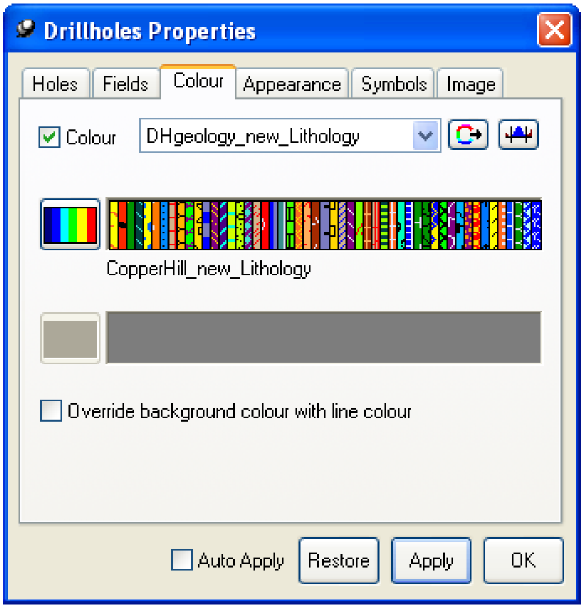
- **No decimation** – all samples are utilised i.e. no compression applied (the default option).
- **Fixed rate decimation** – utilises 1 in every n samples, where n is a userdefined value.
- **Compressed** - a user-specified percentage of the downhole dataset is compressed. Use the slider bar to control the compression percentage applied.



Move the slider bar to increase the compression of drillhole data

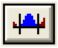
Colour Tab

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). Press the **Edit** button to open the Colour Scale dialog. Select a custom Legend if one has already been created 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 (created using the *Colour Table Editor*) are also available.

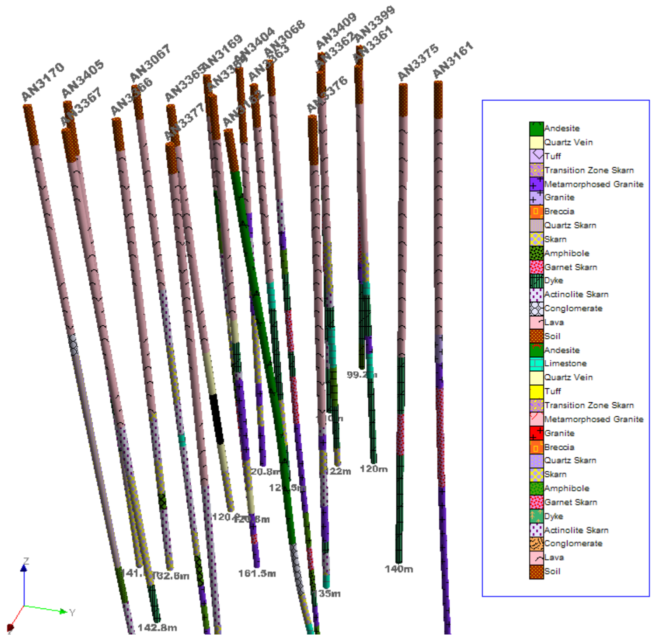


The **Field Data Conditioning** button is used to apply conditions to the selected colour field. Conditions available include data capping (ie. Setting upper data limits to minimise nugget effect for assay data) or excluding unwanted categories from the display (e.g. overburden such as soil and alluvials). See *Appendix G: Field Data Conditioning* for use of this tool.

Note Activating the Override background colour option in the *Colour* tab will replace any background colours applied via a Legend in the **Colour** tab with the colour specified in the **Appearance** tab.

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.



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

Appearance Tab

The **Appearance** tab allows the trace style of drillholes to be altered, as well as the application of labels and symbols.

- *Trace Style*
- *Thickness Modulation*

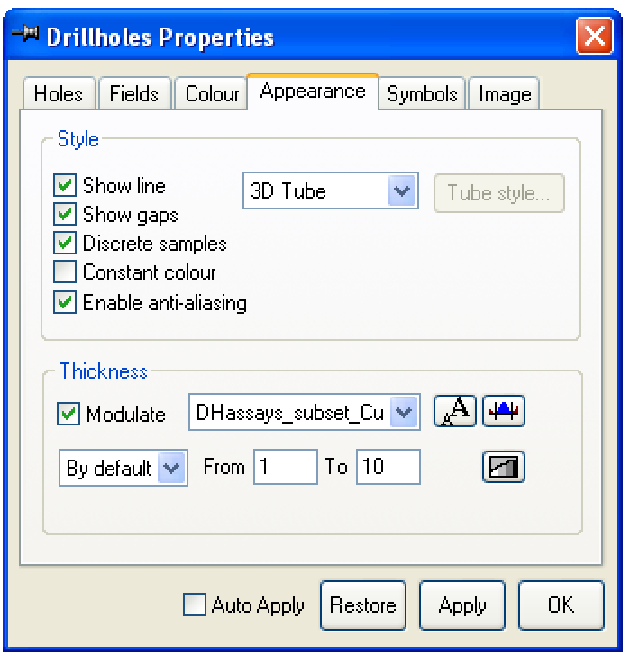
Trace Style

To alter the Trace Style, ensure the **Show Trace** option is enabled, then 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 pattern options under the **Line Style** button. Colour modulation is possible but thickness modulation is not.

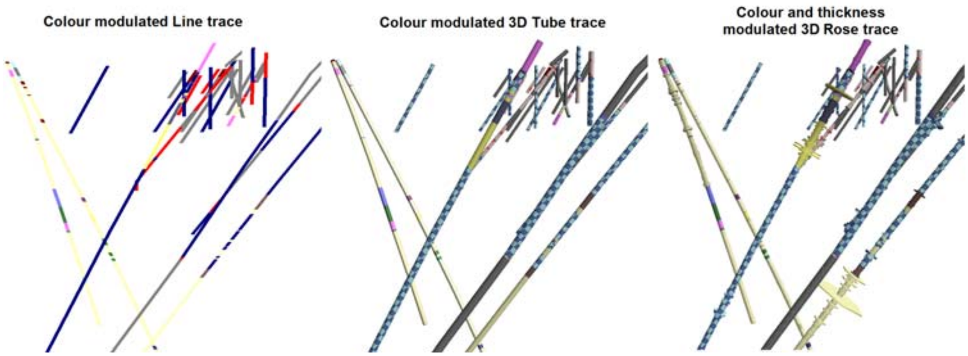
- **3D Tube** – Drillholes can be displayed as solid tubes, with a cross-section profile (and colour) set via the **Tube Style** button (default Tube_Medium style). The constant Thickness of the tube can also be specified. Colour modulation is possible but thickness modulation is not.

- **3D Rose** – This is the most memory intensive of the trace styles, allowing the width of the trace to be modulated based on one or more downhole numeric data fields (such as an assay or geophysical field); these are set under the *Thickness Modulation*. It requires specification of the Thickness range (minimum and maximum widths). 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.



Appearance tab of Drillholes Properties

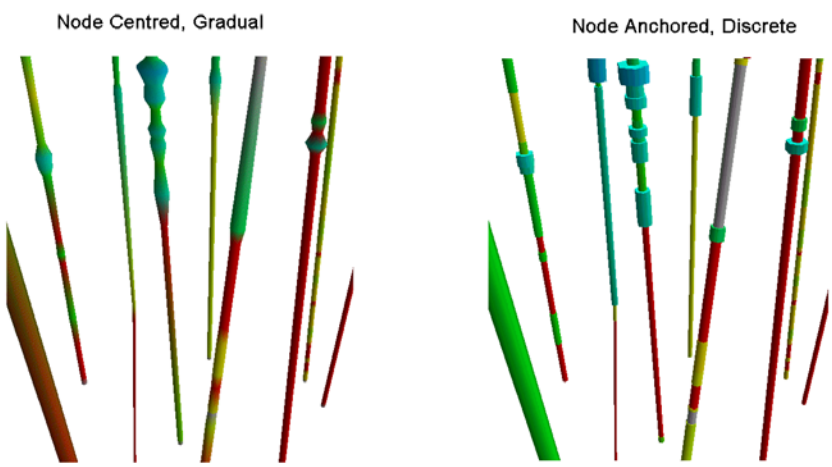
Different drillhole appearances are shown here:



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 plotted; ie drillhole traces will be discontinuous if sampling gaps exist.

Discrete samples are only applicable to the 3D Rose trace style. When enabled, each sample interval is displayed as a uniform thickness across the interval. 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.

Thickness Modulation

Drillholes can be thickness modulated in the Appearance tab either by:

- Single Field Modulation using the 3D Tube style, or
- Multiple Field Modulation using the 3D Rose style.

Single Field Modulation

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 to remove any null values from the dataset, or to limit the data range to that of interest (e.g. all Au values above 50 ppb).

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).
- A custom legend of data ranges and associated line thicknesses selected using the adjacent **Legend** selector button, created with the Legend Editor

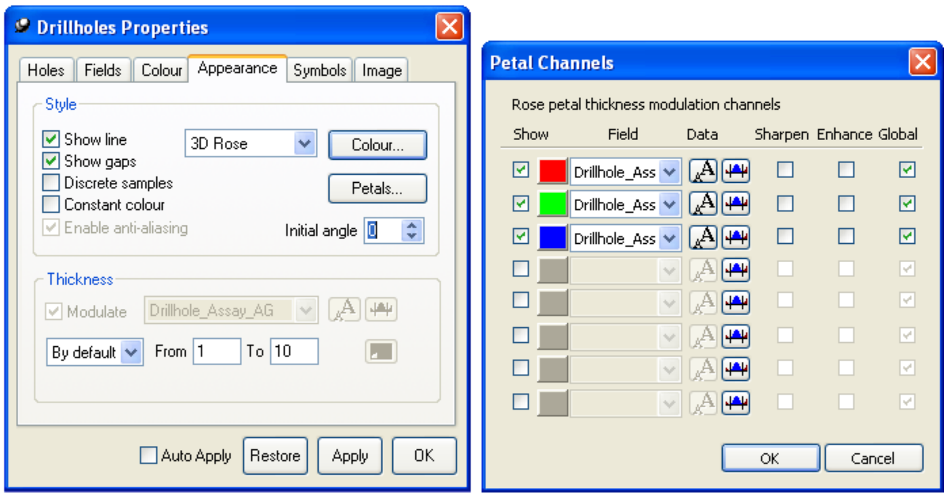
Multiple Field Modulation

To vary thickness of a drillhole by multiple numeric field 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).

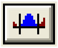
3. Press the **Petals** button (top right of the dialog) to open the **Petal Channels** dialog.



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.



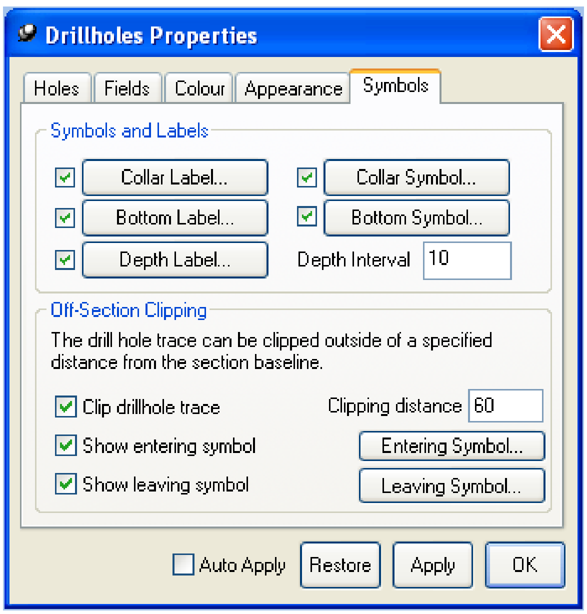
In addition each field has a **Field Data Conditioning** button which can be used to condition the source data. Type of conditions available include data capping (removing nugget effects), null value handling and invalid data assignment (eg. removal of large negative values representing BDL, SNR, etc, that affect thickness distribution). See *Field Data Conditioning* for use of this tool.

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 **Petal Channels** dialog will be overwritten if a colour **Legend** has been selected in the **Colour** 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.

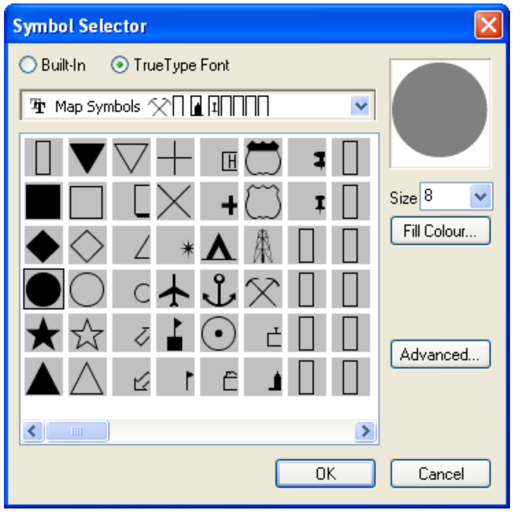
Symbols Tab

The collars and/or bottoms of drillholes can be annotated with a symbol by enabling the **Collar Symbol** or **Bottom Symbol** options in the **Symbols** tab.



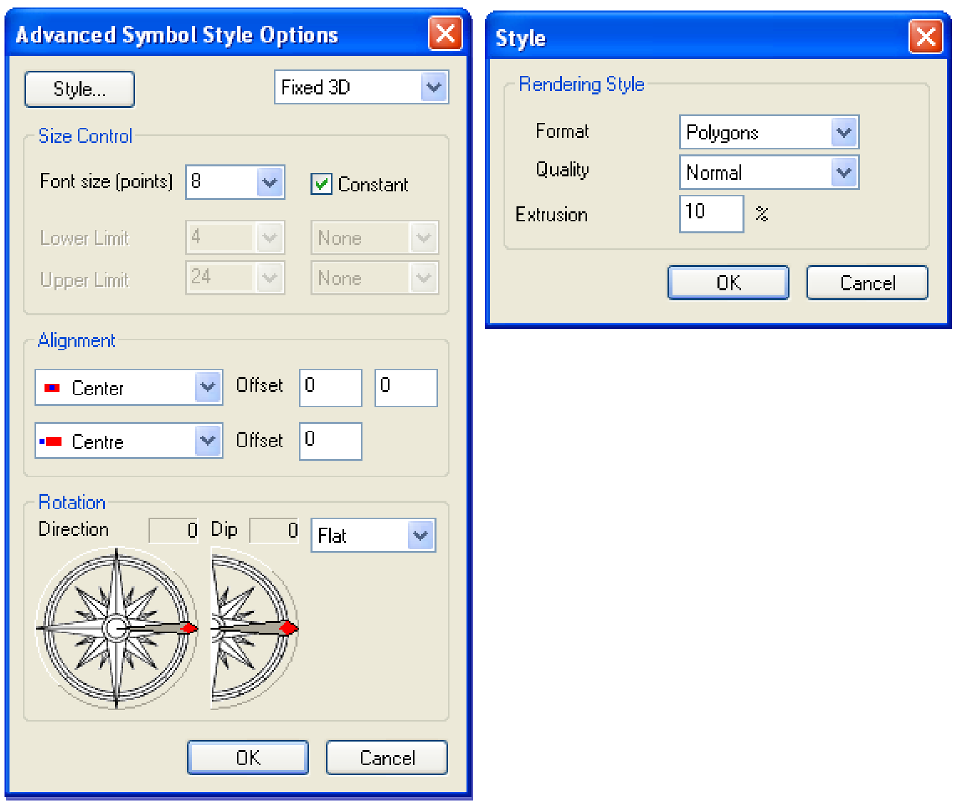
Symbols tab of Drillholes Properties

The adjacent buttons open the Symbol Selector dialog enabling the user 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 similar to those for Labels 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:

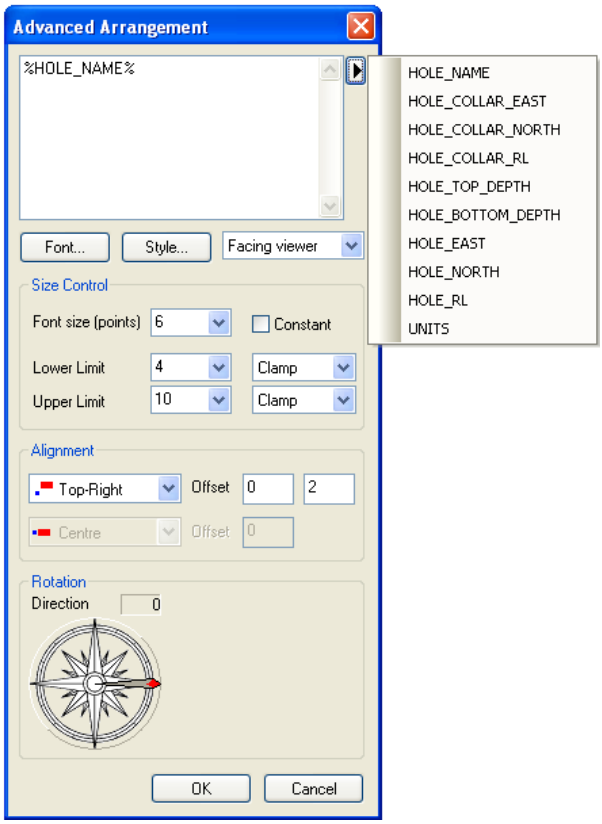
- **Block** - symbols will disappear when the applied zoom level takes the symbol past the specified limit
- **Clamp** - symbols will locked to the specified limit when the applied zoom level takes the symbols past the specified limit

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.

Labelling

Drillhole collar and bottom symbols appearance and annotation can be altered by enabling the appropriate tick box at the bottom of the **Symbols** tab. To change or customise the label display, press 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 press the **Field Selection** button (shown left). Select the desired field from the following 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 replaces 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.

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 locked to the specified limit when the applied zoom level takes the label past the specified limit

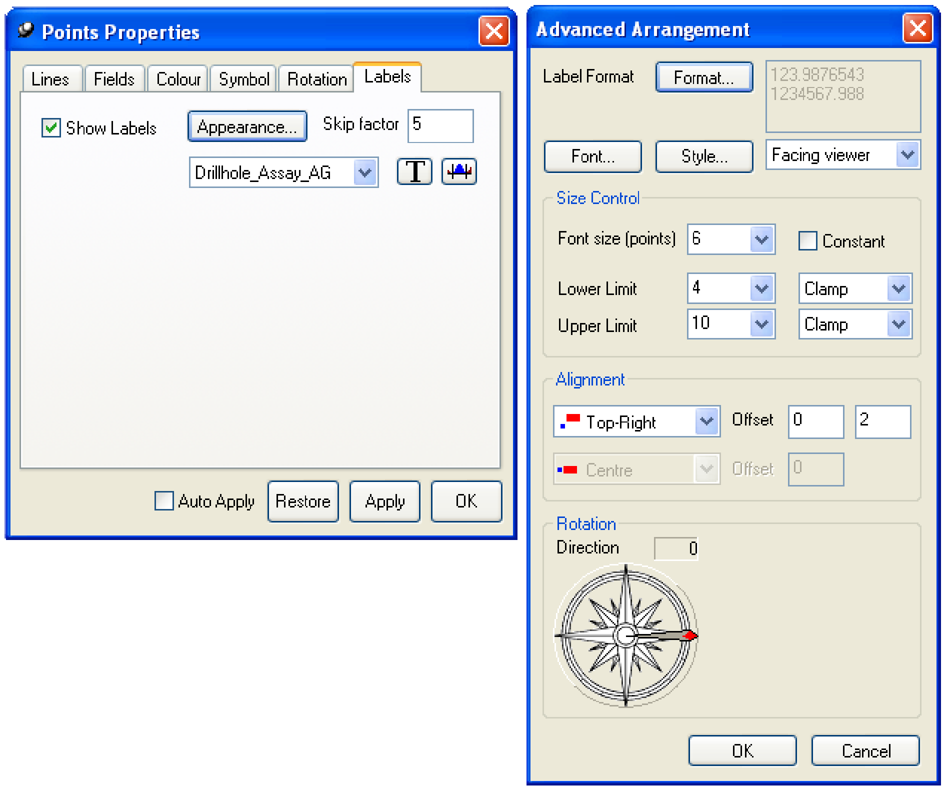
The label position relative to the data (collar) location can be altered using the **Alignment** controls (either preset or manual positioning). The first row of controls concern label positioning in the label plane (i.e. the relative XY components), whilst the second row controls the vertical height 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.

Plotting Data Beside Drillholes

Text values for drillhole data can be displayed adjacent to a hole by using a 3D Points object associated with a Drillhole Group. If you display the **Points Properties** dialog, you can disable the **Show Symbols** attribute in the **Symbol** tab and enable the **Show Label** drawing (from the **Labels** tab).

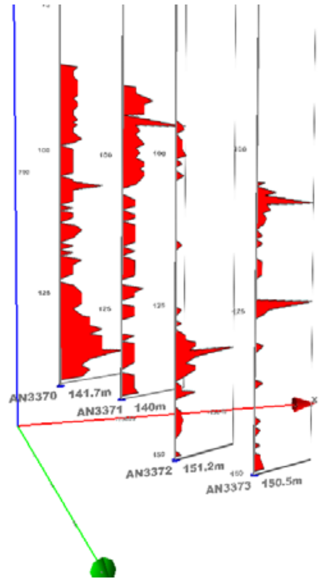
Select the data channel for plotting and configure the text for font, size and format. You may need to top every data value being plotted and for this, use the **Skip Factor**.



Using a Curve control to plot text values beside drillholes.

Display Downhole Logs

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 in a 2D Graph (Vertical Log) display or in Encom Discover to a 3D display. The Drillhole Log is created as a located image (.egb) using the Batch Hard Copy wizard utility and is rendered as an image curtain next to the 3D Drillhole.



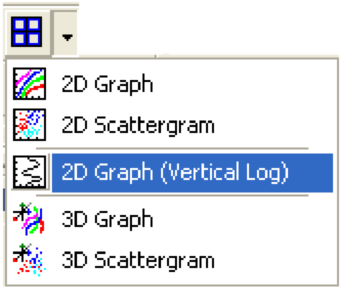
Multiple drillhole logs displayed in Discover 3D.

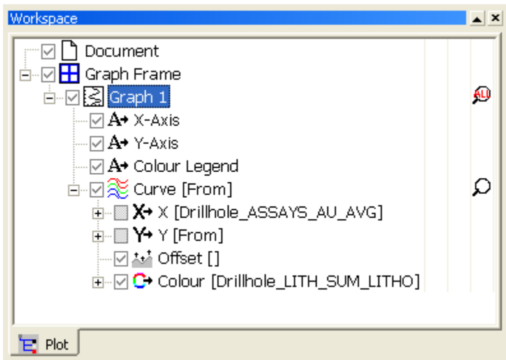
Step 1

Open a drillhole dataset using the **File>Open>Downhole** menu option.

Step 2

Open a 2D Graph display from the toolbar button or the **File>New>Graph>2D Graph (Vertical Log)** menu option. A 2D Graph display will appear with the Curve Properties dialog open on the **Data** tab. By default the first drillhole will be assigned to the display. Select the **Fields** tab and select a log field, such as an assay (Au, Ag, etc) or mag susc field to be assigned for the X field and for the Y field select the From field.

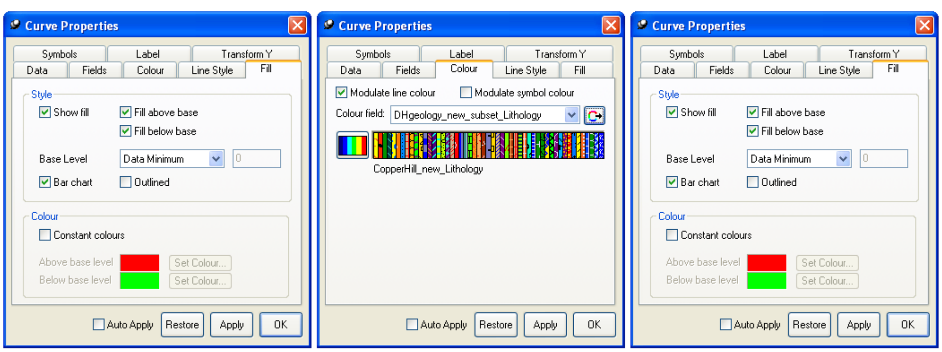




Representation of Workspace tree for a drillhole log graph

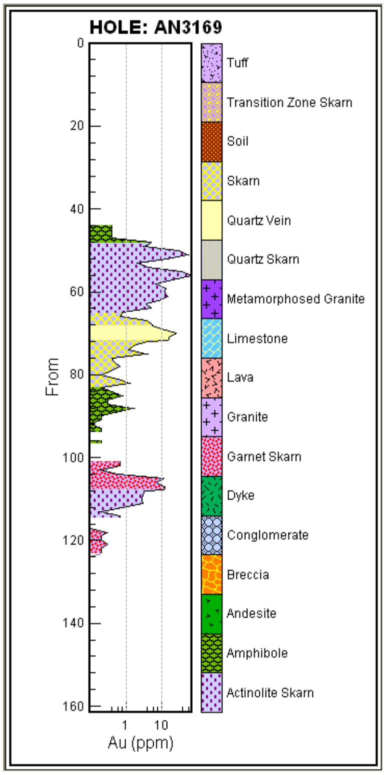
Step 3

In the **Line Style** tab dialog activate the **Modulate Line Colour** option and select an assay or lithology field from the drop down list. If choosing to colour modulate by lithology select a previously created legend file (.leg) from the Lookup Table/ Legend drop down list. Refer to *Legend Editor* for information on how to create this legend file. Also in the **Line Style** tab dialog activate the **Show Fill** option in the **Fill Settings** section. Select to **Fill above the base** option and specify to fill from the **Data Minimum** in the drop down list.

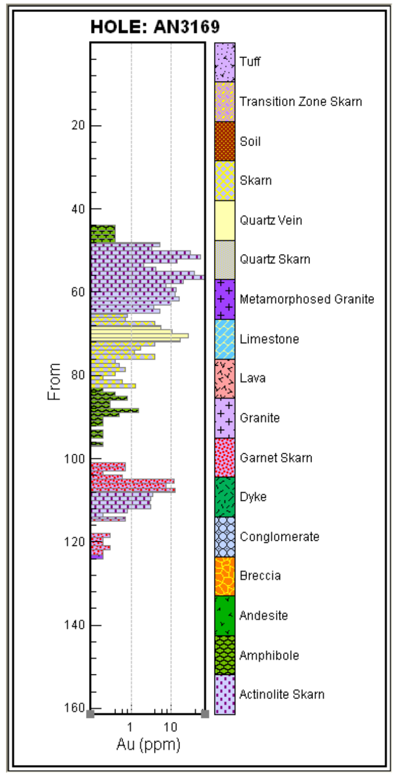


Use the Line Style, Fill, and Colour tabs to alter the appearance of the drillhole log

An option also exists in the **Fill Settings** section to display the drillhole log as a bar chart. Two examples of the resulting log display are illustrated below.



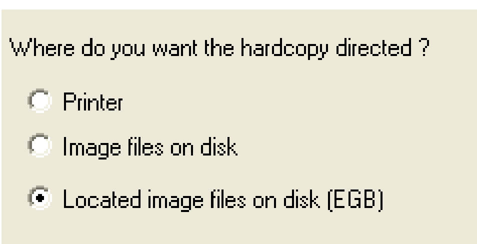
Example of line trace drillhole lo



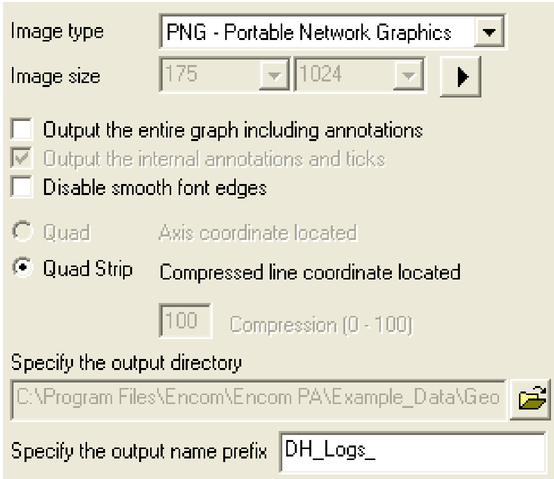
Example of bar chart drillhole log

Step 4 – Batch Printing Drillhole Logs

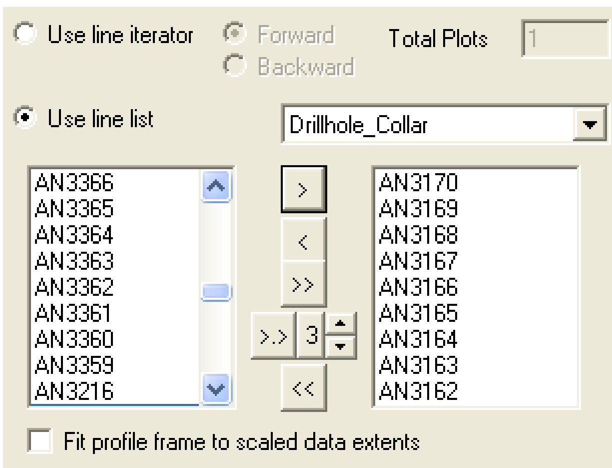
With a drillhole log display completed a batch printing of all drillholes can be performed using the **File>Batch Hard Copy Wizard** menu option and selecting to print the output to Located image files on disk (EGB).



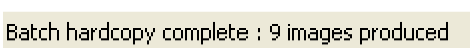
Specify the image size, format and name, including whether or not to display graph annotations and which folder to save the located images to.



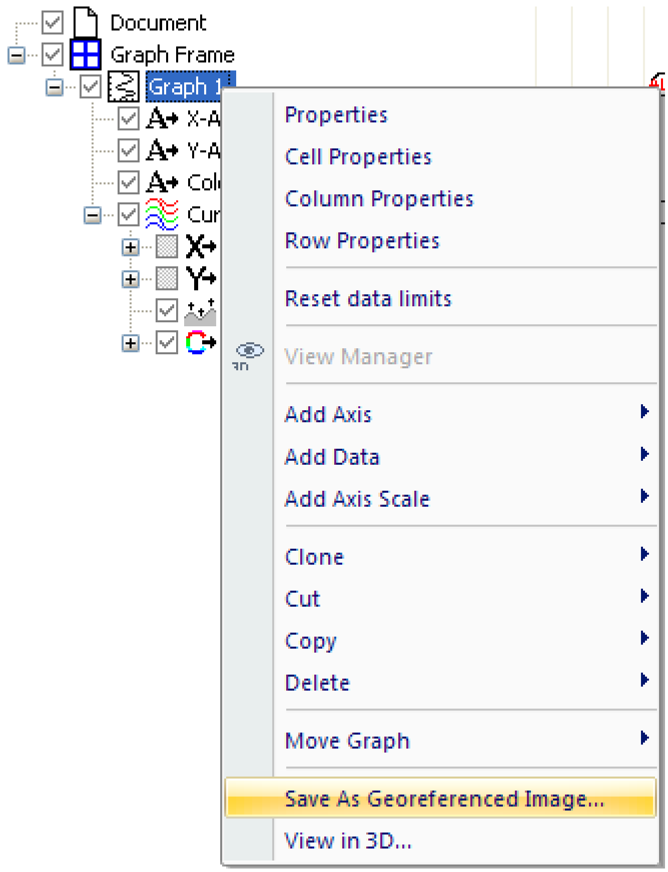
Then choose which drillhole logs to plot by using the line iterator (number of plots in forward or reverse direction) option or else select the individual drillholes from the list.



Once the **Finish** button is selected the batch printing will be executed and the status of the printing will be displayed in the status bar in the bottom left hand corner.



Note If only a small number of drillhole log displays are required to be output to located images, individual .EGB files can be created by clicking the right mouse button on the *Graph 1* branch of the workspace tree and selecting the **Save as Georeferenced Image** pop-up menu option.



Use the pop-up menu option in the Workspace tree to create a single located image of the currently active drillhole log

With a 3D display active and a drillhole group added to this use the **Image** tab of the Drillholes Properties dialog to control the orientation, scale and quality of the view.

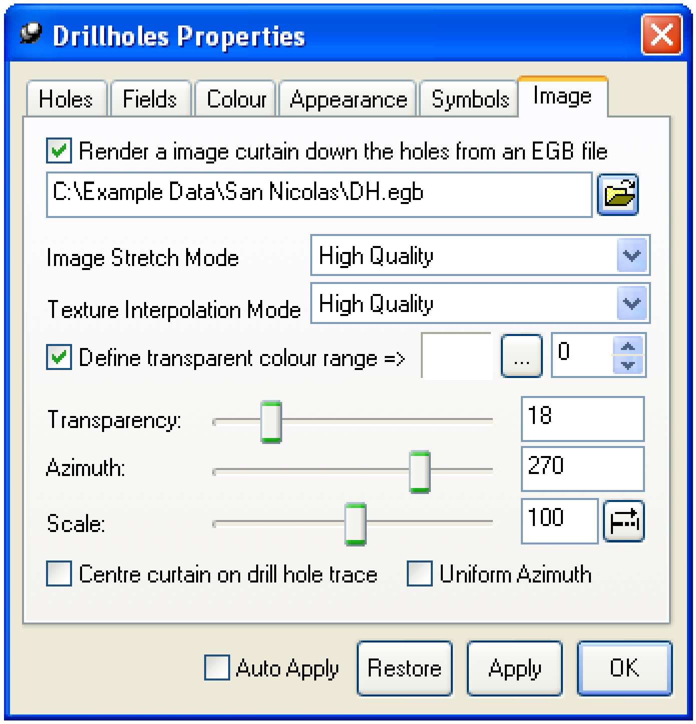
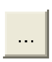
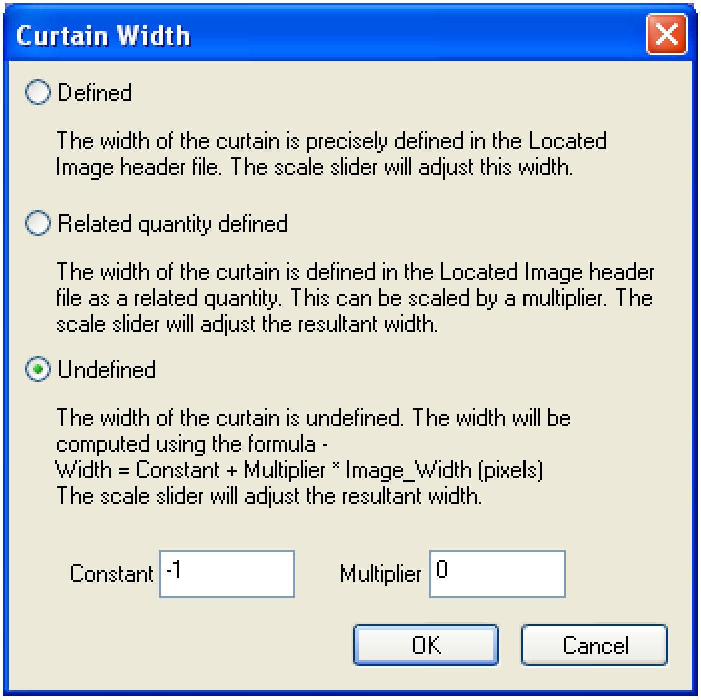


Image tab of the Drillholes Property dialog



The bitmap created by Discover is automatically listed, and the colour white is selected as 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 Encom Discover PA 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.

21 Voxel Models

Encom Discover PA allows block or voxel models (i.e. 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 using the 3D Gridding utility in the Voxel Toolkit.

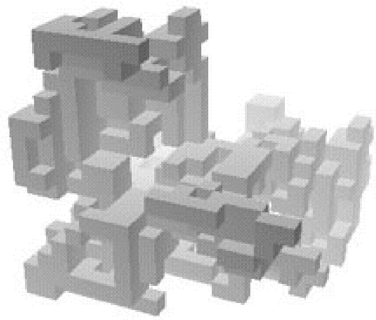
In this section:

What are Voxel Models?

The term voxel refers to a volume element and is the three-dimensional equivalent of the two-dimensional pixel. As used in 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, gravity density, conductivity or IP property (eg chargeability or phase) can be used. Certain geophysical software from inversion applications (e.g. UBC) and software products (such as Vulcan, Gemcom, Surpac) can create 3D voxel files.

Individual cell volumes for display by Voxel Modeller are prismatic in shape but voxel models need not be. For example, complex mesh designs are possible whereby individual cell dimensions vary for each of their 12 side lengths. Software such as Pitney Bowes Software's 3D Gridding utility (see 3D Gridding) or EM Gui are capable of creating and displaying these types of models.

Note Encom Discover PA *Voxel Toolkit* and *Array Data 3D Gridding* can also create 3D voxel model outputs, for example for a 3D drillhole or EM (conductivity-depth) dataset.



Example of complex voxel shape

Supported Voxel Model Formats

Encom Discover PA currently has complete support for six voxel/block model formats.

- **UBC** (University of British Columbia)
- **CEMI** (Consortium of Electromagnetic Modelling and Inversion – University of Utah)
- **Encom3D Grid**
- **ASCII XYZ Simple** (centred and uniform mesh only).
- **Noddy** (developed under AMIRA grant by Dr Mark Jessell at Monash University)

The files used as input to the Voxel Model are the result of modelling and inversion simulation software programs created by the above research organisations. 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.

Additionally, Encom Discover PA has limited support for the following voxel formats:

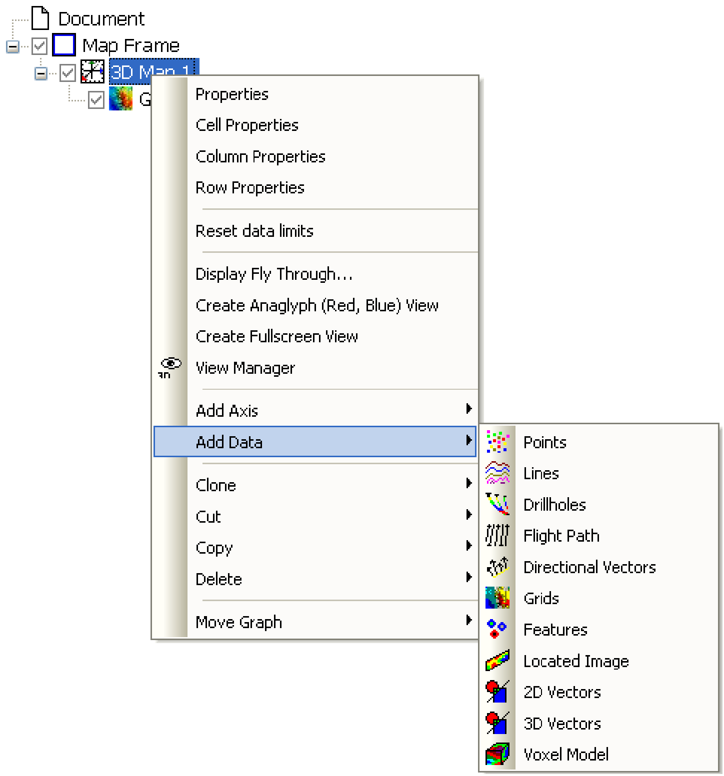
- **Datamine** – single precision only
- **Gemcom** – 16-bit integer, 32-bit integer and 32-bit floating point cell data types
- **Geosoft** voxel
- **GoCAD** voxel
- **Micromine** block model – ASCII file and single precision only. Sub-blocks are not supported.
- **Surpac** – version 3 supported, with 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.

For examples of these supported voxel/block model file formats, refer to *Appendix B: Data File Specifications*.

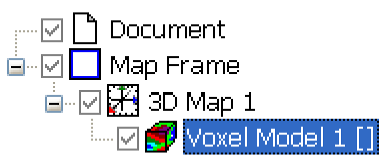
Creating a Voxel Model Display

To add a Voxel Model to Encom Discover PA, either a 3D display, curve profile display or 2D map display window must be active. Refer to*Profiles*, *Two-Dimensional Maps*, and*Three-Dimensional Displays* for information on how to create one of these display windows.

To add a Voxel Model to a display window highlight either the **Profile 1**, **Map 1**, or **3D Map** branch of the workspace tree, click the right mouse button and select **Add Data>Voxel Model** from the pop-up menu. Alternatively click on the **Add Voxel Model** button in the *Data Objects toolbar*. The resultant Workspace tree is shown below:

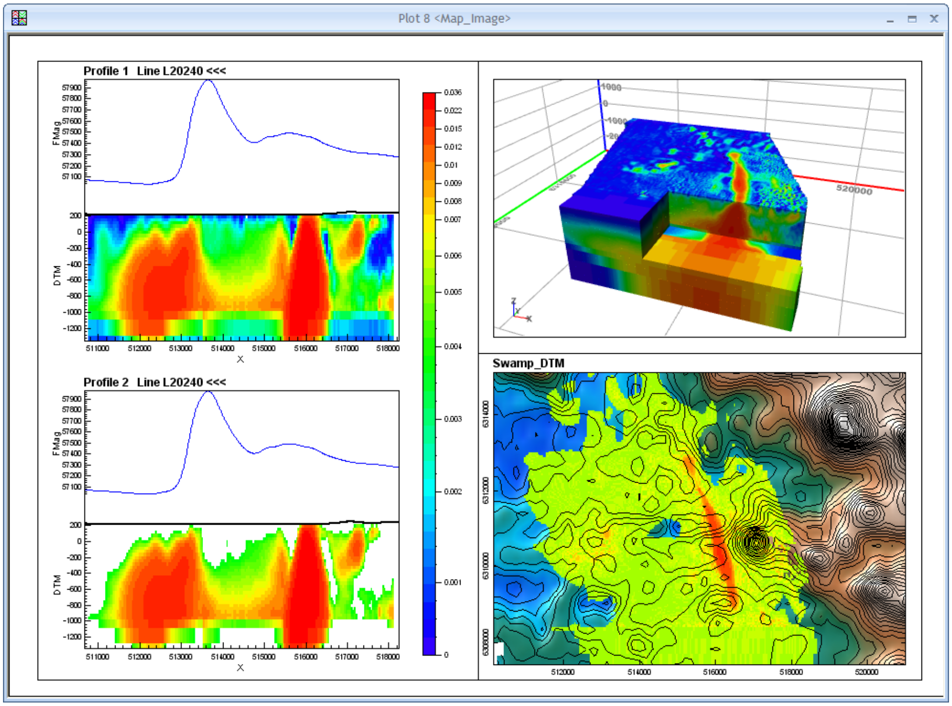


Adding a Voxel Model branch to the Workspace tree



Once the Voxel Model branch exists in the tree, double clicking it or highlighting it and right mouse clicking allows you to display the Properties dialog. For control of the Properties dialog (e.g. remaining on screen etc.), refer to *Display Properties*.

After the data for a voxel model is loaded into Encom Discover PA (see *Importing a Voxel Model*), you can add this to other displays by either using the cut/paste command or "drop" a voxel model into a display window by using the drag and drop operation from the Data Manager window.



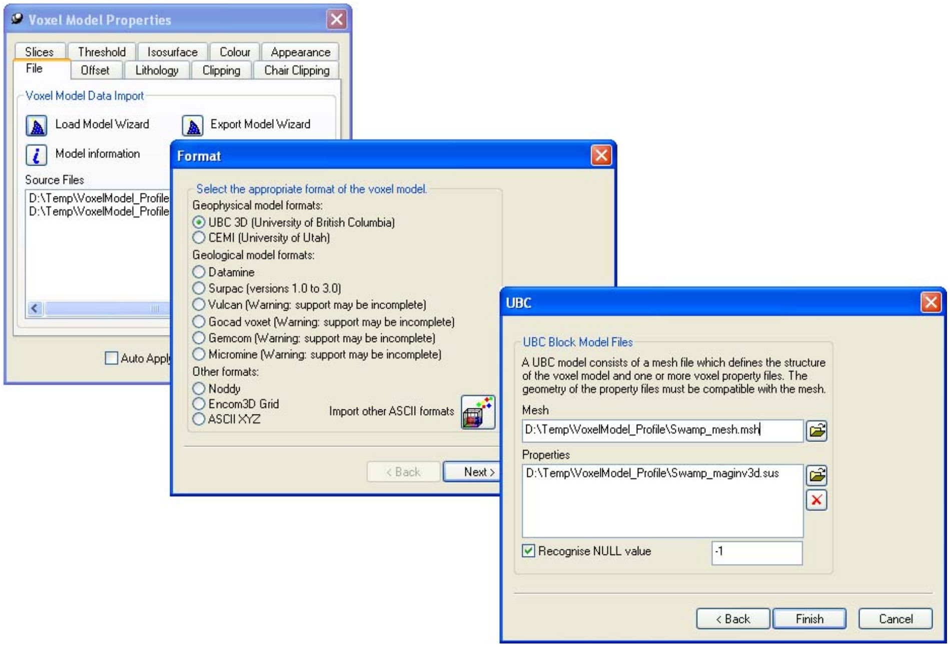
An example display of a voxel model shown in curve profiles, a 2D map with a grid image and a 3D display."

Importing a Voxel Model

The **File** tab of the **Voxel Model Properties** dialog contains the **Load Model Wizard** button . This wizard allows the selection of the model type and navigation to and selection of the relevant model file(s).

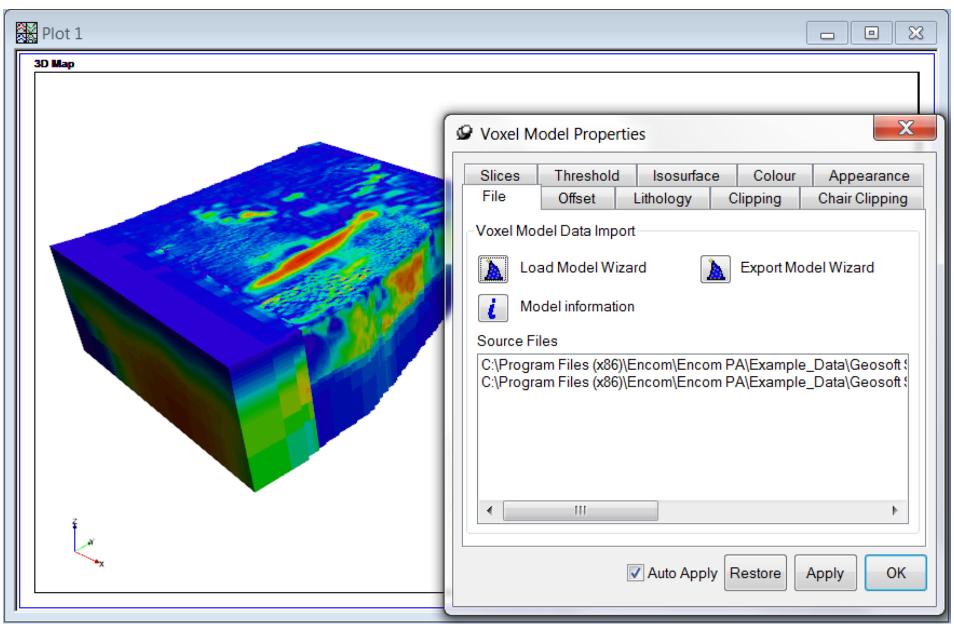
The style of file specification varies depending on the format. For example, the UBC format requires specification of two files (a mesh and a properties data file). The CEMI format and the various mine simulation package options (e.g. Datamine) require only a single location and property definition file.

Note that only a single model can be specified at one time for each voxel model branch in the workspace tree. This is listed in the **File** tab after being specified. If the **Auto-Apply** option (at the base of the dialog) is enabled, upon reading the model file, Encom Discover PA automatically displays the model.



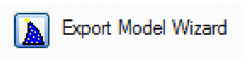
Use the Voxel Model wizard to select the model type and relevant model files

Additional options on the **File** tab include **Export** and **Information** relating to a voxel model. Refer to *Exporting a Voxel Model* and *Voxel Model Information* for additional details.

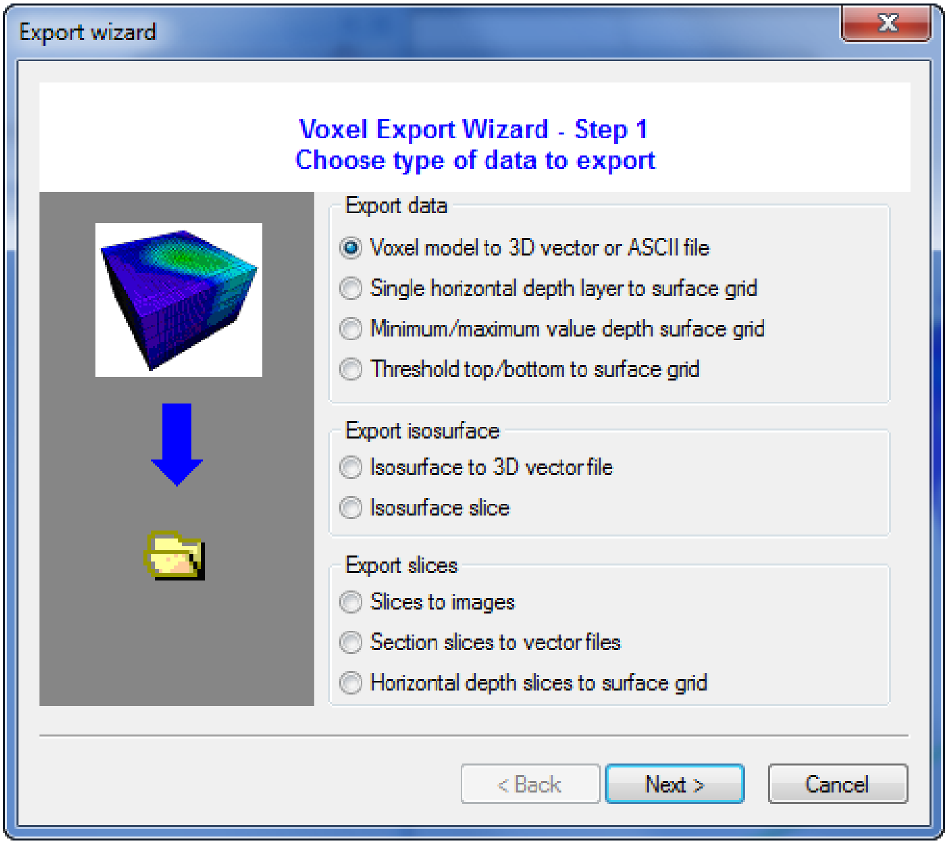


Initial display of the Voxel Model with internal annotation

Exporting a Voxel Model



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

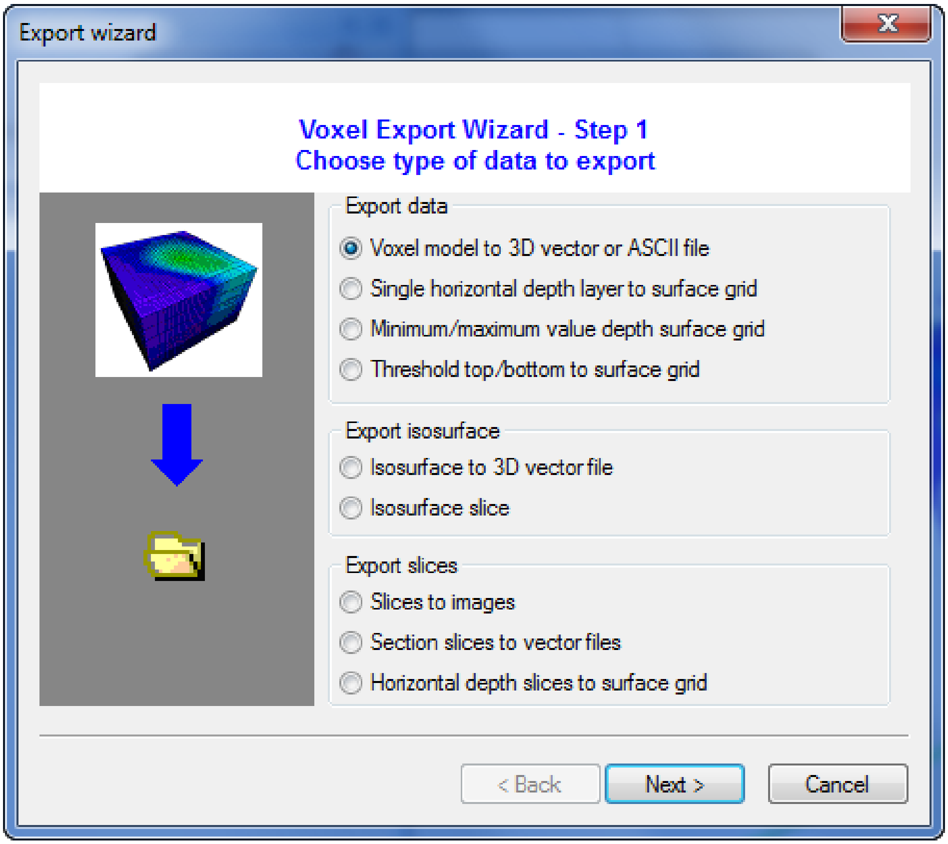
The export options are:

- **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 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.

- **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.

- 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.

- **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.

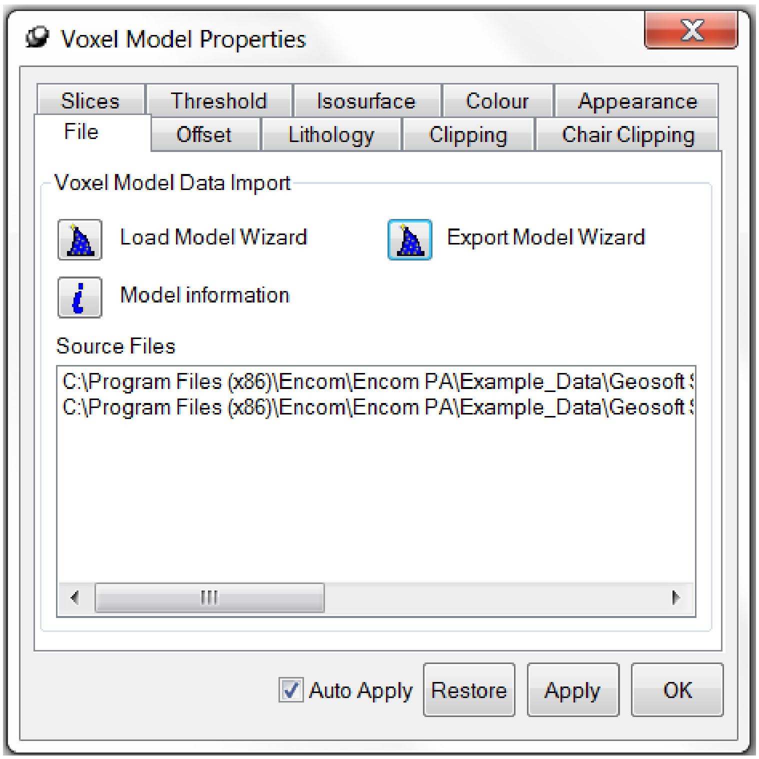
Voxel Model Information

The **Model Information** button in the Data tab opens a text report which can be saved for the model geometery and properties. As well as checking to voxel mesh/cell size, this is also a useful tool for seeing the lithology layers included in the model.

Voxel Model Properties

The display of volume elements (voxels) in the Voxel Model is controlled by the properties of the Voxel Model branch in the 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.

The property dialog for a 3D display is shown below:



Voxel Model Properties dialog

The Properties dialog uses different tabs to control various aspects of the voxel display. These are:

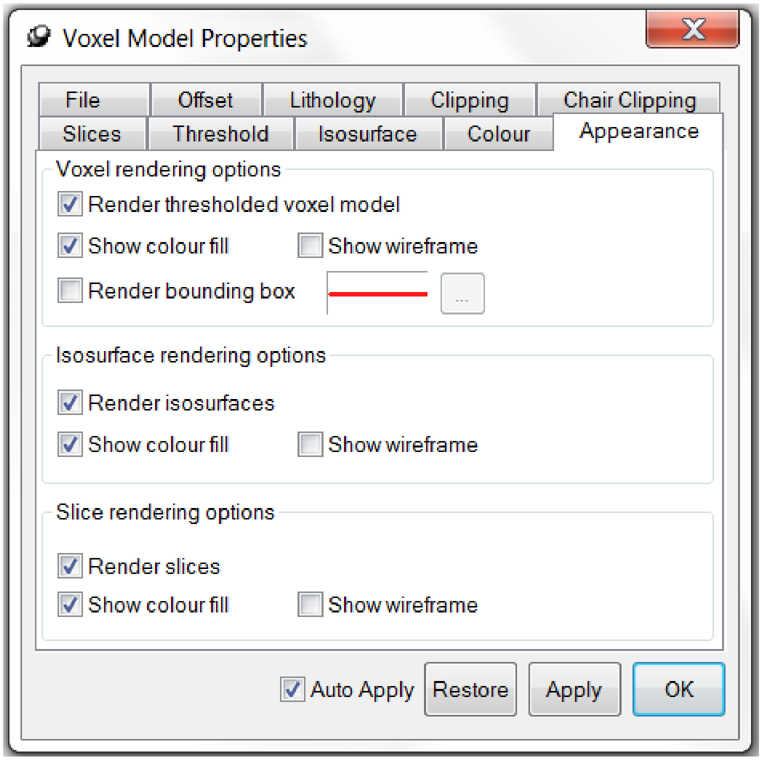
- **File Tab** – Used for access to the supported format models. Refer to *Importing a Voxel Model*.
- *Appearance Tab* – Used to enable and disable thresholding, colour fill, isosurfaces, slices and wireframing.
- *Colour Tab* – Controls the colour look-up table and the colour transform
- *Lithology Tab* – Specify a lithology table for colour mapping of attribute ranges of a voxel model.
- *Clipping Tab* – Control the interactive slicing of a model to view the internal distribution of the model's attributes
- *Chair Clipping Tab* – Control the primary axis slicing to reveal internal attribute distribution in 3 dimensions
- *Threshold Tab* – Specify an upper and lower range of attribute vales that control which voxels are displayed
- *Isosurface Tab* – Nominate a data value that is used to create a 3D surface through the voxel model
- *Offset Tab* – This tab allows the entire model or just the upper surface to be offset to account for elevation and topography
- *Slices Tab* – Controls the display, position and transparency of single or multiple slices.
- *Voxel Model Display Properties* – Properties of voxel models in curve profile and 2D map displays

Appearance Tab

The Appearance tab of the Voxel Model provides the following options:

- Enable thresholding of models.
- Show the look-up table for colour fill.
- Enable colour filling.
- Enable wireframing on the model.
- Enable isosurface creation with or without colour fill and wireframing.

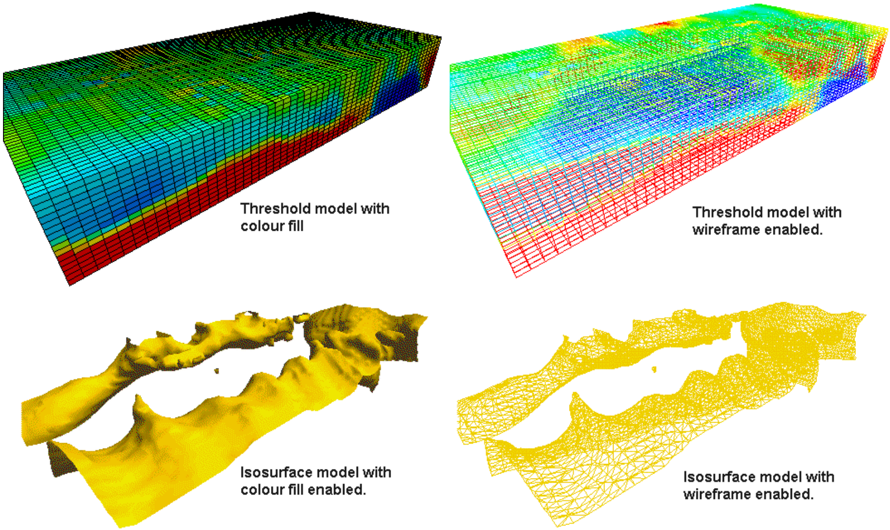
- Display slices at regular distances on any of the three axis directions.



Appearance tab of Voxel Model properties dialog

These controls are useful to quickly change a presentation from one style to another when you wish to retain various setting of the thresholding, isosurface or slice controls. For example, if you spend time setting the values and display of a threshold display (and wish to retain these), you can then switch to an isosurface display (by disabling the threshold display on the Appearance tab) and later return to the same threshold presentation (enable on the Appearance tab) without having to reset the control parameters.

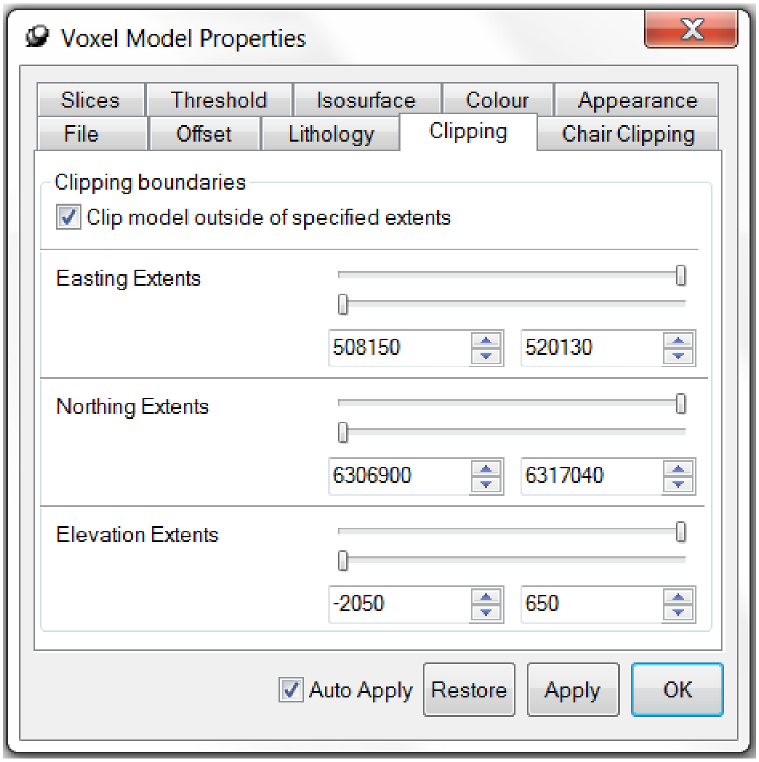
Examples of the above options are shown below:



The various Voxel Model appearance settings

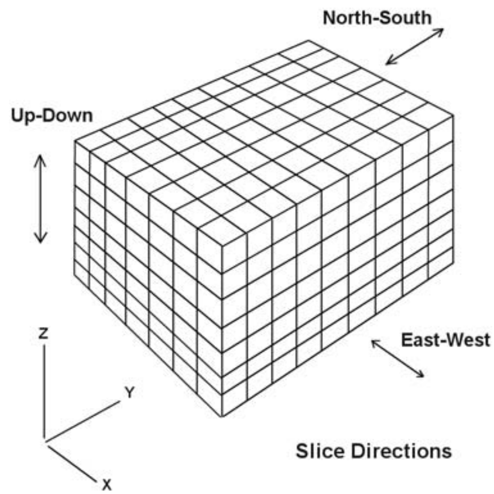
Clipping Tab

The Voxel Model display allows you to interactively slice 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.



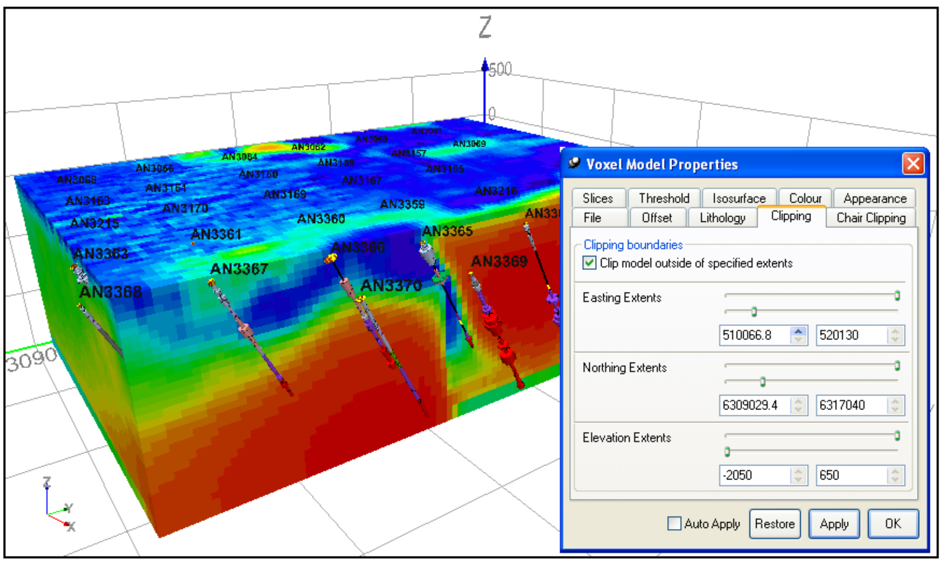
Clipping tab of the Voxel Model Properties dialog

Enable the clipping process by selecting the **Clip model** option in the **Clipping** tab. For each of the **Easting**, **Northing** and **Elevation** directions, two slider bars are provided. So long as the **Auto-Apply** option is checked on, changes to any of the slider bars removes or makes slices 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.



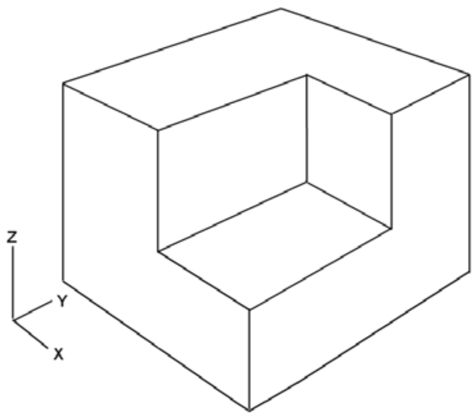
Clipping layers from the model reveals internal structure and intersecting drillholes.

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 Tab

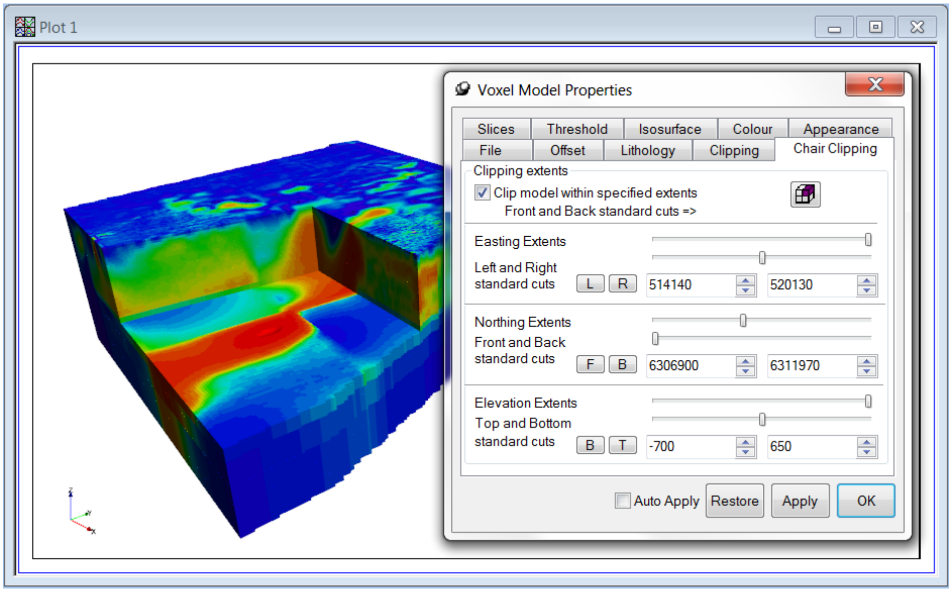
Note This option is only available in 3D Displays.

The **Chair Clipping** tab provided in Voxel Model properties dialog is an extension of the logic used for Clipping (see *Clipping Tab*). 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.



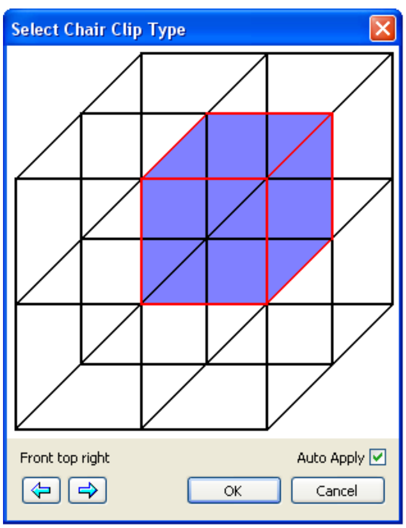
The Chair Clipping showing a portion clipped in each coordinate direction

The Chair View 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. This dialog is shown below with a resultant Chair View.



Chair Clipping of the model to reveal internal structure.

When pressed, the Front and Back standard cuts button at the top of the Chair Clipping tab dialog will display the Select Chair Clip Type dialog as illustrated below. This dialog allows the selection of preset standard cuts in the three coordinate directions based upon a guiding plan of a voxel model. Use the arrow buttons to iterate through the standard cuts.



Chair Clipping can be used in combination with model clipping to further add interactive viewing control (see *Clipping Tab*).

Colour Tab

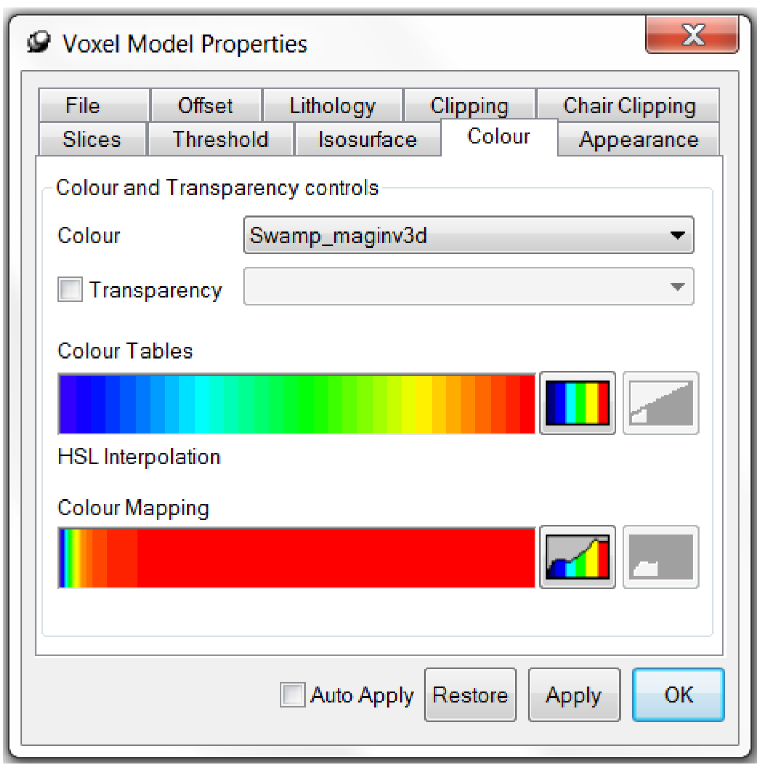
The **Colour** tab allows the colour and transparency of a voxel model to be modified.

Select the data field to **Colour** modulate from the top pull-down list; this can either be a numeric field (such as a UBC Property file Maginv3D.sus) or the **Lithology mode** when handling discretised voxel models (i.e. voxel models comprising a number of unique lithological codes).

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:

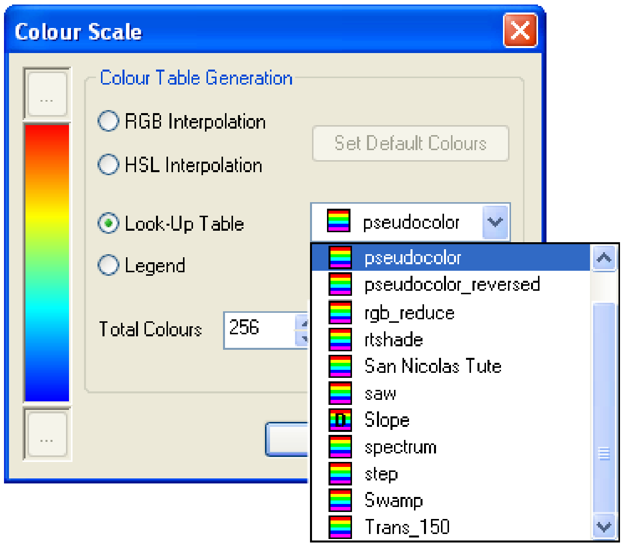
- A choice of *Colour Table* or *Transparency Table*
- Transformation of the selected table with *Colour Mapping* or *Transparency Mapping*



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.

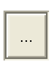


Colour Scale dialog

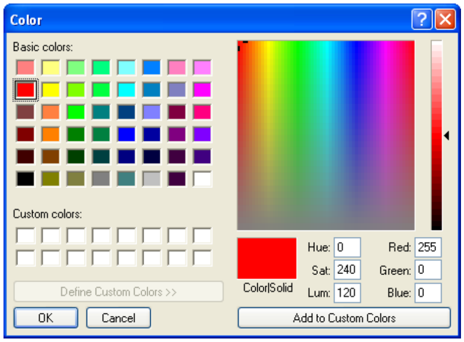
Four methods of colour scale definition are available:

- **RGB Interpolation** interpolates between two colours in Red:Green:Blue colour space.

- **HSL Interpolation** interpolates between two colours in Hue:Saturation:Luminosity colour space.



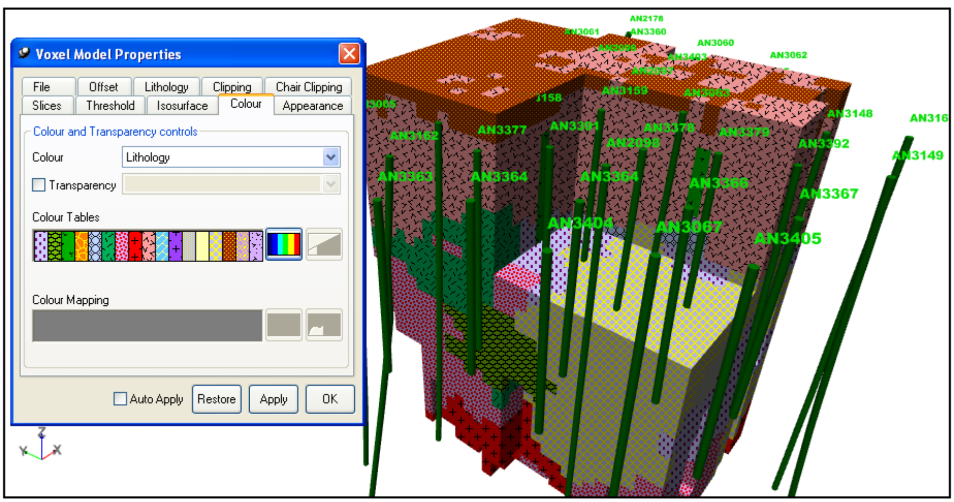
Set the first and last colours of the colour scale by selecting the **Colour Browse** buttons at the bottom or top of the colour bar. When clicked, a standard Windows colour selection dialog is displayed allowing colour specification. These can be reset by clicking the **Set Default Colours** button.



Windows colour selection palette

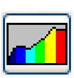
- **Look Up Tables** - the standard look-up table formats are supported and are installed as part of your Discover PA 3D installation. These can be created or edited using the *Colour Table Editor*.

- A **Custom Legend** created using the Legend Editor in Encom Discover PA or the Legend Editor in the Drillhole module of Encom Discover. This is of particular use for colouring Discretised voxel models created using, for example, a series of lithology/rock codes.

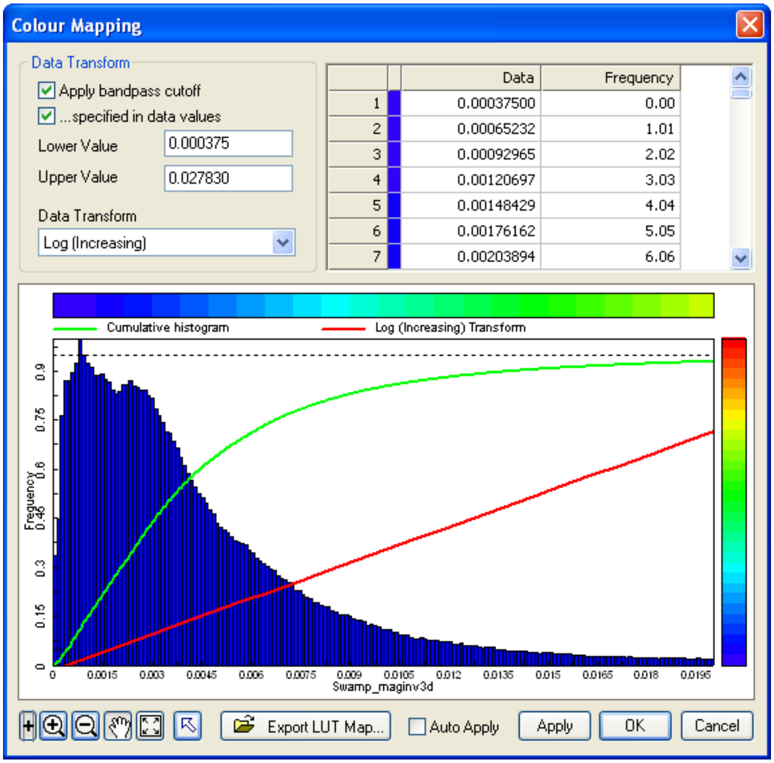


Thresholded lithology voxels coloured by applying a custom legend table

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.



The Colour Mapping dialog configured for an increasing logarithmic data transform, with bandpass cutoff limits set in data values

A preview screen of the data distribution is presented, with the Cumulative histogram displayed as a green line, and the applied data transform a red line. 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 cutoff), choosing this option allows the user 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.

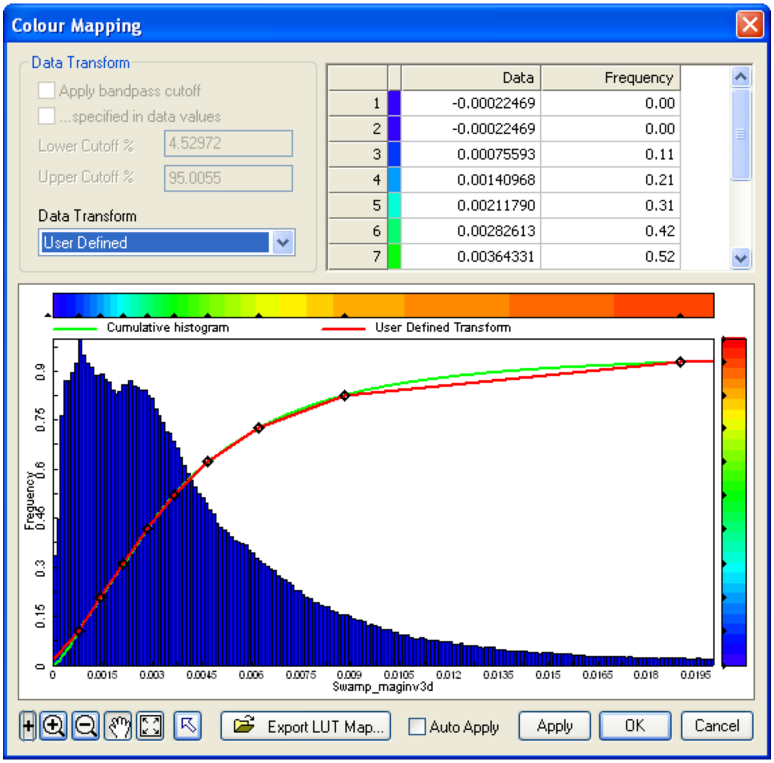
  - Clicking on a control point with the CTRL key held down and moving it moves the selected control point only. Points cannot be moved horizontally beyond the two adjacent control points. Directly editing a control point in the spreadsheet applies the same rules. This mode allows control over the exact 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.

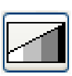


Creation of a User Defined transform

For each of these options (except User Defined), a **Bandpass cutoff** can be applied either as percentage or data values. Bandpass cutoff limits are displayed as dashed lines in the preview screen. 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.

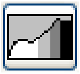
Note The **Colour Mapping** dialog is unavailable for voxel models in **Lithology mode**.

Transparency Table



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.

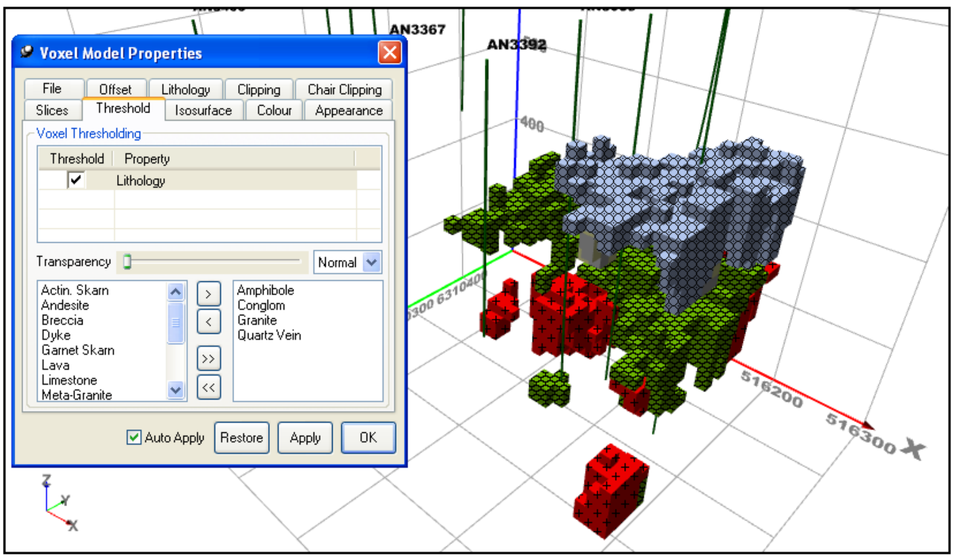
Transparency Mapping



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.

Lithology Tab

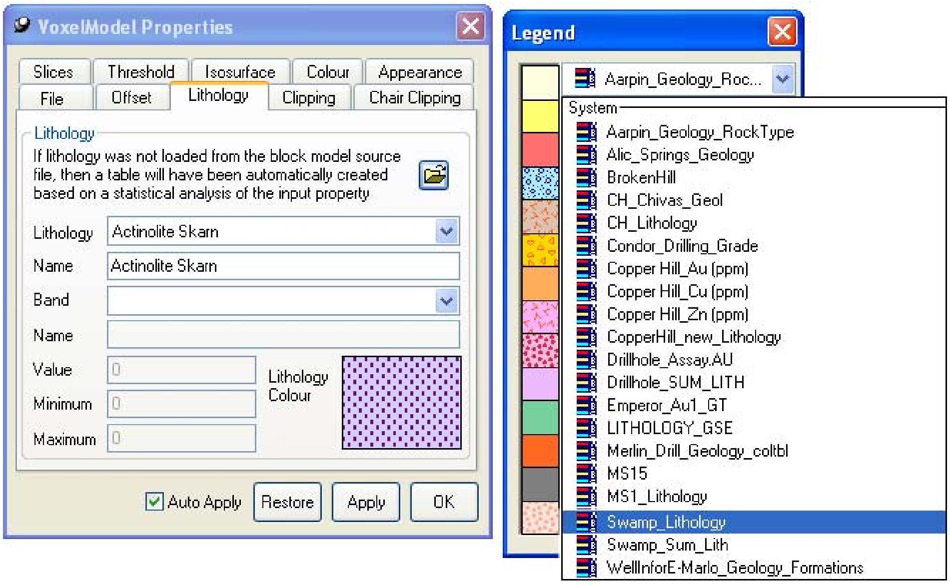
Discretised voxel models (where a series of unique attributes (e.g. rock or alteration codes) have been used to create the block model) can be coloured using an appropriate Colour Legend via the Colour Scale Selection dialog under the **Colour** tab. It is also possible to control and display individual attributed components via Lithology mode controls on either the *Isosurface Tab* or *Threshold Tab*.



Displaying multiple lithology layers from a discretised voxel model (and drillholes) 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 you have selected the Colour Legend table, press the **Apply** button to ensure the changes are applied to the threshold/isosurface tabs.



Colour Legend selection dialog accessed via the Lithology tab.

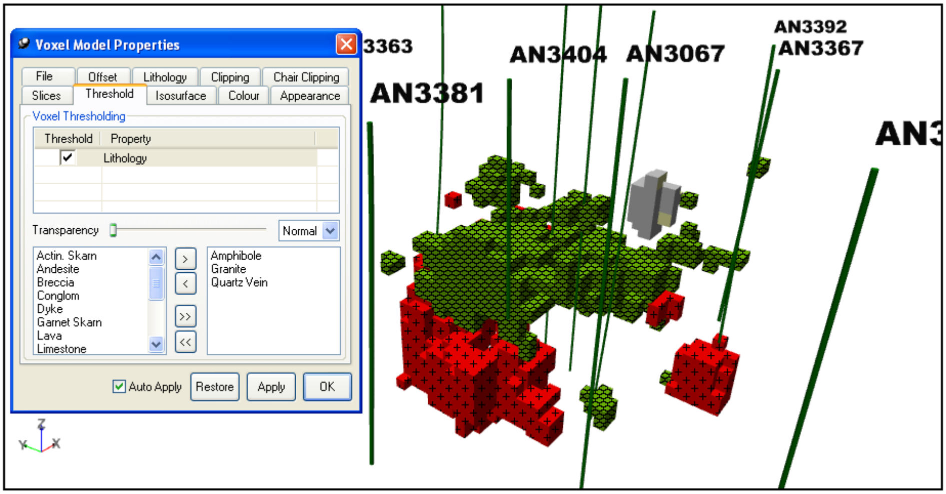
Threshold Tab

Thresholding of voxel models allows volume elements (voxels) to be displayed only if their specified attribute lies within the chosen data range. The tresholding feature operates in two modes:

- *Lithology Mode* –displays voxels with only predetermined data ranges/ unique attributes i.e. Discretised voxel models.
- *Interactive Mode* – Use slider bars to control the displayed voxels by dynamically altering the upper and lower data range.

Lithology Mode

The Lithology mode allows predetermined data ranges to be specified to named bands. These bands may represent rock types or attribute ranges relating to conductivity, magnetic susceptibility etc. When selected for display in the **Threshold** tab of the properties dialog, you can select which nominated lithology layers are to be displayed. All other voxels of the model remain transparent. To use this method of display, initially enable the checkbox labeled **Threshold** and then select **Lithology** from the pull-down list of items.



Lithology control of thresholding and viewing only those items selected

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.

From the Lithology table listing provided, choose the lithology types to be displayed (and place in the right hand list area). If the **Auto-Apply** option is enabled, the items with the specified attribute ranges should immediately display.

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.

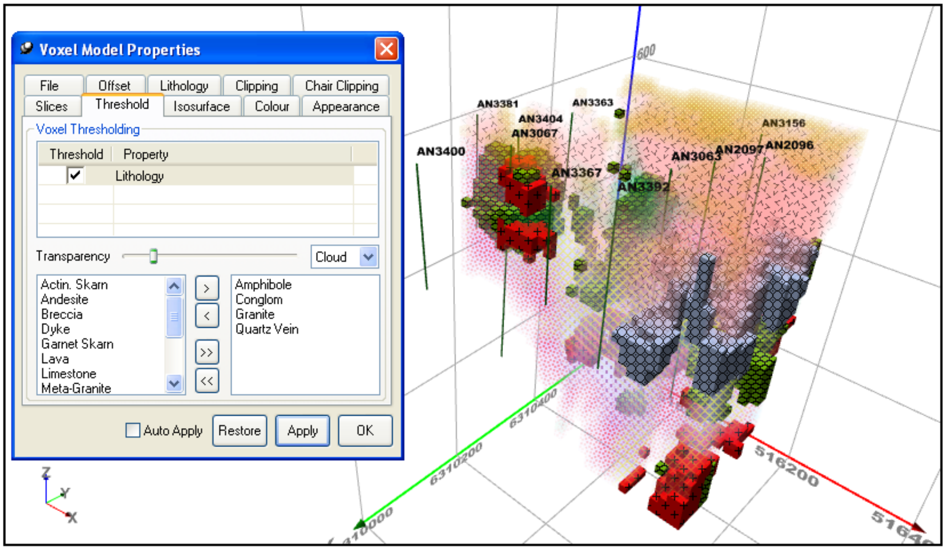
Two **Opacity** modes are available:

- **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.



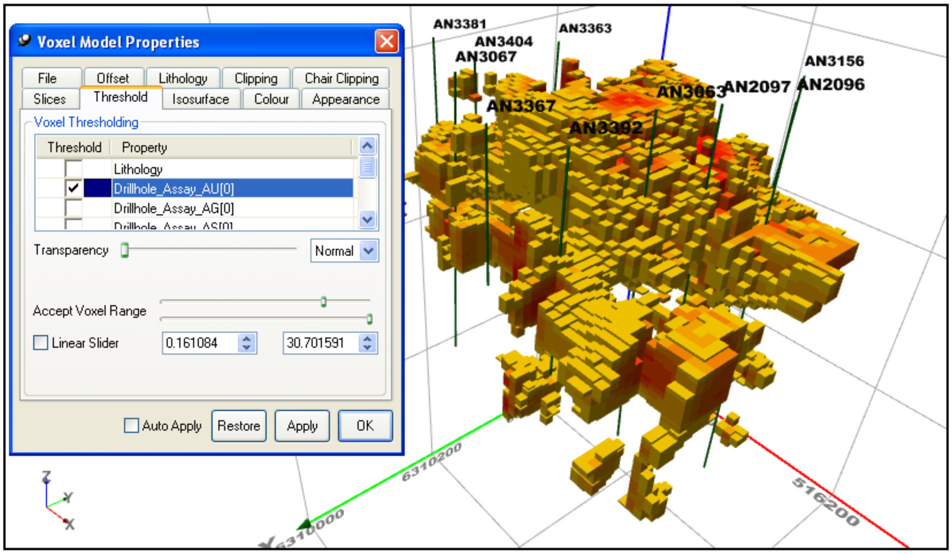
Lithology control of voxel thresholding using cloud opacity

Interactive Mode

The interactive mode of thresholding allows you to dynamically specify an upper and lower range of data attributes and their corresponding voxels for display.

Note If you change from the **Lithology** mode to **Interactive** mode for the thresholding display, you may need to turn off the previously used mode by deselecting the **Threshold by** checkbox.

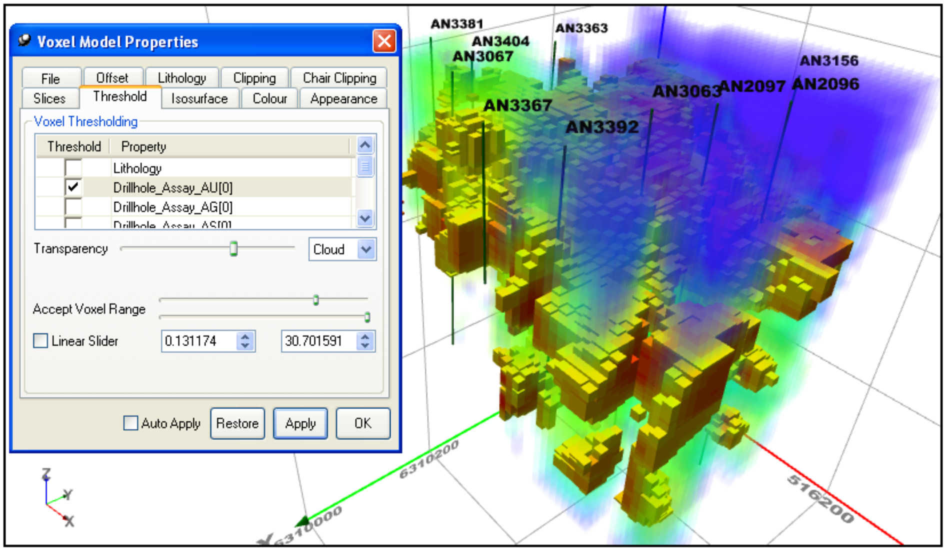
To use this form of thresholding, initially enable the **Threshold by** checkbox and choose the attribute model name available in the drop-down list. When selected, two slider bars and two entry fields become available. With the **Auto-Apply** option on, if you move either of the upper range or lower 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.



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 opaque, whilst the Rejected Voxel Range is displayed with the level of transparency set by the adjacent slider bar. In this mode you can ONLY adjust the rejected transparency – the accepted voxels are always opaque. This is why as soon as you reduce the rejected transparency from 100% (i.e. move the slider it to the left) the accepted transparency gets reset and locked to 0 (ie opaque). An example of the Normal mode with Cloud mode is shown below:



Normal mode thresholding plus Cloud mode to indicate the background outline of the voxel model

Isosurface Tab

Note This option is only available in 3D Displays.

Similar to threshold settings, the Voxel Model can be made to display a surface of a single attribute data value. The created isosurface 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:

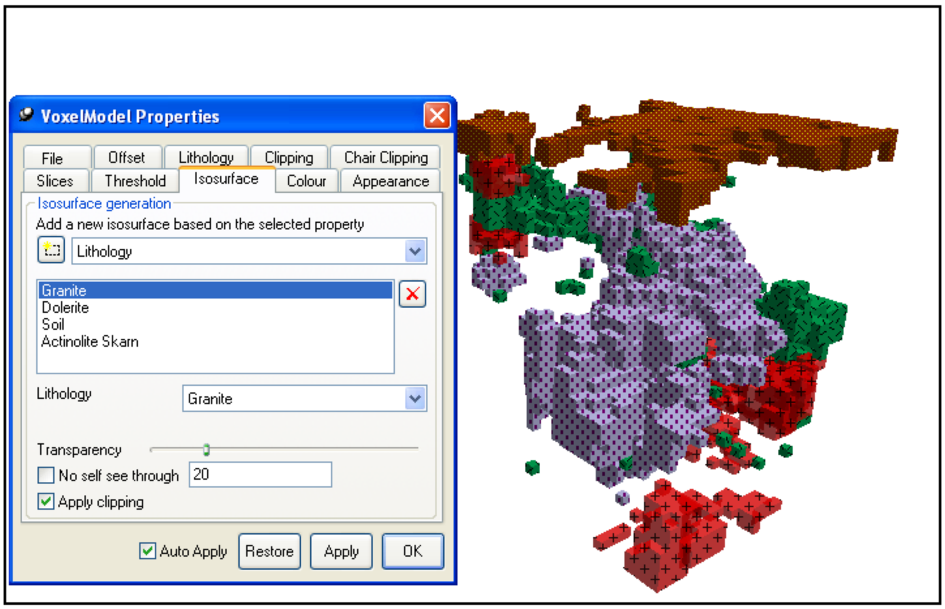
- *Lithology Mode* – The isosurface is mapped by the data ranges of the lithological bands. Note that the created isosurface uses only the lower of the data range values.
- *Interactive Mode* – This mode allows a particular attribute value to be used to create the isosurface.

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.

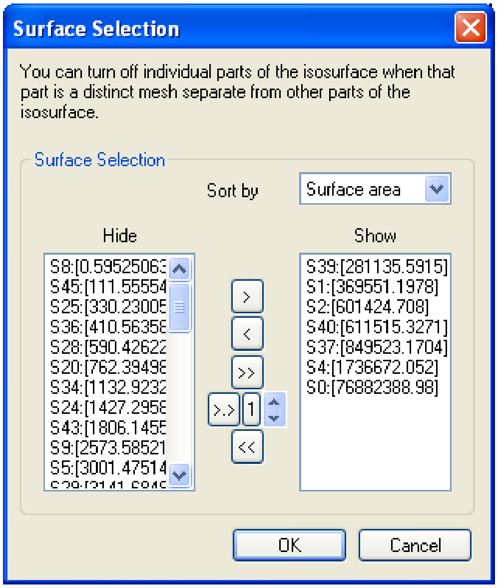
Note Multiple lithology table entries can be added (as pictured beloww); the transparency of individual layers can be controlled via the **Transparency** slider bar.



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**.

To remove any satellite isosurfaces or smaller, insignificant isosurfaces click on the **Show** button and chose from the Show list of isosurfaces that are sorted by either Number, Surface Area, Easting, Northing, or RL.



The isosurface Surface Selection dialog, allowing display control of individual omponents of an isosurface

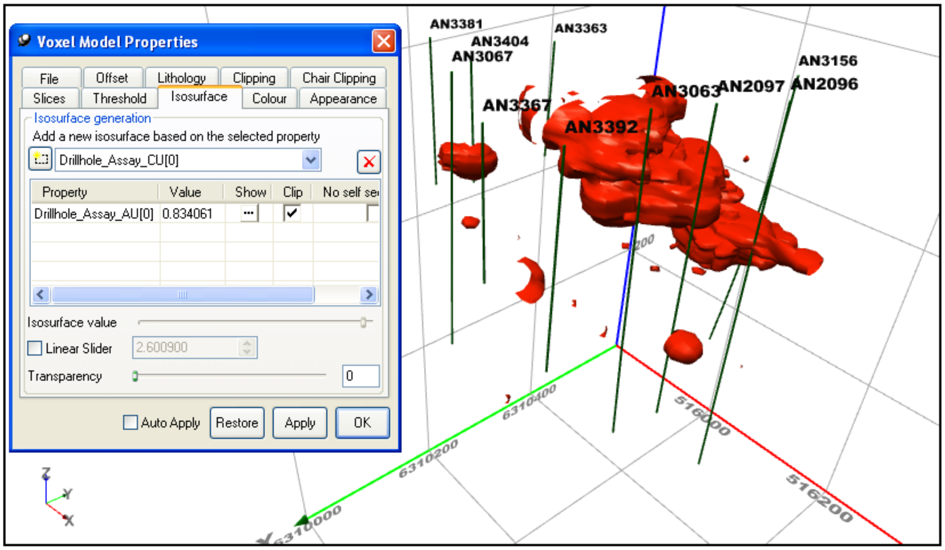
It is also possible to combine both Lithology and Interactive modes by adding layers derived from either the Lithology table or specific values of the model.

Note Displaying lithologies (e.g. rock types) as isosurfaces is identical to displaying them as thresholds (see *Threshold Tab*), 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 property **Colour** tab.

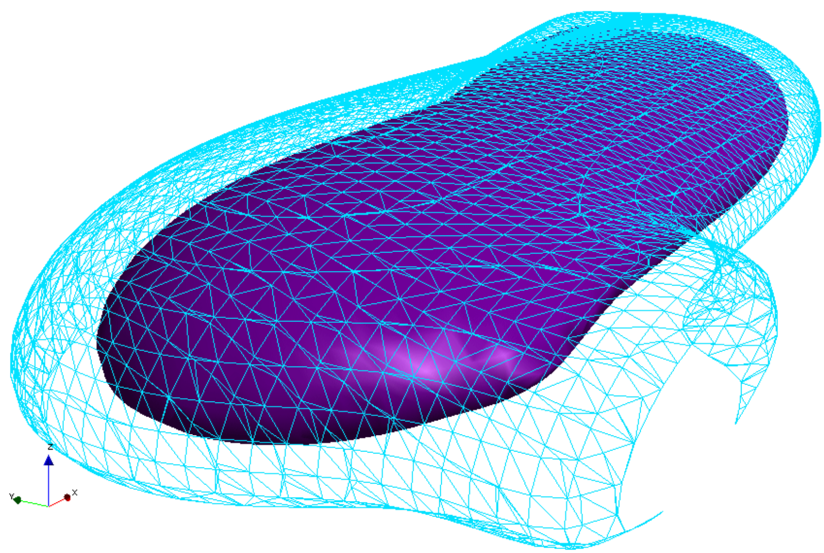
To create an **Isosurface**, select the required voxel model data (or properties) from the pull-down list. Click the **Add** button (shown left) and a default attribute value is added to the list. Additional values can be added if more than one isosurface is to be created. Once a layer is available, 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.



Displaying multiple Property isosurfaces in Interactive mode

Using Wireframes

Occasionally it is useful to use wireframes instead of isosurfaces alone. Wireframing provides a means of seeing through one surface at a second (or other objects) that would otherwise be obscured. An example is shown below:



Two isosurfaces used with the outer one represented only as wireframed, the inner solid

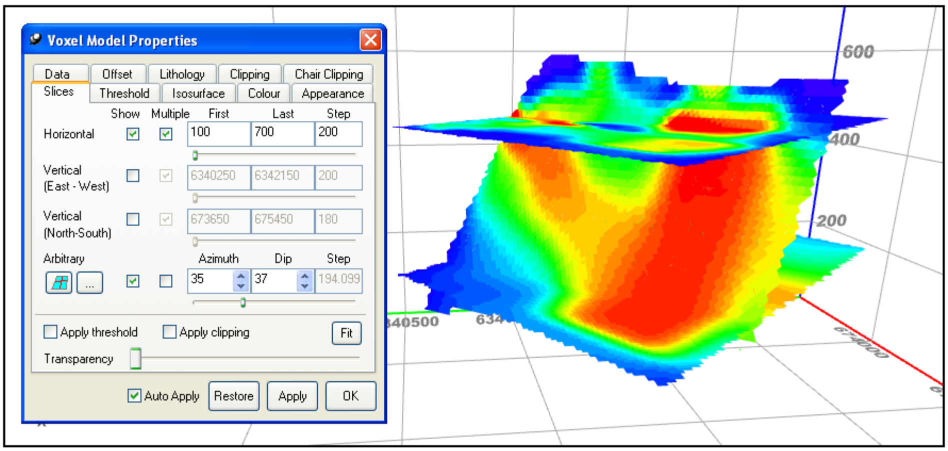
The wireframes also provide information about the control points used in the triangulation process to create the isosurfaces.

To apply wireframes to Isosurfaces, enable the **Show wireframe** option under the **Isosurface rendering options** of the **Appearance** tab.

Slices Tab

Note This option is only available in its entirety for 3D Displays.

Displaying slices of the voxel model permits individual slices of voxel cells to be displayed. The slice locations, regularity and direction are controlled from the **Slices** tab of the Voxel Model Properties dialog.



Specifying the slices to be drawn, their direction and location.

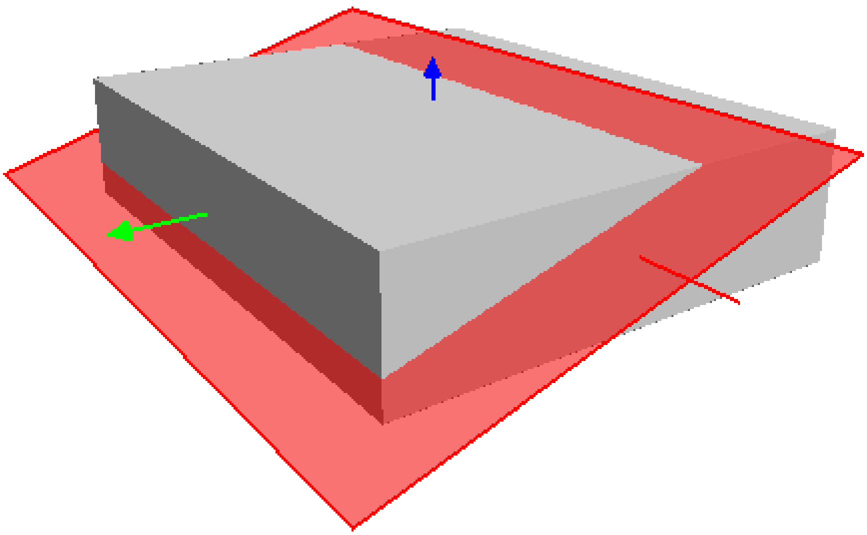
The slices can be in the east-west, north-south or horizontal orientations. Specify a start, end and interval **Step** between each slice (in metres). The Step is incremented by interpolating the display colours of the intersected voxel cells.

The Step setting determines the number of slices created in any direction. Note that if you set the Step value to be large (bigger than the dimensions of the model), only a single slice in the required direction is drawn. What this means in effect, is that with the **Position slider** located beneath each direction setting, you can move the slice interactively forward and back through your model. This is a powerful method of investigating the property distribution within a block model.

You can also specify a Transparency using the slider bar to view other 3D objects displayed in conjunction with the voxel model

To slice on any user-defined plane:

- On the **Slices** tab, select the **Arbitrary** option. This provides two methods for controlling the slice location/orientation:
  - **Cursor Plane**: Enable and orientate the cursor plane to the desired slice orientation (see Cursor Plane). Then click Synchronize to capture the cursor plane parameters. Click Apply to display the slice.
  - **Manually**: 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.



Using the Arbitrary Slice Configuration dialog to set the location and orientation of a slice.

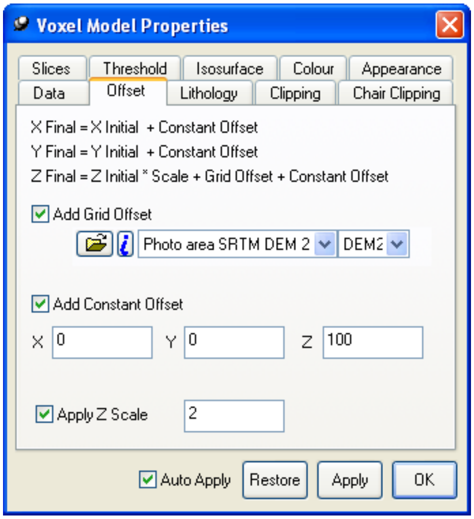
Offset Tab

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.

The **Offset** tab provides two options to control the offset of the voxel model.

- **Add DTM offset** – add Z offset values based on a specified grid, or

- **Add Constant 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 that can be used to exaggerate the vertical size of the voxel model.

Note that the DTM offset from a grid or constant offset is only applied to the voxel mode AFTER the vertical (Z) scale (if activated) is applied to a Voxel Model i.e.

Z Final = Z initial scale + DTM Offset + Constant Offset

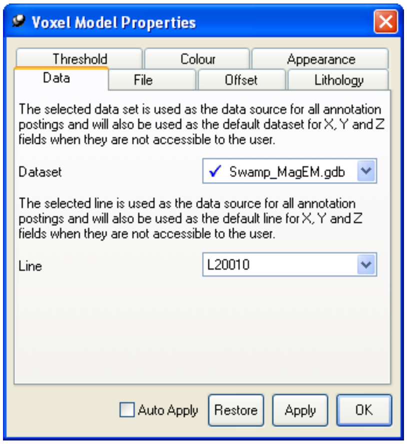
Therefore there may not be an obvious change in the final Z offset. To see a greater enhancement of the voxel model with respect to the DTM offset it is advised that you use the **Scale** tab options in the **3D Map Properties** dialog to apply a Z scaling and then use the DTM offset in the Voxel Model Properties **Offset** tab dialog.

Voxel Model Display Properties

- *Voxel Model Properties in a Curve Profile Display*
- *Voxel Model Properties in a 2D Map Display*

Voxel Model Properties in a Curve Profile Display

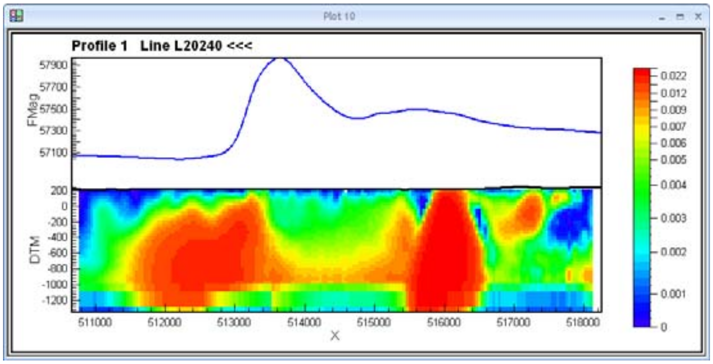
In a curve profile display a limited number of properties are available, although an additional tab dialog, called **Data**, is available, which is consistent with other curve profile data objects to allow the specification of associated dataset and line.



Voxel Model Properties available for a curve profile display.

In all curve profile displays only a voxel model slice can be displayed and the position of the slice is determined by the projected distance parameter of the line in the associated dataset specified in the Data tab dialog. The slice is limited to a vertical plane, centred on the line and parallel to it.

The slice can have a threshold applied to it using the Threshold tab dialog and can also be clipped in a X, Y or Z orientation using the Clipping tab dialog.



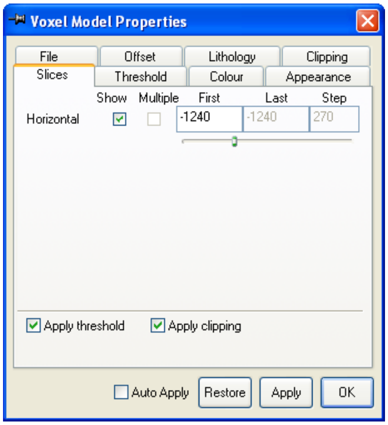
Example off a voxel model in a curve profile display.

Voxel Model Properties in a 2D Map Display

In a 2D Map display a limited number of properties are available, which are suited to a two-dimensional display.

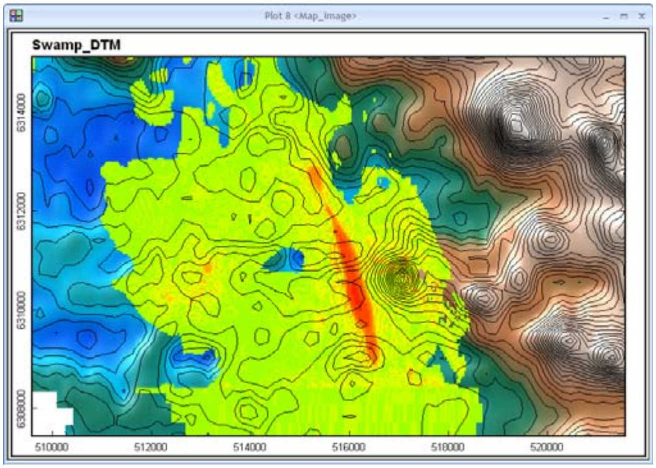
The offset page is retained in its entirety for compatibility with a 3D display.

You can display a single horizontal slice at a specified depth and voxel model clipping in the X, Y or Z plane can be applied.



Slices tab dialog for a voxel model shown in a 2D map display.

The Threshold option allows the first occurrence of any voxels in a model that satisfy the threshold criteria, looking from the top downwards.



2D map display showing a voxel model threshold overlying a grid image of terrain.

The below table shows a summary of which voxel model properties are accessible in the three available display windows.

Property Tab	3D Display	2D Map	Profile
File	✔	✔	✔
Data	✘	✘	✔
Appearance	✔	✔	✔
Colour	✔	✔	✔
Clipping	✔	✔	✘
Chair Clipping	✔	✘	✘
Threshold	✔	✔	✔
Isosurface	✔	✘	✘
Slices	✔	✔	✘
Offset	✔	✔	✔
Lithology	✔	✔	✔

Additional Voxel Display Features

Some additional features are provided for use with Voxel Models. These include:

- *Multiple Voxel Model Displays*
- *Integrating Voxel Models With Other Objects*
- *Voxel Model Legends*

Multiple Voxel Model Displays

In the Isosurface section (refer to Isosurfaces), multiple voxel displays showing the distribution of 2 or 3 voxel layers has been described. In this case, the multiple displays are derived from the same voxel model.

Using the Voxel Modeller within Encom Discover PA can also be used to display voxel displays from multiple model sources. This is useful particularly 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 Voxel Display it was shown how to insert a layer for a Voxel Model into a display window. 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:

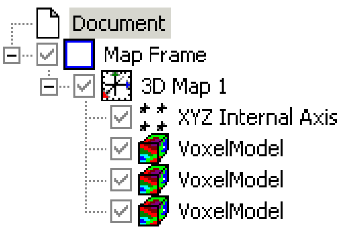
1. Create the Curve Profile, 2D Map, or 3D Map and add a Voxel Model object to this as described in *Creating a Voxel Model Display*.

2. Select the appropriate model as described in *Importing a Voxel Model*.

3. Repeat step 2 to add extra model levels.

Alternatively, once you have created the first Voxel Model Layer, you could select the **Clone Voxel Model** option instead of the **Add** option (highlight the 3D Map item and right mouse click to display the pop-up menu). This duplicates the same model that could then be reassigned to an alternative model if required.

The Workspace tree should appear similar to below:



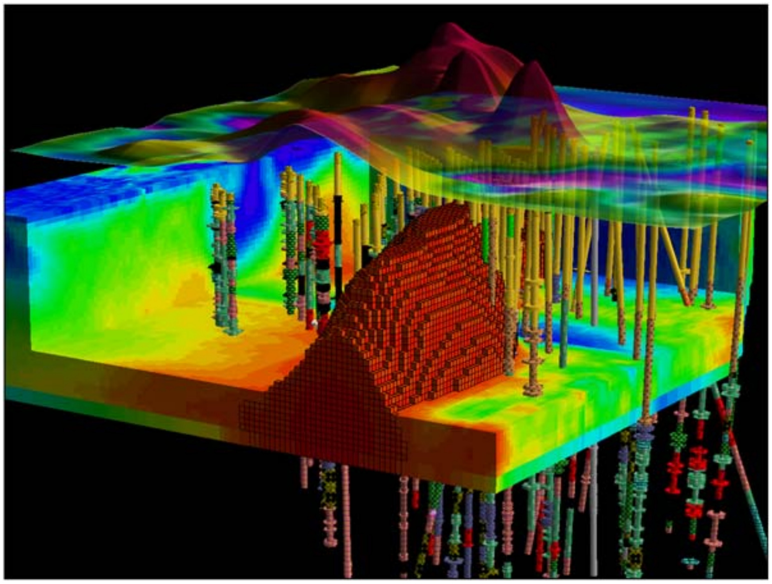
Integrating Voxel Models With Other Objects

Once the Voxel Modeller display is used for importing and displaying models as described, you can commence taking advantage of the ability of Encom Discover PA 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
- Flight path and survey locations
- Located bitmaps derived from scanned or captured imagery

- GIS layers derived from MapInfo Professional or ArcView
- Models derived from third-party packages such as ModelVision, EM Gui etc.

An example of the complexity of display with integration that is possible with the Voxel Modeller and Encom Discover PA is shown here:



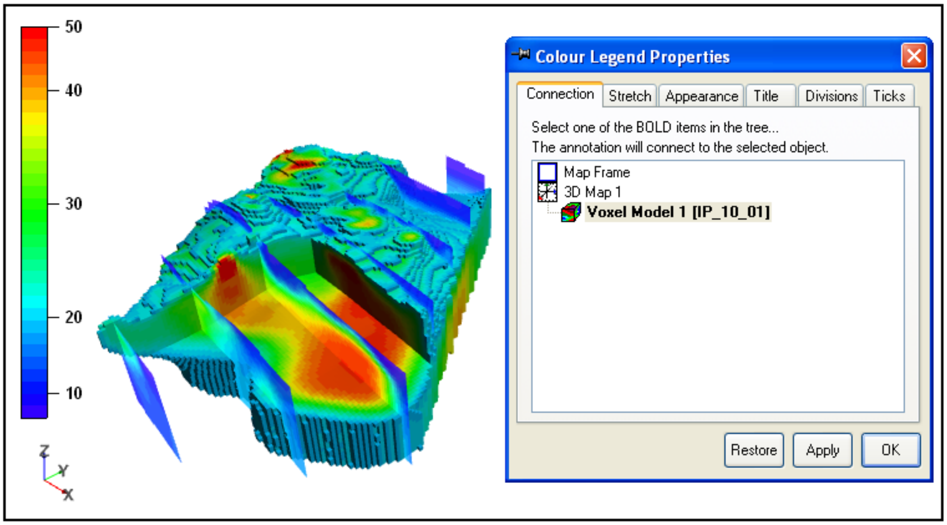
Display of drillholes, located bitmap, model and isosurface using the Voxel Model.

For information on importing and displaying any of these additional objects, refer to the relevant sections of this guide.

Voxel Model Legends

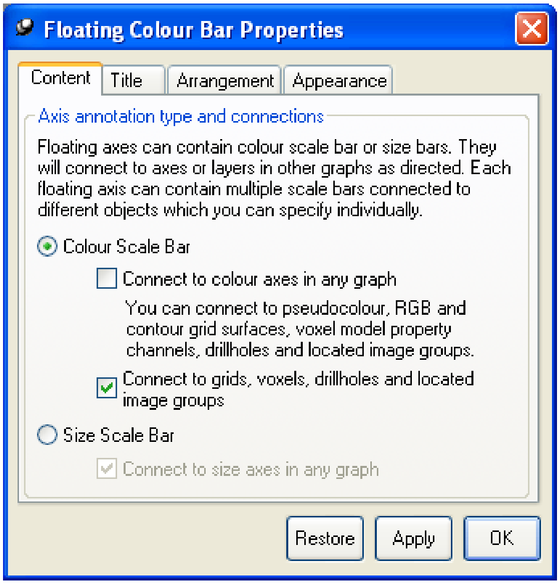
A *Floating Colour Bar* can be added to a display window and referenced to a voxel model, allowing easily visualisation of the relationship between colours and numeric data ranges.

This object can be positioned and sized anywhere within the 3D display. The axis can be connected to the data and so obtain its colour from the displayed model. An example of a Floating Colour Bar is shown below:



Example of Floating Colour Bar with colour bar derived from a Voxel Model display

Note from the Workspace tree that a Floating Colour Bar branch is associated with the vertical colour bar of the axis. The connection of the bar can display colours or sizes and connect to different axes or layers individually. This connection is specified from the Content tab.



Controlling the Content of the Floating Axis

Controls of the size, title, centreing etc of the colour bar are specified in the Appearance, Title and Arrangement tabs. Their operation is identical to axis controls described in *Axis Properties*.

Individual control for colour bars is controlled from the properties of the various Colour Axis branches of the Workspace tree beneath the Floating Colour Bar layer (see the tree above). Refer to *Floating Colour Bar Properties* for further information.

Voxel Display and Processing Tools

Apart from the Voxel Model display in Encom Discover PA some additional tools are provided to assist in creation and use of Voxel Models. These are:

- **Array Builder** – This utility is useful for datasets that may require arrays to be created from single data channel databases. Refer to the *Build Array Field* for details of its use.
- **Array Data 3D Gridding** – Voxel Models (in UBC format) can be created using this utility from databases that have survey data lines with arrayed data channels of depth and property (eg conductivity, resistivity etc). Examples of this are inverted databases from the processing software EM Flow. Refer to *Array Data 3D Gridding* for additional information.
- **3D Gridding Toolkit** – This tool contains a number of gridding and manipulation tools. Various 3D Gridding options are available including discrete, continuous and distribution gridding. Other operations include the merging of 3D models, resampleing, convolution filtering and arithmetic combinations of models. Refer to *Voxel Toolkit* for additional information.
- **Depth Interpolator** – This module is used to provide interpolation of data to account for topography in situations where layered output of software (such as EM Flow) is affected by elevation. See *Depth Interpolation*.
- **Voxel Slice Generator** – A processing utility to create a series of located images from a voxel model for rapid display. Only the UBC model type can use this utility. See *UBC 3D Model to Geosoft GDB*.

22 TrackMaps

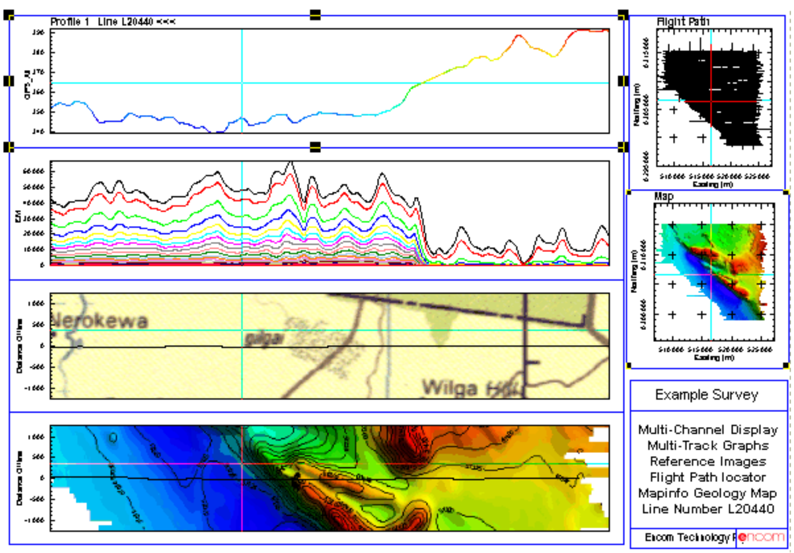
In this section:

- *About TrackMaps*
- *Creating a TrackMap*
- *Rotated Raster Images*

About TrackMaps

TrackMaps are profile tracks with a located image used as a backdrop. The TrackMap object is identical to a 2D map displayed within a track. The difference between a 2D map is that the TrackMap knows about which data line is being used and can be updated using the line navigation tools of Encom Discover PA. The scaling of a TrackMap is isotropic, that is, the same in the vertical and horizontal directions.

An example of a TrackMap display is shown below:



TrackMap example showing the flight path location over a scanned image and over a map of the magnetic response

In the above example a number of valuable interpretational features about the TrackMap module can be shown:

- The traverse of the selected profile (shown in the top two tracks) can be configured to show in the bottom two tracks

- TrackMap tracks (the bottom two in the above example), can have different background images

- The TrackMap track can be combined with other profiles or sections. The TrackMap track has all the display attributes of a normal profile. It can be sized vertically and scaled horizontally for hard copy purposes

- The background displayed in a TrackMap uses an RGB raster image file (such as ER Mapper or GeoTiff)

- TrackMap track display is isotropically displayed. That is, the vertical and horizontal scaling is identical such that no distortion to the image is introduced.

- The orientation of the traverses of a TrackMap are irrelevant. The TrackMap module automatically resamples the background image as required to position the traverse and display the image. If the image or traverse are rotated, the program extracts and rotates only the required swath of image necessary to display the selected track.

- Positioning a cursor in a track indicates the location of the point in adjacent tracks (including the TrackMap tracks) plus any linked maps

- Data zooming of any of the profiles causes all tracks to zoom and pan or smooth scroll

- Profiles used by TrackMap obey the same rules for flight line navigation as other profiles. For example, you can use the Line Iterator toolbar, graphical line selection from a flight path map or line selection from line lists.

Creating a TrackMap

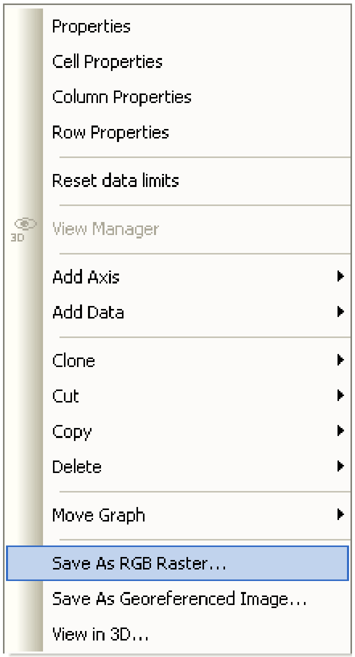
A TrackMap uses a raster file for its backdrop image. Raster images have Red, Green and Blue (RGB) format as used by ER Mapper and GeoTIFF. Raster images result from scanning maps (geology, topography, cadastral etc) or by using the capture option within Encom Discover PA or other applications.

- *Creating a Raster Image*

- *Adding a TrackMap Display*

Creating a Raster Image

Within Encom Discover PA, a map can be saved as a raster by displaying the pop-up menu (position the Pointer cursor in the map and select **Save as RGB Raster** menu option.



The pop-up menu for a 2D map to Save as RGB Raster image.

Any RGB image format can be used but ER Mapper multi-band format is ideally suited.

The option of capturing a raster of a displayed map is especially useful since you can configure the map and displayed image as required with contours, annotations and related data simply and quickly.

Adding a TrackMap Display

The TrackMap object is created by adding it to a Profile branch in the Workspace tree.

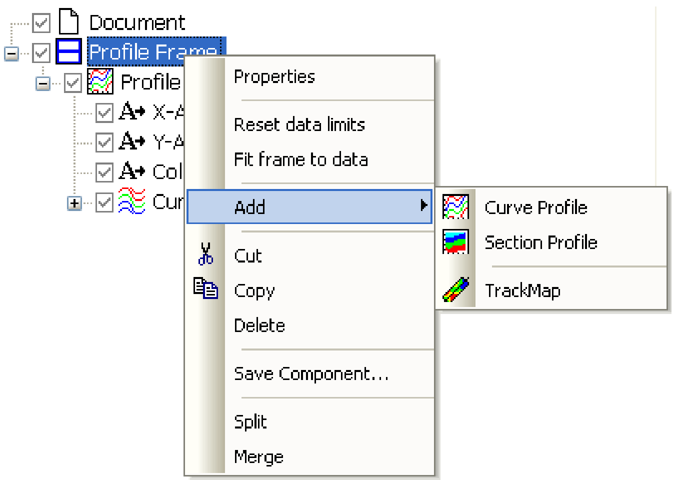
The steps required to add a TrackMap are:

Step 1

Access the database required for use with the TrackMap. Assume a scanned or raster image of the area is already available. If it is not, refer to *Creating a Raster Image*.

Step 2

Create a Profile display from the **File>New>TrackMap** or Document Wizard. Configure the profile with the required data line and with data as required.



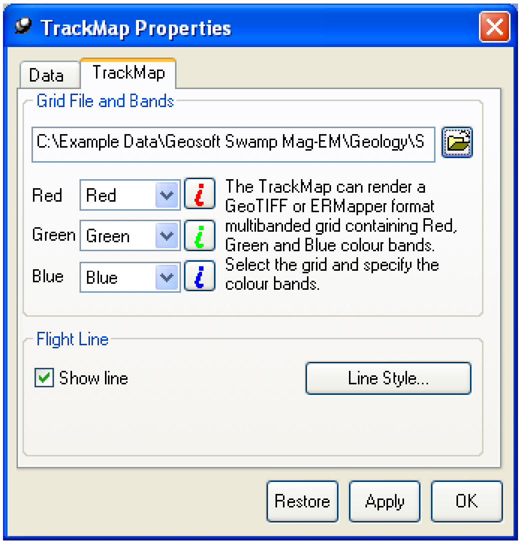
Step 3

Highlight the **Profile Frame** branch of the Workspace tree and click the right mouse button to display the pop-up menu. Select the **Add>TrackMap** option. A new branch is added to the Profile and a new track containing a TrackMap for the originally displayed traverse is shown.

Step 4

Display the properties of the TrackMap. The dialog has two tabs. Ensure the Data tab displays the appropriate database and data line. Select the TrackMap tab and using the **Open** button, navigate and access the required raster image file.

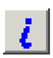
Note As described above, the raster file to be accessed should be in RGB format, as supported by ER Mapper or GeoTiff..



The TrackMap raster file specification dialog

The parameters of the dialog allow:

- **Grid File specification** – Open the RGB image file

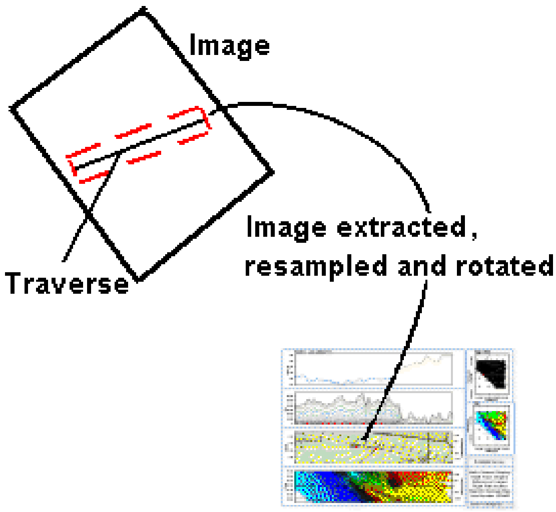


- **RGB Bands** – You can choose which bands to use for assigning the Red:Green:Blue colour. The **Information** buttons display grid coverage and data ranges of the selected image file.

- **Flight Line** – Within the TrackMap a flight line path is drawn by default. This can be disabled or the appearance of the trace can

Rotated Raster Images

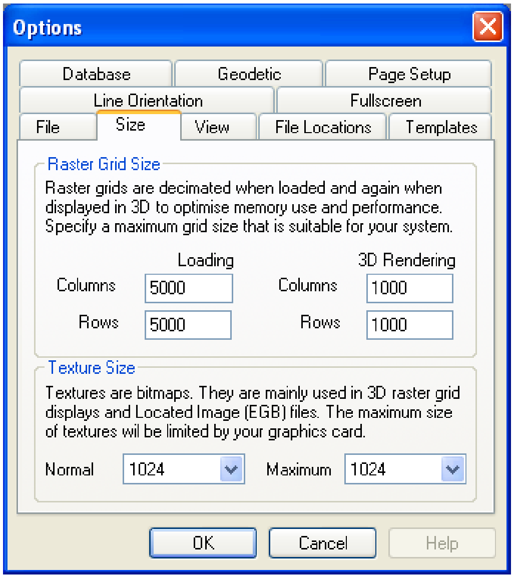
The orientation of the traverses of a TrackMap are irrelevant. The TrackMap module automatically re-samples the background image as required to position the traverse and display the image. If the image or traverse are rotated, the program extracts and rotates only the required swath of image necessary to display the selected track.



In some circumstances the raster image file (especially for high resolution, scanned images) can be very large. When this occurs, the resampled TrackMap result, especially for a rotated image, may be poor. This loss of quality results from the need to store the entire image in memory before the required portion can be extracted. On machines with low memory availability, the problem is worse due to image cell decimation.

To restore the image to a higher resolution (even on machines with low memory), you can alter the default resolution used by images.

This is done from the **Tools>Options** menu item of Encom Discover PA. The dialog displays the **General** tab. Set the **Grid Loading Columns** and **Rows** thresholds that cause grid loading decimation to occur to be larger than the default (2,000). Usually a value of about 8,000 x 8,000 is sufficient to retain the TrackMap resolution.



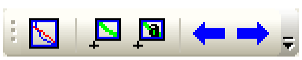
Dialog to modify the grid decimation threshold for loaded images.

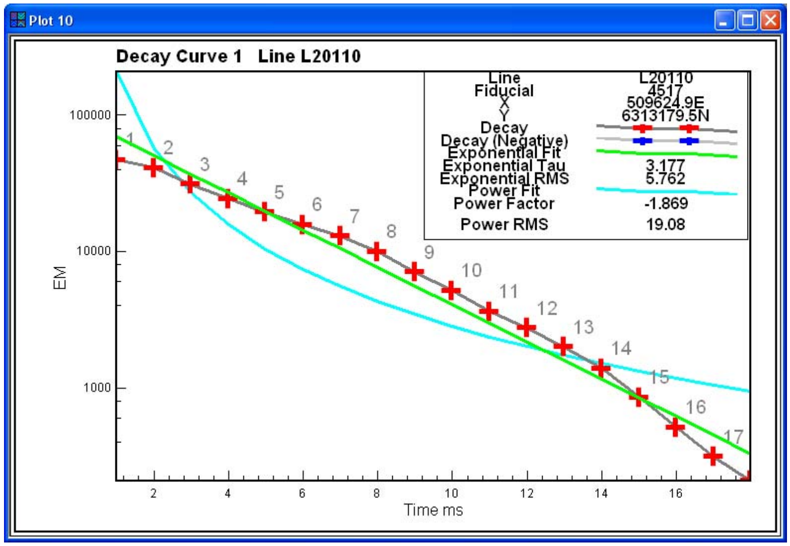
23 EM Decay Curves

In this section:

The Decay Tool

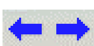
Encom Discover PA contains an interactive display tool for the presentation and analysis of multi-banded data called the Decay Tool. Typical use of the Decay Tool is with time domain EM geophysical data. The Decay Tool is controlled by the Decau Curve toolbar as shown below:





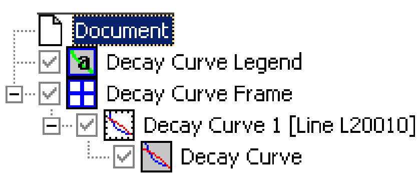
Decay Tool display in Encom Discover PA

Important The Decay Tool is directly related to a profile of data that contains multi-band time-related data fields. A decay display is only presented when a multi-banded, time related EM dataset is contained in a Profile within a document and a particular reading is selected by the **Select/Navigate** cursor . This selection informs Encom Discover PA which reading to use in the decay display. As the cursor is moved, the decay curve display window updates.



Movement of the cursor in the profile window updates the decay display in real-time. If a more controlled location of readings is required, the backward and forward arrows located in the **Decay** toolbar can be used

Characteristics of a decay display are:



- The decay curve display area, by default, displays a reading decay across the time channels with a time axis along the base. Also displayed are the Exponential or Power Fits (if enabled).
- Control of the Decay area is made from the Properties dialog accessed from the Decay Curve branch of the Workspace tree.

- The legend information has separate Properties and again is accessed from the Decay Legend of the Workspace tree.

Banded Data

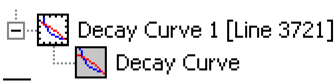
In Encom Discover PA, banded data and especially banded geophysical EM data can be displayed and analysed. For time domain EM, this form of data has channels in the various bands that are representative of time delay windows. Each channel represents a period of time in which received data is recorded from an EM sensing receiver. As Encom Discover PA has some operations that can make use of the time information for analysis operations (for example, the decay analysis tool), a file called ATEM.TXT is used to define the time channels.

The format of an ATEM.TXT file is provided in *Appendix B: Data File Specifications*.

Frame Properties

The Decay Curve Frame Properties dialog displays standard Axis, Options, Appearance, X Scale and Y Scale tabs (see *Profile Frame Properties*). In particular, if scaling of axes is to be altered (as is often the case with decay tool analysis), the X and Y Scale tabs can be used.

Decay Curve Properties

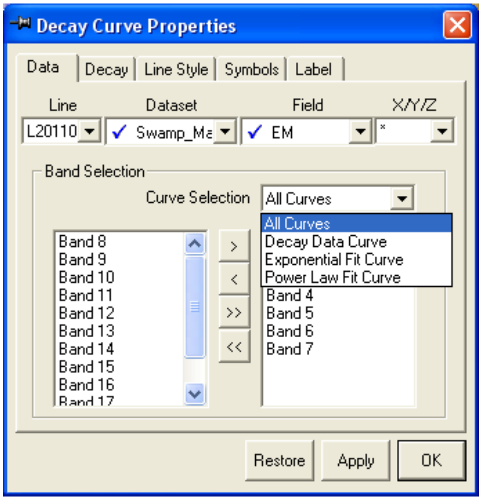


The properties of the main Decay Curve branch of the workspace tree contains the following tabs:

- *Data Tab*
- *Decay Tab*
- *Line Style Tab*
- *Symbols and Label Tab*

Data Tab

The Data tab is used to specify the Dataset, line, data field and measurement component (if it exists).

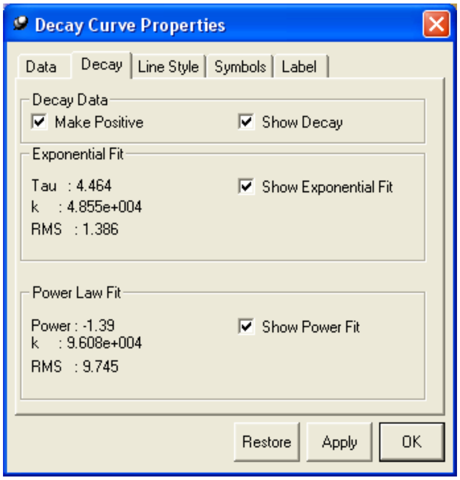


Data tab

When the appropriate data, data channels and line are specified (component is not mandatory), you can select the required data channels to be used in the display and any subsequent curve fitting.

Decay Tab

The Decay tab is used for control of decay data and controls the presentation of exponential or power law data fits.



Decay tab

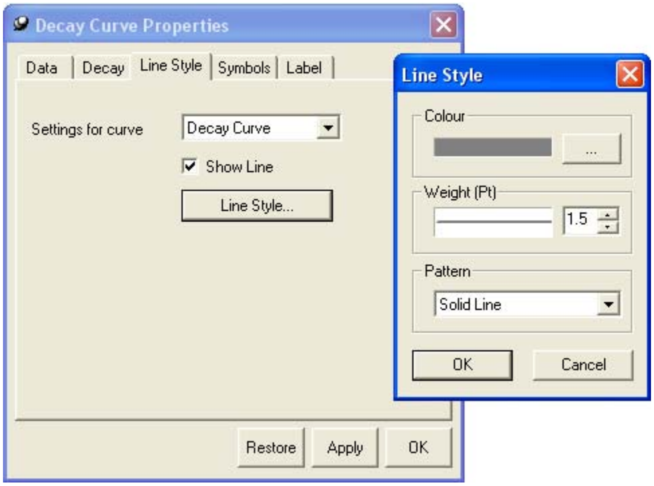
You can specify which data curves to display. Also displayed are the fit parameters derived from the decay fit over the channel range specified in the Data tab.

Line Style Tab

Different data curves can be displayed concurrently for each reading along a profile. The curves available include:

- Decay curve
- Positive decay
- Negative decay
- Exponential decay
- Power Law decay

Each of the curve types can be displayed or disabled. Use the pull-down curve list to select which trace is to be modified in this tab. Individually, you can also control the curve colour, thickness and line style (solid, dotted or dashed).



Controlling which decay curve is to be displayed and its style

The fit parameters are also shown in the Status Bar as the selection in a profile is changed.

Symbols and Label Tab

Two tabs to control the symbols and labels of curves in a decay curve plot are provided. The operation and functions of the controls in these tabs is identical to that of profiles (refer to Profile Display Symbols Properties and Labels Properties).

Curve Fitting

Fitting of decay curves occurs over a defined channel range specified in the Data tab. By default, Encom Discover PA initially selects all available channels, including channels with values that may be negative.

Fits based on the nominated channels can be either Exponential or Power Law.

Exponential Fits

The exponential response of electromagnetic data is given by:

$$Exponential\_response = ke^{-t/tau}$$

where k is a computed value determined by the intercept of the least squares fit of the selected channels. Parameter t is the relevant delay time for each time window used to compute the fit. Values derived for Tau (Time constant) and Y axis intercept (at time zero) are displayed. Additionally, the Root Mean Square (RMS) is also estimated.

Power Law Fits

The power law response of electromagnetic data is given by:

$$PowerLaw\_Response = kt^a$$

Values derived are Power (a) and Y axis intercept (at time zero). The Root Mean Square (RMS) is also displayed.

Decay Display

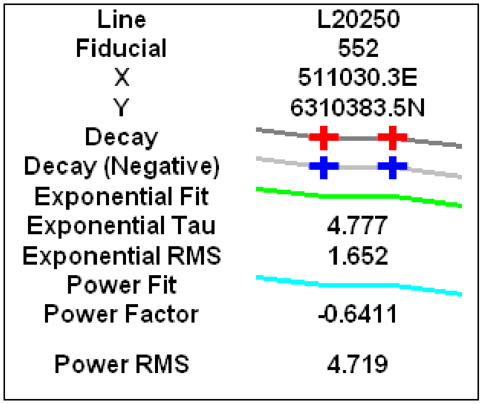
The decay display presents the decay over the available channels for individual readings only. The specific reading chosen is indicated:

- By a vertical cursor in the profile window associated with the decay.
- By an updated entry in the Status Bar showing flight, line and location (see below).

Decay Legend

A decay legend can be placed anywhere in the decay display. Each entry of the decay legend is linked to changes made in the reading position.

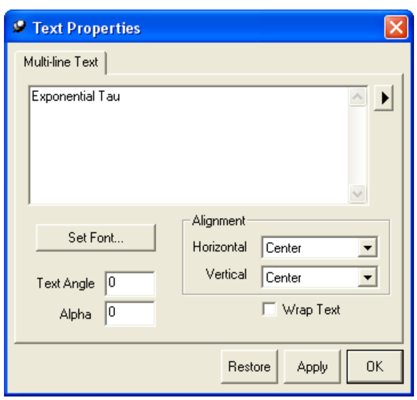
Curve toolbar available. Select the **Add Legend** button from the Decay Curve Toolbar and once clicked, move the cursor into the Decay Curve plot. Position the legend where you wish, click and hold the left mouse button and drag an area for the legend. When you release the button, the legend is populated with the data for the selected data reading (line, location, tau, fit etc).



Decay Curve legend and associated annotation

Each legend text entry has individual properties that can be specified from the Decay Legend branch of the Workspace tree. Items that are graphic (non-text) have no accessible properties but are controlled by the settings of the Line Style tab.

To edit or examine the properties of any of the text items of the legend, position the cursor over the item and click it. Selection handles are placed around the text. Once selected, by right clicking, a Properties pop-up menu can be selected. If you have difficulties selecting the text item, press the CTRL key at the same time as clicking the left mouse button.



Text Properties dialog for the decay legend

The text content can be modified in the text entry field. Font, position, rotation and alignment information are detailed at the base of the dialog. You can reposition the text, if desired by modifying these controls.

Line Navigation in Decays

In the Decay Tool, you can individually select the line to be displayed by:

- Using the **Line** pull-down list in the **Data** tab of the Decay Tool properties dialog.
- The **Line Iterator** (refer to the *Line Iterator Tool*) can be used to navigate individual lines
- Flight Map selection is available.

Where more than one line is displayed in a Profile window of the associated decay, readings displayed in the decay and legend are updated automatically if the cursor is moved from the graph of one line to another.

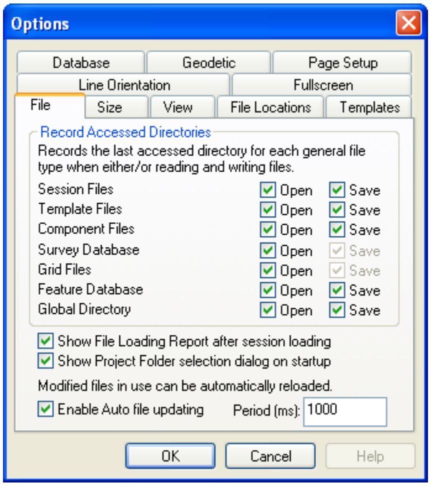
24 Display Utilities

In this section:

- *Options* – default settings for the operation of Encom Discover PA.
- *Image Registration Wizard* – utility for geo-referencing an image such as a map or geological cross-section.
- *Georeferenced Image Export* – create a located image (.EGB) for a slice in the 3D view specified by the clipping plane.
- *Grid Flipper* – interpretation tool for comparing features in grids and raster images.
- *3D View Manager* – allows the creation and use of custom viewpoints in a 3D display.
- *3D Animation and Fly-Through* – creates a movie by recording transitions between view positions.
- *Drillhole Planner* – dynamically position and plan new drillholes in a 3D environment.
- *Seismic Depth* – direct access to SEGY seismic files for velocity modelling and conversion into a format supported by Encom Discover PA.
- *Section to Map Utility* – convert a series of vertical section profiles to located images in plan view.
- *Coordinate Reprojection* – interactively convert points between any two coordinate systems.
- *Colour Table and Legend Editors* – utilities for creating or editing colour look-up tables and legend tables.
- *Tube Shape Manager* – manages the appearance of data lines when displayed in 3D as tubes.

Options

The Tools option provides access to the Options properties plus a number of utility programs. The Options item allows default settings for Encom Discover PA to be maintained.



The default Options Property dialog

Options that can be specified include:

- Paths for data, templates, components, sessions and feature databases
- Default page size (in the absence of an installed printer)
- Grid decimation
- Preferred view options
- Specific file locations and project directory definition
- Global data directory specification
- Line iterator controls
- Line orientation preference by envelope of angle
- Default geodetic settings (projections and spheroids etc). Refer to Map Projections.
- Default printer page size and orientation
- System template defaults

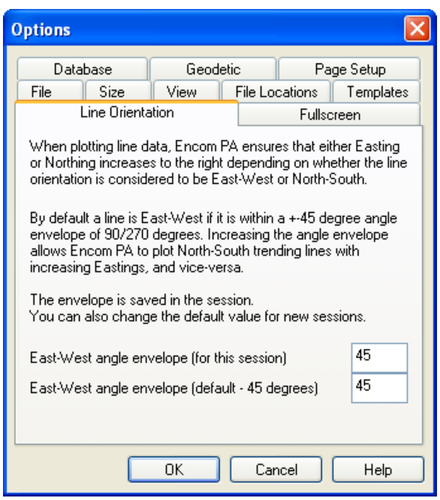
Other applications such as Colour Table Editor, Legend Editor and Customise menu items are described in their relevant sections.

Specific controls available from the Options tab dialogs include:

- *Line Orientation*
- *Page Setup*
- *Templates*
- *Size Settings*
- *View*
- *Database*
- *Files Locations*
- *Setting the Global and Project Directories*

Line Orientation

The **Line Orientation** tab allows control of the orientation of profile drawing in situations where lines are oriented such that they could be plotted in reverse-sense when close to southwest-northeast or northwest-southeast. In these cases, the azimuth of the lines (close to 45 or 225 degrees) can plot such that north or east is to the left end of the profile. To overcome this line orientation difficulty for these surveys, increase the East-West angle envelope.

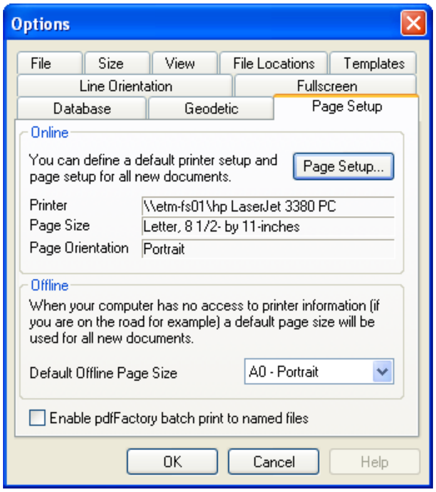


Line Orientation tab of the Options dialog

Page Setup

The Page Setup tab of the Options dialog provides support for default printers when Encom Discover PA is connected 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 if Encom Discover PA 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.

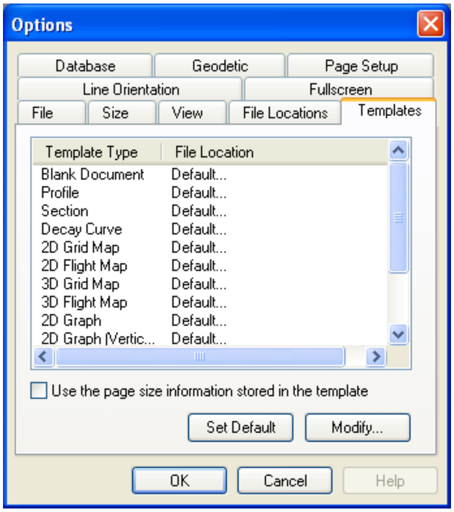
To modify the printer default setting for Online printers, select the Page Setup button.



Page Setup tab of the Options dialog

Templates

Encom Discover PA operates by using templates to display the various presentation formats. A set of default templates are provided within the software to this and they have reasonably universal settings to appear with maps, profiles etc as they do. However, you do not have to use these system defaults. You can define your own templates for basic maps, sections, profiles etc. This is done from the Default Templates tab of the Options dialog (**Tools>Options**).



Options tab to default templates

To specify an alternative template, select the display item to be modified, click the **Modify** button and path to the required template of your choice. To revert to the system supplied default template, again select the item and click the **System Default** button.

Size Settings

The **Size** tab allows the default size settings for raster grids and bitmap images.

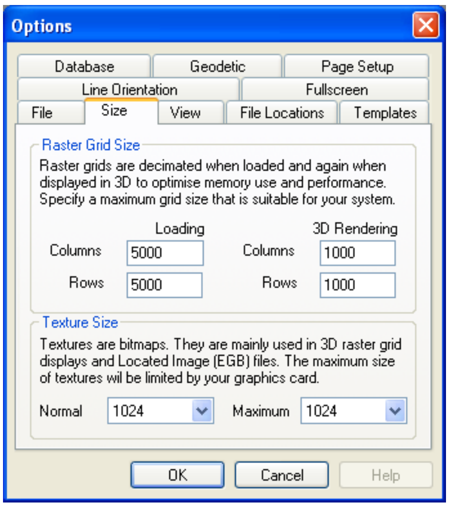
With Raster Grid images the maximum size (in columns and rows) allowed for a grid file to be loaded into Encom Discover PA can be specified and then the decimation for 3D rendering of this image can also be specified, according to the user operating system.

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 Encom Discover PA will use large grid handlers to load the grid into Encom Discover PA. 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 Encom Discover PA will use large grid handlers to load the grid into Encom Discover PA. 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 Encom Discover PA 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 *Surface Compression* in the *View Surface Images* section of the *Three-Dimensional Displays* 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 Encom Discover PA 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 *Surface Compression* in the *View Surface Images* section of the *Three-Dimensional Displays* chapter for more details on compression.



The Size tab in the Options dialog

The **Texture Size** refers to the default resolution setting for images created in a Encom Discover PA session. The maximum size specified is typically limited by the users installed graphics card.

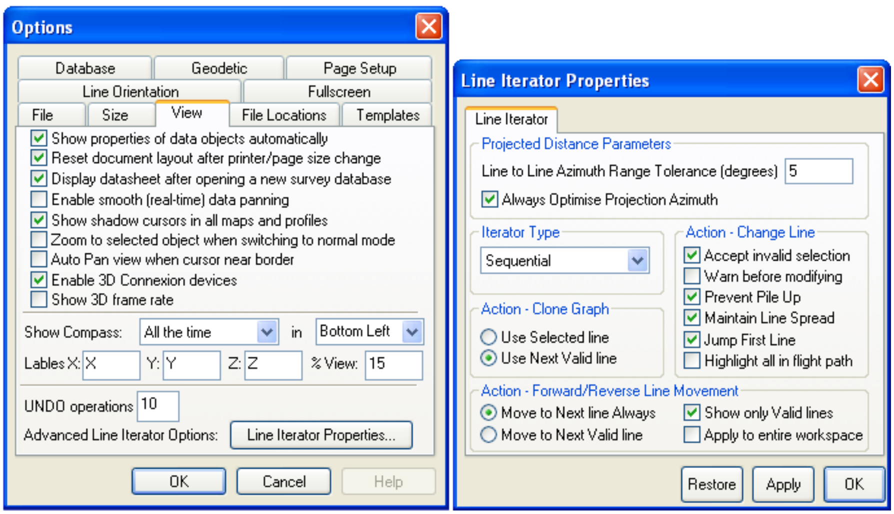
Normal texture size - Pixel resolution of rendered surface in Encom Discover PA. When a surface file is imported into Encom Discover PA the display is actually composed of a bitmap image, the resolution of the bitmap image is determined by the normal texture size setting, a higher resolution setting 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 Encom Discover PA, 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 Encom Discover PA 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.

View

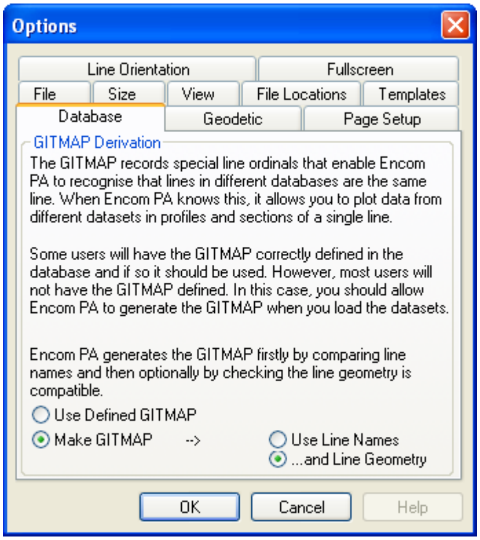
The View tab provides the user with control over when property dialogs and spreadsheet windows appear plus other data panning and zoom options.

The View tab also controls the appearance of the **3D Compass** object, the number of **Undo** operations and advanced **Line Iterator Options**, such as line movement, behaviour when cloning and changing lines the distance selection controls.



The View tab dialog with the Line Iterator properties dialog

Database



The Database tab dialog of the Options dialog

The Database tab controls the use of GITMAP generation in a loaded Geosoft database. The GITMAP field usually contained in a database allows Encom Discover PA to recognise that lines in different databases are the same line. This is done by recording a “primary” line index if the same line is found multiple databases. When Encom Discover PA knows about this GITMAP field it allows the user to plot data from different datasets in profiles and sections of a single line.

If a GITMAP field is not included in a database then the default selection in the Database tab of the Options dialog of “Make GITMAP” should solve this problem. The default option to “Make GITMAP” by using line names AND Line Geometry should account for the majority of databases loaded into Encom Discover PA however sometimes if lines of one database begin or end at a different location to lines in a second database, even though these are in fact the same lines then the “Use Line Names” option should be selected.

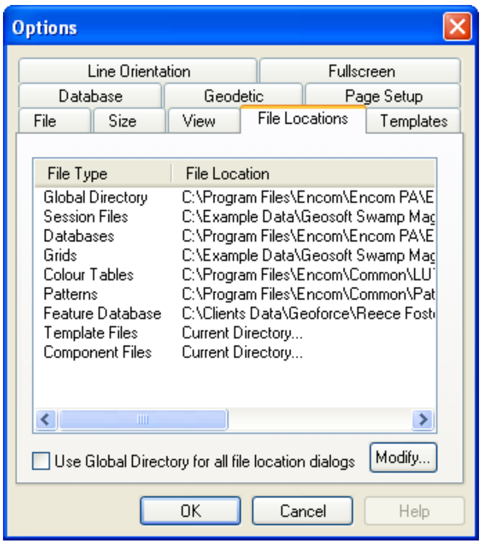
Note The GIT library only supports station rate data and does not allow for variations in station rate sampling so Encom Discover PA does not currently support this type of data.

Some users already have the GITMAP defined in the databases and so the Use Defined GITMAP should be selected. Most users will not have this defined already so the Make GITMAP option should be selected.

Files Locations

The File Locations tab permits frequently used files and directory locations to be defined. The dialog lists the various file types specified for frequent and specific use.

The **Global Directory** is used to open project files such as databases, surface grids and feature databases. Refer to *Setting the Global and Project Directories* for additional information.



File Locations tab of the Tools>Options menu item

To alter any of the specified file location pathings, select the item by highlighting the File Type and then click the Modify button to reset the navigation path.

Setting the Global and Project Directories

Encom Discover PA can use either a Global Directory or a Project Directory to access data. The Global Directory determines the folder that is automatically navigated to when a file of any type is opened. The Project Directory can be used when working on files in a specific project folder.

By default, this location is updated to be the location of all subsequent file locations until it is changed by:

- Opening any data file in a different folder with the **Use Global Directory for all file locations** option enabled

- Selecting the Global Directory in the **File Locations** tab of the **Tools>Options** menu item and clicking the **Modify** button.

Note If you wish to specify a project directory for use but sometimes want to access files from different locations, define the Global Directory but disable the **Use Global Directory for all file locations** option. This allows the Global Directory to always remain at the defined folder but does not prevent you accessing files at different locations.

The dialog used to control the Global Directory is located on the **File Locations** tab of the **Tools>Options** menu item (see above). You can reassign the Global Directory or Project Directory at any time.

For additional information, refer to *Preferred Directory Access*.

Image Registration Wizard

The .EGB files used to locate and define associated bitmaps of Located Images (see *Appendix B: Data File Specifications*) can also be created by a wizard provided in the **Tools>Image Registration Wizard** menu option or from the **Wizard** button associated with the properties dialogs of Located Images.

The wizard is used when images (such as a scanned section or airphoto) already exist and require their locations to be defined for use in other displays. Note that single or multiple images can be used with this wizard. For example if multiple geological sections were scanned and need to be located, this wizard can create a single .EGB file to display the multiple bitmaps.

Step 1

After selecting the **Tools>Image Registration Wizard** option (or **Wizard** button), an introductory screen is presented. Click the **Next** button to progress to the second dialog. Select the bitmap files required to be defined as Located Images. The files are added to the screen as shown below:

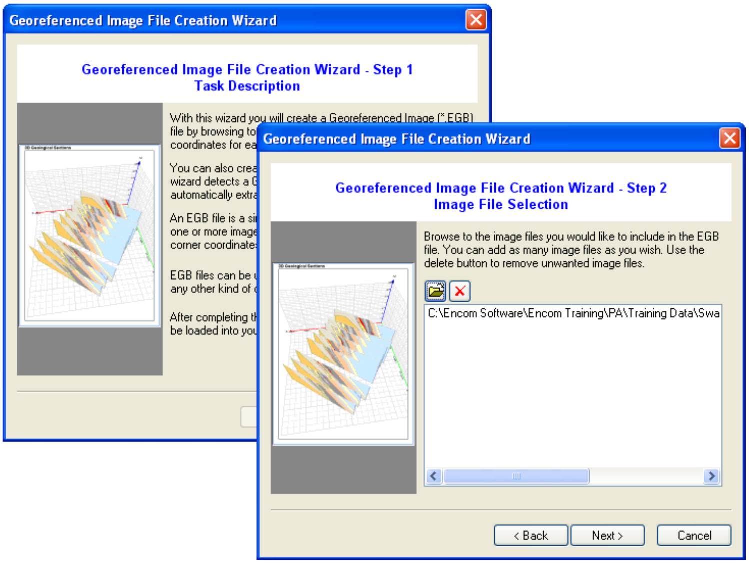
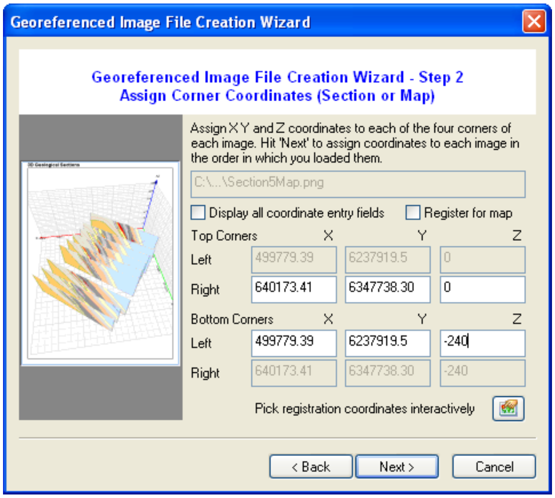


Image Registration Wizard screens 1 and file selection screen 2

Step 2

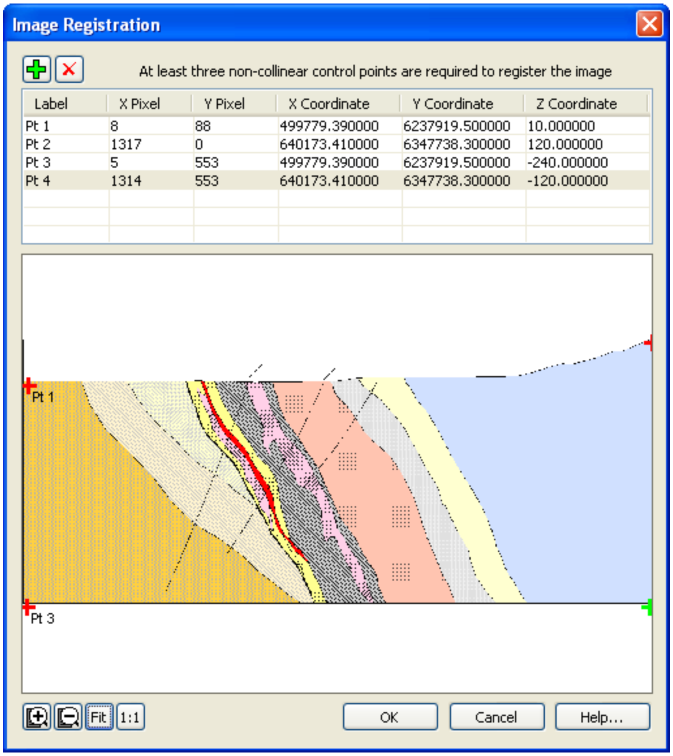
With the image files chosen, click the **Next** button to access Screen 3. This screen allows entry of the corner points of the images. If the bitmap is rectangular in shape and to be displayed on a flat plane, then opposite corners have the same X/Y and Y/X coordinate combinations. In this case, only two corners need be defined (botton-left and top-right). If the bitmap is to be displayed as inclined or non-planar, then you need to specify each corner point easting (X), northing (Y) and elevation (Z). Enable the **Display of coordinate entry fields** to enter the four corner points. The elevation of the corner points relates to the absolute reference level that the image is displayed in the 3D display.



Screen 2 of the Image Registration Wizard

When you have completed specifying the first bitmap, click the **Next** button. This screen cycles through each of the included images of Screen 2. To assist in defining their location, the corner coordinates are retained from the previous image entries.

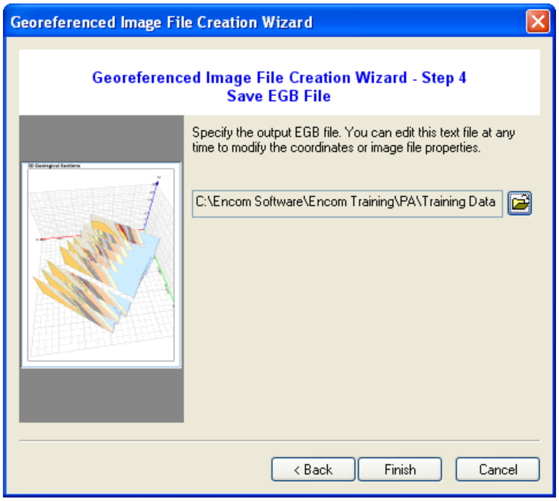
To interactively specify coordinates for an image press the associated button at the bottom of this dialog to view the Advanced Image Registration dialog. This dialog allows you to specify the coordinates for any known location within the image using the interactive map. Press the Add coordinate button to select a location on the image, which will then automatically enter pixel coordinates into the table above. Then manually enter the XYZ coordinate values into the cells of the location.



The interactive Image Registration dialog allows you to enter coordinates for any known location for an image.

Step 3

Screen 3 of the **Image Registration Wizard** requests you specify the .EGB file and path name. 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 for the Located Image(s).



Output the EGB file for use in a 3D display

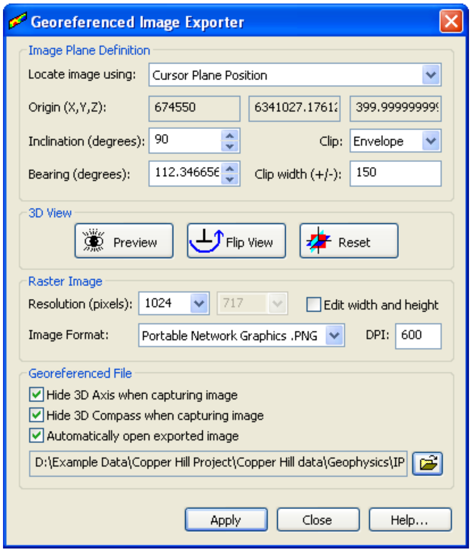
Georeferenced Image Export

The **Georeferenced Image Export** tool allows the user to create a located image (.EGB) for a slice in a 3D Display window, which has been specified by the clipping plane for any arbitrary position.

This is useful for capturing a view to be used in another 3D display, a profile display or simply an image capture for a report.

The Georeferenced Image Export tool is accessed from the **Tools** menu. When selected the below parameters are available.:

- *Image Plane Definitions*
- *3D View Options*
- *Raster Image*
- *Georeferenced File*



The Georeferenced Image Exporter utility.

Image Plane Definitions

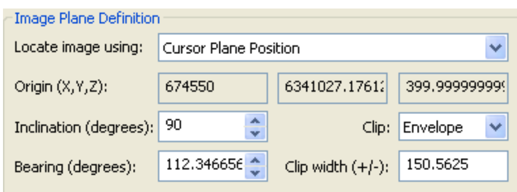


Image Plane Definitions of the Georeferenced Image Export utility.

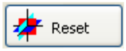
The **Origin**, **Inclination**, **Bearing**, **Clip** and **Clip width** parameters are automatically populated based upon the Cursor Plane properties set prior to opening this utility. These can however, be edited from this utility.

Note If there is a voxel model in the view then the cursor plane will have to be synchronised with the voxel model arbitrary slice feature in the *Voxel Model Properties*.

3D View Options

Preview – view a preview of the located image that will be generated.

Flip View – view the 3D display from "behind" the 3D model and looking back towards the original viewing position.



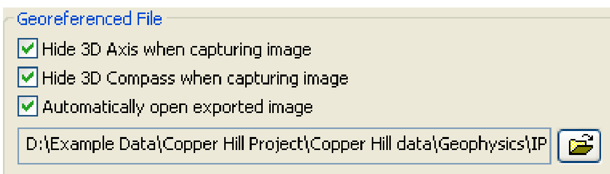
Reset – reset the orientation of the 3D Cursor Plane to be 0 inclination and 0 bearing.

Raster Image

Raster Image options of the Georeferenced Image Export utility.

The Raster Image options allow the specification of the image resolution, the accompanying image file format and the DPI setting.

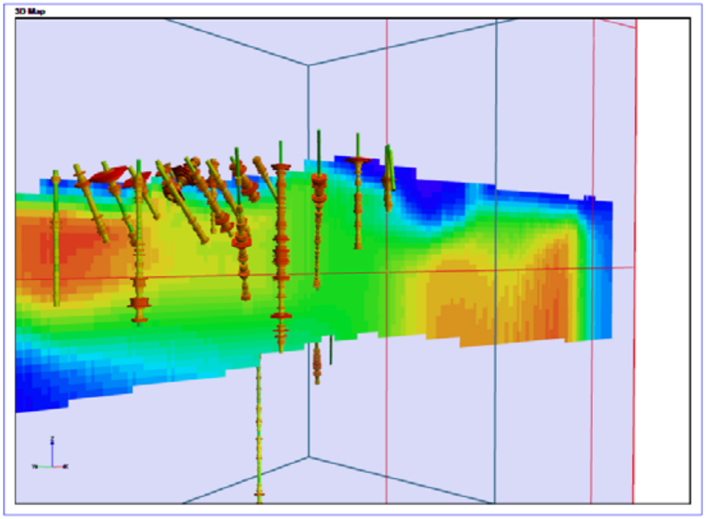
Georeferenced File



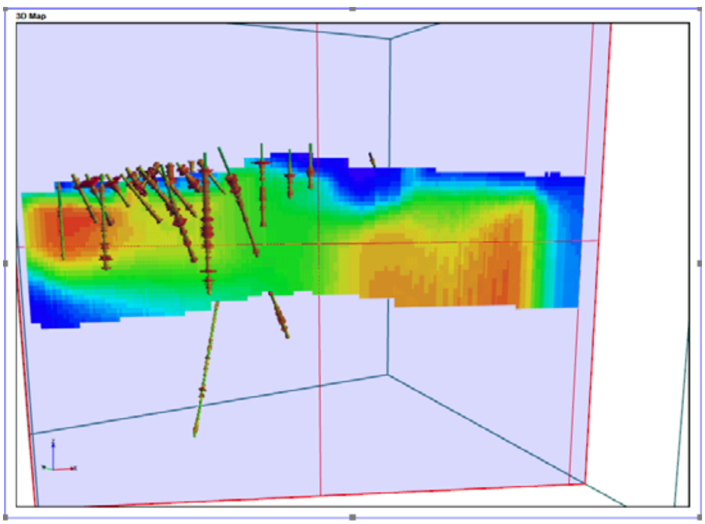
Georeferenced File options of the Georeferenced Image Export utility.

The georeferenced file options provide simple options for what accompaniments will be captured with the image from the 3D display, whether the output will be automatically opened into the same 3D display and the file name and location of the output.

To create the output georeferenced image file press the **Apply** button once the above parameters and output file name and location have been specified. If the *Automatically open exported image* option is enabled then the output located image file will be opened automatically into the current 3D display.



Original 3D display consisting of 3D drillholes and a voxel model slice.



Resultant georeferenced image of original display viewed in 3D.

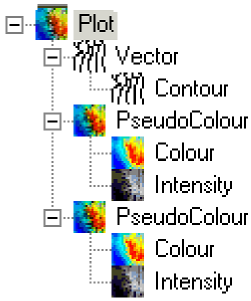
See also:

...*3D Cursor Plane*
...*Voxel Model Properties*

Grid Flipper

During interpretation it is often necessary to compare images from one data source with another. Encom Discover PA provides a method to do this quickly and easily. It allows any number of images to be loaded in a Grid surface and then cycled using a tool commands to rapidly replace one image with the next. Grid flipping can function for both 2D and 3D image displays.

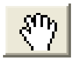
To flip a grid:



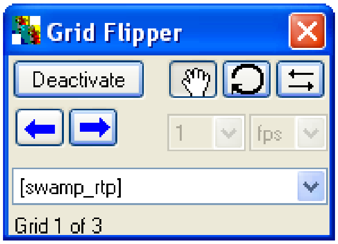
1. Create an image layer in a map and add two or more surfaces to the image. Any of the ER Mapper, RGB or Pseudocolour algorithms can be used. If contours are to be used, they can (and should be the uppermost layer in the tree). They are not flipped but are always displayed.

2. You need to have the Workspace tree visible but you don't need the Grid Group Properties dialog open.

3. From the Tools menu, select the **Grid Flipper** option. A small toolbar is displayed. The toolbar controls whether the grid images are to be cycled through their display from the **Activate/ Deactivate** button. The Deactivate button name must be shown for the grid flipping to be enabled. The name of the displayed grid is shown in the pull-down entry. Using the pull-down list to select an image, displays the selected image.

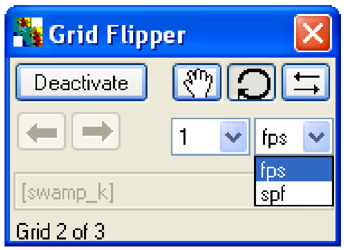


4. To rapidly cycle through the grid displays manually, select the **Manual Grid Flipping Mode** button and click on the blue **Left** or **Right** arrow buttons.



The Grid Flipper dialog in Manual mode

5. To flip through the grid images automatically select the **Continuous Grid Flipping Mode** button and then choose the units for the cycle rate from the drop down list on the far right of the dialog. Choose between *seconds per frame* (spf) or *frames per second* (fps), where the latter is the faster of the two cycle rates. Then enter the value for this unit from the drop down list to the left of this.



The Grid Flipper dialog in Continuous mode

The grid flipping tool is available for both two-dimensional and three-dimensional maps.

Encom Discover PA stores all grids in memory to provide a responsive interactive interpretation system. You may choose to have many grids loaded and this may cause you to exceed physical memory. Large grids may be decimated at load time to improve performance. ER Mapper algorithms can process any sized images supported by ER Mapper.

3D View Manager

Allows the creation and use of custom viewpoints in 3D displays. A full description of the operation for this tool is provided in *3D View Manager*

3D Animation and Fly-Through

This tool of Encom Discover PA is a powerful means of creating animation of 3D displays. A full description of the operation for this tool is provided in *3D Animation and Fly-Through*.

Drillhole Planner

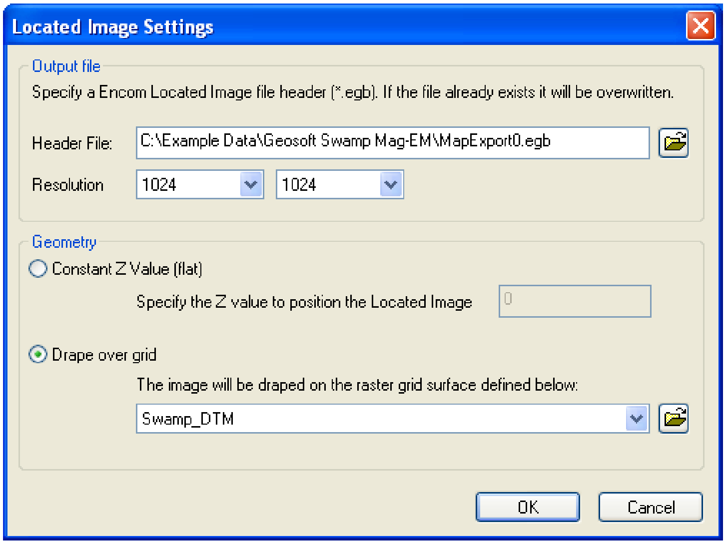
Encom Discover PA provides the powerful capability to dynamically position and plan new drillholes directly within the 3D environment. This allows the user to target 3D objects such as voxel isosurface grade shells, extruded quartz veins, .DXF solids (such as alteration zone volumes generated from digitised section boundaries) or feature object interpretations.

The Drillhole Planner is accessed from the Tools menu of Encom Discover PA. You need an Encom Discover PA Professional licence to use this tool.

- Planning Drillholes in 3D
- RL or Z Value Assignment
- Drillhole Types
- Targeting Example
- Drillhole Planner Projects

Planning Drillholes in 3D

1. Open all required targeting data into the 3D window (e.g. existing drillholes, grade shell (voxel model) isosurfaces, mineralisation solids, etc).



Use the View in 3D menu option to display a grid in 3D as a located image.

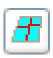
2. If a topographic/DEM grid is to be used for collar positioning, display this in 3D as a located image; in a 2D map display open the DEM grid as a Grid surface, and use the pop-up menu (right mouse click) from the Map 1 branch of the workspace tree to choose the **View in 3D** menu option to capture a view of the grid draped over itself (capture dialog pictured above). In 3D, alter the transparency of this image to 40 or 60%.

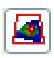
3. Create a new Feature Database by selecting the **Create** option from the **Features** menu. When prompted to enter a name for the Feature Set enter `DHPlanner_Collars`. The collar and survey information will be stored in this database before the data is finally converted to MapInfo .TAB files for displaying as drillholes. Add the `DHPlanner_Collars` feature set to the 3D window containing the located image grid and other targeting data and make this set editable (select the editable “pencil” icon located next to the branch of the feature set in the workspace tree.

4. While still in the 3D window, open the **Tools> Drillhole Planner** menu option.

5. Use the **New** button to create a new Drillhole Planner Project to add the new drillholes to. Specify a project name and location. This will eventually create two files: a `projectname_collars.TAB` and `projectname_surveys.TAB`.

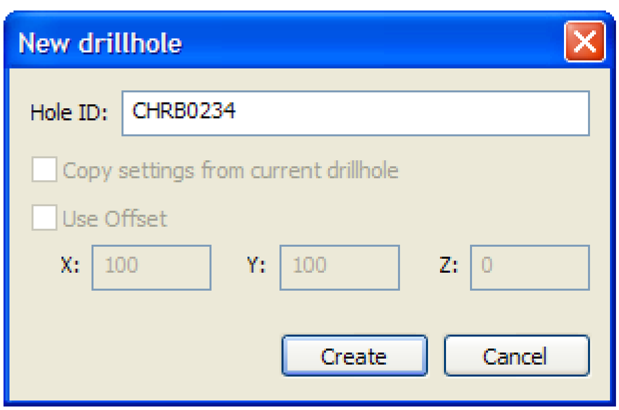
6. Set the required **Dip Sense** (i.e. whether down dip is negative or positive) and **Drillhole Units** (default metres)





7. If the Cursor Plane has not already been enabled click on this button in the Cursor Plane toolbar. If using a located image of the grid to nominate Z values for the collars click on the select “arrow” icon next to this located image branch in the workspace tree and then click on the **Bond to Image** button in the Cursor Plane toolbar.

8. Use the **3D Navigation** cursor controls to orientate the view to the first proposed drillhole location and press the **Create** button in the Drillhole Planner dialog.



9. The New Drillhole dialog will open, with a default Hole ID ‘DH1’ populated. Change this if required and press **Create**. The new hole will be listed in the Drillhole Planner dialog, with various default attributes.

10. The drillhole can be ppositioned either:

    - **Manually** by entering the Collar X & Y coordinates (and Z value if no grid was imported in Step 2 above). Press the **Apply** button to visualize the planned hole location (a point symbol will be added to the Feature Database)

    - Dynamically by pressing the Create New Point feature button on the Drillhole Planner dialog, and clicking on the desired location in the 3D window with the Select/Navigate cursor (see left). If no topographic grid was imported (Step 2 above) the collar Z value will need to be manually entered. A point symbol will be added to the Feature Database called `DHPlanner_Collars`. The hole location can be then manually edited or dynamically repositioned again using the **Create New Point feature** button.

11. Select the hole **Type** as either Straight, Constant Deviation or Variable Deviations. Drillhole Types are discussed fully below; this example will use the straight type.

12. Manually enter the desired **Azimuth**, **Dip** and **Total Depth** 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.

13. Press the **Apply** button at the bottom of the dialog to create the drillhole. A new Drillholes object will be added to the workspace tree, called `projectname_collars` (i.e. the project name specified in Step 4), and the planned drillhole will be displayed.

14. If any aspect of the drillhole is incorrect, modify as necessary in the Drillhole Planner (e.g. Total Depth/Azimuth/Dip) and press **Apply** again to recreate the hole.

15. Repeat steps 8 to 14 for each additional drillhole required.

RL or Z Value Assignment

Z or RL values can be assigned to your planned drillholes in 3 ways:

- By utilizing a topographic/DEM grid imported as an image, as detailed in Step 2 of the *Planning Drillholes in 3D*. When the **Create New Point Feature** button 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).

- Manually by entering the required value in the Collar Z field. This also allows an existing Z value (e.g. one captured using a grid in first method above) to be modified

- By specifying an alternative grid from within the Drillhole Planner dialog via enabling the **Use surface for collars** option. This allows grids that have not been imported into Encom Discover PA to be utilized for the Z value. When using the **Create New Point Feature** button, Encom Discover PA 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.

Drillhole Types

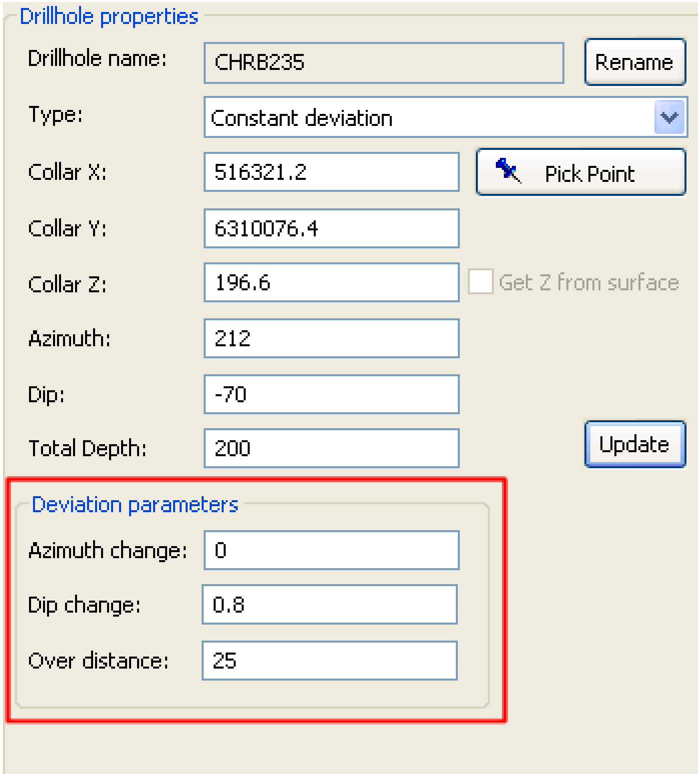
The Drillhole Planner allows 3 types of drillholes to be created. All types require a collar Azimuth and Dip value to be specified, as well as the Total Depth of the drillhole.

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).



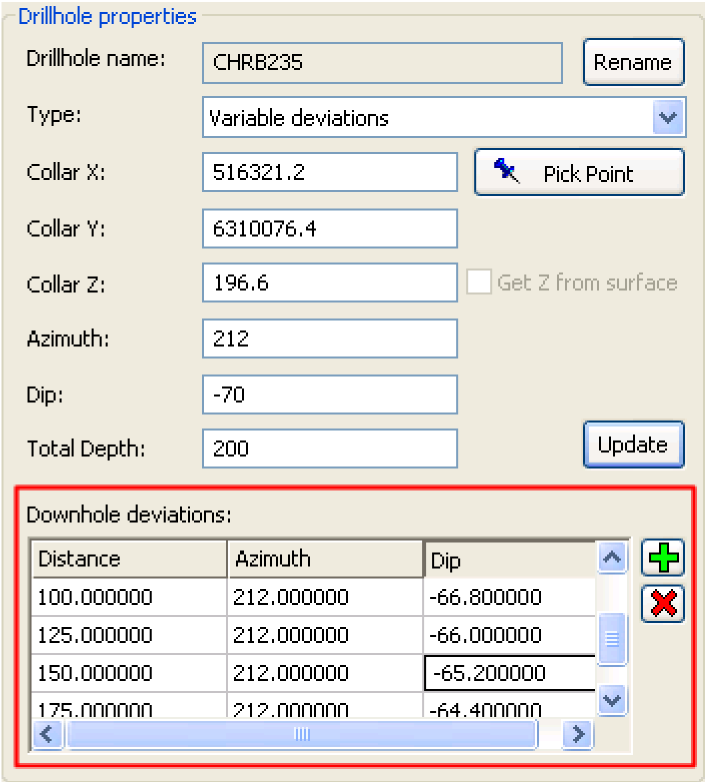
A constant deviation hole set to rise 0.8 degrees every 25m.

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.

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 mineralisation 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).

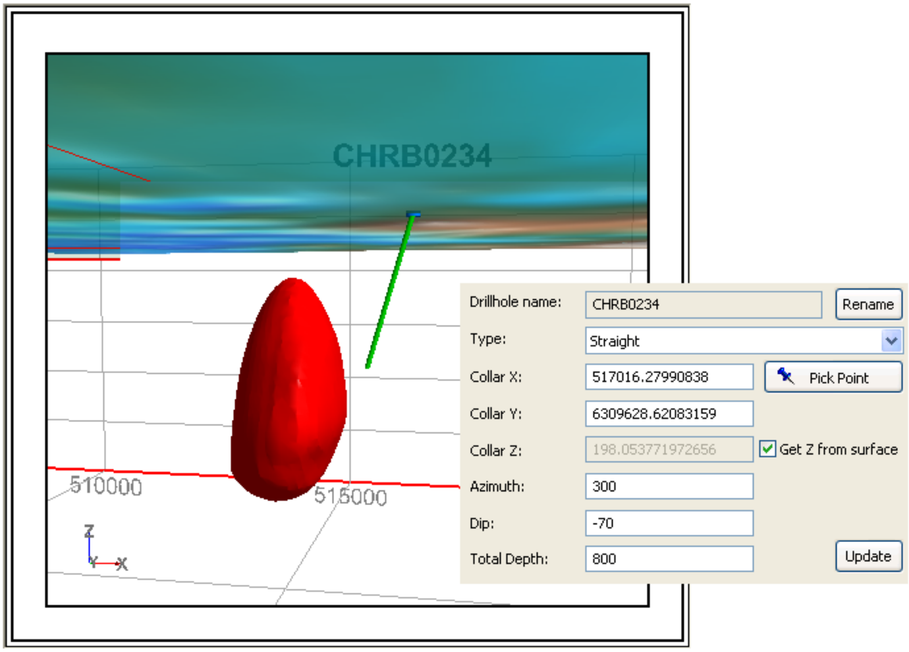


Image 1: Initial hole generation as a straight hole, using the standard collar dip of -60 degrees orientated towards the target.

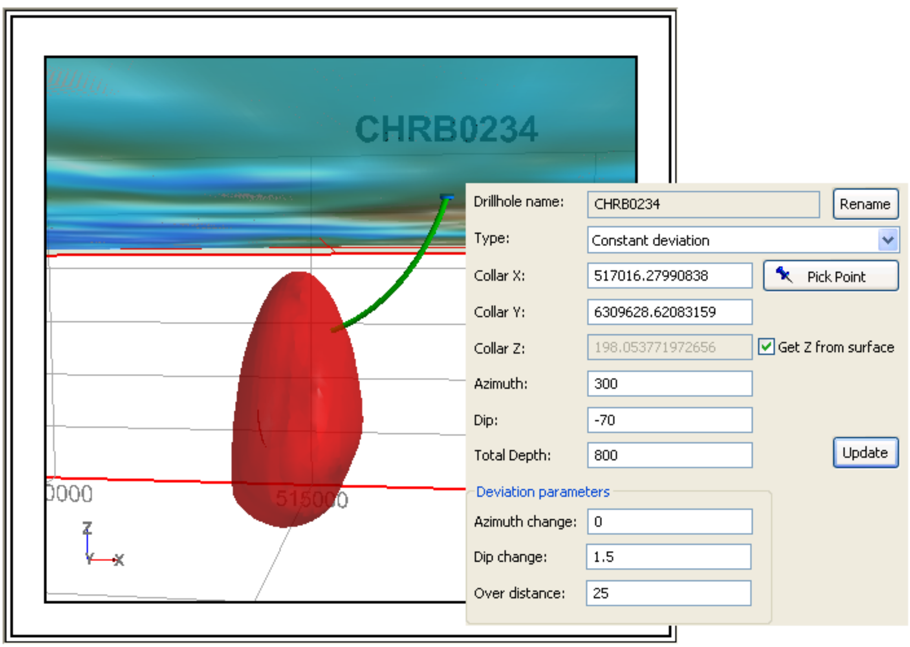
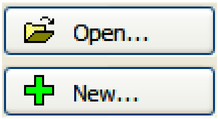


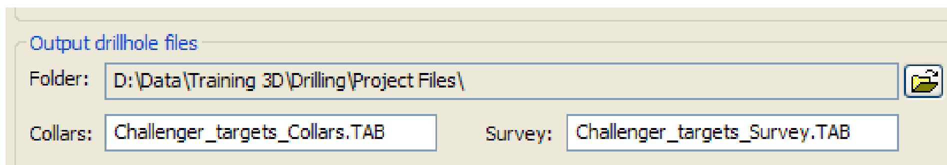
Image 2: Using existing drillhole variations as a guide, the proposed hole is changed to a Constant Variation type, and set to rise 1.5 degrees every 25m. The hole now intersects the target.

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.

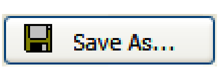
After a hole's attributes have been specified in the dialog, pressing the Apply button uses the .DPP file to generate the necessary MapInfo .TAB collar and survey tables; these can then be used to visualize these holes in Encom Discover PA or MapInfo-Discover. The output .TAB names and location are displayed in the main dialog:



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 utilized 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:

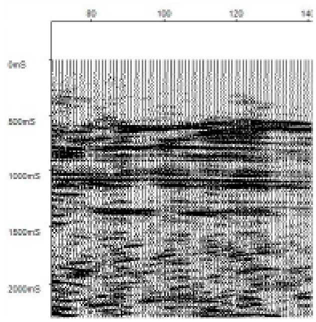
1. Press the **Import** button and browse for the source collar and survey files (these must be part of an existing Discover drillhole project).
2. A drillhole **Selection** dialog will then be displayed- select only the required drillholes to be used as templates
3. Specify a new .DPP file to store these templates in.
4. The selected holes will be displayed in the Drillhole Planner dialog. 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. Alter the holes as necessary and visualize.



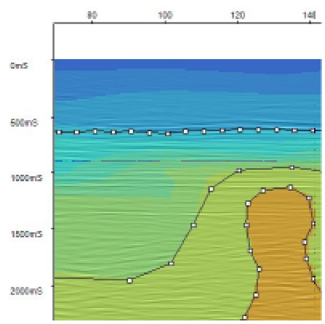
An open drillhole planner project can be renamed and saved as a new project.

Seismic Depth

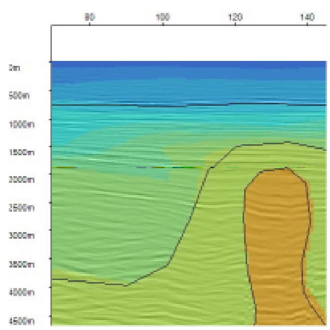
The Seismic Depth tool allows you to create projects by directly opening SGY files (also known as SEG-Y), along with support for various navigation or velocity models files. Sections can be viewed as either two way travel time or a depth axis. You can then manually or auto-pick horizons and define velocities for each facies or region within the horizons. Sections can be exported as 3D geo-located images or a 2D Plan map of the survey lines.



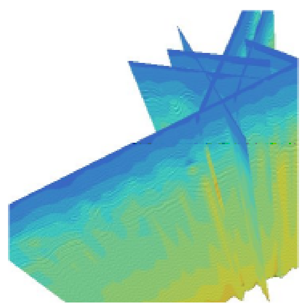
SGY wiggle trace



Velocity tint (time)



Depth



3D

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 2D plan maps or 3D depth image of this data.

The key steps are:

1. *Create and configure a new seismic project and add SEG-Y files:*

2. *Assign navigation and coordinate information*

3. *Visualise section data*

4. *Digitize horizons on a section*

5. *Assign velocity values to a section*.

6. *Generate 2D plan of the track lines*

7. *Generate 3D registered images of the depth sections*

Create and configure a new seismic project and add SEG-Y files:

1. Select **Tools>Seismic Depth**

2. Start a new project with **File>New Project**, or open an existing project with the **File>Open Project**. When the Seismic Depth tool 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

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.

---

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 *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. Click OK to create the project and add any available files from the project directory.

Note If new SGY or metadata files are added to the project, the **File>New Project** Setup can be re-run at any time to create a new project, which will allow batch importing and editing of Section Properties.

---

13. 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).

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>New Project** Setup 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 **Lines to Skip**. Also specify any comments by the **Comment** character. Select the applicable **Delimited** characters (**note not consecutive**) 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

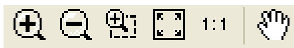
Visualise section data

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.



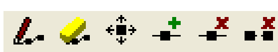
5. The view can also be scrolled using the horizontal scrollbar located below the section.

6. A dynamic profile of individual traces can be viewed by select **View>Profile**.

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.

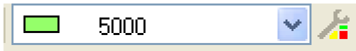


3. You can undo any changes by selecting **Edit>Undo** or **Edit>Redo**.

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.

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 Encom Discover PA.

3. Click **Make TAB file** to generate a MapInfo vector file for the seismic lines.

Generate 3D registered images of 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**.

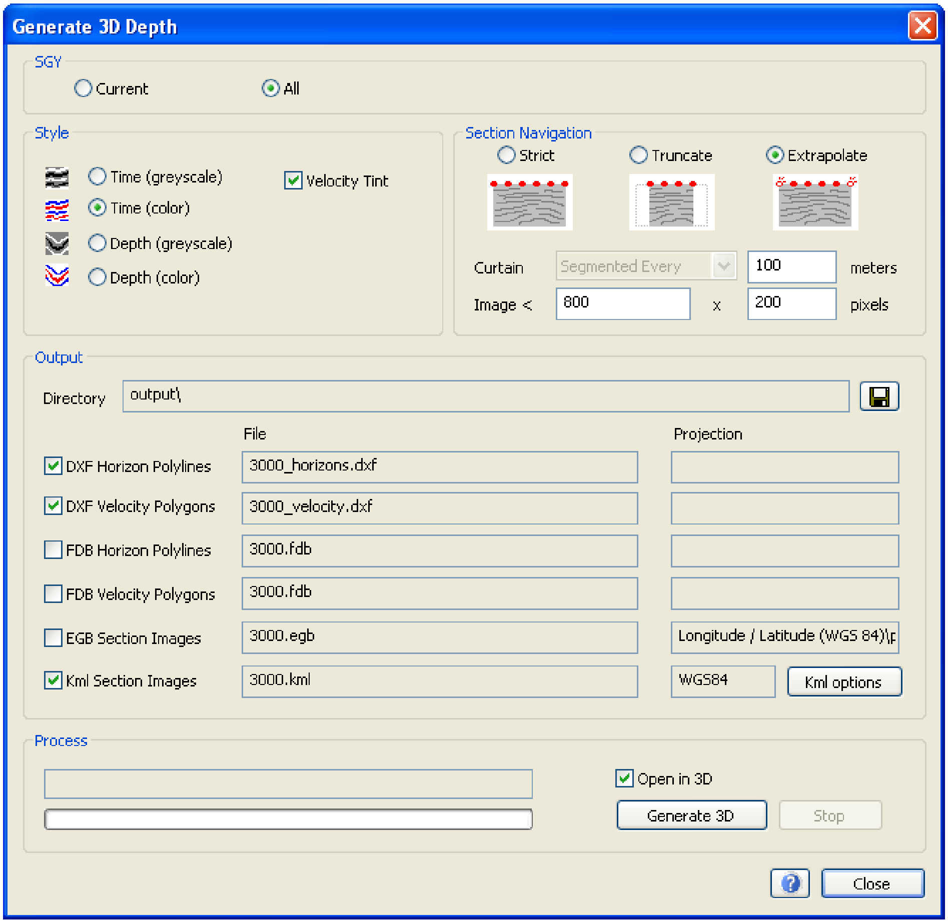
3. In curtain segments, select **Every 10 metres**. This will generate images which bend every 10metres to ensure it follows the bends in the tracklines.

4. For the **Colour**, select **Velocity Tint**.

5. Select an output directory for the files—the located image .EGB file will be automatically named by the section or the project.

6. Tick **Open in 3D** to automatically open the image files in Encom Discover PA.

7. Select **Generate 3D** to create the image files.



Options available for generating a 3D depth section.

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 SGY, .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 unstacked data)
- X
  The source of Longitude or Easting
- Y
  The source of Latitude or Northing
- SKIP
  All other columns should be labelled SKIP

Digitizing Horizons

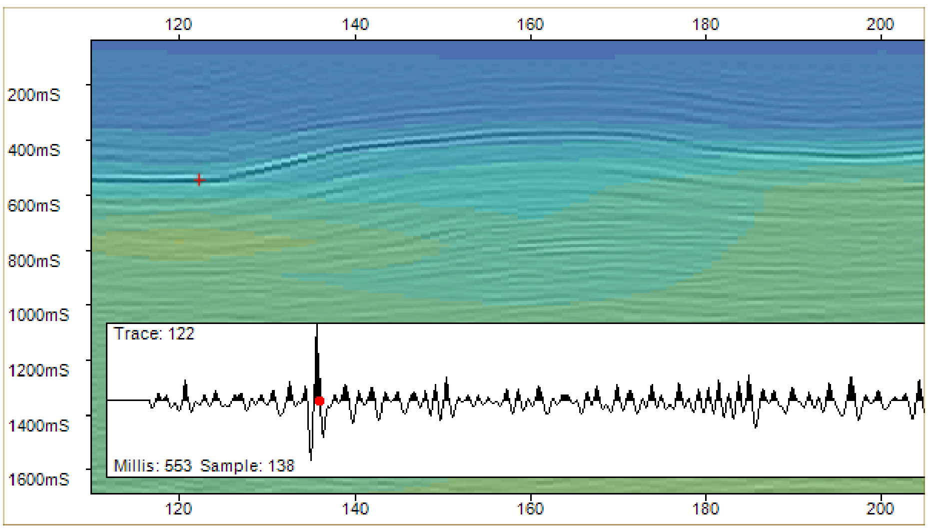
You can create horizons by drawing and editing polylines with the editing tools. You can draw the lines manually and colour fill between them with a velocity value/color. Make sure horizons are drawn past the edges of the data or cut through another horizon so that they define an enclosed region. You 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.

- **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.
- **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.
- **Move Pick**: Move a pick. Moving the end of a horizon onto the end of another horizon joins them as one continuous horizon.

- **Add 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).
- **View>Undo**: Undo any edit operation above.
- **View>Redo**: Redo any edit operation above.

Viewing Profiles

Use the **View>Profile** tool to dynamically displays the wiggle trace horizontally for the current pointer location. Also display the point time location along the trace and the sample number.



Example depth section profile with wiggle 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.

General Project Properties

- **Trace Width**: The pixel width of a trace at 1:1 zoom (when one SEGY sample (receiver) is one pixel).
- **Wiggle Scale**: The full scale amplitude of wiggle traces at 1:1 zoom.
- **Velocity Transparency**: The transparency of the velocity tint over grayscale (0-100)
- **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

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.

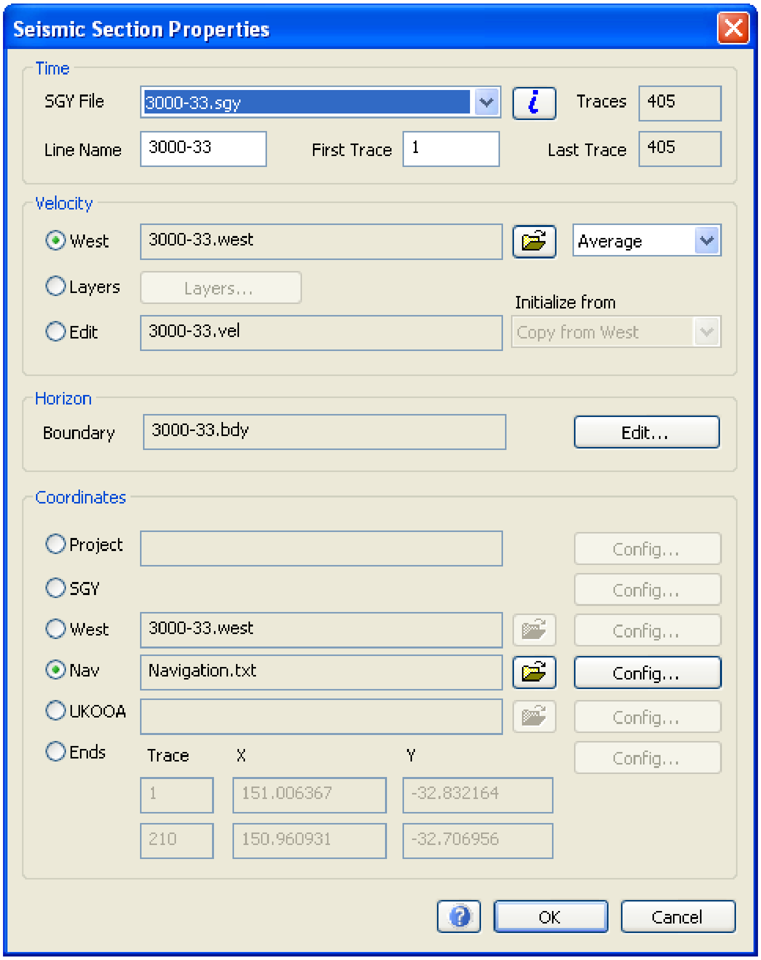
Auto Pick

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 SEG-Y 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.



Seismic Section Properties dialog box

Velocity

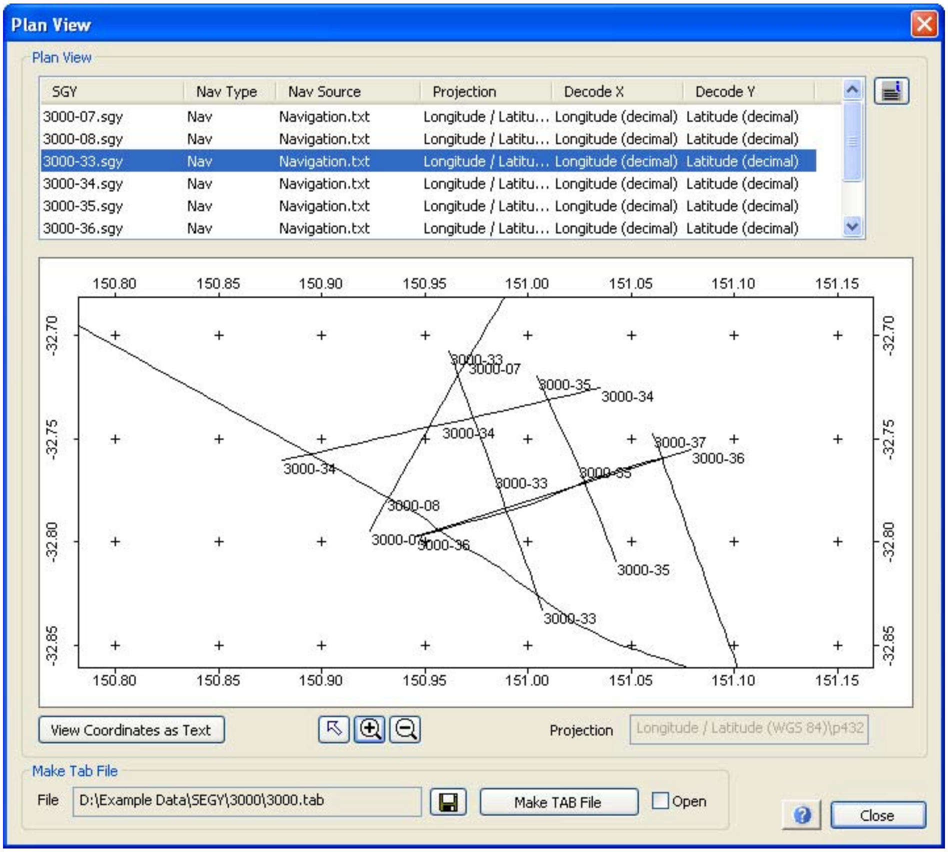
- **West**: The velocity is provided by a .west file. Specify whether the source contains Average (from surface to the bottom horizon) or Interval (within the current horizons) velocities.
- **Layers**: Use the project's layer cake for velocity.
- **Editable**: The velocity is editable and stored in 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

Plan View Dialog Box

The plan view shows the cruise/track plan of the SEG-Y lines and you can make a TAB file of the plan for viewing in Encom Discover PA. 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.



Plan view of seismic lines

Generate 3D Dialog Box

Use the toolbar button to generate 3D images and EGB files. For each SEGY line used in the generation a .PNG file will be created of the depth image. A located image .EGB file will be generated which contains the 3D registration of the PNG image. Encom Discover PA will use the located image to display the .PNG images in 3D space.

- **Current**: Generate 3D files for the currently viewed section.
- **All**: Generate 3D files for all sections in the project, which will consist of sub-layers for each section.
- **Colour**:
    - Greyscale: The generated images will be grayscale.

    - Velocity Tint: The generate images will be grayscale with velocity tint.

    - Lookup: The generated images will be coloured with the lookup table chosen.

- **Curtain Sections**:

    - One Full Length: The output SEGY line will be a straight line between two end points.

    - Every x metres: The output SEGY line will consisted of sections following the navigation coordinates. The EGB file will specify break points (a quad strip) along the PNG image with coordinates for each vertex.

- **Projection**: The output projection of the 3D result.

- **Output directory**: The directory to write the output files. Project folder is used by default.

- **Open in 3D**: Output images will be opened automatically in Encom Discover PA.

Section to Map Utility

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

The Section To Map plug-in utility for Encom Discover PA enables users to display the content of 2D section profiles or graphs as geo-located images in plan view in a 2D map display.

The procedure is.

1. *Prepare the Destination 2D Map*

2. *Prepare the Source Sections or Profiles*

3. *Batch Print to Located Image (.EGB) File*

4. *Run the Section To Map Utility*

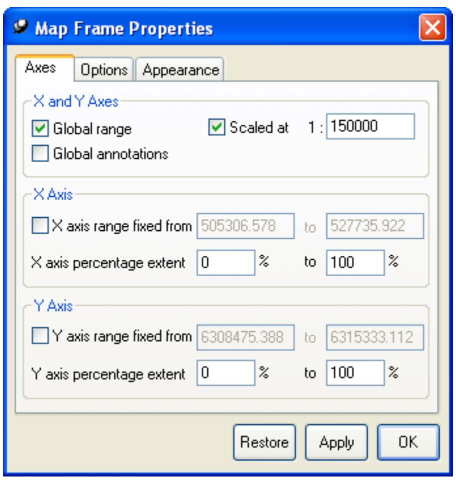
5. *Add Processed Located Images to the Map*.

If you experience problems with using this utuility, see *Trouble-shooting*.

Prepare the Destination 2D Map

The destination map is likely to be either a 2D map or a 2D graph but it is not limited to these because the resulting data can also be displayed in 3D displays. However the most likely target is a scaled 2D map.

The X and Y scale of the map should be determined prior to creating the section output. In a 2D map the X and Y scales will be equal but in a 2D graph they may be different. The X and Y axis scales can be set from the Map Properties dialog of a 2D map display.



Specify the Axis Scaling for the destination 2D map

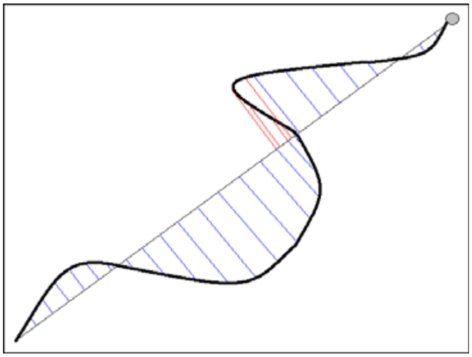
Prepare the Source Sections or Profiles

It is likely you will want to display data from one or more survey lines. The data may be CDI or IP section data, profile data or any other kind of content.

You can either build a profile/section graph or a general 2D graph. Typically you will use a profile/section graph but there are some instances when a 2D graph will be required.

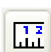
Profile and section graphs convert X and Y coordinates to “projected distance” coordinates which are then used as the X coordinates for the graph. This is appropriate when the survey lines are straight or at least not highly curved.

If the lines are highly curved then it is possible that, when projected onto the line, some parts of the data may overlap other parts. If this is the case or if there is significant distortion because the line direction deviates from the projected line, you may have to use a 2D Graph. In this case you will use the Chordlength field as the X coordinate.



Example of a section traverse where a 2D Graph is required

Set the X scale of the section equal to the scale of the destination map. To display a section using scaling you need to display the document in layout mode. In normal mode all scaling is disabled.



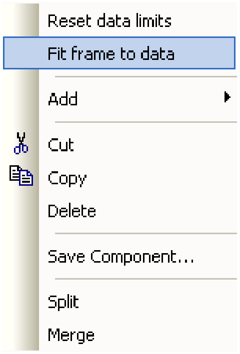
Now set the Y scale of the section. It is assumed there will only be one Y axis. If there are multiple Y axes the procedure is more complicated. Often, it is appropriate to choose a Y scale that is different to the X scale. The sections will be displayed on the map using this Y scale as the vertical scale. You will have to choose a Y scale that ensures that adjacent sections on the map will not overlap. To do this, work out the separation distance between lines. You can do this in Encom Discover PA using the measure tool.

With the distance known, use the following formulae to choose a valid Y scale:

1 mm at a scale of 1:X = X/1000 data units

1 data unit at a scale of 1:X = 1000/X mm

So if the lines are separated by 1000m and the scale is 1:10000 then they will be 1000 x 1000 / 10000 = 100 millimetres apart. If the section is 250m deep then a vertical scale of 100 * VX/1000 = 250 => 1:2500 would fit perfectly. In practice increase the scale slightly to allow some space.

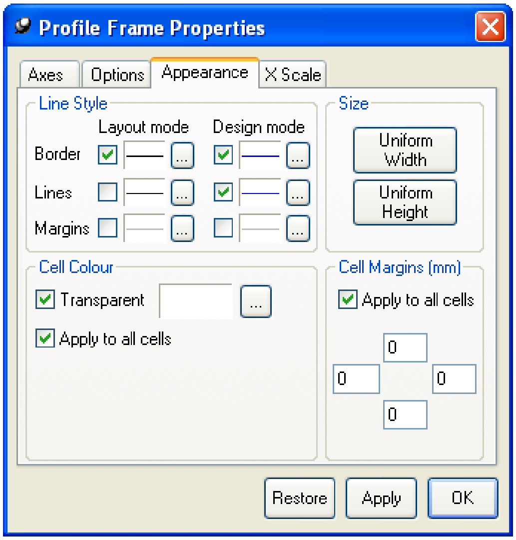


When you scale a graph in Encom Discover PA the graph data area may change size so that all the data is displayed at the appropriate scale. Often, the graph will then overlap the boundaries of the frame it is embedded in. This can be fixed manually by resizing the frame. Also, select **Fit frame to data** on the right click menu for the frame to automatically fit the frame (horizontal size only).

When you output the sections you have the option of outputting just the graph data area or the entire graph including titles, frames and axes. Note that this does not include any objects in the document that are floating over the graph. You will need to adjust the properties of these objects to ensure they display appropriately in the final destination map. As you change the section scale it is likely you will want to adjust font sizes, line thicknesses and other properties.

Note that the **Appearance** tab properties of the graph allow you to turn on or off the inner and outer borders and also allow you to change the margins of the data area. The margins are important if you want the top or bottom of the graph to align with the actual survey line on the map (assuming you select the hang or sit options in the Section to Map plug-in). If you are outputting annotations you may need to set the bottom or top margin to zero to ensure the data lies on the survey line.

Generally you will want the sections to be transparent on the map. In other words, you can see through the white space to the data underneath the section in the map. To ensure this is possible you need to set the **Show background** property in the graph **Appearance** property page off.



Deselect the Show Background option in the Profile Properties Appearance dialog to make the background transparent

Batch Print to Located Image (.EGB) File

Once the section is prepared you can print each line you require to a located image .EGB file. This process generates an ASCII EGB header file which contains the coordinates of all the sections and an image for each line. It is achieved via the **File>Batch Hard copy Wizard** menu option. Before activating the wizard ensure that you are viewing the section in **Page Layout** mode.

When using the wizard select the **Georeferenced image file on disk** (EGB) as the hard copy destination.

It is important to select the **PNG** (portable network graphics) option as the image type. This image file format employs a lossless compression algorithm ensuring file sizes are small and that there is no bleeding of information in the image. This bleeding will cause quality issues when the sections are displayed in the map.

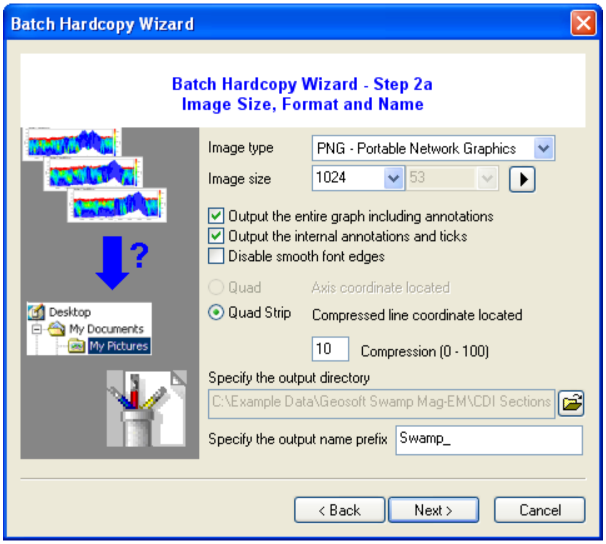
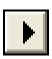


Image size, format and name dialog for the Batch Hard copy Wizard

The image size is an important consideration. If the lines are all the same length you can set a fixed size but if the lines have variable lengths then best results will be obtained if the size of the image is allowed to vary according to the length of the line (and possibly the height of the section).

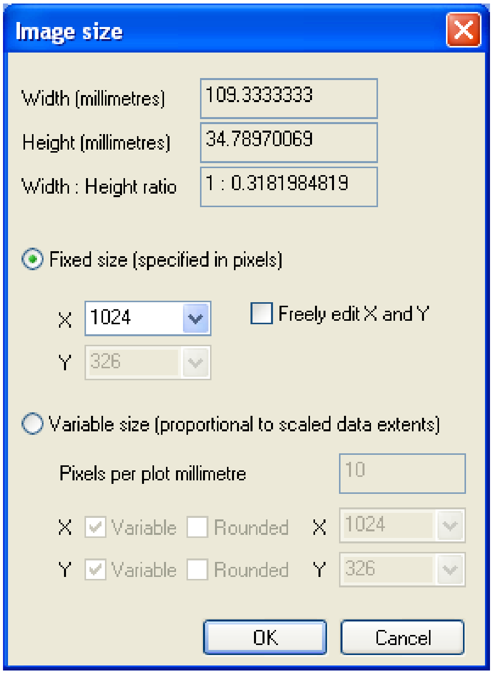
It is important to make sure the image has a high enough resolution so that quality hard copy results are obtained. However, it is expensive to set the image resolution higher than necessary as it taxes computer and graphics resources. It is also necessary to set the X and Y image size appropriately so that the X and Y scales of the section are honoured.



To set these options select the **Advanced** button, located to the right of the Image Size options, access the image size dialog. The dialog will report the width and height of the current section in millimetres as well as the width:height ratio. If all lines are the same length and all sections are the same height use the **fixed size** option.

Decide what resolution you want to use in the destination map. Resolution is often expressed in dots per inch and common figures include 150, 300 and 600 DPI. Your printer may report it can print to resolutions of 2400 DPI or more. In the Image size dialog pixels per millimetre are used instead, bearing in mind 10 px/mm = 254 DPI.

The length and height of your section in millimetres is reported at the top of the dialog. Otherwise you can compute it yourself using the formulaes given previously. Multiply these figures by the required resolution (eg 10 px/mm) to acquire the number of pixels required in the X and Y dimensions. Set the X and Y sizes in the dialog appropriately. You will notice that one of the dimensions is usually greyed out and is set automatically when the other is changed. You can override this by freeing the X and Y extents.



The Advanced dialog for setting Image Size

If line lengths or section heights are variable you can use the **Variable size** option. This requires you to set the formula that the system will use to determine the image dimensions or each output line. Firstly, set the required resolution in px/mm. Then, if the line lengths vary turn the **X Variable** switch on. If the section heights vary turn the **Y Variable** switch on. The image resolution will be displayed in the drop edit box (and will be greyed out if variable). If one of the dimensions is not variable you will need to select an appropriate resolution.

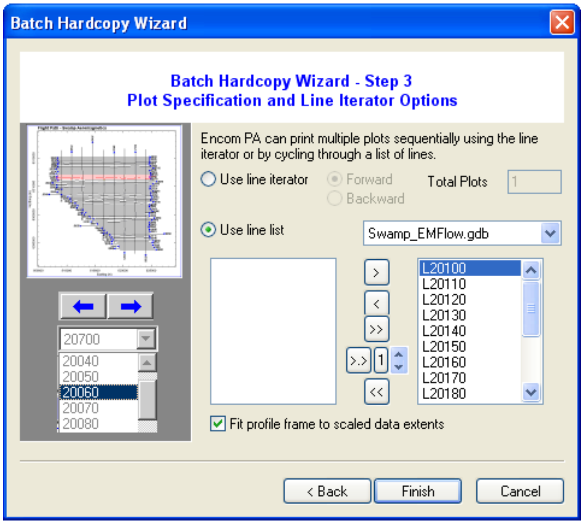
There is one final consideration regarding image size. The sections will be rendered using 3D texture mapping technology and for this purpose it is necessary that the images be a size that is a power of two, e.g. 512, 1024, 2048 etc. Generally, images larger that 1024 pixels will be tiled to NxM 1024 pixel size images. So, above 1024 you can choose a power of two or an integer multiple of 1024. Otherwise the image will be resized. This may involve re-interpolation of the images which can result in image detail and colour bleeding. A higher quality result can be obtained if no resizing is required. That is why the size edit boxes all offer a list of appropriate sizes. If using variable images size you can turn on the **Rounded** option to ensure the image is rounded up or down to the next appropriate size.

If you have decided to output the entire graph including axes, titles and annotations you can specify this in the batch hard copy dialog now.

If the sections will be displayed over the top of some other images – for example a topographic ECW or magnetic image - it is important to turn on the **Disable smooth font edges** switch. By default it will always be turned off. If not switched on then fonts will be anti-aliased (or smoothed) by adding grey pixels on the edges to make the text appear smoother. This will result in unfortunate image processing effects when the sections are displayed in maps and rendered transparently over other data. You will see light haloes around the text. If the sections will be rendered against an empty background then smooth fonts can be retained.

Output the graph as a **Quad Strip**. If you are outputting a profile or section there is no other choice. If it is a 2D graph then you will need to set this now. Also, ensure that line data is being exported rather than actual X coordinates which in the case of the 2D graph will be Chordlength. Finally, set the **Line Data Compression** (0 – 100). It can be difficult to know what level of compression is appropriate. You can experiment by opening a 2D map with a flight path. Choose the line dataset you are creating sections of. You can vary the compression level via the flight path properties and the effects will be instantly reflected in the flight path lines. Unless the line data is curved, a high level of compression of 50 – 100 is usually appropriate.

Select the lines you want to output from the line list. Make sure you turn on the **Fit profile to scaled data extents** switch. The wizard will now generate an EGB file containing all the coordinate information and a bitmap (PNG format) for each line. Check the images to make sure they look alright.



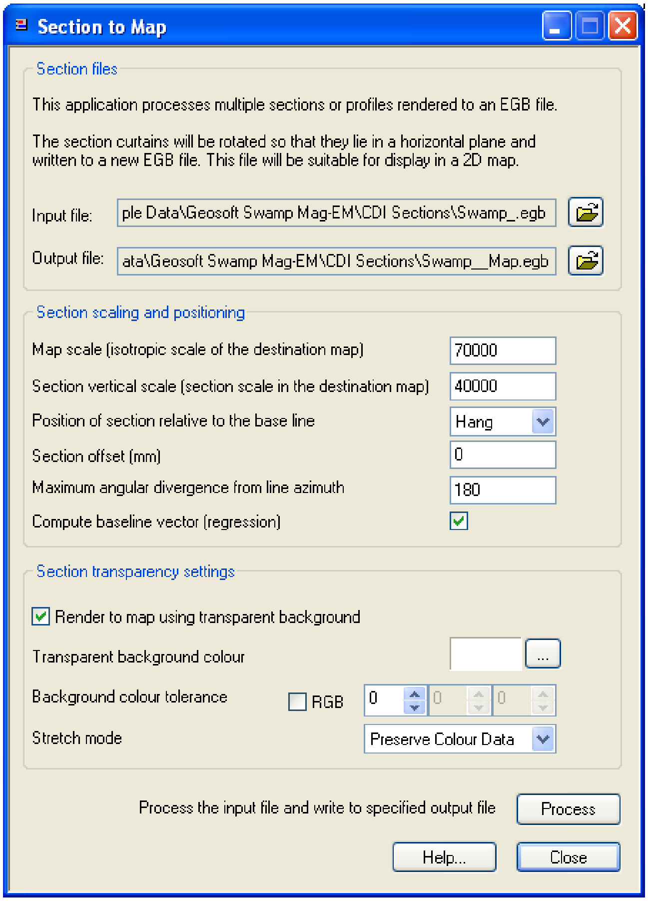
Choose the lines in an active database to be printed by using either the line iterator option of the line list in Step 3

Run the Section To Map Utility

The task of the **Section To Map** plug-in is to read the EGB file and convert the coordinates of the sections from vertical to horizontal. The images are not modified at all and will be re-used by the output located image (.EGB) file.

Access the **Section To Map** utility from the **Tools** menu. Select the .EGB file created by the batch hard copy wizard as the input file and set an appropriate output file. (Do not overwrite the input file as you may need to run the plug-in multiple times.)

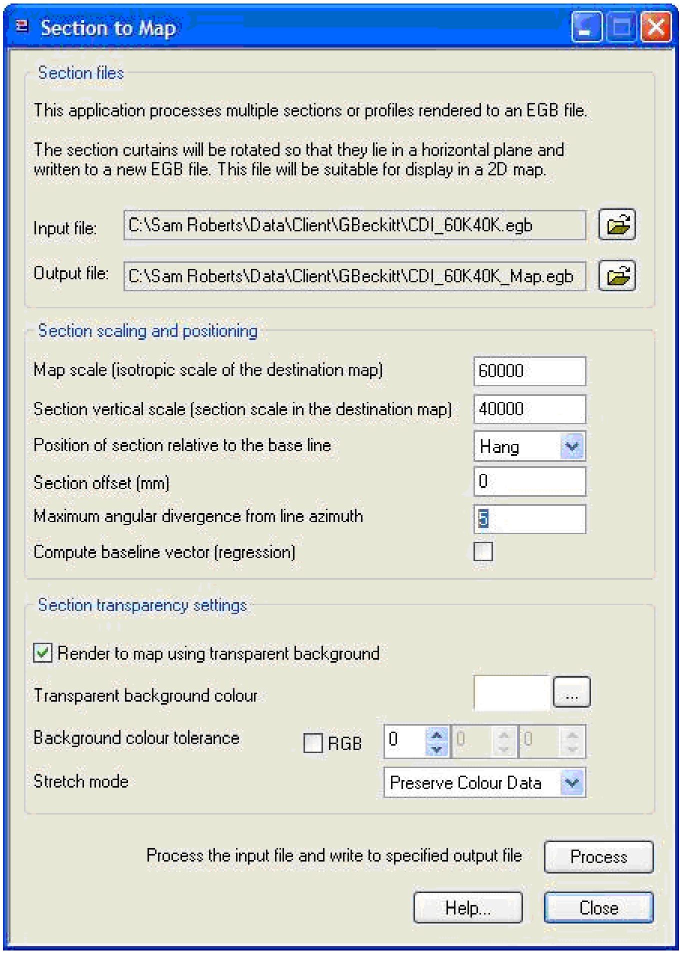
Enter the scale of the map (and horizontal scale of the sections) and the vertical scale of the sections in the appropriate edit fields. These are required to correctly compute the coordinates.



The Section To Map Utility dialog

To set the position of the sections relative to the survey lines you can choose to **Hang** the sections from the survey line so that they lie underneath it, **Sit** them on top of the survey line or **Straddle** (centre) them on the survey line.

You can also specify an additional offset in millimetres along the vertical axis of the section. The sections cannot be offset horizontally along the line. If you are hanging the sections and you have a top margin in the section you can align the data area with the survey line by setting the offset to the negative of the top margin size. This is usually 6 mm.



The Section To Map utility dialog.

To accommodate survey lines which are not linear in their orientation a 'Maximum angular divergence from azimuth" option has been added to the Section To Map tool.

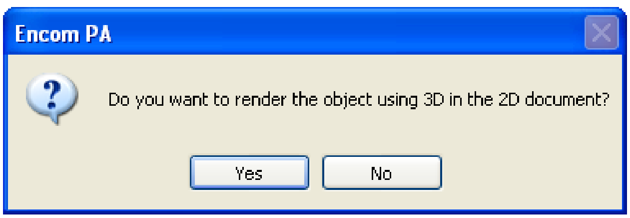
This ensures that the angle of the vector for each node does not depart from the line azimuth by any more than the user specified angle anywhere in the line. In this case we have a 'straight' line that is locally quite divergent from the global line azimuth. If you set the variable to zero in this case it is quite a reasonable thing to do – so the whole thing will be plotted at the line azimuth rather than the local azimuth.

Another option has been applied to the utility to re-compute the position of the baseline by projecting it onto a linear regression of the supplied survey line coordinates. This option is a check box called **Compute baseline vector (regression)**. By selecting this option any spatial deviations caused by local line azimuth deviations will be eliminated.

When the sections are rendered in the map they must be rendered into the map using transparent bitmaps. This means that a colour ought to be defined as the transparent colour. Any pixels that have this colour in the bitmaps will not be rendered. Generally this colour is white. You may have specified a non-transparent graph with a coloured background – if so use that colour instead. When colour bleeding occurs this can be partially cleaned up by specifying a range of colours that are transparent. You can do this by setting a range value for all colour components or for each colour component individually. Finally, the stretch mode chosen will also affect image quality. The **Preserve colour data** option produces good results in this case as it prevents colour bleeding.

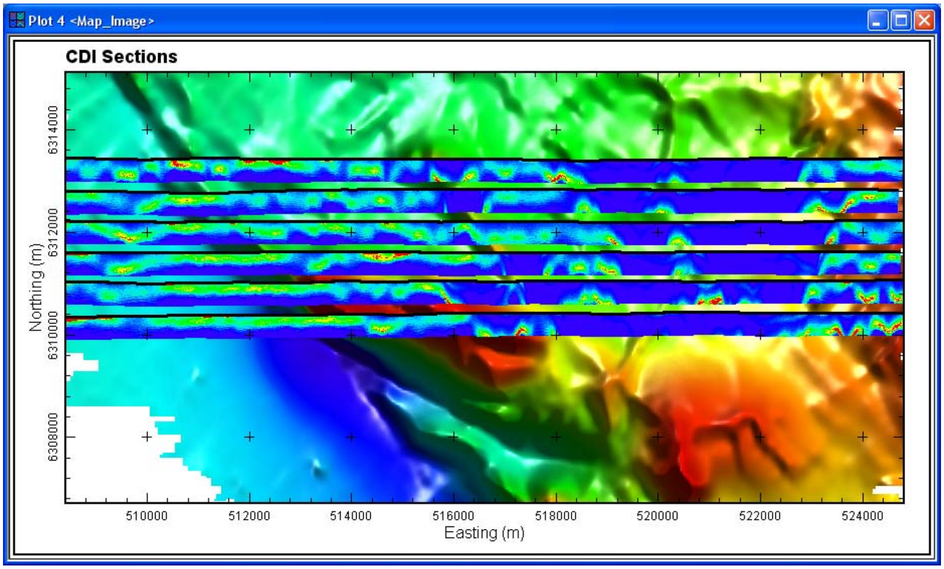
Add Processed Located Images to the Map

The easiest way to do this is to drag and drop the processed located image .EGB file from Windows Explorer into the 2D map document or else open it into the Data Manager window first before dragging the file into a 2D map window. When Encom Discover PA asks you whether you want to render the .EGB file using 3D technology click on the Yes button. Alternatively you can add a located image to the map and load the processed EGB file.



3D rendering option when adding processed sections to a 2D map

It can be useful to display a flight path in the map and to check that your sections are correctly located along the survey lines.



CDI sections displayed in a 2D map overlying a grid image and with reference survey lines

Once displayed in the map there are some options you can modify to improve or change the appearance of the sections.

1. You can move the located image series up and down in the Workspace Tree to ensure that is drawn on top or below other content. Encom Discover PA automatically categorises and sorts content to achieve good outcomes without user intervention.

2. You can change the bitmap rendering options. These are not exposed as properties of the series but are written into the .EGB header file at the top. The settings can be directly edited there. Alternatively run the Sections2Map plug-in again and modify the **Section transparency** settings.

3. You can change the image processing options. These are accessible via the series properties. You can change the input and output stretching options. Generally it is best to use **Preserve colour data** as the input stretch and **None** as the output stretch. However, these may not be the default settings. Ensure that you apply the changes to ALL images.

The map is now ready for printing. Alternatively you can export the map content to a located bitmap suitable for display in other packages such as MapInfo Professional. In this case it is best to use a located image format that employs lossless image compression.

Trouble-shooting

Most problems will be related to image processing options so it is useful to understand the full image processing procedure employed to get sections to maps. Other problems are likely to be related to scaling, image sizes and batch hard copy. Therefore you are advised to follow the instructions above carefully, to think carefully in advance about what you are trying to achieve and to check your work every step of the way. Likely areas where problems may occur are:

The batch hard copy wizard creates the section source image.

This process should not generate any image processing issues itself as the image data is created at or as a native resolution. However, if there are data series in the graph that render via source images themselves this may cause an issue. For example, the section may contain a ModelVision TKM model file displayed as a 3D view in the section. This is rendered as a bitmap into the section. By default text is rendered with smoothing turned on – this generates grey pixels around the edges of text which may be unsightly if rendered against a dark background. Turn it off.

The source images are converted to texture images.

This requires the image size in X and Y to be a power of two, or a multiple of the texture tile size. If this is not the case the image will be stretched up or down to the closest valid size. The stretch mode employed is important. The modes available are Preserve block data, Preserve white data, Preserve colour data and High quality. In most cases you would choose high quality but this mode employs pixel colour interpolation which can cause bleeding of colour and loss of apparent resolution.

The textures are rendered to the source geometry in 3D.

Texture mapping always involves scaling of the image up or down and consequently there must be some interpolation involved. In this case the best results are obtained if a nearest neighbour assignment scheme is used. Any kind of interpolation will cause sharp boundaries to become blurred and this can cause unsightly effects. The None mode uses nearest neighbour colour assignment.

The 3D viewport is grabbed and rendered into the map.

Generally speaking this will not involve any image stretching as the source bitmaps are usually produced at the exact resolution of the output device. However if the output device has a very high resolution there may be stretching of the source bitmap into the output device.

The images are usually rendered with a transparent colour so that content in the map can be seen through transparent areas of the sections. Colour bleeding can be cleaned up at this late stage by choosing a range of transparent colours.

If thin line work is disappearing from view this may be caused by the stretching or the texture mapping. Change the stretch to Preserve black data in the *Prepare the Source Sections or Profiles* step. Sometimes the lines are visible at high resolution but not visible on the screen. Use the Page Zoom In to see if this is the case. They may print correctly if this is the case.

Coordinate Reprojection

This plug-in utility is a geographic point converter allowing you to interactively convert points between any two coordinate systems that are supported in the Encom projection library.

Since the Point Coordinate Transform utility uses the same projection library as Encom Discover PA does, it is therefore also a useful way to explore coordinate system problems.

- *Interactive Conversion*
- *Text File Conversion*
- *Longitude to UTM Zone Converter*

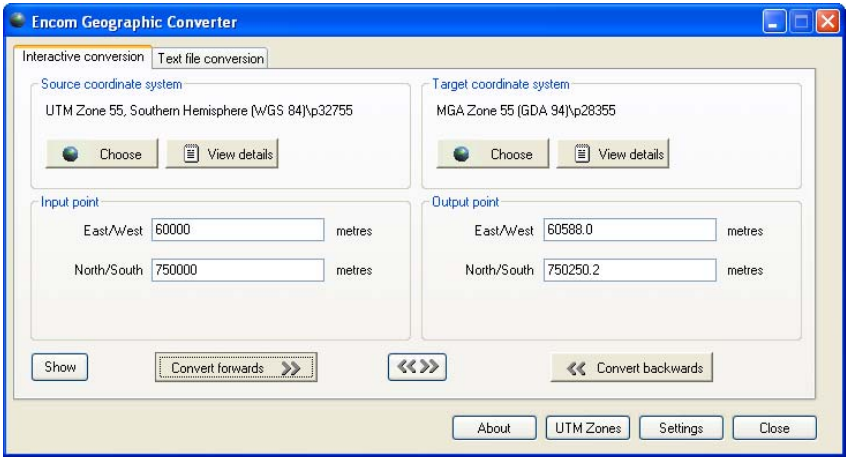
Interactive Conversion

After executing the **Point Coordinate Transform** utility from the **Tools** Menu a dialog containing two tabs will be displayed with the Interactive conversion tab dialog displayed by default. Load a source coordinate system (Datum and Projection) by clicking on the **Choose** button on the left hand side of the dialog.

A second coordinate system can then be loaded into the **Target coordinate system** section on the right hand side of the dialog by clicking on the **Choose** button for this section.

Differences between these two coordinate systems for a specific point location can then be compared by entering a value for East-West and North-South into the **Input Point** boxes on the left hand side and then clicking on the **Convert forwards** button.

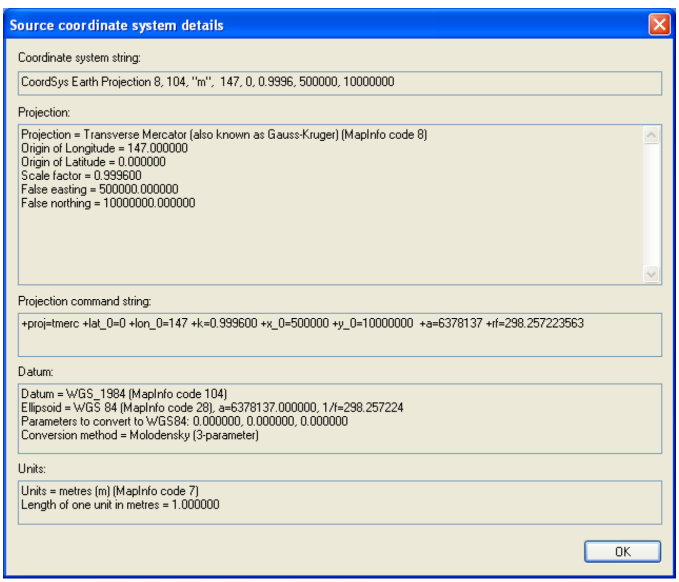
The **Output Point** entry boxes for the second coordinate system, displayed on the right hand side of the dialog, will then be updated to display the equivalent X and Y location of the input point in the second coordinate system.



Interactive Conversion tab dialog of the Point Coordinate Transform utility

This same procedure can be reversed by clicking on the **Convert backwards** button on the right hand side of the dialog.

Note With a coordinate system loaded you can click on the **View details** button to see the full set of parameters for the coordinate system.

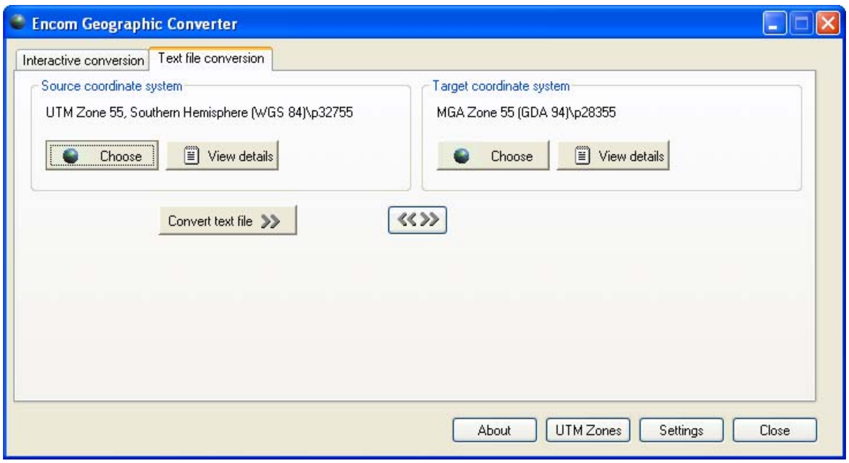


Details of a selected coordinate system loaded into the Point Coordinate Transform utility

Text File Conversion

The Point Coordinate Transform tool also allows “batch” conversion of coordinates in a loaded text file. This second feature is accessed from the **Text file conversion** tab dialog and utilitises the ImportAscii plugin utility with the appropriate source and target coordinate systems chosen by the Point Coordinate Transform function.

To load an ASCII file into the Point Coordinate Transform dialog click on the **Convert text file** button and browse for the text file using the navigation window that appears.

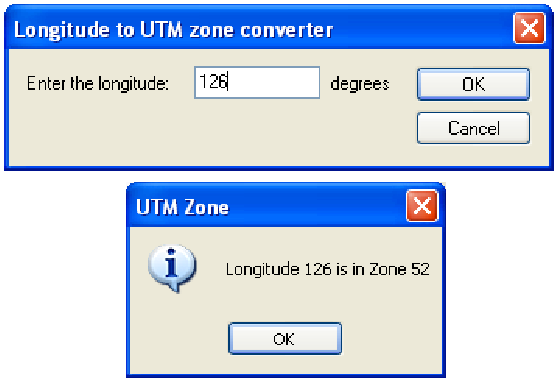


Text File Conversion tab dialog of the Point Coordinate Transform utility

Assign column header information and specify which fields are to be imported in the **Import ASCII** interface that appears and click on the **Convert** button. You will then need to save the converted file as a new ASCII file. The Easting and Northing fields will then be converted and updated in the newly saved ascii file and can then be imported into Encom Discover PA in the usual manner using the **File>Open>Text Import** menu option.

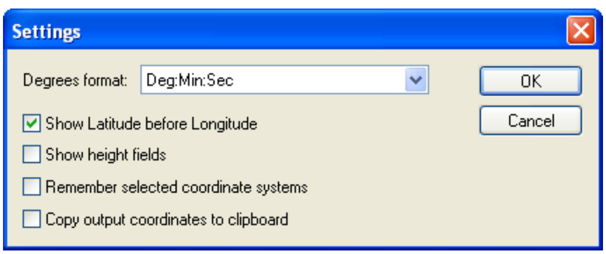
Longitude to UTM Zone Converter

A simple querying tool exists in this processing utility for locating in which UTM zone a specific Longitude degree belongs. Click on the **UTM Zones** button at the bottom of the Point Coordinate Transform dialog and enter the degree of longitude into the dialog that appears.



Use the UTM Zones button to query which zone a Longitude degree is associated with

Click the OK button and a smaller dialog will appear reporting the UTM Zone associated with this Longitude degree.



Select the **Settings** button on the Point Coordinate Transform dialog to apply a small number of settings that can be applied to this utility. Turn on the **Remember selected coordinate systems** option to have the utility remember the loaded coordinate system for the next time this utility is accessed.

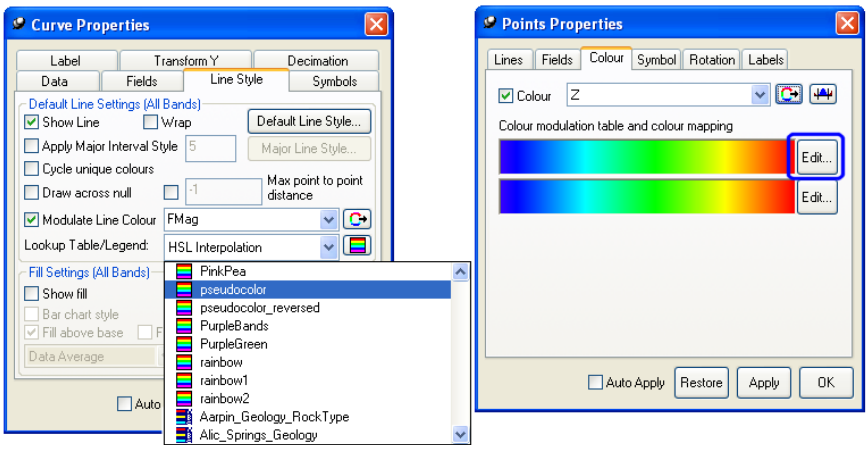
Colour Table and Legend Editors

Encom Discover PA has the ability to create, edit, save and restore Colour Look-Up tables and Legends that can be applied to surface, point or line data in all displays to enhance their appearance.

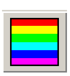
- A **Colour Look-Up Table** distributes colour across the displayed objects according to the value of a specified data field.

- A **Legend** is a different look-up table that uses textural or ranged data from a column in the data source table to create the legend categories. For example, legend categories may be displayed as solid colours or contain patterns that can be used to enhance a line styles, assay or lithologies in drillhole displays.

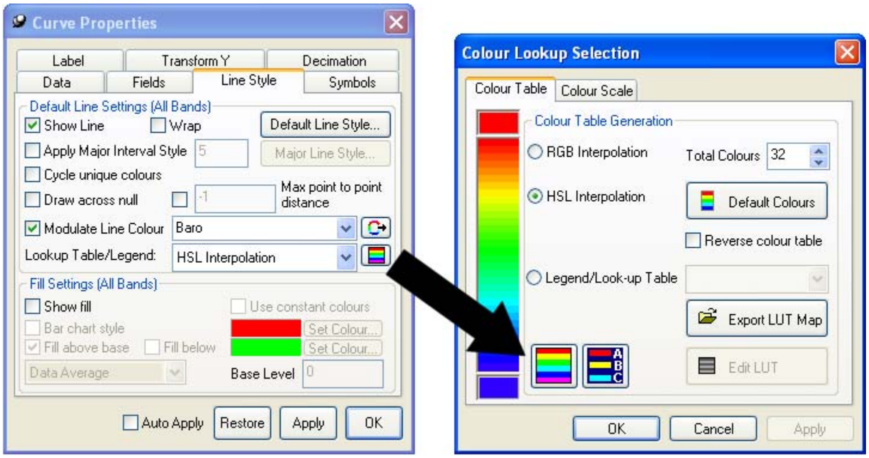
To use an existing Colour Look-Up Table or Legend check the **Modulate Line Colour** box in the **Colour** tab of the Points or Lines Properties dialog or the **Line Style** tab of the Lines Properties dialog and select a colour or legend table from the pull-down list. For further information on this, refer to the appropriate display object type section in this guide.



Curve Properties and Points Properties dialog showing access to Colour and Legend Lookup tables



To create or modify a look-up colour table or legend click on the **Lookup Table/ Legend** button. The **Colour Lookup Selection** dialog is displayed.



Colour Lookup Selection dialog

Choose the look-up table to edit from the pull-down list in the Colour Lookup Selection dialog.

If the look-up table selected is a **Colour Look-Up Table** click the **Rainbow** button to display the *Colour Table Editor*.

If the look-up table selected is a **Legend Table** click the **Aircraft** button to display the *Legend Editor*.

Look-up tables can be created, edited, deleted or imported from an external source via the LUT and Legend Editor dialogs.

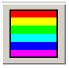
Colour Table Editor

Colour look-up tables can be created and modified using the Colour Table Editor. The look-up tables are stored in

`..\\Program Files\Encom\Common\LUT directory.`

The format of LUT files varies depending on the saved format type. Supported types include:

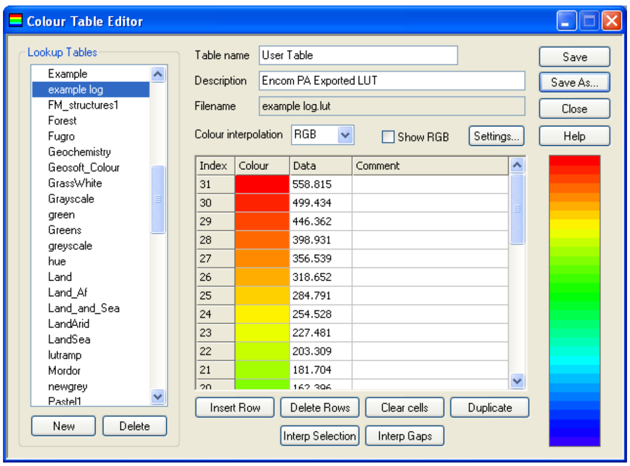
- ER Mapper (.LUT)
- MapInfo Professional (.CLR)
- Geosoft Oasis (.TBL)
- Engage3D Legend (.LEG extension)



To open the Colour Table Editor (LUT Editor) dialog click on the Lookup Table/ Legend button in the Points Symbols tab of the Points Properties dialog or the Line Style tab of the Lines Properties dialog.

In the Colour Lookup Selection dialog check the **Legend/Look-Up Table** control and select an existing Colour Look-Up Table from the pull-down list. Click on the Look-Up Table button to display the **LUT** Editor dialog.

The **Colour Table Editor** can also be accessed from the **Tools>Colour Table Editor** menu command.



LUT Editor dialog.

The Colour Table 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. For information on creating a new colour table, see *Create Colour Table*. 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.

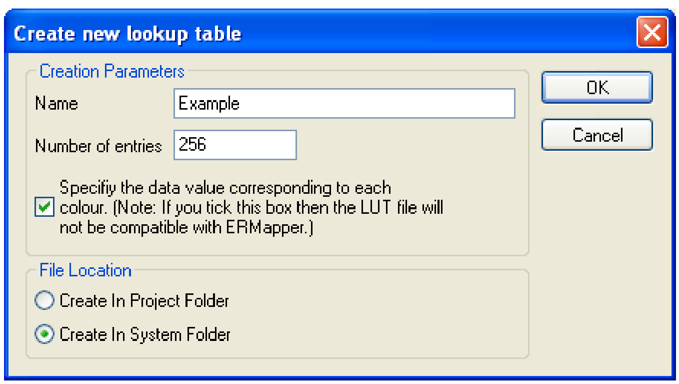
The lookup **Table name**, **Description** and the corresponding **Filename** are displayed at the top of the dialog. Underneath the Colour interpolation can be selected from RGB or HSL with an option to show the individual RGB values for each row. The Settings button shows the file path of the stored 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.

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

Create Colour Table

To create a new LUT click the **New** button in the **Colour Table Editor** dialog. The **Create new lookup table** dialog is displayed



Create new lookup table dialog

Enter a **Name** and the **Number of entries** (rows or colours) to be assigned to the new LUT. 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 of 51,000 to 51,500, the colour look-up table can contain 500 definable colours for each of the 500 data values. Note that if this utility is used, the created LUT cannot be used within ER Mapper.

When a new table is created it is added to the **Look-up Tables** list and a blank colour table is displayed in the centre of the LUT Editor dialog. Select **HSL** or **RGB** from the **Colour Interpretation** pull-down list. Left-mouse click any of the **Colour** cells of a row to select a colour from the standard colour palette. Use the **Custom** option to create additional colours.

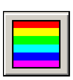
To create a colour spread over a range of cells colour and select two end member cells in non-adjacent rows by holding down the SHIFT or CTRL key. Click the **Interp Selection** button to fill the intermediate blank cells with graduated colour. Use the **Clear Cells** button to remove colour from highlighted cells. To view the Red, Green, Blue values for each row check the **Show RGB values** box.

Selected cells can be all set to match the first selected 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.

If the **Specify the data value corresponding to each colour** box was checked when the LUT was created enter the minimum and maximum data values next to the corresponding rows in the **Data** column. Click the **Interp Gaps** button to populate each intermediate row with the corresponding data value.

Any nominated LUT files in the displayed list can be deleted using the **Delete** button on the bottom left of the dialog.

Legend Editor


Colour Lookup Table


Legend Editor

The Legend Editor is used to create and/or modify individual or ranged data legends manually or from a selected dataset field. The Legend Editor can be accessed from a Properties dialog by clicking first on the **Colour Lookup Table** button to open the Colour Lookup Selection dialog and then on the **Legend Editor** button, or from the **Tools>Legend Editor** menu command.

The Legend Editor is used to maintain legend tables where each individual category or numeric data range has the following display attributes:

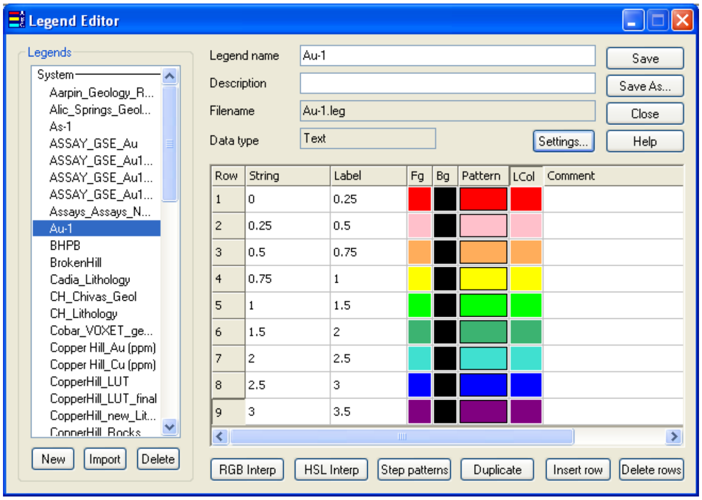
- Foreground colour
- Background colour
- Pattern

- Bounding line colour
- Bounding Line Style
- Bounding Line Thickness

Legend information is stored in a file with a LEG extension. Legend files are stored in the `C:\Program Files\Encom\Common\LUT` directory. In addition to the legend files, associated patterns are stored in the `C:\Program Files\Encom\Common\Patterns` directory.

Use the **Settings** button to view the current file path for the legend and pattern directories.

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.



Legend Editor dialog

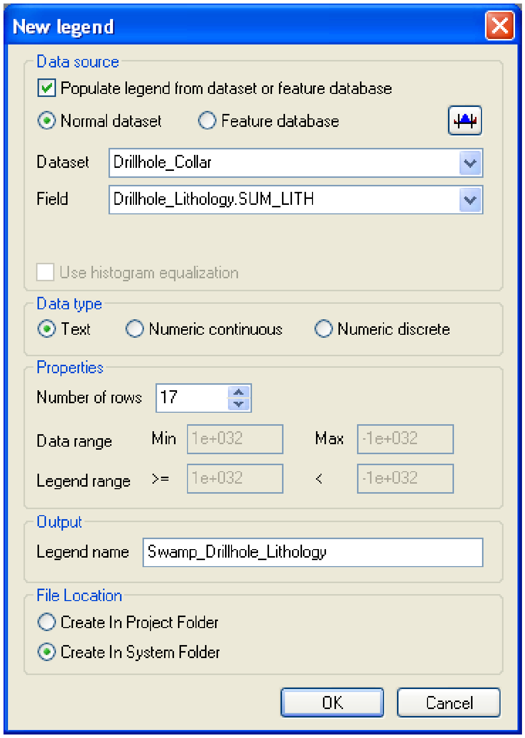
At the top of the Legend Editor dialog the selected Legend name is displayed with a Description, the Filename and the legend Data Type which may be text or numeric. Next to these entries are buttons to save and/or **Close** the dialog. The **Save As** button allows you to save a legend in a new folder and with a different filename if required.

For information on creating and modifying legends, see:

- *Creating a New Legend*
- *Modifying Legends*

- *Additional Legend Modification Tools*

Creating a New Legend



New Legend dialog

1. Click **New** button to display the **New legend** dialog.

2. To use an existing field in a drillhole downhole table as the basis for the new legend check the **Populate legend from dataset or feature database** box. Select the downhole table from the **Dataset** pull-down list and the data attribute column from the **Field** pull-down list. If an existing field is not used to create the legend, the number of rows and data range will need to be selected manually.

   - To create a legend with an even colour spread, select the option **Use histogram equalization**.

   - To condition or filter data from the automatic assignment select the **field data conditioning** option.

3. Select the legend **Data Type** from the following options:

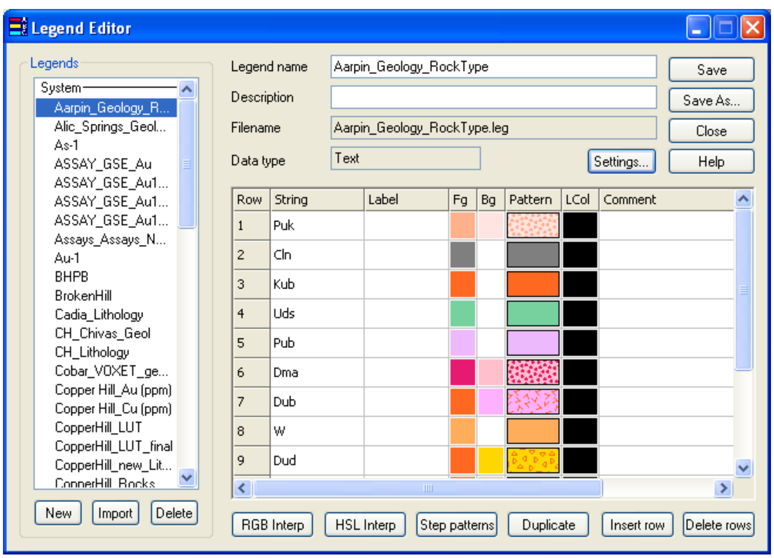
- **Text** – Individual textual categories
- **Numeric** – Numeric ranges whereby the upper limit of one range is continuous with the lower limit of the next range. Only values that are less than the maximum value for each range are displayed using the range colour. If using the **Populate legend from dataset** option the maximum value will need to be increased in order for the maximum value in the dataset to be coloured.

4. Select the **Number of rows** to display in the legend. If a text field is selected as a **Data source** the number of rows is automatically populated with the number of unique data entries.

5. Enter the **Data range** to use for numeric legends. The data range determines the minimum and maximum data values to display in the legend. The data range is divided by the number of rows to create the initial legend from and to values. If a numeric field is selected from a downhole table as a **Data source** the data range is automatically populated although this value can be changed manually.

6. Enter a **Legend name** for the output table. If a field is selected from a downhole table as a **Data Source** the legend name is automatically created by concatenating the downhole table and attribute field name together. This name can be overwritten with a user-defined name.

7. Click OK to display the legend in the Legend Editor dialog.



Legend Editor dialog with assigned properties, colours and patterns

Modifying Legends

Each legend is composed of a number of properties that can be modified at any time. To modify a legend select the legend from the list. The existing legend properties will be automatically displayed in the **Legend Editor** window. For each legend entry the following legend properties are available:

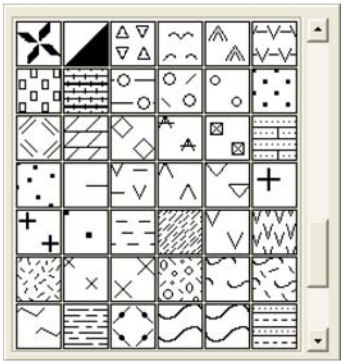
- **Row** – incremental row number
- **String** – a text string to describe the item of the legend row. Double-click to edit.
- **From (>=)** – Numeric legend minimum range value. Double-click with the mouse in the From (>=) column to modify the entry (Only available for Numeric continuous Data type).
- **To (<)** – Numeric legend maximum range value. Double-click with the mouse in the To (<) column to modify the entry (Only available for Numeric continuous Data type).
- **Fg (Foreground colour)** – Colour used for solid or pattern fills. Left-mouse click in the **Fg** colour box in the legend entry to display the colour palette.

  Select from any of the standard colours available or press Custom to create a new colour. In the **Colour** dialog select a colour from the **Basic Colours** palette. Click in the colour spectrum window or enter RGB or HSL values until the desired colour is created. Use the **Add to Custom Colours** button to add the colour to the **Custom Colours** palette. The **Fg** colour can also be set to **Transparent** to display data with this legend entry see-through.

- **Bg (Background colour)** – Colour displayed as solid background fill if a pattern is selected. Selection of background colours is identical to selection of foreground colours (see above

- **Pattern** – Pattern displayed for a legend entry. Left-mouse click in the **Pattern** box in the legend entry to display the available patterns. 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 `C:\Program Files\Encom\Common\Patterns`.



- **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 (Line style)** – Line style used for trace shade or histogram boundaries and structure ticks. Left-mouse click in the **LStyle** box in the legend entry to display the available styles.

- **LThick (Line thickness)** – Line thickness used for trace shade or histogram boundaries and structure ticks.

- **Comment** – a descriptive text entry for each legend row can be stored with the legend if required.

Additional Legend Modification Tools

Various operations can be performed over a selected range of legend rows within a column. Select the 1st 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.

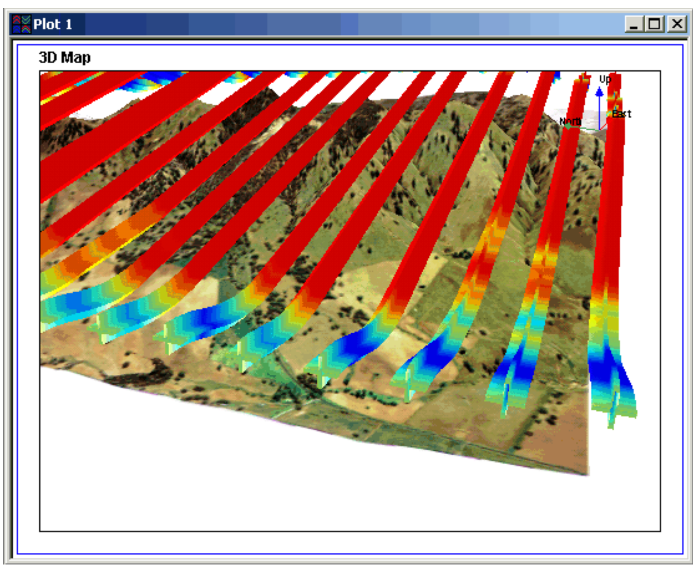
- **Step Patterns** - assigns patterns from the pattern list incrementally from the first selected pattern cell to the last selected cell.
- **Duplicate Patterns** copies the pattern of the first selected cell to the last selected cell.
- **Insert row** places an empty row before a selected row or cell.
- **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 After the legend is saved, deleted rows cannot be restored.

Tube Shape Manager

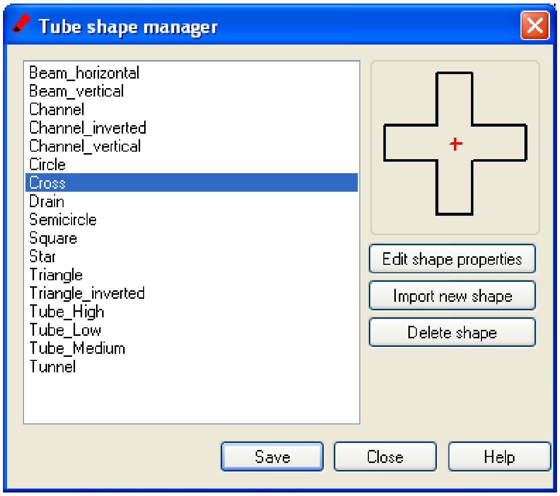
3D data lines can be displayed as tubes with a user-defined cross-section. The cross-section can be either open or closed and any number of tube cross-sections can be defined. Using the Lines data type in the Workspace tree, 3D traces of any linear data can be displayed. A description of displaying linear data as Lines in a 3D window is provided in *View Line Data in 3D Displays*.

An example of this is shown below where a cross shaped tube has been used to indicate a series of flight lines. A range of cross-sectional shapes can be used and these are created and manipulated by the Tube Shape Manager.



Example flight lines using a colour modulated tube shape Note the cross-sectional tube *shape*

The manager edits and manages tube cross-section shapes that are stored in an .ETS file. The dialog for the Tube Shape Manager appears as:



The Tube Shape Manager and selection of cross-section shapes

You can alter the name and description of individual shapes from the **Edit shape properties** button. Cross-sectional shapes are imported from MapInfo Professional.TAB or .MIF files. Remove any shapes using the **Delete shapes** button.

The specification of an .ETS shape file is described in *Appendix B: Data File Specifications*.

25 Database Utilities

The Database menu provides access to the following functionality:

- *Database Maintenance* – performs simple maintenance on Geosoft databases, e.g. delete or rename lines.
- *Build Array Field* (Array Channel Builder) – build multi-channel data (such as Geotem or EM data) from imported single channel data.
- *Depth Interpolation* – used to make vertical adjustments due to topography to depth values derived from interpretation processing of EM data.
- *Database Coordinates Reprojection* – provides the ability to reproject databases to different coordinate projection systems.
- *Geosoft Database Merging*– merge individual Geosoft databases to produce a single database.
- *Line Utility* – selection of tools for processing or manipulating Geosoft survey database files.
- *Line Calculator* – provides the ability to create new fields in a database based upon user-defined calculator expressions.
- *Line Filtering* – performs line-based operations including padding, filtering (convolution, FFT, and user-defined), null filling, noise adding, and threshold.
- *TEM Tau Calculator* – computes the time constant Tau value (and associated error data) along lines containing EM decay readings and then save the results in the database.
- *Flight Path Simulator* – create a simulated airborne survey Geosoft database from a DTM grid surface.
- *Airborne Quality Control Tool* – check that contractual obligations have been met in the collection of airborne geophysical survey data
- *Diurnal Correction Tool* – applies diurnal corrections to magnetic values recorded in a database.
- *Flight Lines to Vector File* – create a vector file of survey lines from an open database.

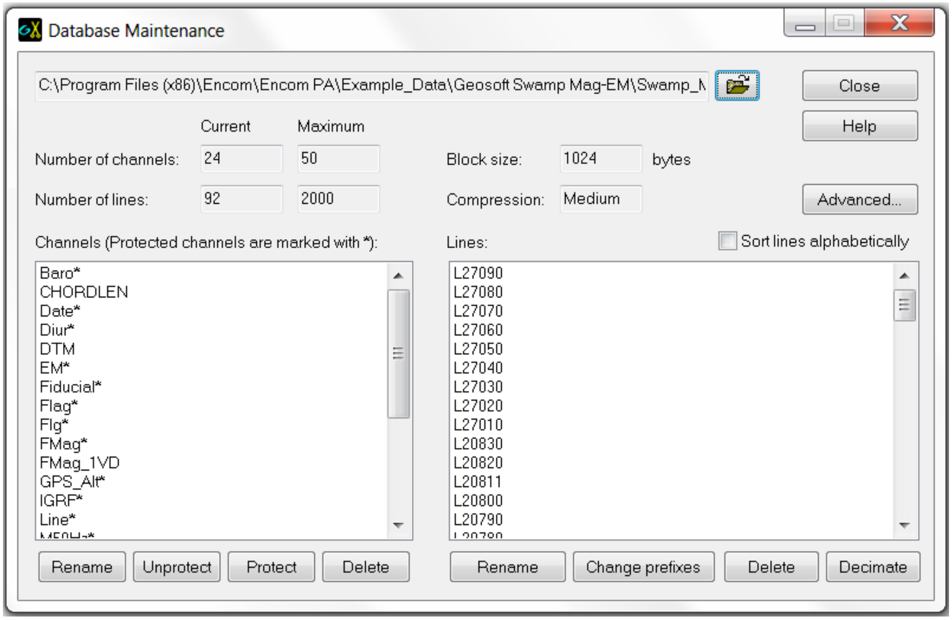
- *3D Symbol Generator* – converts individual depth and property solutions into 3D objects for display in 3D.
- *Stacked Profiles Tool* – creates a graph in 2D of a nominated channel displayed along a traverse base line.

Database Maintenance

The **Database Maintenance** tool is accessed from the **Database** menu and allows you to remove or rename data channels of a database or to delete data lines.

Note The tool operates independently of Encom Discover PA and can modify Geosoft Oasis databases. Geosoft databases can only be accessed by one application at a time, so you cannot make any modification to a database that is concurrently being accessed by Encom Discover PA. If you have a database open in Encom Discover PA and wish to use this tool, you must close the relevant dataset. A simple way of doing this is to use the **File>Close Dataset** menu option and select the database required.

Note Databases may have protected data fields. These fields are indicated in the Database Maintenance tool with an *. No change can be made (e.g. from the Line Calculator or with a Projection Transformation) while the relevant channel is protected. A protected channel can be used in say, the Line Calculator, but not altered. You can reset the protection of a data channel using the **Protect** and **Unprotect** buttons.

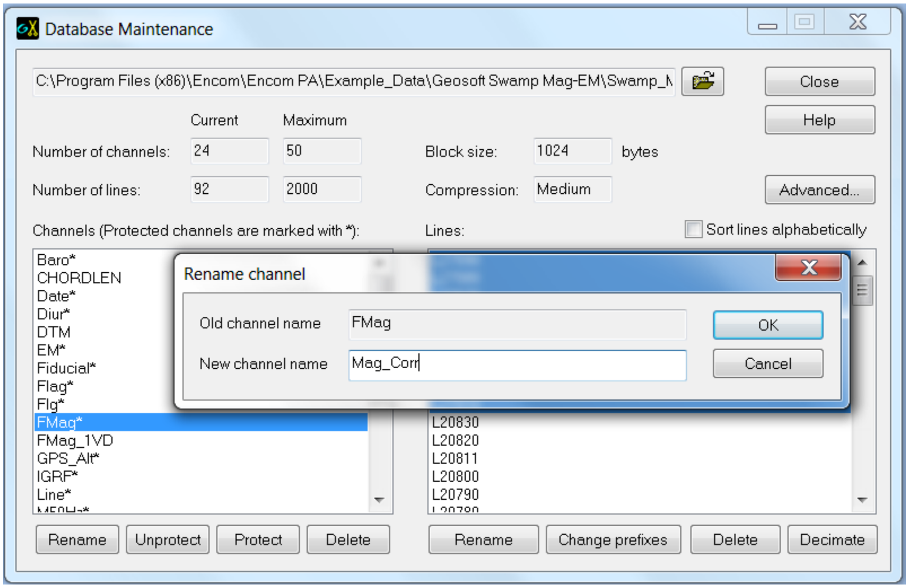


Specify the data channels and data lines that require renaming or deletion.

To use the utility, select a single dataset using the **Browse** button. The dataset is analysed and the number of data channels and lines displayed.

To rename a data channel:

Highlight a single field from the list in the left side of the dialog and press the **Rename** channel button. Specify a new channel name and click the **OK** button. This process alters the data channel name for all lines in the database.



The Rename channel dialog.

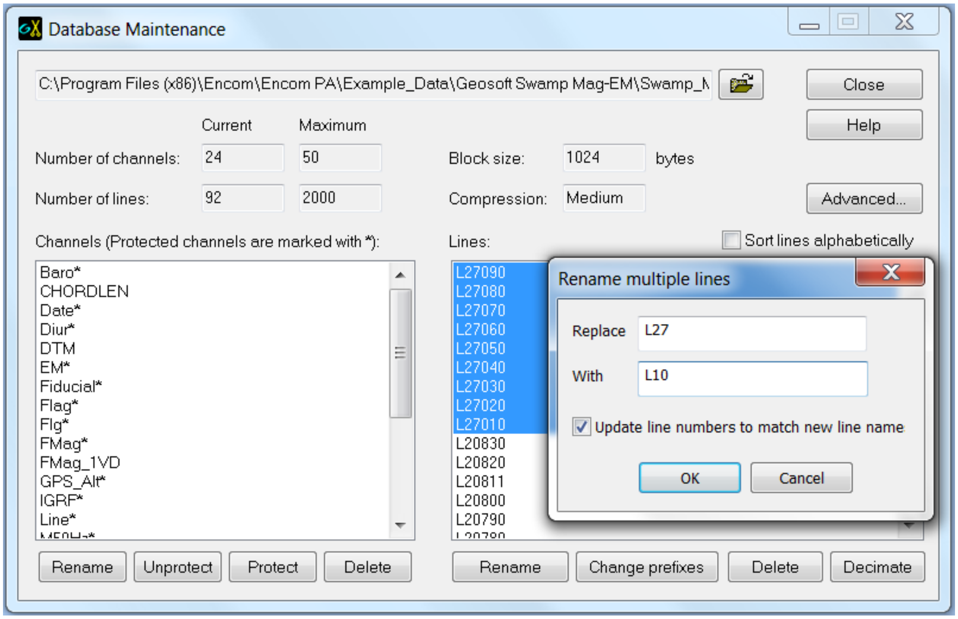
To delete channels:

You can multiply select the required channel names using the mouse cursor and the SHIFT and CTRL keys. Once selected, press the **Delete channels** button. You are asked to confirm the removal of the nominated data channels.

Important Deleted channels cannot be restored.

To rename lines:

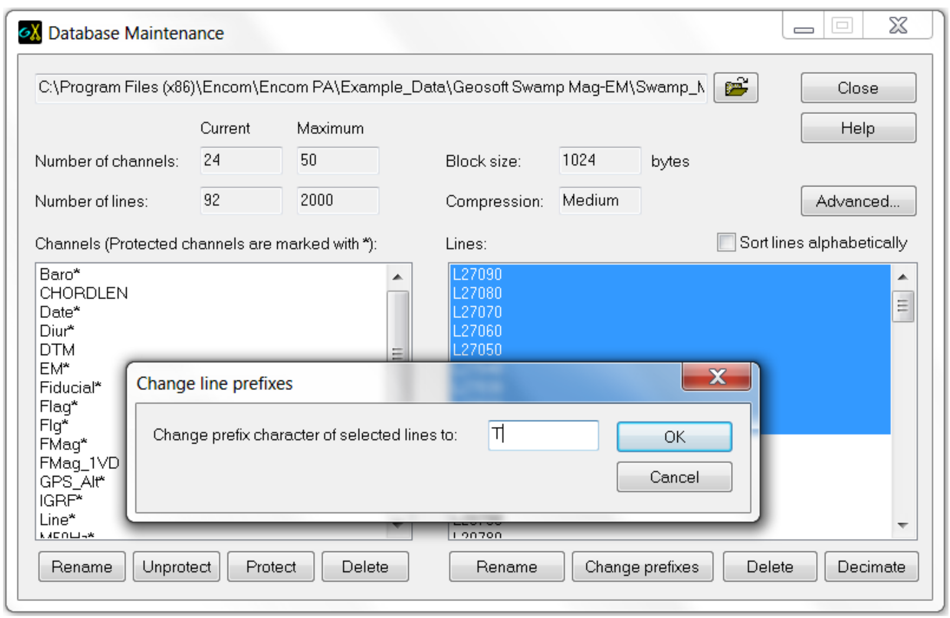
Highlight a single line or multiple lines from the list on the right side of the dialog and click the Rename line button. Specify a new line name in the dialog that appears and click the OK button.



The Rename line dialog.

To change line prefixes:

Press the **Rename** line button and specify a new line prefix in the dialog that appears and click the **OK** button.



The Change line prefixes dialog.

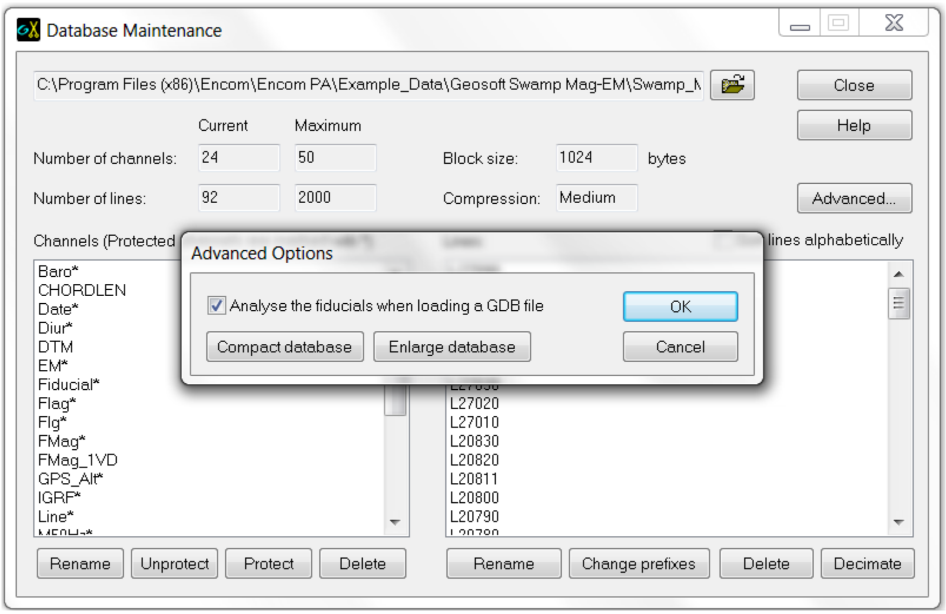
To delete lines:

You can multiply select the required line names using the mouse cursor and the SHIFT and CTRL keys. Once selected, press the **Delete** lines button. You are then asked to confirm the removal of the nominated lines.

Important Deleted lines cannot be restored.

Advanced Options

The **Advanced** button provides additional options on the maintenance of the loaded database.

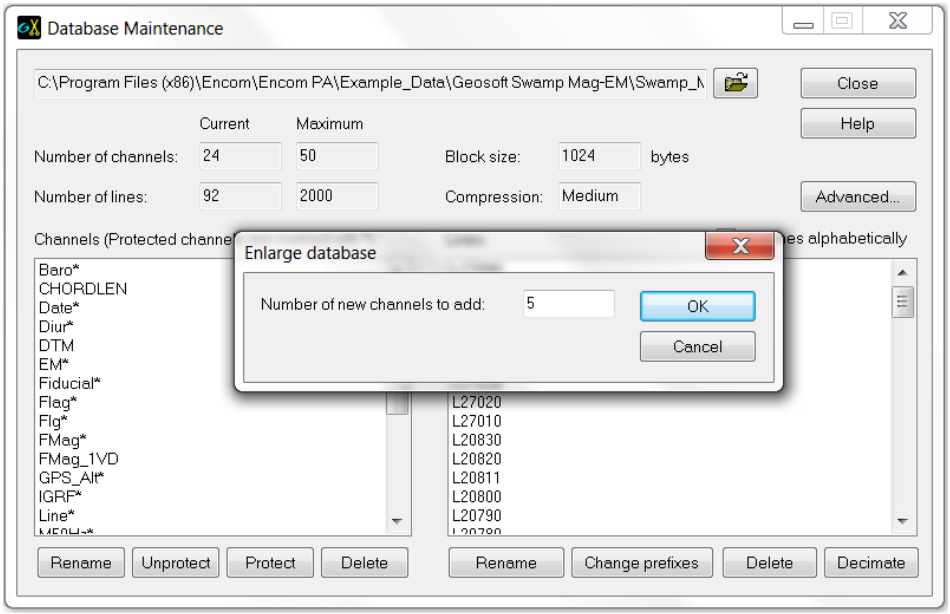


The Advanced Options dialog for the Database Maintenance utility.

The option to **Analyse the fiducials when loading a GDB file** instructs Encom Discover PA to analyse the fiducials of the database when it is next opened into the software.

Pressing the **Compact database** button will produce a compact copy of the original database, leaving the original unchanged. Compacting will remove unused blocks from where channels and lines have been altered or removed and will reduce the size of the database accordingly. When this button is pressed a summary report will be displayed, reporting how much compacting the original database will change the database size by and prompting confirmation that this operation is to be performed.

Pressing the **Enlarge database** button will display the Enlarge database dialog as illustrated below. This dialog allows you to increase the capacity of the database for adding new field (channels) to it by using other Encom Discover PA utilities, such as the Line Calculator, Grid Interpolation, etc.



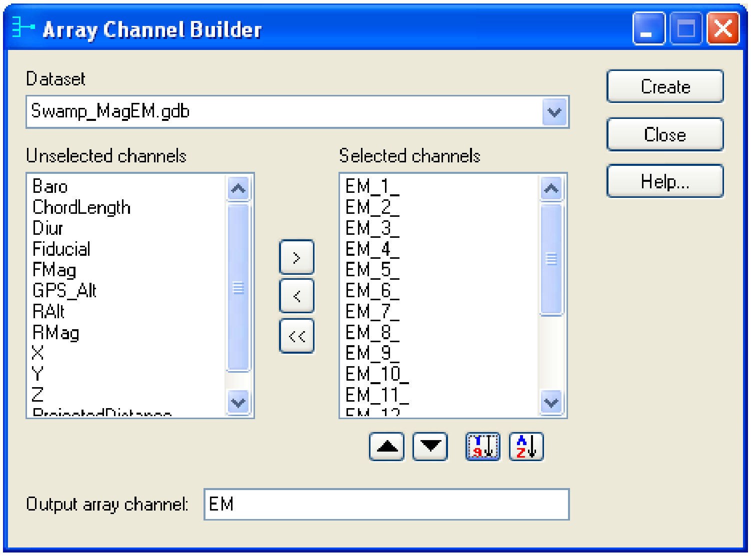
The Enlarge database dialog allows the specification of the number of new channels to be added.

Build Array Field

Typically, the use of Geosoft Oasis Montaj databases in Encom Discover PA requires that multi-channel data (such as Geotem or EM data) are imported as an array. The **Build Array Field** utility can join channels that have been imported into Encom Discover PA as separate data fields. These fields can then be saved as a new, single arrayed data field.

The Array Channel Builder can be run from the **Database>Build Array Field** menu.

The Array Channel Builder interface appears as:



Array Channel Builder dialog with individual channels selected

The Array Channel Builder requires the dataset containing the individual channels to be specified in the **Dataset** field at the top of the dialog. Available channels of the selected database appear in the **Unselected channels** list on the left hand side of the dialog.

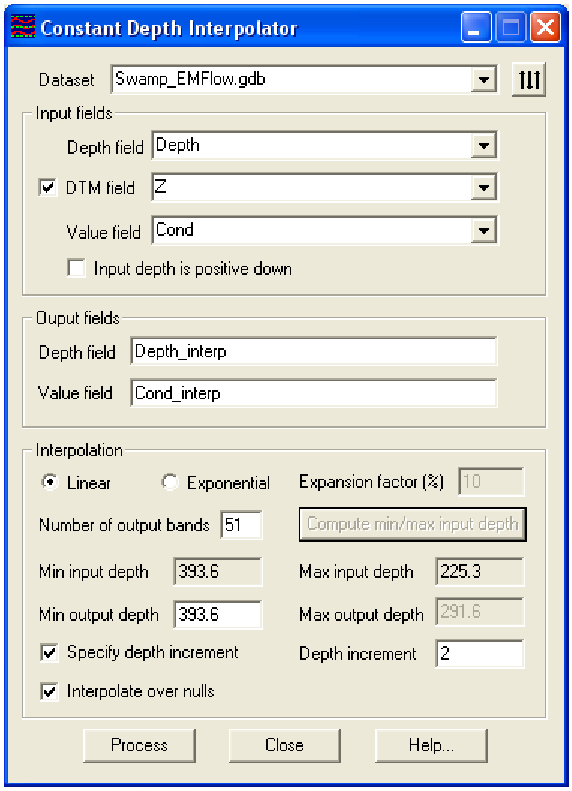
Channels from this list can be highlighted and moved to the **Selected channels** list on the right using the select channel arrow. The order in which the channels are displayed in this list indicates the order in which they will be presented in the array. These channels can be re-sorted individually using the **Move** channel arrow buttons. Alternatively, if you want the selected channels to be sorted numerically, click the **Numeric Sort** button. A third option is to sort the data channels alphabetically using the **Alphabetic Sort** button.

Depth Interpolation

The Depth Interpolator is used to make vertical adjustments to depths derived from interpretation processing such as EM Flow whereby data is contained in a database with depth and property arrays. The depth interpolation is required in the processing of such data to account for offsets due to topography and original data elevation corrections. Refer to *What Does the Interpolator Do?* This option is used as a preconditioning process prior creation of voxel models by using the voxel gridding module (EM Voxel). The constant depth interpolation corrects the resultant voxel grid for display.

The Constant Depth Interpolator can apply the interpolation operations directly to data fields of a Geosoft database. You can initiate the **Depth Interpolation** from the **Database** menu.

The Constant Depth Interpolator interface appears as:



Constant Depth Interpolator dialog

The parameters that can be specified on the control dialog are:

- **Dataset** – This entry details the required database to be used as input and to be modified. The database should contain arrayed depth and property data fields. Within the processing you can also select which lines are to be interpolated using the **Line Select** button.
- **DTM Field** – Specify the Digital Terrain Model data channel to initially create real depth values
- **Input Fields** – These entries specify the required Depth and Value or property specification field. These data fields should be multi-arrayed.

- **Output Fields** – The entry names are automatically derived from the input data fields, but they can be altered is desired. The program appends the text "_interp" after the specified input data field names.

- **Interpolation** – The depth and conductivity data fields (in the above example) are interpolated using either a linear, exponential or expansion factor (see below).

- **Output Bands** – The number of output bands usually matches the number of input bands but if a different depth increment level is required, you can increase or reduce the number of required bands.

- **Find min/max depth** – This button computes statistically the minimum and maximum depth values in the database and automatically populates the Output Depths. The values computed can be overwritten but this option makes estimation of output simpler.

- **Min/Max Input Depths** – These values are automatically computed from the database when the **Find Min/Max depth** option is used.

- **Min/Max Output Depths** – You can have the program automatically enter these fields by adopting the result of the **Find Min/Max depth** option or the depth values can be overwritten to a different data range. In this case the specified input data field is used to interpolate the output property values.

- **Specify Depth Increment** – If a uniform depth increment is required to be created, enable this option and specify an increment in metres.

- **Interpolate over nulls** – Where nulls occur in the input data, the property data are interpolated such that no gaps appear in the output gridded data.

What Does the Interpolator Do?

To prepare layered, array data for 3D gridding (using the Array Data 3D Gridding utility) the Constant Depth Interpolator is used. The Constant Depth Interpolator performs two processes:

- It converts the depths within the inversion database to real depths. In applications (such as EM Flow), the depths derived from the inversion processing are not absolute values. The depths require adjustment due to topography and the height of the aircraft/sensor above the ground and so correction should be applied to create real depths.

- With the real depths for each reading computed, these depths need to have their associated data values interpolated to flat layers such that the 3D Gridding can create correct depth voxel models.

If the first step of the processing is required, it is achieved by specifying a DTM or Z Offset data channel (use the DTM field option). Once specified, real depths are computed. In some inversion routines where true depth layering is derived, this stage of processing is not required.

The depth interpolation procedure adjusts the depths of the output data channel to have a constant depth with adjusted associated values for creation of the voxel model.

Interpolation Processing

For each reading of the specified database, a depth and property data field exist as arrayed readings. As the results are computed from the interpretation source program (for example, EM Flow), the depths may be offset vertically from one reading to another. Therefore, the Constant Depth Interpolator adjusts the depths for each reading up or down such that the final result has a value of the property for constant depths below a reference level.

The interpolation can be:

- **Linear** – An equal spacing of incremental steps between each depth value is computed.

- **Exponential** – An exponentially increasing depth between adjacent depth intervals is computed.

- **Expansion Factor** – This option increases the thickness (depth) increment for each property layer by a specified percentage. For example if the first layer were 10 metres thick and an Expansion Factor of 10% were used, the second layer would be 11 metres thick, the third, 12.1 metres etc. and expanding by 10% for each successively deeper layer.

Database Coordinates Reprojection

The Coordinate Transform tool allows the location data stored in a database to be re-positioned. Different Projection, Spheroid and Datum information can be specified from that originally stored in the database location channels and new positions computed.

When a transformation of position is performed, all associated data channels retain their original values. The database can have new easting and northing data channels created and added to the original database or the new positions can overwrite the original easting and northing values.

The options available from this tool are:

Dataset

With a database opened within Encom Discover PA, access the Project Coordinates tool and specify the required database.

X/Longitude and Y/Latitude Fields

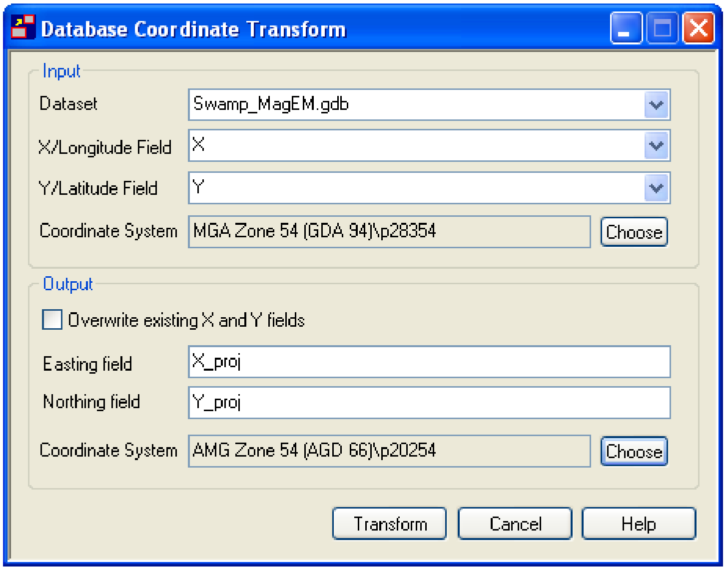
By default, you are required to nominate the X/Y or Longitude/Latitude data fields that contain the database position information. A pull-down list allows you to select this.

Coordinate System

Specify the input coordinate system from the **Choose** button. The **Category** and the **Members** of the Category allow selection of a wide range of datums and projections.

Overwrite existing X and Y fields

The original easting and northing data fields can be optionally overwritten. Alternatively, name new Projected easting and northing data fields. Specify the required output **Coordinate System** to use.



Project Coordinates dialog to nominate the database and specify new coordinate positions.

Warning If output Easting and Northing values are named the same as the input values and the **Overwrite existing X and Y fields** option is on, the original data values are overwritten. In this circumstance, the new data is then re-displayed and reprojected to the new, transformed locations in Encom Discover PA

When the coordinate information is specified, click the **Transform** button. The new database position data channels are computed and replaced or are added to the required dataset.

Geosoft Database Merging

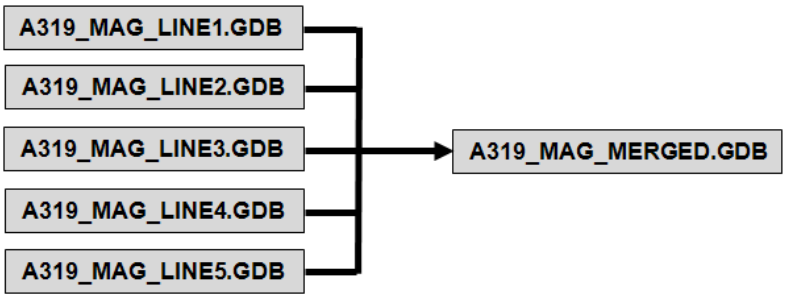
The **GDB Merge** utility is accessed from the **Database** menu and allows the user to merge individual Geosoft databases to produce a single database.

The GDB Merge utility allows the user to select all Geosoft (.GDB) databases to be merged and begins by reading the field list from each database and creates a new field list which is the superset of these.

The utility then reads each database in turn detecting the line names present in each. If two databases are found to have the same line name(s) present then the process will abort.

The utility then creates an output Geosoft database (.GDB) with all fields merged into it and finally reads the lines from the first database and inserts them to the new Geosoft (.GDB) database, filling any fields that are in the output database, but not in the input Geosoft database with default/null values.

The process is then repeated for the second Geosoft database, then the third database, etc.



The GDB Merge utility merges all lines and fields in multiple Geosoft databases into one Geosoft database.

The GDB Merge utility merges all valid fields and lines from multiple databases into one new database.

Line Utility

Database > Line Utility

The Line Utility contains a number of tools to process a survey (Geosoft) database. These contain tools for processing or manipulating database files. Each tool has a Properties (Settings) option dialog. There is also a description, and input/output file description for the selected utility.

To run a utility:

1. Select the utility.
2. Click on the **Properties** button.
3. Select the input file and select the options as required for the utility.
4. Click **OK** to close the properties dialog.
5. Click **Commence** to run the utility.

To run another utility, repeat steps 1–5.

Spatial Sort Database Along Line

The spatial sort utility will sort samples/stations in each line. Three sort methods are available -

- **Sort by selected field** – stations are sorted by increasing value of the selected numerical field. Typically this field will be the fiducial or station/ sample order.

- **Projection onto XY line** – for each line, the stations are projected onto a best fit straight line along the line's bearing. Stations are then sorted according to the order along this line that they occur.

- **Station proximity following** – stations are sorted, starting from the first point in the line, it will then calculate the distance against every other point in the line and place the closest station second. It then iterates to the second station and repeats the process for each station along the line. This is a very slow method, and is only recommend for non-linear lines or lines without a ordering/fiducial field.

Decimate by Station Rate

This utility will create a smaller database containing only 1 in every Nth station/ sample point.

Subset a Database by Lines

This utility will create a smaller database containing selected lines from the input database. You can select the lines individually, every Nth line or by a graphical map selection.

Clip a Database to the Area Within a Polygon or Rectangle

The clip utility will clip a database to the stations that are within a polygon (or group of polygons). This requires a vector file defining the polygons. Alternatively a bounding rectangle can be specified to clip too.

The output lines are automatically suffixed based on the polygon that they intersect, and what portion of the original line they are. The suffix is in format LineName.A:B, where A is the "piece" of the original line, starting from 0.

For example, if a single line is cut into three pieces, they are 0, 1 and 2, numbered from the start of the line. B is the polygon code, starting from 0. If there is just one polygon cutting the line into multiple pieces, then they would all be ":0". But separate polygons will have incremented numbers.

Compute IGRF Parameters

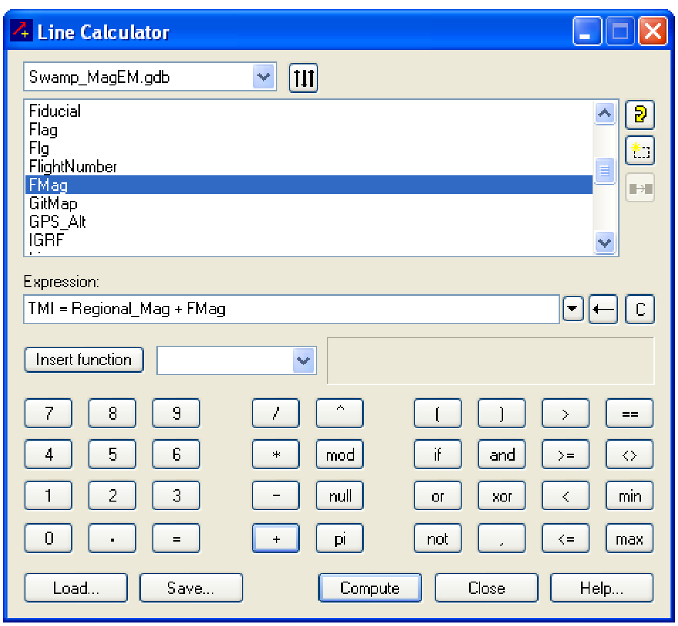
The IGRF utility will calculate the IGRF models inclination, declination and Intensity. The results for each station will be appended in new fields to the input database.

To allow the computation of IGRF values at each station, the stations location in X, Y fields and the projection of the units must be defined. Also a Z field maybe defined, or a constant Altitude/elevation used. As the IGRF model varies over time, and Time field or constant date must also be set.

Line Calculator

The Line Calculator can apply arithmetic operations directly to data fields of a Geosoft or Intrepid database, or between data fields of a database. You can initiate the Line Calculator from the **Database** menu.

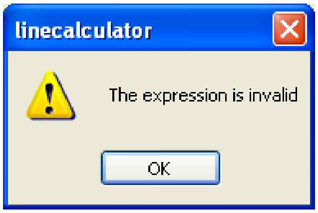
The Line Calculator interface appears as:



Line Calculator dialog with TMI computation formula

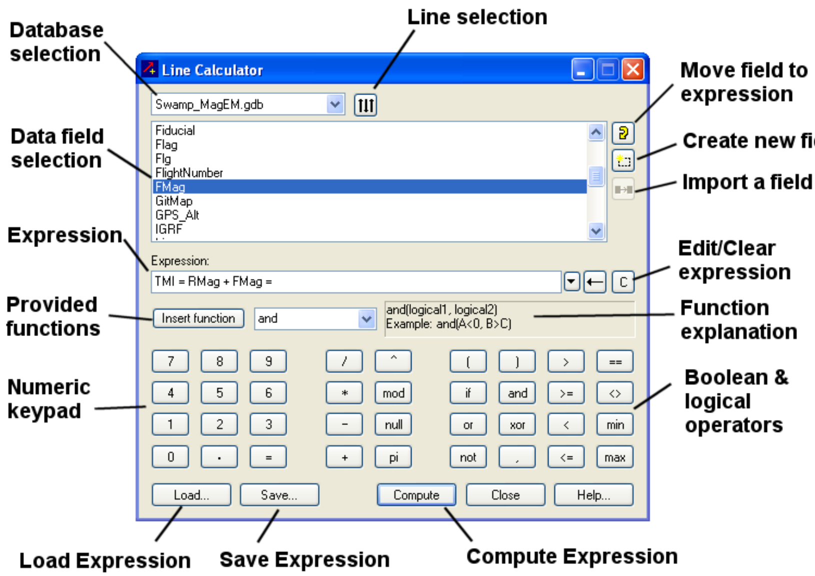
The Line Calculator requires a formula to be specified in the command line at the top of the dialog. Available data fields appear in the left scrollable list from the nominated Geosoft montaj or Intrepid database (GDB). Complex data field operations such Boolean, comparative, logical and decision functions can also be applied. These operations are described in *Calculator Syntax*.

Clicking individual operation's buttons causes their display to be automatically placed in the command line. You can either enter formula directly into the command line by typing from the keyboard, or by selecting the appropriate fields and buttons. Note in all cases that a correct arithmetic formula must be written. If the calculator detects errors in the command entered, a message indicating an invalid expression is displayed.



Buttons along the base of the dialog allow you to **Select** a field (after highlighting), **Load** or **Save** a formula, or **Compute** the displayed formula. The formula specification is as for normal algebraic operation. In the above example, a new data field (TMI) is to be created from existing data fields (FMag added with Regional_Mag). A new TMI data value is calculated and written into the database for every data reading of the database survey.

The **Insert Function** button allows you to construct complex formulae. A wide range of arithmetic, logic and comparative operations are available with the Line Calculator. A subset of the formulae are provided by the pull-down list that provides the various operators and an example of their use. Refer to *Special Line Calculator Operations* for a list of the operators available.



Line Calculator controls and button definitions.

Note too that the Line Calculator can be used for multi-banded, array field operations (refer to *Special Line Calculator Operations*). This means that arithmetic operations of arrayed data fields or between similarly arrayed multi-banded fields can be done

Note If a data field does not exist in a database, the Line Calculator creates one. Similarly, if you request a data field result from a formula but the field already exists, the existing data is lost and replaced with the recalculated data. You can also create a new data channel from the **New Field** button. When this is selected you can assign a new name, and if the new channel is to be multi-banded (arrayed), also specify the number of channels to be assigned.

Note Output channel names should be chosen with caution. The Line Calculator has a number of reserved words that should not be used for channel names.

- *Reserved Channel Names*
- *Special Line Calculator Operations*
- *Array Operations*

Reserved Channel Names

ABS	ACOS	ASIN	ATAN	COS
EQV	EXP	LN	LOG	SIN
RAND	SQRT	TAN		

Special Line Calculator Operations

Some operations with the Line Calculator can be applied for specific uses using specific command line controls. For example, if you wish to create a new channel that contains the number of the channel (eg EM multi-banded data) for labelling each trace in a profile), there is a special operation for this.

For example, you could create a multi-banded field called `Ch_No` which has 10 bands using the **New Field** button.

Example 1

To use a new channel for labelling traces of a multi-banded dataset, create a new channel using the **New Field** button and call it `Ch_No` but have it contain the same number of bands as the channel to be labelled. Now set an expression in the Command Line of the Line Calculator that says:

```
Ch_No=_band
```

This populates the `Ch_No` channel with the numbers 1, 2, 3, 4 etc up to the number of bands in the initial nember set when created..

Example 2

If you type:

```
Ch_No=_band/_bands
```

then for every line and station value band 1 of `Ch_No` will contain 1/10 = 0.1, band 2 of `Ch_No` will contain 2/10 = 0.2, band 3 of `Ch_No` will contain 0.3, ..., and band 10 of `Ch_No` will contain 1.0.

Example 3

If you type:

```
Test=_station
```

then a field called `Test` will be created which for each line contains the station number. e.g. `Test` for Line 1 Station 1 will contain the value 1, `Test` for Line 1 Station 2 will contain the value 2, etc.

Note that:

Variable	**Function**
`_band`	Creates an increasing number based on the number of array values of the requested field (eg `Ch_No` of 1,2,3,4...).
`_bands`	Creates a constant number entry of the total number of array values in the field (eg `Ch_No` of 24, 24 24, 24 etc).
`_station`	Creates a station number for each value entry of the field.
`_stations`	Creates a total number of stations of each line.

Array Operations

For use with the Line Calculator, array data expressions can be used. The array index of zero [0] is not used. An array index one [1] is the first element in an array.

Syntax

```
Output = Ch1[m] operator Ch2[n]
```

where the Output data channel (a non-arrayed channel) will contain the combined operator result of the mth band of channel Ch1 with the nth band of channel Ch2.

Example 1

```
ZCh_ratio=ZCh[1]/ZCh[2]
```

Example 2

```
Spec_Ratio = K[215]*K[168]/Tot[32]
```

Line Filtering

The Line Filter module is a powerful tool that provides the following functionality:

- Numerous line-based operations including padding, filtering, null filling, noise adding, and threshold.

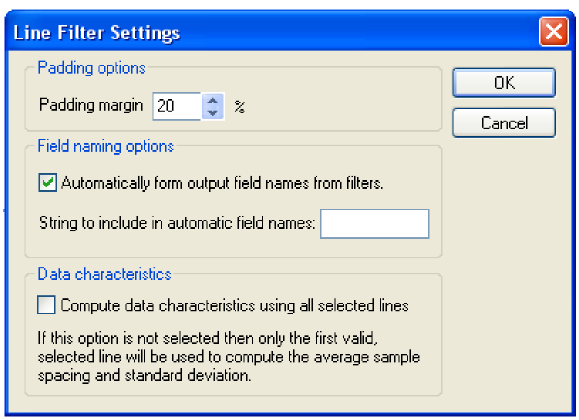
- Convolution line filters can be applied including standard, Gaussian, average and user-defined filters.
- FFT filtering using sophisticated operators to apply geophysical filter results.

Access to the Line Filter module is from the **Filtering>Line Filters** menu item.

All operations of the Line Filter module are applied to individual reading values of a specified data channel of an open dataset. Padding can be applied to fill nulls in the data and to the ends of the line to avoid end effects from the filters. Nulls are skipped in the processing but optionally transferred through to the output channel in the same location as for the input channel.

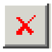
There is no option to apply an operation that overwrites the original, input data channel. You can, however, overwrite a previously created output channel. This is useful when resetting filter parameters and wanting to update processed results.

The Line Filter dialog presents selection fields for the dataset, input data channel plus output channel. A Line Selection buttton is available beside the Dataset selection field that permits filtering operation to only be applied to the chosen lines. As filters are added and applied, the initial channel name is used plus the applied filters to result in a created channel name. If you wish to rename or specify the output channel name, click the Settings button and unselect the option to **Automatically form output names from the filters**.

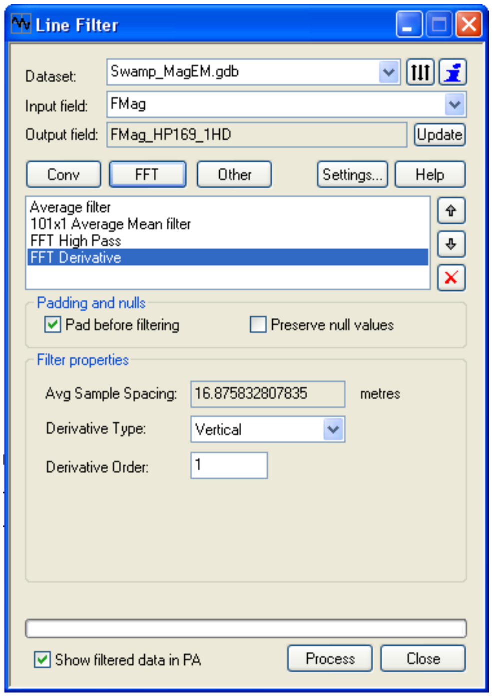


Line Filter Settings dialog

Line filter operations are cumulative in their effect. This means that as a filter type is selected, its presence is listed in a scrollable list. Processing (when applied by clicking the **Process** button) is applied to the data using the first operation in the listed sequence. When completed, the second selected operation is applied to the output of the first, as so on through the selected operations.



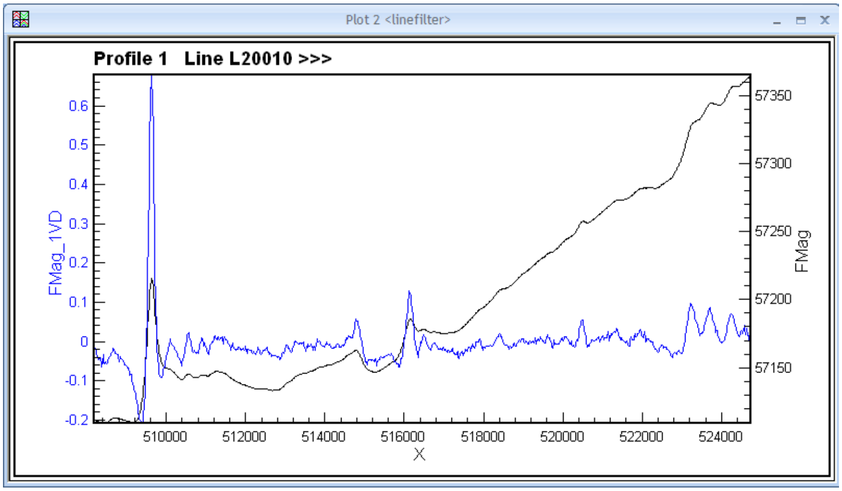
When multiple operations are displayed in the list, their order can be controlled by the **Up** or **Down** arrow buttons after selecting the filter or operation. A selected filter can also be removed using the **Delete** button.



Line Filter dialog with listing of selected filters and operators

When applying line filters, you can choose to pad before filtering to remove end effects and to assist the FFT process. This is achieved by selecting the **Pad before filtering** option. The amount of padding is set in the Settings (accessed by clicking the **Settings** button)

After processing is completed you can have the original and filtered output channels automatically displayed in Encom Discover PA. To do this, enable the **Show filtered data in Encom Discover PA** option. A new profile display window is opened that has one Profile and two Curve branches specified in the created workspace. One of the Curves has the original specified data channel and the second has the filtered output.



Profile display after processing with the Line Filter.

Line filters are grouped into the following categories:

- *Convolution Filters* (**Conv** button)
- *FFT Filtering* (**FFT** button)
- *Other Filters* and Line Filter Utilities (**Other** button)

Buttons are provided to select the various filtering operations from each of these groups.

Convolution Filters

The convolution filter operators define a 1 x n convolution matrix that is convolved along the data of a line. The convolution filters operate by passing a weighted kernel function across a window of data. The kernel weights are displayed in the dialog. For the User Defined filter the filter weights can be edited. Filter weights are normalised during the filter application.

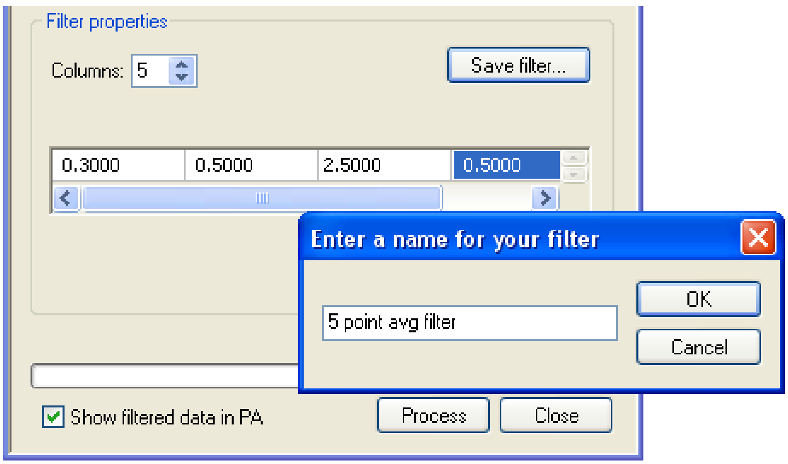
Convolution filters provided include:

- User Defined
  - General user-defined weightings – the user inputs weights a specified number of columns (stations). Weights are normalised during filter application such that the sum of weights are 1.
  - Averaging filter – weights of 1 are assigned over the user specified number of columns (stations).

- Gaussian filter – weights are assigned as a Gaussian function over the user specified number of columns (stations).

- Average – Select from several predefined averaging filters. Filter weights are all 1.
- Gaussian – Select from several predefined Gaussian filters. Weights are assigned as a Gaussian distribution.

The above filters provide examples of convolution filters and how they can be created. If you wish to create a specific filter and so have weights that you assign, select the user-defined **General User Defined** filter. This filter allows you to specify the number of data values in its window from the **Columns** entry. Each cell containing a weight using this filter type can be modified by positioning the cursor over the cell and double clicking the left mouse button. Modify the data value to edit the weight. Once a series of weights have been defined, you can save the filter using the **Save Filter** button.



Specify the number, weights and save the filter for re-use

FFT Filtering

Fast Fourier Transform (FFT) Filters are selected from the **FFT** button. After a filter is selected, it is added to the process list and its properties are displayed to allow you to enter the control parameters appropriate to that filter. Fast Fourier Transform filters provide an alternative method to convolution filters for transforming data.

The Fast Fourier Transformation operation performed in the Line Filter module uses the theory described by Singleton, (1969 - An Algorithm for Computing the Mixed Radix Fast Forward Fourier Transform IEEE Trans Audio and Electroacoustics, AU-17, 2, p.93.). A detailed description of the FFT one dimensional filters is provided in *Appendix C: Grid Filters* (One Dimensional FFT Line Filter Descriptions).

The major advantage of the FFT approach is its ability to look at the energy spectrum and to more efficiently process long wavelengths.

A disadvantage often associated with FFT is that it normally uses datasets with lengths that are powers of 2. To avoid this constraint, a mixed radix FFT that is based on prime factors up to a maximum of 23 has been used. With this algorithm, an FFT can normally be performed on a dataset length that is within 1 percent of the total data length. This computation and adjustment can be made by padding the data lines before processing. To do this, ensure the **Padding** option is enabled before using FFT filters. The degree with which padding is applied (as a percentage) is specified from the **Settings** button.

Selection of a FFT filter.

Note New data channels may be created for the forward transformation process but, as for convolution filters, care must be taken in naming output channels since newly created channels can be specified to overwrite previously generated data with the same channel name.

Frequencies for FFT filters are specified in a range of cycles or as wavelengths (in distance). Cut-off rates control the sharpness of the tapering of the energy spectrum. A high value for the cut-off rate will taper the spectrum more sharply, but may cause ringing on the edges of large amplitude changes.

- **Low pass** filters pass wavelengths longer than a specified wavelength. The controlling parameters required are the wavelength cut-off and the cut-off rate. All wavelengths smaller than the specified wavelength are filtered.

- **High pass** filters pass wavelengths that are shorter than the specified cut-off wavelength. As for low pass filters, the wavelength cut-off and cut-off rate are required.

- **Band pass** filters pass frequencies that lie between two specified wavelengths. Information must be entered for Short Wavelength cut-off, Long Wavelength cut-off and the cut-off rate (between 1 and 15). The result of this is the same as applying a low pass and then a high pass filter using the same cut-off frequencies.

- **FFT Derivative** filters are computed using a definable order and the choice of Horizontal or Vertical. Ensure the correct derivative filter is chosen from the pull-down list.

- **Continuation – Upward and Downward** filters allow either magnetic or gravity data to be filtered in such a manner that it appears to have been collected at a greater or lesser height than was actually the case. This is equivalent to a low pass filter operation. The height of continuation is specified in metres distance. When using the Downward continuation filter, care should be taken not to continue below the top of the shallowest magnetic or anomalous density contrasts, as in this circumstance the process becomes unstable.

- **Reduction to the Equator** performs a transformation on magnetic data to make it appear as though it were recorded at the Earth's equatorial pole. Information is required on the direction of the Earth's magnetic field (inclination and declination in degrees) and the direction of remanent magnetization (if any). Field inclinations should be specified as negative for the Southern Hemisphere.

- **Reduction to the Pole** performs a transformation on magnetic data to make it appear as though it were recorded at the Earth's magnetic pole. Information is required on the direction of the Earth's magnetic field (inclination and declination in degrees) and the direction of remanent magnetization (if any). Field inclinations should be specified as negative for the Southern Hemisphere.

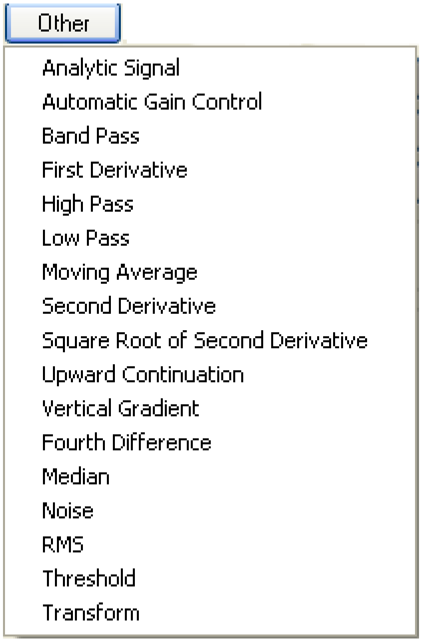
- **Pseudo gravity** filter can be computed from a total field magnetic data channel in a manner similar to that of reduction to the pole. Magnetization Type can be specified as Induced or through defining an Arbitrary Magnetization Direction. The information required is the Earth's field inclination and the field declination in degrees. Default Density and Magnetic Susceptibility values are also required to be used in zones where the pseudo magnetic values cannot be derived.

- **Pseudo magnetic** filter can be computed from a measured gravity data channel. The information required is the Earth's field inclination and the field declination in degrees. Default Density and Magnetic Susceptibility values are also required to be used in zones where the pseudo magnetic values cannot be derived.

Other Filters

A number of line-based filters and other processing operations are available from the Line Filter **Other** button.

For many of the line filters, you must specify **Operator Length**. Operator lengths can be specified in terms of **samples** or **metres** which can be changed for your convenience but are converted to samples for the actual filter operation. Operator lengths should be chosen such that they can properly resolve the longer wavelengths. You should maintain an operator length which is in the order of twice the longest wavelength that you are trying to remove from the data, that is, Operator length = 2 x wavelength/data spacing.



List of Other filters available in the Line Filter utility.

- **Moving average** filters are similar to low pass filters. The major difference is that the cut-off is not as clean as the low pass filter and it allows considerable leakage of high frequencies into the filter output.

- **Low pass** filter outputs wavelengths that are longer than the specified wavelength. Selection of this filter requires specification of the cut-off wavelength and operator length.

- **High pass** filters output wavelengths that are shorter than the specified pass wavelength.

- **Band pass** filters require the specification of an upper and a lower wavelength. Output from this filter gives the same result as applying two low pass filters with cut-offs equivalent to the top and bottom wavelengths of the band pass.

- **Median** filter is a specifically designed filter for the removal of spikes or data having a specific wavelength and amplitude. In particular, this filter can be used for the removal of cultural noise in magnetic data. The filter operates by defining a threshold amplitude. An operator window is also specified to define the wavelength. The filter moves incrementally along the required lines and compares the middle sample of the window with the difference between the median value of the window and the threshold. If the absolute value is greater than the threshold, the median is substituted for the value.

- **Automatic Gain Control** (AGC) filters apply a non-linear amplification process to a dataset intended to increase the amplitude of small anomalies and decrease the amplitude of large anomalies. A benefit of this filter is that it maintains an anomaly's shape and wavelength as far as possible while filtering. That is, small amplitude features are amplified and large amplitude features are attenuated.

  The AGC process is achieved by defining a window width and using the mean amplitude for the line or survey to define an amplification factor that is then applied to the centre value for that window.

  The formulation of the AGC filter is provided by Rajagopalan (1987 - The use of Automatic Gain Control to display vertical magnetic gradient data. Bull. Aust. Soc. Explor.Geophys. 18, No.1/2 pp.166-168) as:

$$f_{out} = \frac{(f_{in} - f_{av}) * f_{sdwin}}{f_{sdsur}} + f_{av}$$

  Where
  f_{out} is the computed AGC output
  f_{in} is the function input
  f_{av} is the average value at point f in the moving window
  f_{sdwin} is the standard deviation for window
  f_{sdsur} is the standard deviation for survey

- **Analytic signal** – The analytic signal filter uses the computation of the Hilbert Transform. The filter creates an amplitude spectrum which differs by 90 degrees from the original input data. This operation is especially useful in magnetic anomaly detection since the anomaly peaks created occur at infection points of the original input.

  The modulus of the (3D) complex analytic signal of a function g(x,y,z) is given by :

$$AS(g) = [(\partial g/\partial x)^2 + (\partial g/\partial y)^2 + (\partial g/\partial z)^2]^{1/2}$$

  This enhancement is popular for use in magnetic interpretation because it reduces complexity due to magnetization direction. The Line Filter module provides the 2D version of the analytic signal given by:

$$AS2D(g) = [(\partial g/\partial x)^2 + (\partial g/\partial z)^2]^{1/2}$$

This function produces output identical to the 3D analytic signal for any profile input which has no off-profile variation. For inputs that vary perpendicularly to the profile, the 2D analytic signal does not record the full variation of the 3D function. The advantage of the 2D version is that it can be computed rapidly along any individual line. While the 2D analytic signal provides only an approximation of the 3D function it does produce a valid comparison of model input and output channels as it is applied identically to each.

- **First derivative** filters compute the horizontal gradient of the selected column. No operator length is required as this is a simple 5 point numerical operator.
- **Second derivative** filters compute the second horizontal derivative of the selected data. A three point operator based on the work of McIntyre (1981) is used
- **Square root of second derivative** is the square root of the amplitude of the second derivative filter. Negative amplitudes are preserved. McIntyre demonstrated that this method provided an effective enhancement procedure for the recognition of small anomalies on the flanks of large amplitude anomalies. Care is required with this filter because it is also a noise enhancement filter. It is often desirable to low-pass filter the selected column prior to second derivative filtering.
- **Vertical magnetic gradients** can be computed for gravitational or magnetic fields. This is the vertical rate of change of the potential field. A high pass filter enhances the edges of shallow magnetic sources. The operator length should be as long as possible for accurate computation.
- **Upward continuation** is a classic geophysical filter for low pass filtering of magnetic and gravity data. It is mathematically exact for two-dimensional geological sections. This is the case if the geology is uniform in a direction perpendicular to the direction of the data profile. Continuation height is specified in multiples of the sample interval or as a specific distance in metres. Operator lengths should be as long as possible to achieve accurate continuation.
- **Fourth difference** filters are useful tools for the detection of noise spikes, especially in aeromagnetic data. This technique is commonly used for quality control of magnetic data recording systems. The amplitude of the operator is divided by 16 to normalise the output to equivalent noise levels.

- **Noise** generator filter can be used to study the application of other filters. You can artificially add white noise from this filter to create a new channel of signal plus noise. This channel can then be filtered to test the design and characteristics of a second filter in removing noise. Parameters of Mean and Standard Deviation set up a Gaussian noise distribution which is added to the input channel.

- **Standard Deviation** filter returns the standard deviation of the values contained within the moving window. This filter can be used as a moving measure of noise along a line. Care should be taken at the ends of the line where the padding will affect the result. It may be better to run this filter without padding.

- **Threshold** filter had two bounds that can be set. It checks the input data to see if it is between the bounds. If the data point is below the lower bound the filter returns -1. If it is above the upper bound it returns 1 and if it is equal to or between the bounds, it returns zero. This can be used as a quality check e.g. to produce statistics on the number of points above a specified bound.

- **Transform** filter performs an offset and/or scaling of the input values. To offset the input values without scaling, set the scaling to 1. To perform scaling without offset, set the offset to 0.

A range of other filters such as Band Pass, High/Low Pass Moving Average etc are described in the *Appendix C: Grid Filters* (Convolution Filter Descriptions).

TEM Tau Calculator

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software..

When interpreting time-domain electromagnetic (EM) data the time constant (Tau) is a direct indicator of ground conductivity and is an excellent method of quickly and simply acquiring an overview of EM survey results. The time constant is computed at every station by fitting an exponential function to the measured TEM decay over a subset of the off-time decay windows. Using data from survey lines contained within a loaded database, the TEM Tau Calculator is designed to compute the time constant Tau value (and associated error data) along the lines containing EM decay readings and then save the results in the input database.

EM Channel Times Specification

Encom Discover PA is the ideal environment for analysing EM time domain survey data as it is designed to easily manage multi-banded EM data and analyse the decay using the Decay Curve display (see *EM Decay Curves*). As with the Decay Curve display the specification of the recorded window times used in the recording of the EM data is necessary for the computation of the Tau values within the TEM Tau Calculator.

An ATEM.TXT file with format as shown below is the mechanism for the definition of these times and must be present in the same data folder as the input EM database.

```
Begin Survey
XChannelCenter[18] = {78.0625, 103.3875, 128.7125, 154.0375,
228.7499, 341.450, 454.15, 566.85, 735.90, 1017.76, 1412.10, 1919.25,
2539.1, 3271.65, 4116.9, 5074.85, 6145.5, 7328.85} # microSeconds.
End Survey
```

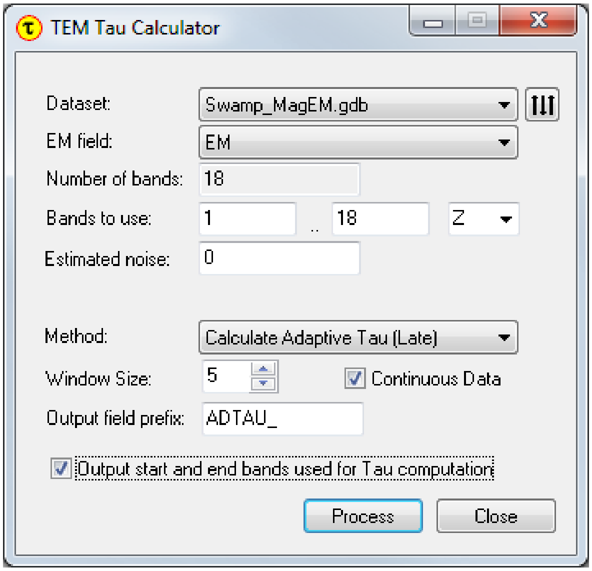
Tau Computation

The Decay Tool available in Encom Discover PA already computes the Tau value of a selected number of data channels, however the **TEM TAU Calculator** provides a mechanism to store the results in a database. In addition to the Tau computation formula, the TEM Tau Calculator allows the specification of a ‘noise’ threshold above which data channels are to be excluded from being used in the tau estimation process. This is specified by the user and the noise rejection is then calculated.

The time constant is computed by fitting an exponential decay function to a selection of off-time decay windows. By carefully selecting the input decay windows you can improve the value of the result. For example, if your aim is to detect good conductors then restrict the windows to late time only. Furthermore, measurement noise can compromise the calculation or even invalidate it. If, within the range of decay windows in question, the measured decay in any window is negative or less than a user specified noise threshold, or if the amplitude of the measured response is not monotonically decreasing with decay time, then the computation can be invalidated or modified.

The TEM Tau calculator contains options to adapt the range of input decay windows at each station to ensure that the input data meets your noise criteria. The range of windows can be modified in response, or expanded to exclude windows that violate the noise criteria. The result will be a more stable and robust time constant computation.

The **TEM Tau Calculator** utility is accessed within Encom Discover PA from the **Database** menu.



The TEM Tau Calculator dialog.

The specifications for performing a calculation with the TEM Tau Calculator utility are as follows:

Dataset

Choose from the available Geosoft (.GDB) databases currently open in Encom Discover PA from the drop-down list. Ensure that a "ATEM.TXT" file defining the channel times is present in the same data folder as the input Geosoft database (see *EM Channel Times Specification*).

Line Selection

Specify which lines in the Geosoft (.GDB) database will be computed.

EM Field

Ensure that the Field visible in the EM Field box is the correct array channel for the calculation.

Bands to use

The user can specify which measurement windows the derived time constant computation should commence from and finish with. You should exclude any on-time windows - if present. If an adaptive calculation method if selected then the decay windows chosen at each station will be restricted to within the specified bands.

Note A minimum of three channels must be used for the calculation to succeed.

Noise Estimate

Specify a value representing the estimated noise. If the response in a decay window is less than this noise estimate then the window may be ignored or the calculation may be invalidated.

Maximum Tau

Specify a value representing the maximum number of channels to be used. The algorithm will find no more than MAX channels that are monotonically decreasing and above the noise threshold.

Tau Calculation Method

A number of methods for determining which data channels to use in the calculation are provided. See *Tau Calculation Methods* for details.

Window Size

The number of station samples used for the adaptive tau calculation.

Continuous data

This option is available for all adaptive methods. If selected, then the algorithm will attempt to find an uninterrupted sequence of the requested number of windows that satisfy the noise criteria. If any window within the sequence does not satisfy the noise criteria then the window selection is modified. If no valid sequence of windows can be found satisfying the noise criteria then no result will be computed. If unselected then the algorithm can ignore invalid decay windows. It will still attempt to find the requested number of windows but they may not be in an uninterrupted sequence.

Output start and end bands used for Tau computation

By enabling this option, the indices of the first and the last stations of the chosen range will be recorded in the database as separate fields.

Tau Calculation Methods

A number of methods for determining which data channels to use in the calculation are provided. The options available are:

- *Calculate Tau at a Selected Range*
- *Calculating Adaptive Tau*
- *Calculate Adaptive Tau (Late)*
- *Calculate Adaptive Tau (Early)*
- *Calculate Adaptive Tau (Optimum)*

The effectiveness of the Tau computation technique is based upon the selection of the data channels (bands) to use in the computation process.

Calculate Tau at a Selected Range

This is a method for a fit to all off time channels in the range specified in **Bands to use** from channel A to channel B.

Calculating Adaptive Tau

This involves the program fitting the data of the five latest channels above the noise estimation value using an exponential decay.

Calculate Adaptive Tau (Late)

This is a method for computing a fit to all off time channels from channel A to channel B, as specified in the **Bands to Use** option, starting from the later stations. The algorithm searches for the specified number of stations with valid data (i.e. above the noise level and monotonically decreasing) starting from the later stations (i.e. the end of the decaying curve).

Calculate Adaptive Tau (Early)

This method is very similar to the Calculate Adaptive Tau (Late) method as described above but the search starts from the earliest data (i.e. the start of the decaying curve).

Calculate Adaptive Tau (Optimum)

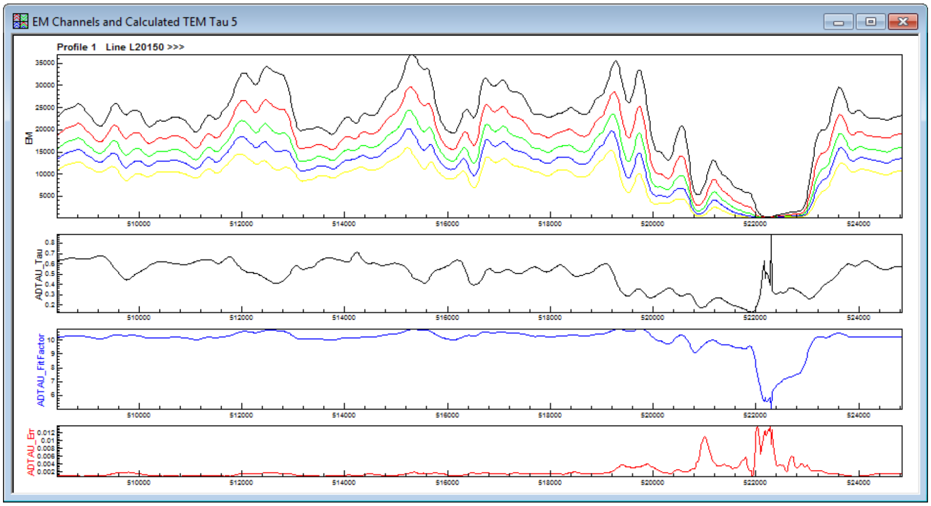
This method will find a sequence of decay windows that return the best fit to an exponential decay curve and return the time constant for those windows. This may vary from early to late time from one station to the next and so the method ought to be used with caution.

Output and Displaying the Results

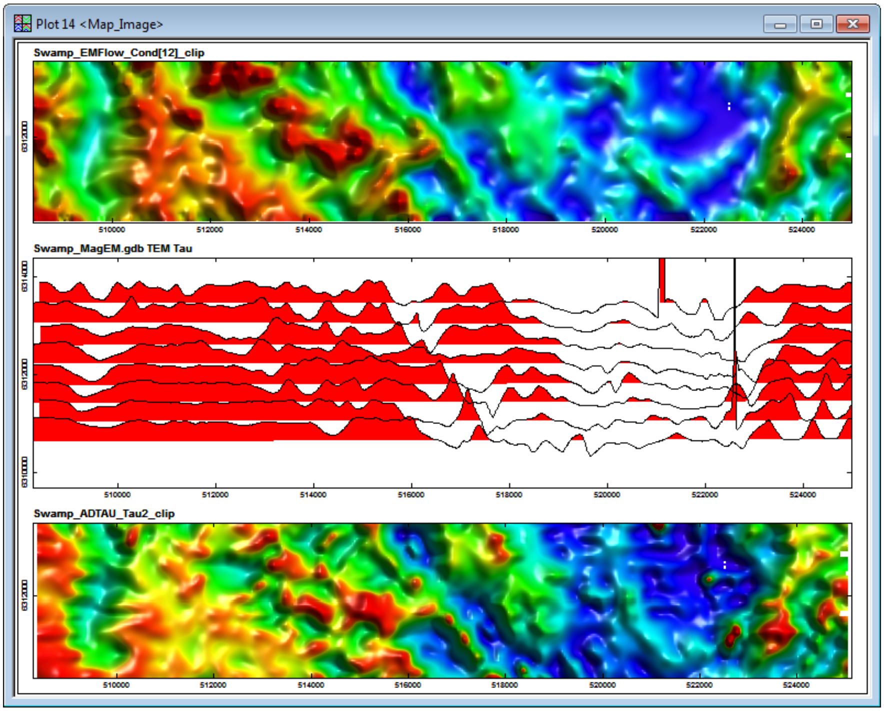
The output from the TEM Tau Calculator can output to an array channel in the same database the following:

- TAU – the time constant in msec
- Fit factor – fitting coefficient
- RMS error – average fitting error
- StartBand and EndBand – outputs as individual fields indices of the first and last bands of the chosen range which the computation used.

Once the Tau values are computed for all available readings, the above output fields of data can be plotted in a **Curve Profile** (see *Profiles*) for line-by-line analysis or alternatively, as a **Line** series in a **2D Map** display (see *Displaying Lines*) or else gridded (see *Interactive 2D Gridding*) to provide imagery of the time constant information.



A profile display of (from top to bottom track) the EM Channels used in TEM Tau calculation, and the output fields of Tau (black), Fit Factor (blue) and Error (red).



A 2D map display of (from top map to bottom map), gridded data of the median conductivity field, output Tau field displayed as stacked profiles, and gridded output of the Tau field.

Flight Path Simulator

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

Disclaimer The flight simulation algorithm used in the SimFlightPath program is intended only to simulate flight surveys and not to be used to plan real world surveys. A number of factors such as the changes in maximum aircraft climb rate depending on weather conditions and altitude are not taken into account.

The Flight Path Simulator allows generation of a Geosoft database containing a simulated, geophysical flight path. This utility is useful in planning flight surveys, minimising flying time and distance covered but ensuring a particular region has adequate flight coverage. An intelligent flight path algorithm is used to predict climb and fall rates of the aircraft over the digital elevation model (DEM) using aircraft performance specifications and even a look-ahead factor to account for pilot control when approaching an elevation rise or fall of terrain. This terrain-following flight-track are used in calculating the resultant distance travelled and in estimating simulated height coverage over the topography.

The flight path can be generated from an input digital terrain (DTM) grid using a combination of editable parameters such as flight path angle, flight line spacing, station spacing, desired clearance and an optional survey area polygon, and input vector file representing flight lines. Aircraft characteristics are read from a Microsoft Access database that has been populated with several common survey aircraft flying characteristics. You can add extra aircraft specifications and modify the various survey aircraft settings in the database.

- *Data Inputs*
- *Run Flight Path Simulator*
- *Using the Simulator Dialog*

Data Inputs

The following data files can be used with the Flight Path Simulator:

- A **digital terrain grid** in a standard grid format with elevation in metres is mandatory. Supported grid formats are the same as for Encom Discover PA and include ER Mapper, Geosoft (Compressed or Uncompressed), Surfer etc.

- The **survey clipping polygon** can optionally be used. The polygon shape can be an ESRI shape file (.SHP), a MapInfo Professional .TAB or .MIF file or a Encom Discover PA polygon from a Features database. If a clipping polygon is not used, a rectangular area defined by the dimensions of the input digital terrain grid is used by default.

- The **use vectors** can also be optionally used. The vector input is a representation of the flight lines and can be in the format of an ESRI Shape file (.SHP), a MapInfo Pro (.TAB or .MIF) file, an AutoCAD (.DXF) plus many others. If a vector input is used to define the survey lines the **Flight lines** section is disabled.

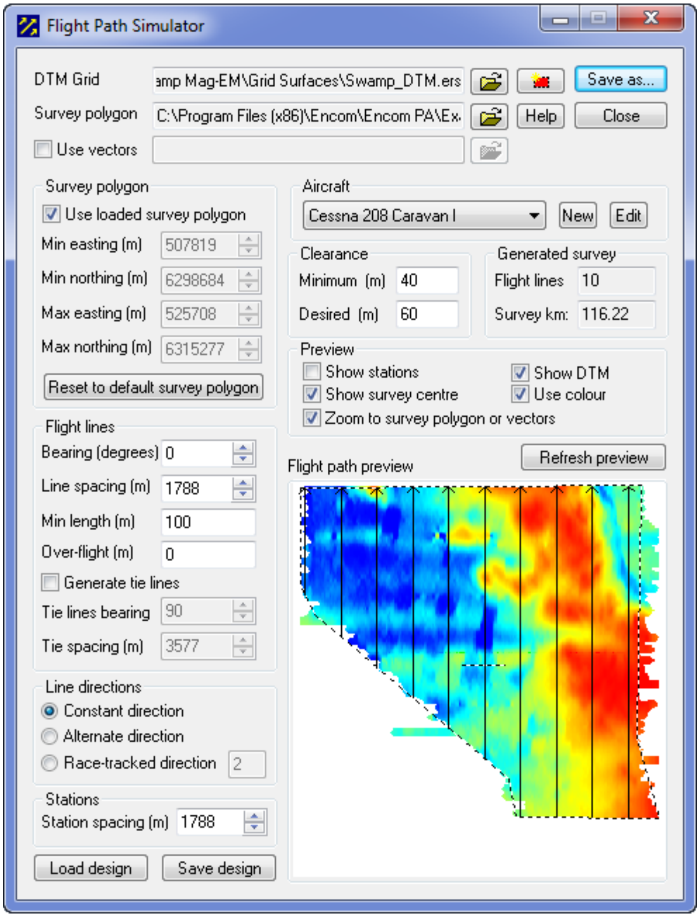
- The **aircraft performance database** in Microsoft Access format. New entries can be added to the database or existing aircraft stored in the database can be modified.

Run Flight Path Simulator

After running the Flight Path Simulator tool from the Database menu, you are required to specify a digital terrain grid since this is mandatory (although you can create an artificial flat grid) for operating the Flight Simulator. A single dialog allows specification of all aspects of the proposed survey flight design (including traverse and tie lines). Use the **Browse** button to navigate to a folder and select the required terrain grid. Modify the **Files of Type** option as required to suit the format if necessary.

By default, the area chosen for the flight simulation is a rectangle defined by the corner points enclosing the chosen terrain grid. If the survey area is to be an irregular shape, a polygonal area defined by a .TAB, .MIF or .SHP file can be used. Ensure the areas of the digital terrain grid and polygon file overlap for correct operation.

Using the Simulator Dialog



Flight path simulator dialog

The figure above illustrates the appearance of the simulator dialog using the input DTM Grid and Survey polygon.

- *Input Files*
- *Creating an Artificial DTM Grid*
- *Survey Polygon Parameters*
- *Flight line parameters*
- *Aircraft Selection*
- *The Aircraft Database*

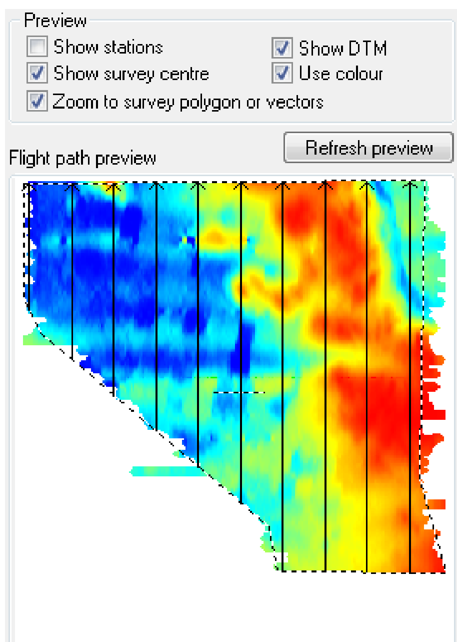
- *Description of the Algorithm*
- *Data Outputs*

Input Files

At the top of the dialog are two **File Open** buttons that allow you to choose an input DTM grid and an optional survey area polygon. The input DTM grid may be any of the Encom-supported grid types. For a list of supported formats, refer to *Appendix E: Supported File Formats*.

An **Additional Input** button is also available if you do not have a DTM grid available. In this case a flat, user-defined grid can be created to generate an artificial flight survey. Refer to *Creating an Artificial DTM Grid* for further information.

The survey area polygon may be either a MapInfo Professional .TAB, .MIF or ESRI shape (.SHP) file. The DTM grid file is mandatory and must be chosen first. After the grid is loaded the preview window shows an image of the flight path with the current flight path settings (assuming the **Show DTM** option is enabled).



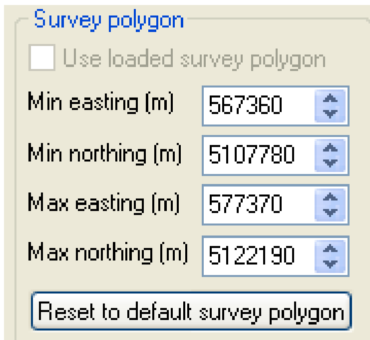
Flight simulator preview window

Select the **Refresh preview** button when survey parameters are changed to force an update of the display

Creating an Artificial DTM Grid

If a DTM grid is not available for the area required, an artificial, flat grid can be created to use for the flight simulation. Click on the **New flat grid** button to display the **Create new grid** dialog. You can specify a rectangular area for the new grid (from the minimum and maximum X and Y plus the grid cell width and height). The grid can contain multi-bands if necessary and be assigned a default elevation even though the created grid is horizontally planar. Any of the supported grid formats can be used to save the output. Refer to *Appendix E: Supported File Formats*.

Survey Polygon Parameters

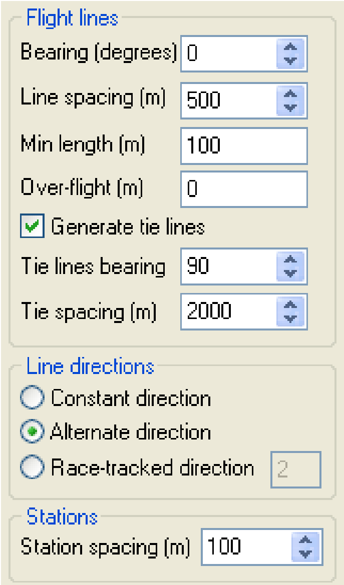


Survey Polygon details derived from a loaded polygon file

This section of the dialog allows you to control the area of the DTM over which the flight lines are generated. If no survey polygon has been provided in the input files section, you can alter the corners of the default rectangular survey area by using the spin buttons or entering new easting and northing values. The **Reset to default survey polygon** button resets these parameters to a rectangle that covers the whole of the DTM area.

If a survey polygon has been specified in the input files section, the **Use loaded survey polygon** check-box is checked. This can be unchecked if you wish to override the loaded survey polygon with a rectangular area.

Flight line parameters

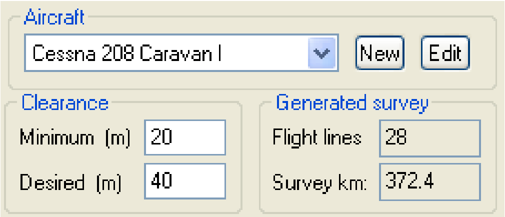


Controls for bearing, spacing and line directions

Controls for the orientation and distribution of flight lines (traverses) and tie lines are provided for when the **Use Vectors** option is not selected. Entry fields in this section allow you to specify the line angle, and line spacing, sample spacing and line direction. You may also want to adjust the start offset of the flight lines (the distance from the centre point of the survey), and the minimum length of flight lines. The minimum length parameter suppresses the generation of flight lines that would be shorter than the specified length.

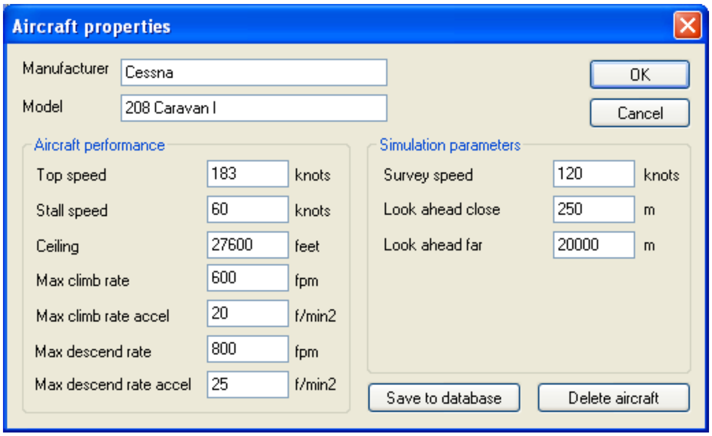
Three different line direction options are provided: constant, alternate, or race-tracked. If race-tracked lines are selected, a number can be entered to control how many flight lines go in the same direction before the line direction swaps (that is, the number of traverse lines in each race-track).

Aircraft Selection



Aircraft selection with links to an aircraft specification database

Select an aircraft from the database and enter the desired mean terrain clearance in metres. The minimum safe clearance value can also be edited. The simulated aircraft is an attempt to maintain the desired clearance and never goes below the minimum clearance. To access the characteristics of a particular aircraft or add a new type, select either the **Edit** or **New** buttons.



Aircraft properties dialog

The Aircraft Properties dialog allows you to select one of a number of aircraft that are stored in the database (AIRCRAFT.MDB). If you want to view or modify the aircraft characteristics you can click the **Edit** button to display the Aircraft Properties dialog. Changes made in the Aircraft Properties dialog can be made permanently by selecting the **Save to Database** button. You can also remove existing aircraft entries by selecting the **Delete aircraft** button. The database format used to retain the aircraft information is Microsoft Access (see *The Aircraft Database*).

A good internet facility to investigate aircraft flight specifications can be found at *www.risingup.com/planespecs/*.

The Aircraft Database

The aircraft database (retained in file AIRCRAFT.MDB) is a Microsoft Access database that is provided with the flight path simulator. It must be placed in the same directory as SIMFLIGHTPATH.EXE. It contains the flight characteristics of a few common survey aircraft. New aircraft can be easily added using the dialog shown above and selecting the **Save into database** button.

The aircraft database contains a single table called FlightParameters. Each row in this table represents the flight characteristics of an aircraft. Descriptions of each of the fields in this table and the units used can be viewed by opening the table in design mode. Most of these fields are self-explanatory, however several parameters are specific to the flight simulation algorithm used by the flight path simulator.

The **Top Speed** and **Stall Speed** are only guides for setting the **Survey Speed**. That is, the Survey Speed must be between these two values. This speed is the constant velocity at which the simulated plane flys. This is a simplification of course as in real-life the aircraft speed changes depending on various factors – such as climb rate, temperatures, wind conditions etc.

The **Ceiling** is a guide only and not used by the program except as an upper extrapolation limit. This is in feet rather than metres because the aviation industry still use imperial units.

The **Max Climb Rate** field contains data that is widely available in aircraft specifications. However, the other parameters related to climbing and descent are more difficult to determine. Max Climb Rate Accel is the maximum rate at which an aircraft can change its rate of climb. **Max Descent Rate Accel** is the maximum rate at which an aircraft can change its rate of descent.

The Maximum climb rate, Climb rate acceleration, Maximum descent rate, and Descent rate acceleration determine how steeply the flight line can climb over peaks and how swiftly it can descend on the other side of a peak. Climb rates are in feet per minute and accelerations are in feet per minute squared.

The **LookAheadClose** and **LookAheadFar** fields provide the look-ahead distances for the terrain following and collision avoidance algorithms respectively. The LookAheadFar distance is the distance ahead the simulation looks to ensure that the aircraft will always have enough altitude to clear any peaks which are coming up. The best value for this is quite large (e.g. 20km) to ensure the plane never crashes. The LookAheadClose distance by contrast needs to be reasonably small (e.g. 250m) to allow averaging of small scale changes in the terrain.

Description of the Algorithm

When you choose an aircraft for the simulation from the aircraft database, a number of parameters are read including the maximum climb rate, descent rate, and maximum accelerations of these. These may be manually overridden for the current simulation to match expected conditions such as air temperature and the average altitude of the survey.

The simulation is performed one flight line at a time and proceeds through the following steps:

Step 1: Determine initial altitude of aircraft.

This altitude is chosen such that the aircraft is as close to the ideal ground clearance as possible while still allowing it to clear any peaks which lie in its path.

Step 2: Determine desired altitude for next aircraft position.

This uses the “look ahead close” distance to determine the average height of a window of terrain (typically a few 100 metres wide) and its average slope. These values are used to adjust the current climb rate of the aircraft to make it “hug” the terrain so that it flies as closely as possible to the user’s desired clearance.

Step 3: Dampen change in climb rate.

A dampening formula is applied to any changes in climb rate so that the aircraft does not oscillate too much above and below the desired clearance.

Step 4: Check for long-range collisions.

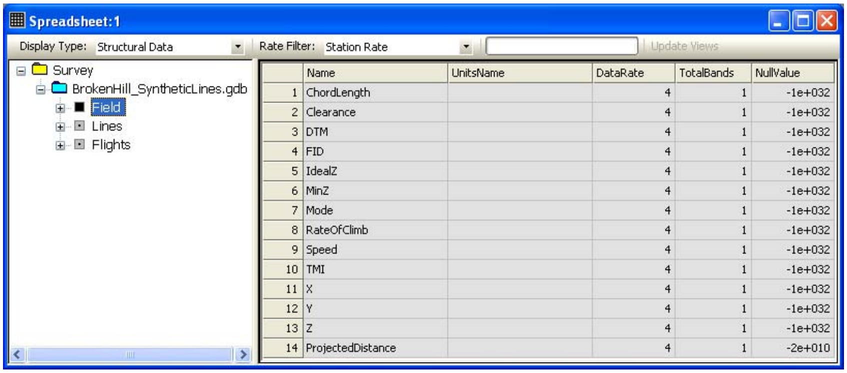
The look ahead far distance is used to determine whether the aircraft can clear all peaks that lie ahead (typically this look ahead distance is 10-20 kilometres). When it becomes necessary to clear a peak by applying the maximum climb rate, the algorithm switches into collision avoidance mode and increases the current climb rate by the maximum climb rate acceleration each simulation cycle. It continues to do this until there is no longer any danger of a collision and then reverts to ground hugging mode.

Step 5: Move the simulated aircraft and record the station position.

The changes in climb rate are applied to the simulated aircraft and its ground position is advanced to the next simulation position. (The aircraft is assumed to move at a constant, user-configurable, survey speed.)

Repeat steps 2 through 5 until the aircraft reaches the end of this simulated flight line.

Data Outputs



	Name	UnitsName	DataRate	TotalBands	NullValue
1	ChordLength		4	1	-1e+032
2	Clearance		4	1	-1e+032
3	DTM		4	1	-1e+032
4	FID		4	1	-1e+032
5	IdealZ		4	1	-1e+032
6	MinZ		4	1	-1e+032
7	Mode		4	1	-1e+032
8	RateOfClimb		4	1	-1e+032
9	Speed		4	1	-1e+032
10	TMI		4	1	-1e+032
11	X		4	1	-1e+032
12	Y		4	1	-1e+032
13	Z		4	1	-1e+032
14	ProjectedDistance		4	1	-2e+010

Encom Discover PA Spreadsheet summary of simulated survey fields

The simulated flight path is saved in a Geosoft Oasis montaj database (.GDB) file that can be accessed from Encom Discover PA.

The database fields created by the flight simulator are listed in the table below

F	**Description**	**s**
X	Easting	m
Y	Northing	m
Z	Aircraft elevation	m
	Terrain elevation at sample location	
C	Ground clearance (radar altimeter)	m
Ie	Target terrain clearance	m
Mi	Minimum elevation	m
	Aircraft climb rate	fe
S	Aircraft speed	ks
C	Cumulative distance along line	
P	Distance projected onto an average baseline orientation	m

Airborne Quality Control Tool

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

Introduction

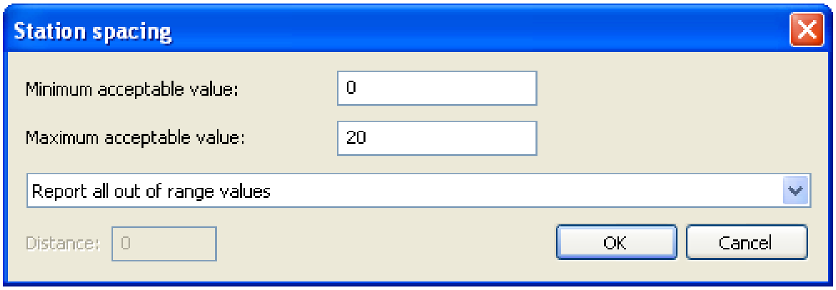
This utility is a plug-in to Encom Discover PA which is designed to help check that contractual obligations have been met in the collection of airborne geophysical survey data.

Before running the tool, the survey dataset must be open in Encom Discover PA. When the tool is run the user must select the input dataset and then choose the lines that are to be processed. Lines are considered to be either normal lines or tie lines. The normal lines are assumed to be in approximately the same orientation as each other (or at 180 degrees to this orientation). All quality control checks except for line spacing are performed on all selected lines. Line spacing is only checked for normal lines.

Quality Control Checks

- *Check station spacing*
- *Check line deviation*
- *Check values for field*
- *Check normal line spacing*
- *Report survey lines which are approximately co-linear*
- *Report on normal/tie-line intersections*
- *Output options*

Check station spacing



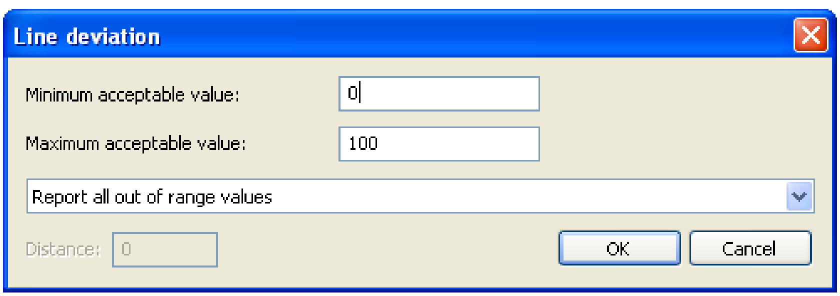
The station spacing dialog.

The **Check station spacing** option is performed along every selected line. Any stations which are less than or greater than the specified distances apart along an individual line are considered to be a problem. The range of acceptable station spacings are specified using the dialog which pops up when the corresponding Properties button is pressed. This dialog also allows the problems to be reported in a variety of ways:

- Report all out of range values
- Report where the values are continuously out of range over a specified distance
- Report where the values are out of range over a cumulative percentage of the line length

These reporting options are designed to mirror the common contractual requirements.

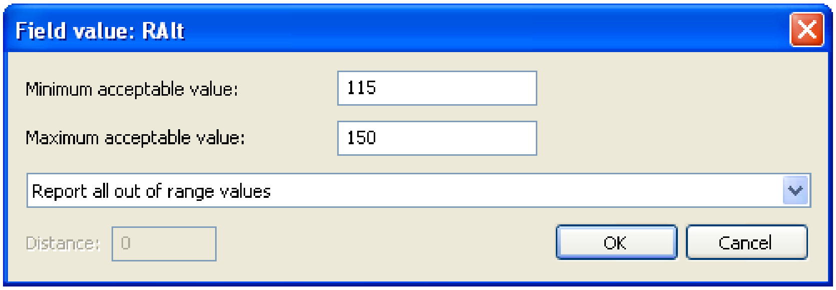
Check line deviation



The Check line deviation dialog.

The **Check line deviation** option is performed along every selected line. First, the best fit line is computed using the least squares linear regression method. Then, the perpendicular distance of every station on the line from the line-of-best-fit is computed. Distances which are outside the range specified using the properties dialog are reported with the same options as for station spacing above.

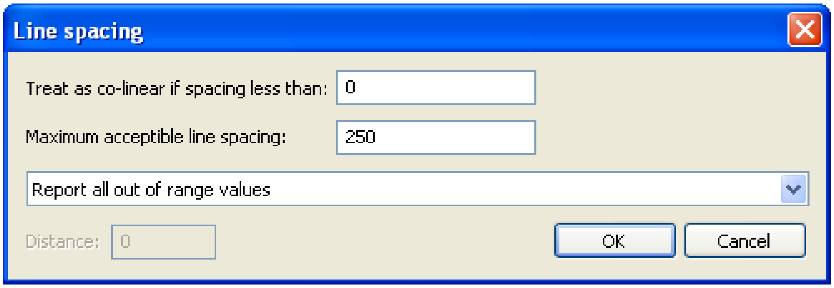
Check values for field



The Check values for field dialog.

The **Check values for field** is a general purpose option that is performed for the specified field on every selected line. Fields which are outside the range specified using the properties dialog are reported with the same options as described above. This check may be useful for finding aircraft clearance values which are outside the acceptable range for the survey. If a clearance field is not available in the dataset this may need to be computed using the **Line Calculator** plug-in before running this tool.

Check normal line spacing



The Check normal line spacing dialog.

The **Check normal line spacing** option is performed between selected normal lines only. First, the lines are sorted spatially (using the best fit lines), and then a list of adjacent lines is generated. For each pair of adjacent lines, the distance between corresponding stations along the lines are computed. Any distances greater than the value specified in the properties dialog are reported using the same options as for the other checks above.

Since flight lines are sometimes flown in multiple passes, a further option on the properties dialog allows a distance to be specified below which adjacent lines are considered to be approximately co-linear. Specifying a non-zero distance here will allow the program to compare only the correct pairs of adjacent lines to check for line spacing. The value you choose depends on the survey. To visually check for lines which are approximately co-linear in PA, open a 2D flight path map and turn on the **Symbols** option at the end of the flight lines under the **Flight Path Properties>Appearance** tab.

Report survey lines which are approximately co-linear

The **Report survey lines** which are approximately co-linear option operates with the Line spacing check and uses the same properties dialog. If this check is enabled, then report entries will be generated when approximately co-linear lines are detected.

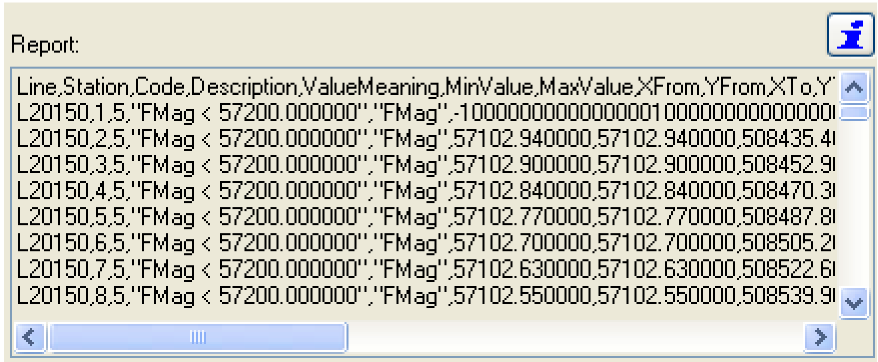
Report on normal/tie-line intersections

The **Report on normal/tie line intersections** option reports all 2D intersections between the selected normal lines and the selected tie lines. The report includes the location of the intersection and the interpolated vertical offset between the two lines. The vertical offset is computed using the selected Z

Output options

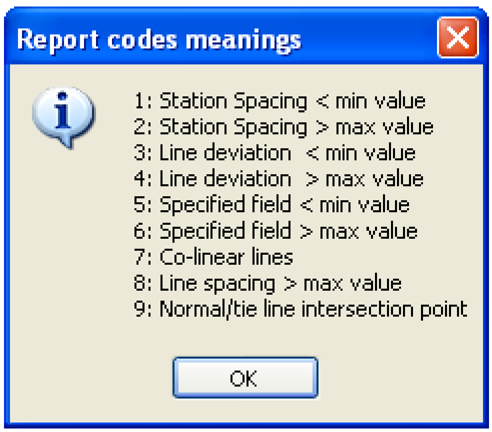
When the **Perform QC checks** button is pressed, a summary of the qulaity control checks are displayed in the top right of the dialog and a report is generated for all the checks selected in the **QC options** area of the main dialog.

The report is shown at the bottom right of the dialog and can be saved to a .CSV file or any of the supported vector file formats (e.g. MapInfo .TAB) using the **Save reported points** button. If the report is saved to a vector file, then this file can be displayed in Encom Discover PA over a 2D flight path map to highlight where the problems have been found.



The Report preview dialog.

The **Information** button provides the definitions for the codes that are displayed in the report (third column).



The definition of report codes dialog.

The best fit lines computed during the run can also be saved to any supported vector file format using the **Save best fit lines** button, and may also be overlaid on a flight path map.

A final output option is to save the per-station values checks to a new database, or append to the input database. This will add the selected checks as new fields in the input database (Station Spacing, Line Deviation and Field Check). This option will only output values for the first three quality control checks since they are the ones performed for individual lines.

Diurnal Correction Tool

The purpose of most aeromagnetic surveys is to measure the local deviation in the magnetic field caused by crustal geology. It is therefore necessary to define the global variation in the earth's magnetic field and then subtract this from observed magnetic field to obtain the time constant magnetic anomaly.

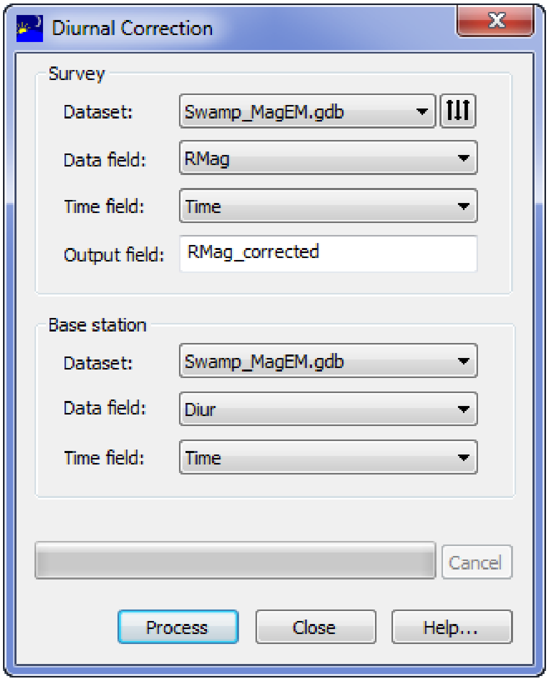
The variations in the total magnetic field with time over time scales ranging from seconds to millions of years have a profound effect on how magnetic surveys are carried out.

Diurnal variations follow a daily cycle associated with the rotation of the earth with respect to the sun. The 'solar wind' of charged particles emanating from the sun, even under normal conditions, tends to distort the outer regions of the earth's magnetic field. The daily rotation of the earth within this sun-referenced distortion leads to variations in the total magnetic. The main variation occurs towards local noon when peaks are observed in mid-latitudes and troughs near the magnetic equator.

Since it is unsuitable for airborne surveys to be flown at night when recorded geomagnetic observations are more accurate, it is therefore required that surveys be planned so as to allow for corrections to be made for diurnal (amongst other) variations.

The **Diurnal Correction** utility, accessible in Encom Discover PA from the **Database** menu with an active Encom Discover PA Professional licence allows for the correction of these diurnal variations on collected magnetic values recorded in a loaded database.

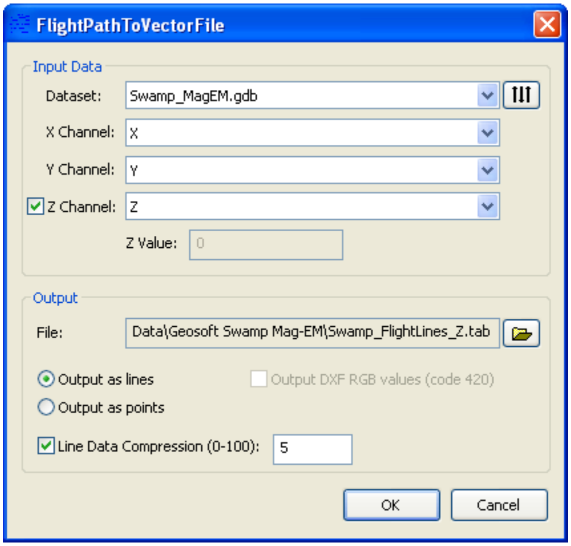
This is achieved by specifying a loaded database containing the fields for the recorded survey magnetic data and Time, then the magnetic field and Time fields in either the same or a secondary database representing the magnetic recording from a base station magnetometer in the same survey region.



The Diurnal Correction utility dialog.

Flight Lines to Vector File

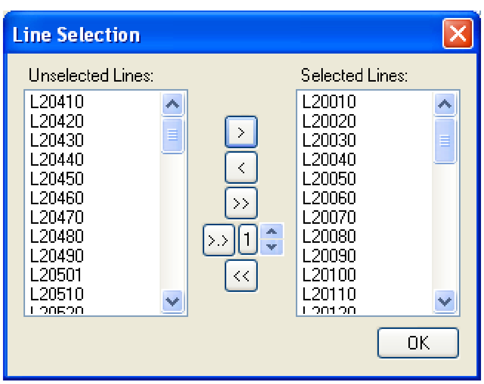
The Flight Path To Vector File plugin utility can be accessed from the **Database** menu and will produce a vector file of survey (flight) lines from a database currently loaded into a session.



The Flight Path To Vector File utility dialog with specified output as a MapInfo .TAB file.

When initiated the Flight Path To Vector File dialog will appear and by default the first (if there is more than one) database currently loaded in the session will be selected as the **Input Data Dataset**. To change to a different database activate the Dataset drop-down list and select the required database.

A subset of the survey lines from the database can be specified for the output by clicking on the **Line Selection** button



The Line Selection dialog allows a subset of survey lines to be converted to a vector file.

The X Channel and Y Channel input fields are compulsory and will be automatically updated when the Dataset is selected. To change to another X or Y field activate the drop-down lists for either of these and select the required field(s) in the database.

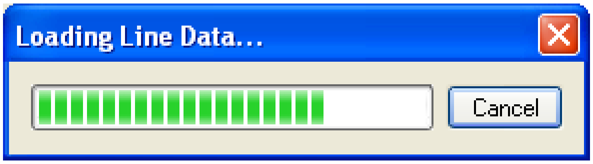
The Z Channel is an optional channel that can be assigned to the flight lines. To assign a Z Channel ensure that the tick box is selected and then select the appropriate field from the Z Channel drop-down list. To assign a constant Z value to the flight lines ensure that the Z Channel tick box is unselected and enter a value into the Z Value box available.

The output file formats supported by this utility are:

- MapInfo TAB files (.TAB)
- MapInfo MIF files (.MIF)
- ESRI Shape files (.SHP)
- AutoCAD DXF files (.DXF)
- Comma Delimited ASCII Files (.CSV)
- Encom TKM files (.TKM)
- Google KML files (.KML)

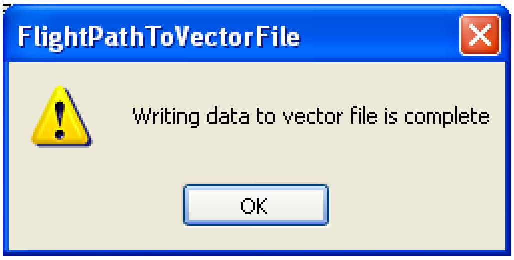
The output file can render the survey lines as either **Lines** or **Points**. When working with large survey datasets a **Line Data Compression (0-100)** can also be specified.

To proceed with the conversion click on the OK button and a status bar will appear.



Status bar for the conversion process.

Once complete a message will appear confirming that the conversion has been successful.



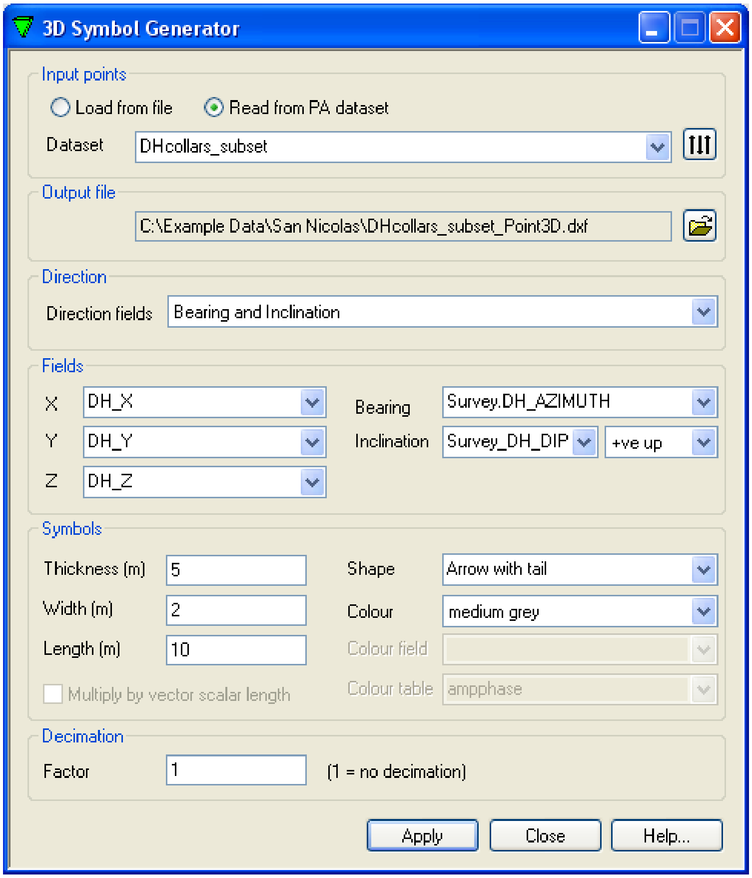
Confirmation message for a successful Flight Path to Vector File conversion.

3D Symbol Generator

Encom Discover PA can be used to display 3D views of data. Frequently, interpretation applications can be executed to create locations in three dimensional space that are associated with a solution that provides information about the physical property at that location. For example, Euler or depth-to-basement techniques (such as the AutoMag option of Encom ModelVision) create database information of solution locations with magnetic source data (such as magnetic susceptibility, depth, dip and azimuth). To display such solutions in 3D views and with other datasets (grids, drilling, modelling etc) is useful for interpretation.

The Symbol Generator utility program can be run to convert individual depth and property solutions for 3D display. The program converts the location, including depth and creates a variety of symbol types as AutoCad .DXF format files. The Symbol Generator can be run from the *Database Menu*.

The Symbol Generator uses any database accessed by Encom Discover PA or an external MapInfo Professional .TAB or .MIF file. The database can contain line or point data but should optimally have easting, northing and depth information. The interface dialog for the Symbol Generator is as shown below:



Symbol Generator dialog specifying a database for location of symbols

For infoprmation on using the Symbol Generator, see:

- *Symbol Generator Controls*
- *Operating the Symbol Generator*

Symbol Generator Controls

- **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.

- **Dataset** – A Encom Discover PA accessed database can be used for symbol creation at each data point selected by line. Line selection is from the **Line** button

- **Output File** – The created symbols are stored in an AutoCad .DXF format file. You can path and specify a filename if required.

- **Direction** – 3D Points can be orientated either via specification of Azimuth and Dip fields, of Vector component (u, v, w) fields.

- **Fields** – Data fields used for location (X and Y) and depth (optional) can be selected. If the symbols to be created are to have azimuth rotation and dip rotation, specify the optional Azimuth and Dip fields.

- **Symbols** – The Width, Length and Thickness of the 3D symbols created can be specified (in scaled metres). The Shape of the 3D symbols can be either:

    - Triangular

    - Rectangular (prisms)

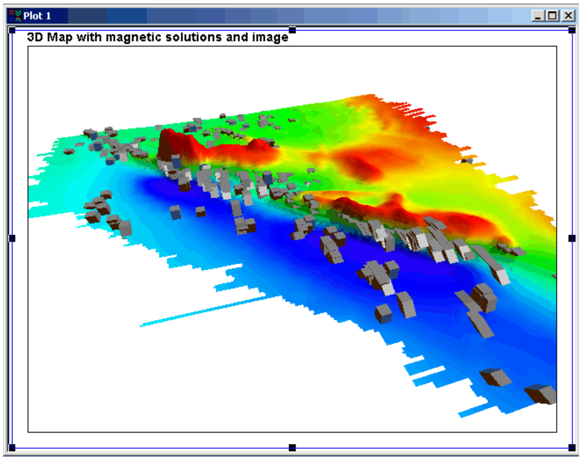
    - Arrows

    - Tabular

Operating the Symbol Generator

When the Symbol Generator is executed (from the **Apply** button), the created 3D symbols are placed in the specified .DXF output file. When completed, a window with the output .DXF file displayed is automatically presented using a provided template. You can modify this template if desired.

The created display uses a 3D Map with the output .DXF file included. To this window you can add surfaces, flight paths, drilling and modelling. An example of a depth solution group of bodies derived from Encom's AutoMag is shown below.



Use of 3D symbols at depth locations from an AutoMag database

Like most Encom Discover PA utilities, while the Symbol Generator is running, you can make modifications to the parameters and reapply these. When this is done, the display updates automatically and the .DXF file is saved. Once the dialog for the Symbol Generator is closed, you need to re-enter parameters if you want to alter the .DXF output.

Stacked Profiles Tool

The Stacked Profile tool creates a graph of a nominated channel displayed along a traverse base line. Stacked profile presentations of line oriented data are frequently used in geophysical data analysis.

Advantages offered by stacked profiles include:

- Anomaly positions can be made to lie in their correct location
- Subtle trends and anomaly correlation from line-to-line can be extremely sensitive and often more so than as presented using contouring or imagery
- Various filter operators can be applied to line data and optimally presented in stacked profile form
- Stacked profiling is independent of traverse line leveling problems.

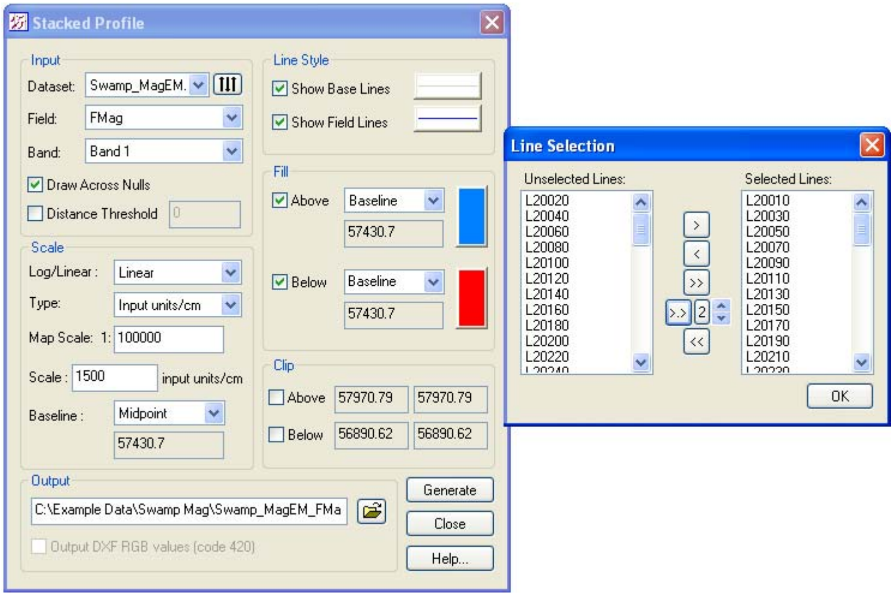
Stacked profiles created within Encom Discover PA offer the following features:

- Control over clipping (high and low) thresholds

- Filling of profile above or below a definable baseline (a variable area presentation)
- Linear or logarithmic scaling
- Clipping of upper and lower thresholds of data.

The option requires a database to be initially accessed. In order to generate a stacked profile the database must contain a column with a unique line identifier attribute (e.g. line number) and at least one numeric data column on which to create the profile.

When the Stacked Profile utility is initiated from the **Database>Stacked Profiles** menu option the following dialog is presented.



Dialog for control of Stacked Profile display with Line Selection dialog

You can select the following options:

- An available dataset, data field and channel (band).
- Individual lines can be selected from display, for example tie lines could be eliminated.
- Both data and baselines can be displayed. Each can have its own line style specified.

- Scaling of the field data can be linear or logarithmic and comply with the Scale Factor of the map in which the stacked profiles are displayed or in units per centimetre.
- The stacked profiles can be filled above or below a threshold value and with different specified colours.
- Colour filled profiles can be clipped at specified upper and lower thresholds.

Note The Stacked Profile tool can also be used for Multi-Banded field data. The drop down box for Band selection allows different bands to be selected.

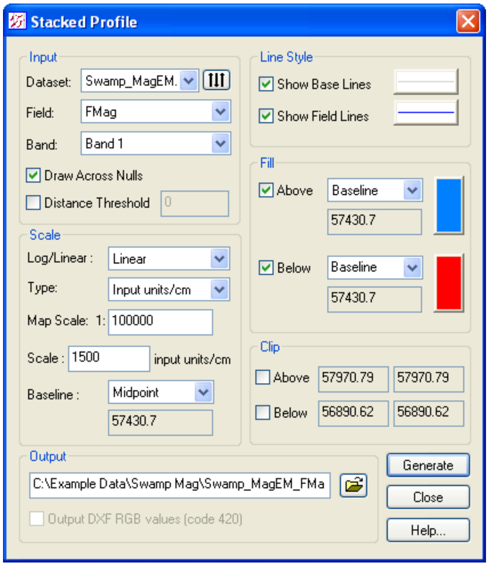
For more information about creating and displaying stacked profiles, see:

- *Creating Stacked Profiles*
- *Displaying a Stacked Profile in a 2D Map*

Creating Stacked Profiles

Initiate the **Stacked Profile** option from the **Display Utilities** menu and in the dialog that appears select one of the open databases from the drop-down list at the top of the dialog.

Select the data column to display in the profile from the **Field** pull-down list. Individual lines can be selected using the **Select Lines** button. Line fill options relative to the baseline can be applied to the stacked profiles and Z scaling can be left on auto apply or alternatively a specific scale factor and baseline value can be specified.



Example Stacked Profile dialog settings.

The controls available from this dialog are:

- Check the **Draw Across Nulls** box to allow Encom Discover PA to continue to draw a profile when a blank data field is encountered.

- In the event of larger gaps in the sampling interval within the one sample line, check the **Distance Threshold** box and enter a distance. When the distance between two sample points is greater than the value entered as the distance threshold, Encom Discover PA starts a new profile on the same sampling line. If a distance threshold is not entered, a continuous profile for the line is displayed.

- Scaling of the field data can be linear or logarithmic and comply with the **Scale Factor** of the map in which the stacked profiles are displayed or in units per centimetre.

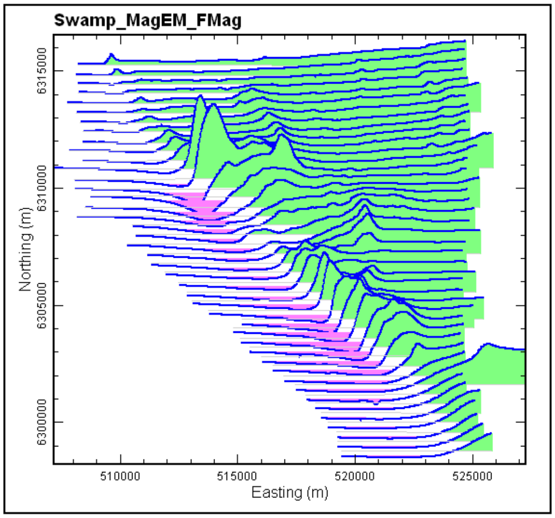
- The baseline or x-axis may be displayed with the profile. Use the **Baseline** pull-down list to select from the range of baseline options. A baseline may be created according to the minimum, maximum, average or median value of the data column. A user-specified baseline value can also be entered.

- When the **Show Base Lines** option is enabled in the Line Styles section the base line or x-axis of the profile will be displayed in the final stacked profile. The appearance of this baseline can be modified by clicking on the style button to the right of this option.

- Check the **Show Field Lines** to display the profile for the selected data column. Use the line style button next to this option to select the desired profile curve style and colour.
- Using the options in the **Fill** section of the dialog each profile may be filled with a solid colour to denote samples with values above or below the baseline or a user specified value. Click on the colour buttons to change the fill colour.
- The minimum and maximum values in the data column are automatically inserted into the **Clip** control section. Enable the **Above** or **Below** box to enter another value in order to clip the data range used to the specified data range.

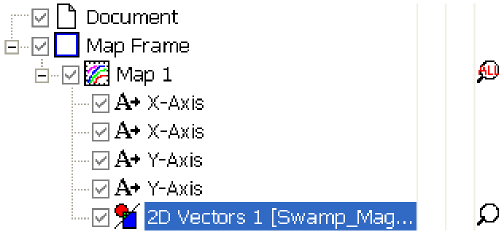
Displaying a Stacked Profile in a 2D Map

Upon selecting the **Generate** button on the dialog a stacked profile is created using the set parameters in the dialog. A new 2D Map display will be automatically created showing the Stacked Profile.

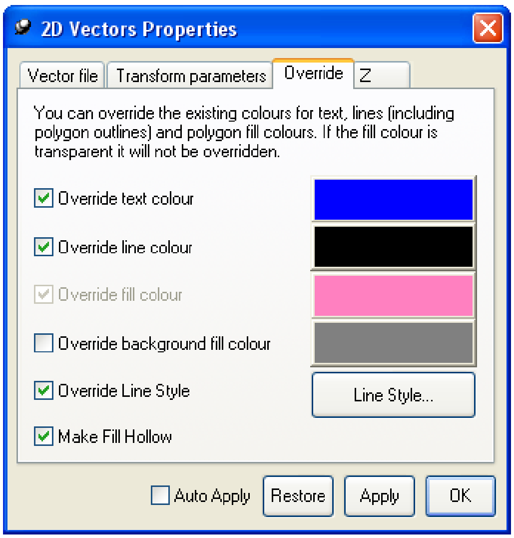


Example of a stacked profile map

The Stacked Profile output file type is referred to as a .GSF file extension and is 2D vector file. Therefore in the workspace tree associated with this display the Stacked Profile file will be shown as a 2D Vector file branch.



To change the colouring and line style of the stacked profile access the 2D Vectors Properties dialog and view the **Override** tab dialog. The line colour, fill colour and line style as well as making the fill hollow can be altered from this dialog.



Use the Override tab dialog in the 2D Vectors Properties to alter the appearance of the Stacked Profile display.

Note To alter the map scale and vertical scaling of the stacked profiles the .GSF file will need to be recreated. It is therefore suggested that the Stacked Profile dialog is kept active until a suitable scaling can be set and then select the Close button on this dialog.

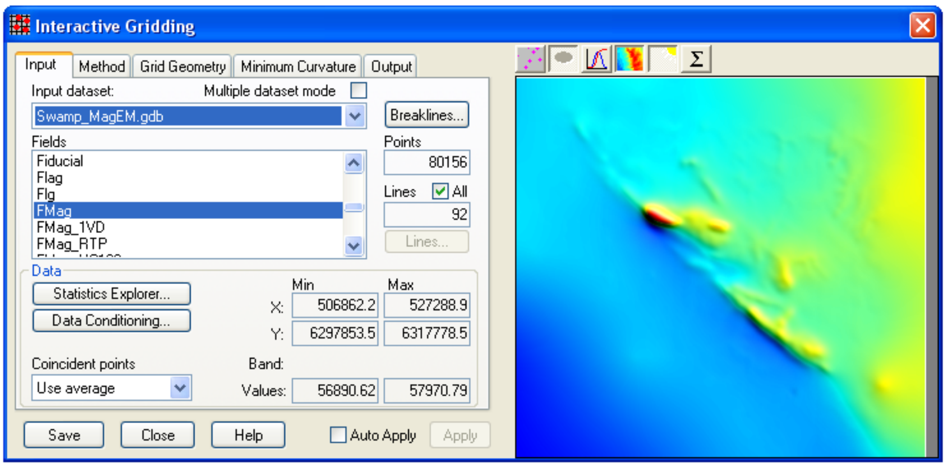
26 Gridding Utilities

The **Grid** menu provides access to the following functionality:

- *Interactive 2D Gridding* – Create a continuous surface grid from a selected database field using variety of interpolation methods: Inverse Distance Weighting, Kriging, Triangulation, Spatial Neighbour, Minimum Curvature, Density or Distance.
- *Large and Multi-file Gridding*: Interpolate multiple large data sets with millions of points or polylines to build a massive grid using methods such as minimum curvature, inverse distance weighting, and triangulation.
- *Section Gridding* – produce section grids from profile data such as EM or IP.
- *Grid Calculator* – Perform grid arithmetic, statistical and Boolean calculations on one or more grids
- *Grid Filter* – Options for filtering gridded data.
- *Grid Utility* – Classify, grid to grid clip, clip, convert, create RGB, cut/fill, edit, fill holes, flip, merge, outline, overlay, replace, reproject, resample, rotate, shift, split, vectorize grids or calculate volume.
- *Assign Grid Value to Dataset* – interpolate and assign data from grid centre points to a Geosoft or Feature database field.
- *Grid Tiler* – Large grids can be 'tiled' (i.e. subdivided) into a number of smaller grids without needing to open the source grid into Encom Discover PA.
- *Vector File to Grid Utility* – convert a 3D .DXF vector file to a grid surface.
- *SRTM HGT to ERMapper ERS* – converts an SRTM .HGT elevation data file to a ER Mapper .ERS grid file.

Interactive 2D Gridding

The **Interactive 2D Gridding** in Encom Discover PA allows you to create two dimensional data grids of a selected database field (including multi-arrayed fields) and then output the grid to a specified grid file in any one of the supported grid formats. The optimised gridding algorithms used in this tool have made it possible to include a real-time preview window to assist you in choosing the best parameters for your data. This preview window allows you to experiment with different gridding parameters in real-time and observe the effect they have on the quality and appearance of your grid. The Interactive 2D Gridding automatically analyses the spacing and distribution of your input data and computes an optimal set of parameters for you. If you are not satisfied with the default parameters and gridding method selected by the tool, you can override these settings by selecting your own method and parameters (grid cell spacing, search criteria, etc).



The Interactive 2D Gridding Tool showing the data selection tab and grid preview

For more information see:

- *Using the Interactive Gridding Tool* (includes descriptions of individual tabs)
- *Previewing the Grid*

Using the Interactive Gridding Tool

The Interactive Gridding is controlled via a series of tabs to the left of the preview window. The number of visible tabs in the dialog varies depending on the gridding method selected (see *Method Tab*), however, the four main tabs are always visible irrespective of the gridding method selected.

- *Input Tab*
- *Geometry Tab*
- *Method Tab*
- *Output Tab*

Additional tabs may be displayed depending on selections made on these tabs. When the gridding parameters have been selected, press the **Save** button. This closes the Interactive 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 Interactive Gridding Tool dialog prior to saving.

- The **Save** button creates an output surface as specified on the **Output** tab. You do not have to wait for the preview window to complete drawing before saving the output grid file. If you are satisfied with the initial appearance of your grid you can click the **Save** button and the grid is saved in the same folder as the input data points. It is then loaded automatically into Encom Discover PA and placed in a display window specified by a default template.
- 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 Interactive 2D Gridding tool:

- **Automatic mode** – This is specified with the **Auto Apply** checkbox selected and is the default behaviour. In this mode, any changes to a control parameter are automatically applied and the grid recomputed when the cursor is moved to another tab or field in the dialog. At any stage you can manually apply a pending change by clicking on the **Apply** button. If the button is disabled then no changes are pending and no grid computation is performed.

- **Manual mode** – Deselecting the **Auto Apply** checkbox places the Interactive Gridding tool in manual mode and grid processing does not commence until you select the **Apply** button or change back to the Auto Apply mode. This mode is best suited to situations where you wish to modify a number of parameters prior to gridding, or have especially large datasets where the gridding operation may take some time.

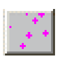
  Manual mode is automatically selected by the program if the dataset is detected as having a large number of data points. Consequently, with large datasets the gridding process does not automatically commence. You can override this by selecting the **Auto Apply** option or forcing a computation by clicking the **Apply** button.

A vertical progress bar is displayed next to the preview window in both modes and shows you the progress of the gridding operation. You can also monitor progress by watching the grid growing in the preview area (see *Previewing the Grid*).

Previewing the Grid

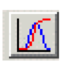
The appearance of the preview display is controlled by five toggle buttons and a pull-down list located above the preview area.

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. This displays a graphic illustrating the search ellipse used by the Inverse Distance Weighting gridding method. The size, shape and orientation of this ellipse is displayed in the preview window and will change as you change the search parameters. This control is only available for the Inverse Distance Weighted gridding method.



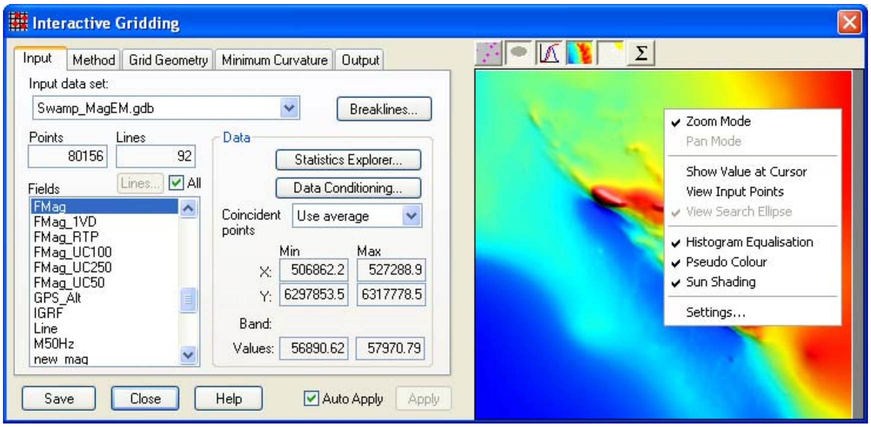
Apply a Histogram Equalisation stretch to the image in the preview window. This button can be used to distribute colours more evenly across the image and is particularly for data with poor dynamic range.

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 northeast direction.

Interrogate the interpolated surface using the Statistics Explorer. The individual grid nodes of the interpolated surface can be extracted and then examined in a spatial or frequency mode using the Statistics Explorer. Refer to *Appendix F: Statistics Explorer* for details on how to use this utility.

Additional controls are available by positioning the cursor in the preview window and clicking the right mouse button. A pop-up menu as shown below is displayed:



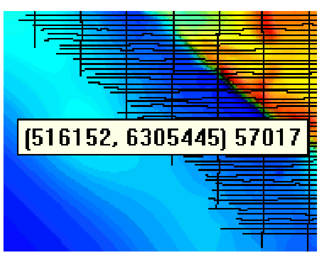
Preview area of the Interactive Gridding tool with pop-up menu .

Options available from the pop-up menu include:

- **Zoom Mode** – two levels of zoom are available – Normal and zoomed. The cursor changes format to indicate the level of zoom.

- **Pan Mode** – This mode is only available when zoomed in. Again use the right mouse button to select pan mode

- **Show Value at Cursor** – when enabled, a text reading of the location (row:column) plus the gridded value at the location is displayed. As the cursor is moved within the Preview area, the data location and value update in real-time.



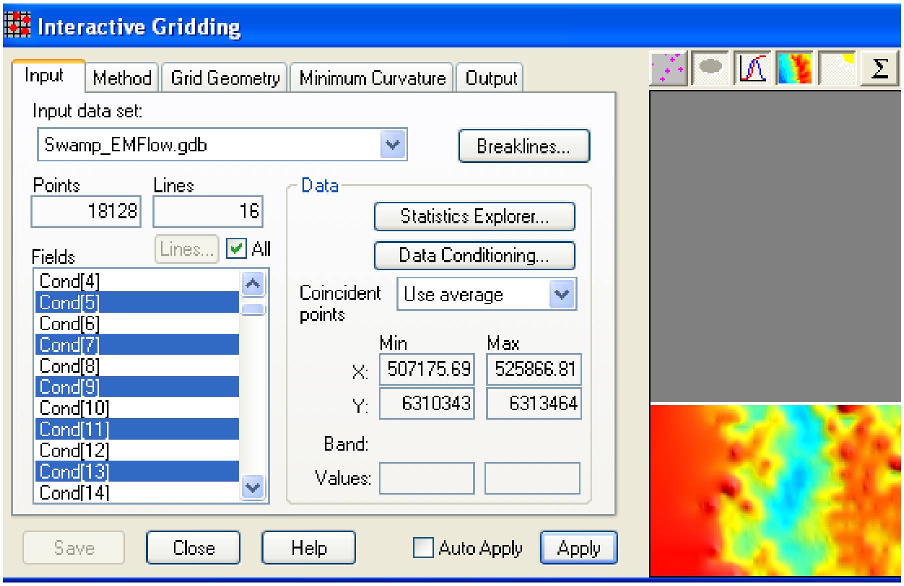
- **View Input Points** – Display the location of the data points used to create the surface

- **View Search Ellipse** – This option is only enabled if the Inverse Distance Weighting grid method is used.
- **Histogram Equalisation** – Apply a normal distribution to the colouring of data values across the area of the grid.
- **Pseudocolour** - You can select either a Pseudocolour colour table to be applied to the image in the preview window or a greyscale.
- **Sun-Shading** – An artificial north-east sun location can be applied in the display of a grid to indicate gradient and shading which enhances the presentation of the surface.
- **Settings** – By default a zoom level of 4 pixels per grid cell are used in the display of the preview area. You can control this if higher or lower resolution is required. Setting of the resolution is from a displayed dialog.

Input Tab

The **Input** tab summarises the input data and provides a selectable list of all numberic data columns that are available for gridding in the source database. Basic summary information is provided for the selected data including minimum and maximum easting (X) and northing (Y) values for the minimum bounding rectangle that fully encloses the input data and a count of the total number of input points.

Numeric data columns from the Input tab appear in the Field (bands) to grid list. To change the data field to be gridded, select the field name so it is highlighted. The data range for the selected column 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 you need to click the Apply button to generate and display the new grid.



The Input Tab dialog of Interactive 2D Gridding.

For information on using the other options available from the Input tab, see:

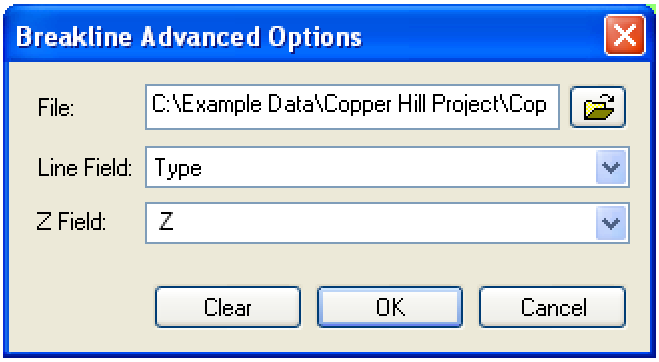
- *Breaklines*
- *Coincident Points*
- *Statistics Explorer*
- *Data Conditioning*

Breaklines

The **Breaklines** button enables support for break lines. Break lines are defined as multi-segment lines along each segment of which the slope is monotonically increasing or decreasing. Examples include stream or river traces, topographic ridge lines or roadways.

To load break line data, access the Breakline Advanced Options dialog from the **Breaklines** button on the **Input** property page.

The **Breakline Advanced Options** dialog enables you to browse and select an input TAB file. Only one input file can be selected. 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.



Breakline reference to a MapInfo Professional .TAB file

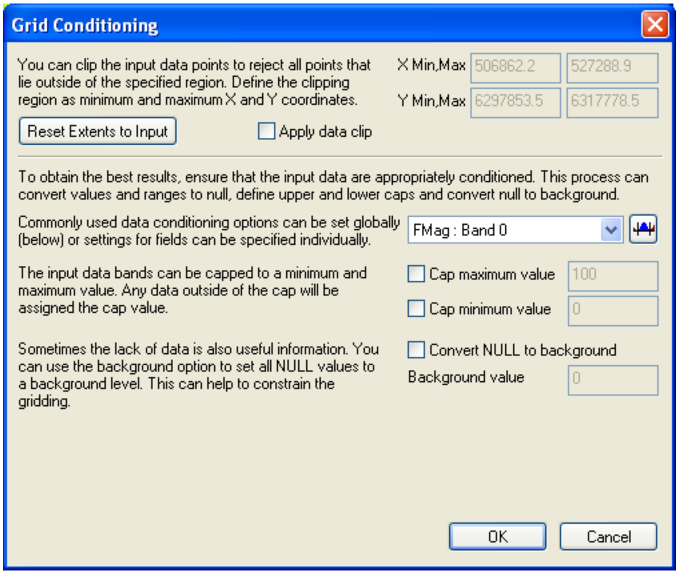
Coincident Points

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 the *Appendix F: Statistics Explorer*.

Data Conditioning



The Grid Conditioning dialog

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 Field Data Conditioning dialog (see *Appendix G: Field Data Conditioning*).

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

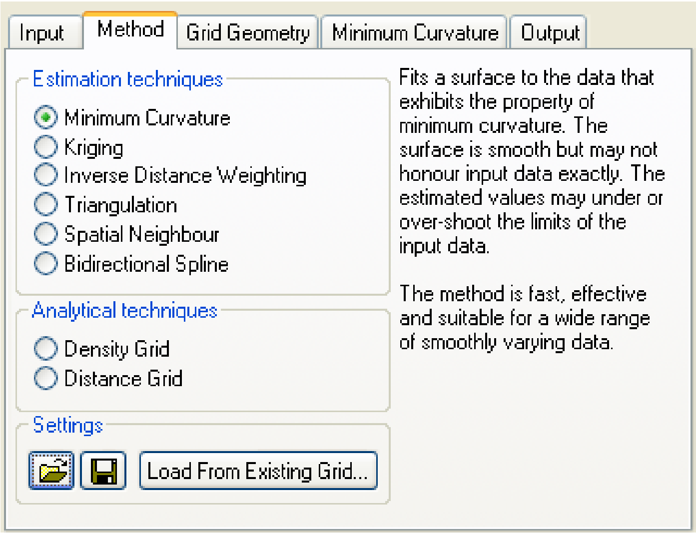
It is also possible to check the **Convert NULL to background** box and enter a user specified value in order to constrain the gridding. For example, if gridding drillhole geochemical assays, much of the hole may not have been sampled and in these areas the assay result may be assumed to be equal to 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 Input Tab (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 to the right of the listed input data field. If multiple data fields have been selected in the *Input Tab*, the **Advanced Settings** button can be used to open the Field Data Conditioning dialog, used to set individual capping and background values for each field by selecting the required field from the pull-down list. For more information on this refer to *Appendix G: Field Data Conditioning*.

Method Tab

Eight gridding methods are available: six estimation techniques and two analytical techniques.



The Method Tab with Minimum Curvature method selected.

The corresponding tab is added to the Interactive 2D Gridding tool when a method is selected:

- *Minimum Curvature Tab*
- *Kriging Tab*
- *Inverse Distance Tab*
- *Triangulation Tab*
- *Spatial Neighbour Tab*
- *Bi-Directional Spline Method*
- *Density Gridding Tab*
- *Distance Gridding Tab*

The last two analytical techniques produce grids of parameters derived from the spatial distribution of the input data:

Gridding parameters can be saved and restored or applied from another grid. For more information, see *Save, Restore, and Apply Gridding Parameters*..

Save, Restore, and Apply Gridding Parameters

The Gridding Tool computes the optimum gridding parameters by analysing the spatial distribution of the input data. You can recompute the optimum parameters at any stage by using the **Compute Best Values** button located on the Grid *Geometry Tab*.

You can also save grid parameters or apply pre-defined parameters:

The **Load Settings** button will present the user with a list of user-defined grid parameters. The grid configuration file GridTool.set, containing these settings, is located in the user configuration folder, by default this will be
C:\Documents and Settings\<user_name>\Local Settings\Application Data\Encom\IGridding\GridTool.xml
on Microsoft XP and older operating systems and
C:\Users\<user_name>\AppData\Local\Encom\IGridding\GridTool.xml
on Microsoft Vista operating systems.

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 grid configuration file GridTool.set is located in the user configuration folder, by default this will be
C:\Documents and Settings\<user_name>\Local Settings\Application Data\Encom\IGridding\GridTool.xml
on Microsoft XP and older operating systems and
C:\Users\<user_name>\AppData\Local\Encom\IGridding\GridTool.xml
on Microsoft Vista operating systems.

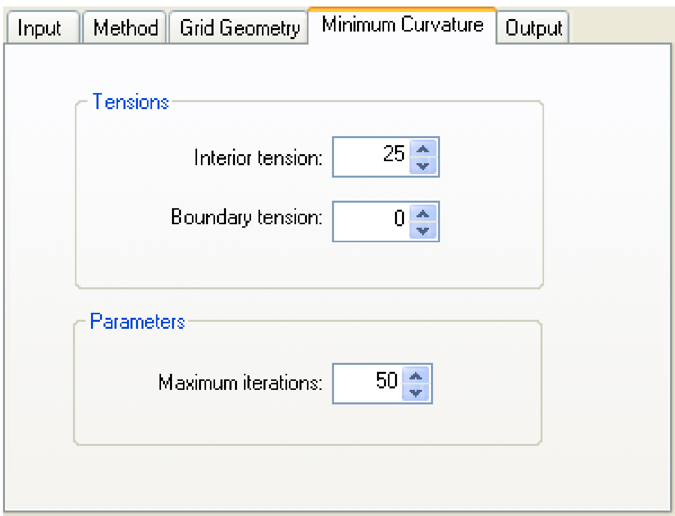
The **Load From Existing Grid** option is designed to load gridding parameters from an existing grid file. By default an Encom Discover PA-created grid will have an associated XML configuration file located in the same folder as the parent grid; to load these settings navigate to the associated XML file. Note that this option should only be used on the same dataset, as it will load all geometric settings for the grid such as grid extents and, cell size and band used.

Minimum Curvature Tab

The Minimum Curvature gridding method is widely used in many branches of science and research. The 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 criteria 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 but also near the edges. Minimum curvature gridding generates the smoothest possible surface while attempting to honour the data as closely as possible. Like all gridding methods, minimum curvature gridding is not an exact interpolation and therefore some error at a majority of data points is observed.

The minimum curvature method produces a grid by repeatedly applying an equation over the grid in an attempt to smooth the grid. Each pass over the grid is counted as one iteration and the number of iterations can be specified in the Gridding Tool. The grid node values are recalculated until successive changes in the values are minimised, or the maximum number of iterations is reached.

Minimum curvature attempts to fit a surface through all of the observations without putting any abrupt kinks in the surface. Between the fixed observation points, the surface bends according to the internal 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.



Minimum Curvature parameters specification

Kriging Tab

The geostatistical estimation method of kriging is a two stage process:

1. Study the gathered data to establish the predictability of values from place to place in the study area. This study results in a graph known as a semi-variogram which models the difference between a value at one location and the value at another location according to the distance and direction between them;

2. Estimate values at those locations which have not been sampled. This process is known as kriging. The basic technique “ordinary kriging” uses a weighted average of neighbouring samples to estimate the unknown value at a given location. Weights are optimised using the variogram model, the location of the samples and all the relevant inter-relationships between known and unknown values. The technique also provides a “standard error” which may be used to quantify confidence levels.

Over the past several decades kriging has become a fundamental tool in the field of geostatistics.

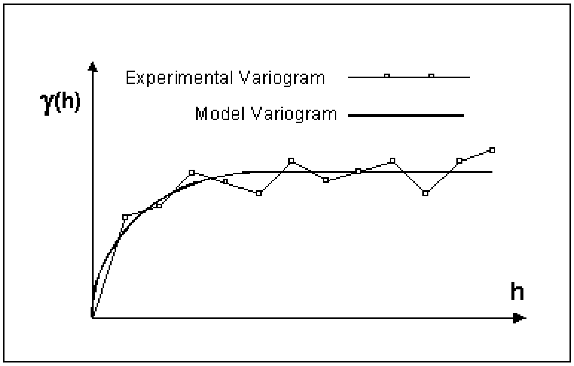
Kriging is based on the assumption that the parameter being interpolated can be treated as a regionalized variable. A regionalized variable is intermediate between a truly random variable and a completely deterministic variable in that it varies in a continuous manner from one location to the next and therefore points that are near each other have a certain degree of spatial correlation, but points that are widely separated are statistically independent. Kriging is a set of linear regression routines which minimize estimation variance from a predefined covariance model.

The first step in kriging is to construct a variogram from the scatterered points of the data to be interpolated. A variogram consists of two parts:

- An experimental variogram
- A model variogram.

Suppose that the value to be interpolated is referred to as `f`. The experimental variogram is found by calculating the statistical variance (`g`) of each point in the dataset with respect to each of the other points and plotting the variances versus distance (`h`) between the points. Several formulas can be used to compute the variance, but it is typically computed as one half the difference of f^2.

Once the experimental variogram is computed, the next step is to define a model variogram. A model variogram is a simple mathematical function that models the trend in the experimental variogram. As shown below, the shape of the variogram indicates that at small separation distances, the variance in f is small. In other words, points that are close together have similar f values. After a certain level of separation, the variance in the `f` values becomes somewhat random and the model variogram flattens out to a value corresponding to the average variance.



Experimental and Model Variogram used in kriging

Once the model variogram is constructed, it is used to compute the weights used in kriging. The basic equation used in ordinary kriging is as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i$$

where n is the number of scatter points in the set, f_i are the values of the scatter points, and w_i are weights assigned to each scatter 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*.

Selection of Kriging on the method tab opens both the *Search Tab* and *Kriging Tab* 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 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 Tab

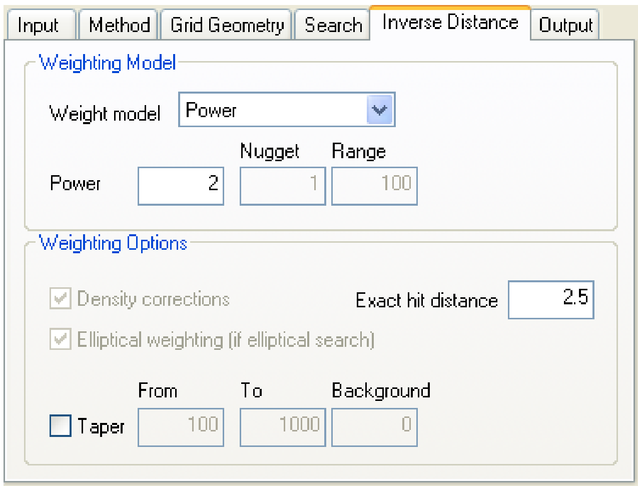
Inverse distance gridding is a universal technique that can be applied to a wide range of spatial data. IDW uses weighted average interpolation to estimate grid cell values and can be used as either an exact or a smoothing interpolator. Each grid cell value in an output surface is calculated using a 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 Tab* and Inverse Distance tab.

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*; 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.



Dialog controls of Inverse Distance tab of the Inverse Distance Weighting method

For descriptions of these controls, see:

- *Weighting Model*
- *Weighting Options*

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.

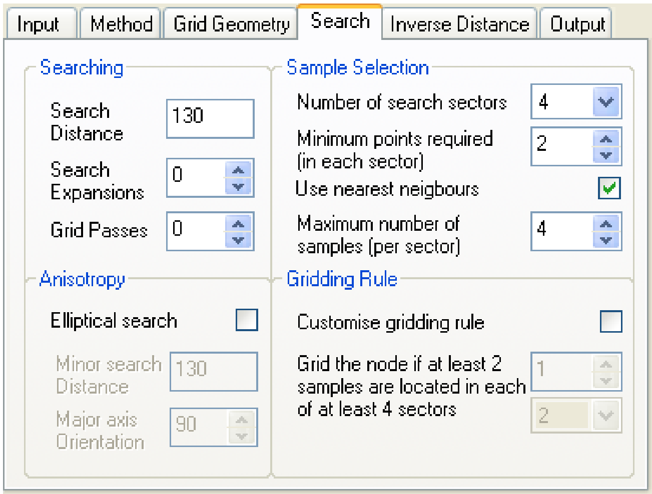
- The **Exact hit distance** is a tolerance distance for assigning actual input data values to coincident grid nodes. As the inverse distance gridding technique is attempting to interpolate a continuous surface through the data, a certain number of grid nodes coincide with the input data points. Where 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 Inverse Distance Weighting methods, when selected on the *Method Tab*, 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.



Dialog controls of Search tab of the Inverse Distance Weighting method

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 Encom Discover PA 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, and 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, 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 on 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.

Encom Discover PA 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. When this option is selected, Encom Discover PA 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.

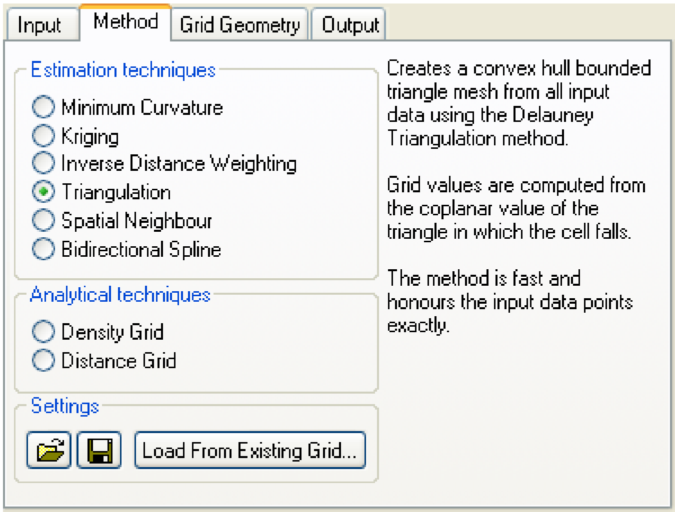
Gridding Rule

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.

Triangulation Tab

The Triangulation method produces a regular gridded surface through a set of data points by first creating a triangular mesh where each input point is located at the vertex of a triangle. Triangle edges are not allowed to intersect. The triangular mesh is created by drawing lines between adjacent input data points which form an irregular network such that no two triangle edges intersect. A regular grid is then computed from the triangular irregular network by linear interpolation at each grid node. Because the original data points form the vertices of the triangles, this method is useful in situations where the data must be honoured very closely (for example, elevation data in a digital terrain model).

The triangulation method is best applied to data that is evenly distributed over the gridded area. If you have large areas of sparse or missing data you may end up with distinct triangular facets in the output grid. Because triangulation uses all the input data to construct the triangular mesh, the only variable value that can be adjusted is the grid cell size.

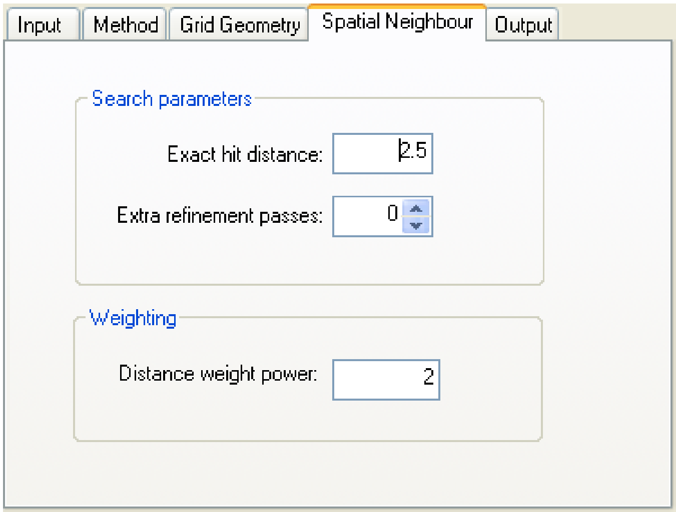


The Method tab with the Triangulation method selected.

Spatial Neighbour Tab

This method is similar to the Inverse Distance Weighting method 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 acerage 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.



The Spatial Neighbour weighting and search specification tab.

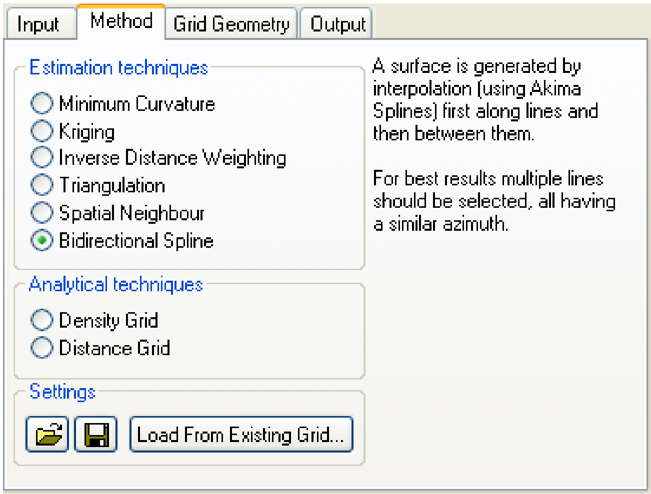
A triangular mesh is first formed. Then for each grid node the nearest data point is determined and contributions from that point and its immediate neighbours (the points that are joined directly to it via triangle edges) are combined using inverse distance weighting parameters to determine the value of the grid node.

The main difference between this technique and Inverse Distance Weighting is the fact that you don't need to specify a search radius. The search and weighting parameters can be specified on the Spatial Neighbour tab, displayed when this grid method is selected.

Bi-Directional Spline Method

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.



The Method tab dialog with the Bi-directional method selected.

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.

Density Gridding Tab

The Density gridding method produces a grid which records a measure of the input point density at every grid node. The density at each grid node is determined independently using an estimator function. Two estimators are available:

- 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 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 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.

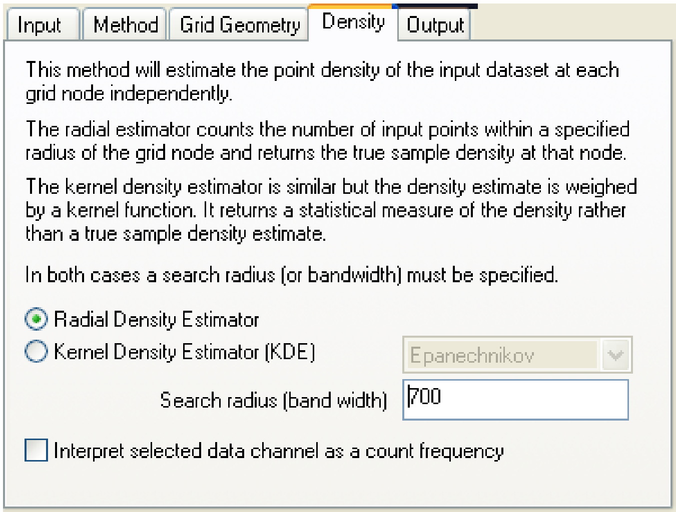
Given a kernel function K and a search radius (or bandwidth) h, the estimated density at any point x is given by:

$$\hat{f}(x) = \frac{1}{n}\sum_{i=1}^{n} K(\frac{x - x(i)}{h})$$

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 **Input** tab, ensure the frequency or count field is selected as the input data field.



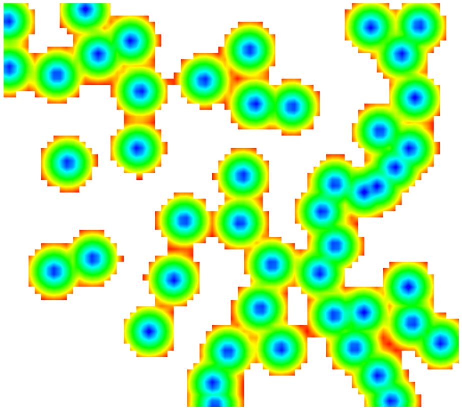
Density tab dialog options

Distance Gridding Tab

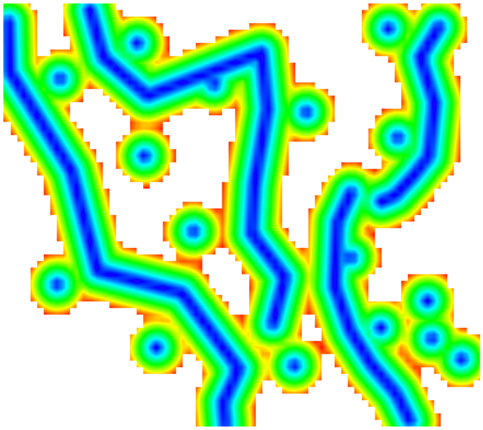
The Distance gridding method produces a grid which records the minimum distance to the nearest input data feature at every grid node. A distance envelope can be specified to clip the grid at a maximum distance from the nearest feature. Any grid node that is outside this envelope will be assigned a null value.

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 is 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 considers all input data as point locations. Optionally, you can add to this the input polyline data. In this case the method also checks the perpendicular distance to the nearest polyline in addition to looking at all point data.

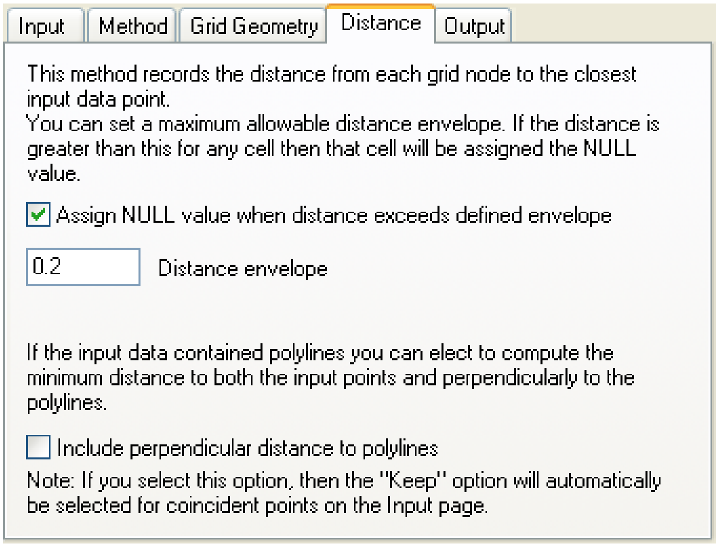


Point data classification



Polyline data classification

In the illustrations above, 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 dialog options

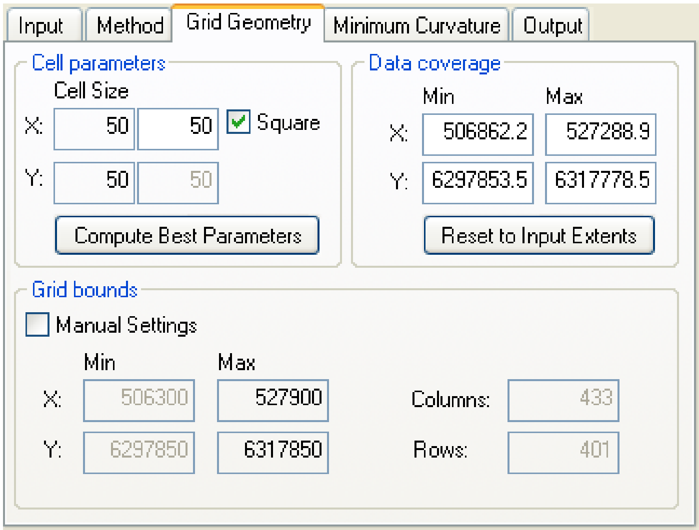
Geometry Tab

This tab controls the main parameters for determining the geometry of the output grid. The **Cell Parameters** determines the size of each grid cell in the output image and is measured in metres. During the loading process, the Gridding Tool automatically computes a grid cell size based on the distribution and density of the input data. If you wish, you can alter the cell size to better suit your needs.

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 minimum and maximum 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. To return to the original data extents, select the **Reset to Input Extents** button. 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.



The Grid Geometry tab indicating the cell size, extent and number of rows and columns

When creating a surface, the grid cell size that you choose 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 introduce unwanted irregularities in the output image. Assigning an excessively small grid cell size also increases computation time and file size. The Gridding Tool computes a grid cell size for your data but you may want to experiment with alternative values.

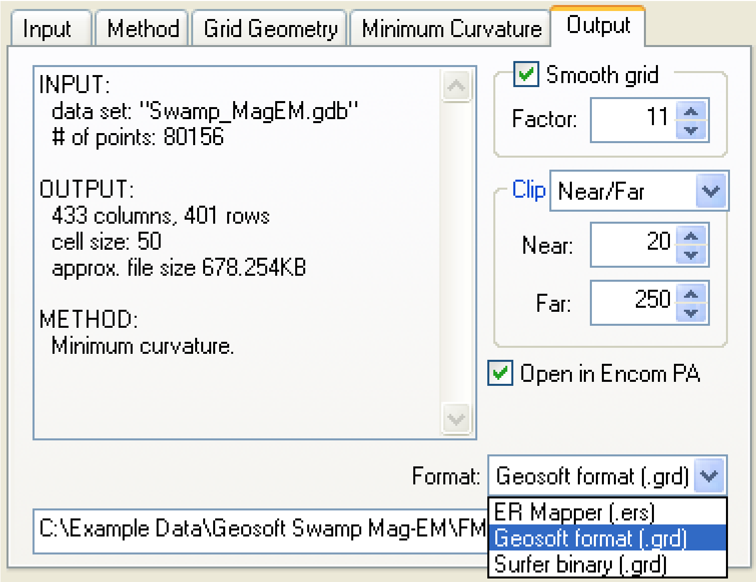
Output Tab

The Output tab provides controls for saving the gridded image. The top section of the dialog provides a summary report of the gridding process and lists the primary parameters chosen. You can copy this information to the Windows clipboard by selecting it with your mouse and using Windows' copy command and paste it into Notepad for saving or printing.

Below the summary box is a pull-down list containing the supported grid formats. The Gridding Tool currently supports ER Mapper (.ERS) and Geosoft (.GRD) uncompressed grid formats. To save a grid, select the appropriate format from the list and click the **Save** button. The output grid file will be assigned a default name and is saved in the same directory as the input data. You can alter 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 gridding so it is not necessary to wait for the preview window to complete drawing before saving the final grid.

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.



The Output tab dialog with statistics and grid saving options

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. MAG, 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. The RGB grid is first loaded into the **Grid>Grid Utility>Create RGB Grid** dialog before the final RGB GeoTiff image is created.

Large and Multi-file Gridding

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

The **Large and Multi-file gridding** tool is a powerful grid interpolator which can construct gridded surfaces from extremely large datasets. Interpolated grids can be created from a single input file or from multiple input files of point data or gridded data in any of the following formats:

- ASCII text (.txt, .asc, csv, xyz), including zip compressed ASCII
- MapInfo .TAB
- ASPRS .LAS
- LizardTech MrSID compressed Lidar (MG4)
- Gridded formats (Arc ASCII, Arc Flt, ER Mapper, BIL, MapInfo, Geosoft, MinEx, Surfer, Vertical Mapper)
- Geosoft database .GDB

The Large and Multi-file gridding tool is best suited to gridding very large, closely spaced point datasets, such as those commonly acquired by airborne Light Detection and Ranging (LIDAR) data surveys. The tool can also be used to assemble a large number of adjoining grid tiles into a single large contiguous grid. An example of this use case is assembling multiple Shuttle Radar Topography Mission Arc ASCII grid files into a single continuous grid. This method of interpolation works best with relatively even spaced points that cover semi-rectangular survey areas containing minimal internal holes or areas of missing data.

For moderate-sized point datasets (e.g. < 2 million points) or situations where some experimentation with interpolation methods is required the Interactive gridding tool is a more flexible and suitable choice.

- *When to Use Large and Multi-file Gridding*
- *Using the Large and Multi-file Gridding Tool*
- *Input Data Files*
- *Gridding Methods*
- *Interpolation Settings*
- *Output Grid*

When to Use Large and Multi-file Gridding

As a rule of thumb if the number of points in the input dataset exceeds 3 million points or the number of cells in the output grid is greater than 6,000 columns x 6,000 rows (360,000,000 cells) or there are multiple input files the Large and Multi-file gridding tool is the appropriate tool to use. For all datasets which fall below this size threshold the interactive gridding tool is the most flexible and best tool to use. For practical purposes the maximum size of the combined input datasets that the Large and Multi-file gridding tool can process is 2 billion points and the maximum size of the output grid(s) is 1 terabyte.

Using the Large and Multi-file Gridding Tool

- *Running the Gridding Tool*
- *Re-running the Gridding*

Running the Gridding Tool

For more information about the settings available from dialog boxes, see *Interpolation Settings*.

1. On the **Surfaces** menu, point to **Create Grid**, and click **Large and Multi-file**.

   The **Large and Multi-file Gridding** dialog box is displayed.

2. Under **Input**, to select input files that are stored in a single directory:

   - Press the **Browse for files** button on the top right of the dialog. On the open file dialog adjust the file type filter at the bottom to the appropriate type (.TAB, . ASCII, .LAS, .SID, .ZIP or one of the supported grid formats) and select the files to grid.

   Or for input files that are stored in a multiple directories:

   - Press the **Browse for folder** button, second button down on the top right of the dialog. On the **Select file format** dialog which appears choose the file type filter which is appropriate for the type of input data, press OK and select the top level directory from which to search for files to grid.

3. Depending on the format of the files chosen you may be prompted to assign the column field order for the input dataset. To do this press the Assign **Column order** button. On the **Select data fields** dialog choose the field for the X coordinate and Y coordinate. If the input dataset(s) are LIDAR LAS files these fields will be automatically assigned. From the field list select the data column(s) you wish to grid. If the chosen input files are zip compressed files and the zip file contain more than one file (e.g. an ASCII data file and a metadata or .PRJ file) you will need to specify a file name mask to filter out the correct file to grid. Under Input data format enter a file name mask to uniquely identify the name of the file in the .zip archive that you wish to grid. The mask can contain characters that are part of the file name and/or the extension. For example *.txt or myfile.* or *.asc are all valid masks. The mask will be applied automatically and when a valid file is found you will see a preview of the content in the File preview region.

4. Assign the appropriate projection for the input data files by pressing the **Choose Projection** button in the bottom right of the dialog.

Note If you are unfamiliar with the distribution of statistics of your input files you can examine the spatial statistics of any of the selected input data files using the Statistics Explorer. To launch the Statistics Explorer select an input file and then press the **Statistics Explorer** button  (on right of file format dialog).

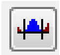
---

5. If desired, modify the name that has been assigned in the **Group alias** property for the group in the input manager grid control by double-clicking in the cell and typing in a new name.

6. To change the properties of a group of input files, first select the group you wish to modify in the input manager and then press the **File format** button on the right of the dialog. On the format dialog you can add or remove files from a group as well as change the input data format and projection.

7. If you are familiar with the spatial distribution and coverage of the input data enable the **Define cell size** control and change the cell size from <Auto> to an appropriate value that is suitable for the input data and for your analytical needs. Should you need assistance in choosing an appropriate cell size you can press the **Compute cell size** button and the software will attempt to analyse a small subset of the data and estimate an appropriate value. However it is generally recommended that if you are not sure what cell size to assign for the output grid that you leave the Define cell size disabled so the software can compute an appropriate value once it has completed an analysis of the data distribution.

8. Optional steps:



- Press the **Data conditioning** button to configure any null values or data ranges you may wish to exclude during the gridding process.
- Define the grid geometry by specifying the **Origin X** and **Origin Y** coordinate of the lower left corner of the output grid as well as the grid dimension in rows and columns.
- Press the **Advanced settings** button and modify the Options as necessary. For a large gridding job you may need to set the Temporary directory path to an alternative hard drive which has plenty of free space available to store an entire copy of the input dataset. You may also wish to adjust the Scan data extents control to perform a quick scan of the input data as well as constrain the maximum memory that will be used by the software during the gridding process.. Use the Output grid Data type to change the numeric data type that will be used for the output grid. Choosing an appropriate data type for the output grid could have a large impact on the size of the output grid file. See *Output Grid* for more information

9. Press the **Save** file button and select an appropriate file path for the final grid file(s). Ensure that you select a location with plenty of free space to store the output files. It is recommended that for large datasets the output grid(s) are stored on a separate disk drive to the input data.

10. Press the **Projection** button and choose an appropriate projection for the output grid file. It is recommended that the projection of the output grid file is set to the same projection as the input file(s). By default the output projection will be set to the first input file(s) projection.
11. (Optional) Disable the **Auto open output file(s)** if you do not want the output grid file(s) to be automatically opened into MapInfo on completion of the gridding process.
12. Press the **Process** button to commence gridding.

Re-running the Gridding

At the completion of gridding a control file containing a complete set of parameters used by the gridding engine are saved to a text file in the same location as the output grid. The control file is created with the same name as the output grid file and has .final.txt appended to the end.

Should you wish to create another grid using a different field from the same input data or recreate the same grid with slightly modify settings you can reload the saved control file. To do this first open the Large and multi-file gridding tool. Next press the load control file button (the last button at the top right of input manager). When the control file is successfully loaded you can modify the settings as necessary and then press the Process button to create a new grid.

Input Data Files

The main features of input files that can be read are:

- One or more input files are supported.
- A range of commonly used input formats can be read including .TAB, .LAS, .CSV, space or tab delimited text files (including zip compressed files),, MrSID MG4 LIDAR files and a number of common grid files.
- Input files can be located in a single directory or hierarchically in multiple directories.
- Input files can be specified in groups. A group of input files share common properties such as format (i.e. file structure) and projection. If the file format or projection of any of the input files is different they should be separated into their own group.

Input file formats

Input files can be provided in any of the following formats:

- **MapInfo .TAB**
  The TAB reader can read simple MapInfo .TAB files which contain mapped objects. The coordinate and projection information is extracted from the mapped objects and the X,Y point coordinates are obtained for every node of simple or multi-part objects. The data field information which is used for interpolation can be obtained from any corresponding attribute field in the .TAB file.
- **ASPRS .LAS**
  LAS is an industry standard public format used for the interchange of LIDAR data. The file format is defined and maintained by the ASPRS organisation (*www.asprs.org*). Discover supports versions 1.0, 1.1 and 1.2 of the LAS specification and can extract "Z" (elevation) as well as RGB (red, green, blue) and intensity information.

- **MrSID MG4 compressed LiDAR**
  MrSID MG4 LiDAR files are a new version of the MrSID format that supports compressed LiDAR point data. The files are created using the LizardTech LiDAR Compressor™ software. For more information on this format please refer to LizardTech (*http://www.lizardtech.com*). These files have an extension of *.MG4 and may be loaded directly into the gridding tool.

- **Geosoft database (GDB)**
  Standard geophysical data format.

ASCII data readers

- **ASCII Space, Comma or Tab Delimited (Fast)**
  This is the faster of the two text readers. It expects data to be stored in columns with a space, tab or comma separating each value in a row. The text reader can skip multiple header lines at the start of a file but once it starts reading data it expects the remainder of the file to be clean and void of inline comments or other formatting changes. For well structured, clean data this is the best method to use.

- **ASCII Generic Text (Slow)**
  This is a general text reader which handles a wider range of delimiters. It can also detect and skip header lines as well as empty fields (e.g. consecutive commas with no data between them) and can adjust to formatting changes within the input file(s). It will also read files with unusual delimiters such as a pipe (|).

Grid file readers

- There are several readers available for loading gridded data files (e.g. *.ERS, *.GRD, *.BIL, *.ASC) as input to the Large and Multi-file gridding tool. All the common industry grid formats that are supported by Discover can be loaded as input.

Input file groups

- Projection information can be assigned to each group of input files. The projection information specified must correspond to the spatial reference the data is stored in. If no projection is assigned to the input data it will be assigned the projection of the output grid.

- You may modify the group alias name for each group of input files to help distinguish them in situations where a lot of groups have been added.

Gridding Workflows

The following gridding methods are available:

- *Triangulation*
- *Inverse Distance Weighting*
- *Minimum Curvature (Full)*
- *Minimum Curvature (Stamped)*
- *Data Density*
- *Data Point Separation (Distance)*
- *Stamp Only*

Triangulation

The *Triangulation* gridding method consists of eight operational phases:

Phase 1. A quick analyse of your computer is performed to establish the amount of RAM and free disk space available and all input files are validated.

Phase 2. The first 65,000 data stations are loaded to determine spatial statistics of the data.

Phase 3. Each input data file is scanned to determine the data extents and spatial distribution statistics. Some data will be skipped during this phase if the Scan data extents property is set to any value other than complete

Phase 4. Preliminary gridding parameters and temporary data storage requirements are estimated from the spatial statistics obtained during the initial data scan (phase 3)

Phase 5. Gridding properties and memory requirements are established

Phase 6. The entire input data set is read and spatially sorted into a collection of tiles. Unwanted or null readings are cleansed and coordinates are reprojected into the output grid coordinate system if required.

Phase 7. The spatially sorted input data is triangulated and the interpolated values are written to the output file.

Phase 8. The triangulated grid(s) are finalized and corresponding header information and.TAB files are written to disk. All temporary files an allocated memory is then released.

Inverse Distance Weighting

The *Inverse Distance Weighting* gridding method consists of seven operational phases:

Phase 1. A quick analyse of your computer is performed to establish the amount of RAM and free disk space available and all input files are validated.

Phase 2. The first 65,000 data stations are loaded to determine spatial statistics of the data.

Phase 3. Each input data file is scanned to determine the data extents and spatial distribution statistics. Some data will be skipped during this phase if the Scan data extents property is set to any value other than complete

Phase 4. Preliminary gridding parameters and temporary data storage requirements are estimated from the spatial statistics obtained during the initial data scan (phase 3)

Phase 5. Gridding properties and memory requirements are established

Phase 6. The entire input data set is read and unwanted or null readings are cleansed and coordinates are reprojected into the output grid coordinate system if required. The points are stamped into a spatially indexed file and an inverse distance weighting algorithm is applied to interpolate new grid values from the set of input points that lie within the specified search radius (*Inverse Distance Weighting Properties*).

Phase 7. The interpolated data file is finalized and written to the output grid file along with the corresponding header information and.TAB files. All temporary files and allocated memory is then released

Minimum Curvature (Full)

The *Minimum Curvature (Full)* gridding method consists of thirteen operational phases:

Phase 1. A quick analyse of your computer is performed to establish the amount of RAM and free disk space available and all input files are validated.

Phase 2. The first 65,000 data stations are loaded to determine spatial statistics of the data.

Phase 3. Each input data file is scanned to determine the dataset extents and spatial distribution statistics. Some data will be skipped during this phase if the **Scan data extents** property is set to any value other than **complete**.

Phase 4. Preliminary gridding parameters and temporary data storage requirements are estimated from the spatial statistics obtained during the initial data scan (phase 3)

Phase 5. Gridding properties and memory requirements are established

Phase 6. The entire input dataset is spatially sorted into a collection of tiles. Unwanted or null readings are cleansed and coordinates are reprojected into the output grid coordinate system if required.

Phase 7. The spatially sorted input data is now stamped into the grid.

Phase 8. A multi-resolution grid pyramid is constructed and filled from bottom to top.

Phase 9. The clip mask is built, based on the clipping parameters defined in the minimum curvature properties dialog.

Phase 10. The source mask pyramid is focussed in each level to allow more grid cells to be unconstrained in the gridding procedure.

Phase 11. The pyramid is filled and refined from the top down applying minimum curvature at each level.

Phase 12. The final phase of minimum curvature interpolation is performed upon the output grid.

Phase 13. The output grid(s) are finalized and written to disk along with their corresponding header information and.TAB files. All temporary files are removed and allocated memory is released

Minimum Curvature (Stamped)

The *Minimum Curvature (Stamped)* gridding method consists of twelve operational phases:

Phase 1.	A quick analyse of your computer is performed to establish the amount of RAM and free disk space available and all input files are validated.
Phase 2.	The first 65,000 data stations are loaded to determine spatial statistics of the data.
Phase 3.	Each input data file is scanned to determine the dataset extents and spatial distribution statistics. Some data will be skipped during this phase if the **Scan data extents** property is set to any value other than **complete**.
Phase 4.	Preliminary gridding parameters and temporary data storage requirements are estimated from the spatial statistics obtained during the initial data scan (phase 3).
Phase 5.	Gridding properties and memory requirements are established
Phase 6.	The entire input dataset is now stamped into the grid. Unwanted or null readings are cleansed and coordinates are reprojected into the output grid coordinate system (if required).
Phase 7.	A multi-resolution grid pyramid is constructed and filled from bottom to top.
Phase 8.	The clip mask is built, based on the clipping parameters defined in the minimum curvature properties dialog.
Phase 9.	The source mask pyramid is focussed in each level to allow more grid cells to be unconstrained in the gridding procedure.
Phase 10.	The pyramid is filled and refined from the top down applying minimum curvature at each level.
Phase 11.	The final phase of minimum curvature interpolation is performed through the pyramid.
Phase 12.	The output grid(s) are finalized and written to disk along with their corresponding header information and TAB files. All temporary files are removed and allocated memory is released.

Data Density

The *Data Density* gridding method consists of seven operational phases:

Phase 1. A quick analyse of your computer is performed to establish the amount of RAM and free disk space available and all input files are validated.

Phase 2. The first 65,000 data stations are loaded to determine spatial statistics of the data.

Phase 3. Each input data file is scanned to determine the dataset extents and spatial distribution statistics. Some data will be skipped during this phase if the **Scan data extents** property is set to any value other than **complete**.

Phase 4. Preliminary gridding parameters and temporary data storage requirements are estimated from the spatial statistics obtained during the initial data scan (phase 3)

Phase 5. Gridding properties and memory requirements are established

Phase 6. The entire input dataset is now scanned and unwanted or null readings are cleansed and coordinates are reprojected into the output grid coordinate system (if required).The points are stamped into the grid file and accumulated densities are computed within the specified search radius using chosen the kernel weighting function.

Phase 7. The output grid(s) are finalized and written to disk along with their corresponding header information and.TAB files. All temporary files are removed and allocated memory is released

Data Point Separation (Distance)

The Data Point Separation gridding method consists of seven operational phases:

Phase 1. A quick analyse of your computer is performed to establish the amount of RAM and free disk space available and all input files are validated.

Phase 2. The first 65,000 data stations are loaded to determine spatial statistics of the data.

Phase 3. Each input data file is scanned to determine the dataset extents and spatial distribution statistics. Some data will be skipped during this phase if the **Scan data extents** property is set to any value other than **complete**.

Phase 4. Preliminary gridding parameters and temporary data storage requirements are estimated from the spatial statistics obtained during the initial data scan (phase 3)

Phase 5. Gridding properties and memory requirements are established

Phase 6. The entire input dataset is now scanned and unwanted or null readings are cleansed and coordinates are reprojected into the output grid coordinate system (if required). The points are accumulated and separation distances are computed within the defined search radius. The interpolated values are stamped into the output grid file.

Phase 7. The output grid(s) are finalized and written to disk along with their corresponding header information and.TAB files. All temporary files are removed and allocated memory is released.

Stamp Only

The Stamp Only gridding method consists of seven operational phases:

Phase 1. A quick analyse of your computer is performed to establish the amount of RAM and free disk space available. All input files are validated. Note depending on the number, size and type of input files selected this phase can take some time.

Phase 2. The first 65,000 data stations are loaded to determine spatial statistics of the data.

Phase 3. Each input data file is scanned to determine the dataset extents and spatial distribution statistics. Some data will be skipped during this phase if the **Scan data extents** property is set to any value other than **complete**.

Phase 4. Preliminary gridding parameters and temporary data storage requirements are estimated from the spatial statistics obtained during the initial data scan (phase 3)

Phase 5. Gridding properties and memory requirements are established

Phase 6. The entire input dataset is now scanned and unwanted or null readings are cleansed and coordinates are reprojected into the output grid coordinate system (if required). The points are then stamped directly into the output grid file(s).

Phase 7. The output grid(s) are finalized and written to disk along with their corresponding header information and.TAB files. All temporary files are removed and allocated memory is released.

Interpolation Settings

- *Data Conditioning*
- *Gridding Properties*
- *Options*

Data Conditioning

Data conditioning is a pre-processing operation which is applied to the input data prior to gridding. It allows you to define a NULL value as well as ranges of data, with upper and lower caps, in order to exclude them from the gridding process. The data conditioning process only applies to valid input data. Should any of the fields in the input data set for a given point (i.e. X, Y or the field values) have a NULL (or missing) value then the entire point is discarded. For more information, see *Data Conditioning*.

Gridding Properties

Because the gridding process is relatively autonomous it is possible to leave the gridding properties set to their default settings and the software will attempt to automatically compute an appropriate set of parameters for the output grid by examining the field and spatial statistics of the input file(s). If you are unfamiliar with the distribution or range of the input data sets then leaving the gridding properties on their default settings is recommended.

If you are familiar with the input dataset and know in advance what the spatial extents and distribution of the data is like and you have a good understanding of the field range then you can manually adjust the gridding properties to best suite the input and output data requirements.

Define cell size

This property controls the size (or resolution) of each cell in the output grid file. The size is measured in the spatial units of the output grid coordinate system. The default cell value is "Auto" which indicates that the software will attempt to compute an appropriate cell size for the output grid by analysing the spatial statistics of the input data. Providing the source data is regularly spaced and evenly distributed across the entire survey area the automatic cell size estimation algorithm will choose a reasonable cell size. However if the data distribution is clustered or skewed then it is strongly recommended that you set the cell size to an appropriate value manually.

As a general rule of thumb the output grid cell size should not be set to a value that is less than $^{1}/_{5}$ the average spacing of the input data. A cell size of $^{1}/_{3}$ the average data spacing is good starting point. Another important aspect of the cell size which must be taken into consideration, particularly on large datasets, is that halving its size will have the effect of double the file storage requirements of the output grid. For example if you set a cell size of 2 m and the output grid requires 100 MB of disk space, then reducing the cell size to 1 m will increase the storage space requirements for the output grid to 400 MB.

Compute cell size

The compute cell size button will compute a default cell size for the input data. It attempts to do this by analysing the spatial statistics of a small subset (approximately 65,000 stations) of the first input file. Depending on the regularity and spatial coherence of this small subset of data the estimated cell size may or may not be assigned an optimal value.

If you are unfamiliar with the dataset and you are not sure what value to set for the cell size then it is recommended that you leave the property set to "Auto". When configured this way the software will attempt to adjust the cell size to an appropriate value after it has examined the complete spatial statistics of the input data.

Define grid geometry

The define grid geometry properties control the spatial extent and size of the output grid file. By default these values will be automatically computed by the software to fully encompass the input data once it has scanned all the data files and established the spatial statistics and data distribution. If you need to constrain the output grid to a larger or smaller area than the extents of the input dataset then you can override the default settings and specify the extents manually. By configuring the grid extents to an area which is smaller than the extents of the input dataset you will effectively clip it to the defined grid geometry. If the output grid geometry is smaller than the input data then any input data that does not overlap with it will be discarded prior to gridding. The grid geometry is controlled by the Origin X and Origin Y coordinate values, which specify the centre coordinate of the lower left grid cell and the height and width are specified by the number of rows and columns measured in grid cells respectively.

Advanced settings

The advanced settings allow you to control a number of properties relating to the memory and temporary storage location that is used during gridding, the resolution at which the initial data scan is done, and output grid data type. Details of each of these properties are described below.

Options

Maximum memory use

This control is used to constrain the amount of physical memory (RAM) that the software will attempt to use during the gridding process. The default control setting is disabled and in this state the software will attempt to use up to 80% of available physical memory (to a maximum of 2 GB) during gridding. While it is possible to constrain the amount of RAM that is available to the software during the gridding process; doing so will reduce the gridding performance significantly if the software has to repeatedly page tiles of data between disk and memory in order to perform the gridding operation.

To achieve best performance on large datasets it is advisable that you close all running applications and free up as much physical memory (RAM) as possible before commencing gridding. For very large datasets (>50 million points) it is recommended that the software is run on a machine with between 2 and 4 GB of RAM.

Note The gridding tool is a 32-bit program, and therefore cannot address more than 4GB of RAM on a 64-bit operating system.

Scan data extents

Controls the resolution at which the software initially scans the input data files to establish the preliminary spatial statistics during the first phases of gridding. The default behaviour is to scan all lines of the input data. It is possible to speed up the initial scan of the input files by adjusting the scan data extents control to one of the following settings:

- **Complete**: scans every line of each input data files
- **Fine**: scans approximately 12% (1-in-8) of the lines from each input data file
- **Course**: scans approximately 3% (1-in-32) of the lines from each input data file
- **Overview**: scans approximately 0.75% (1-in-128) of the lines from each input data file
- **Bounds**: acquires the data extents from information stored in the files (e.g. LAS or grid formats such as .ERS and, GRD) if available or performs an Overview scan if unavailable.

For datasets that have a relatively even spatial distribution of input points setting the scan data extents control to Overview will provide the best compromise between speed and a representative statistical sample.

Temp directory

The Temp directory is used to temporarily store the spatially sorted input data tiles which are used during the gridding process. If all of the input data can fit into system memory then no temporary files will be created and the entire process will occur in RAM. If the input dataset is very large (>10 million points) then it will be necessary to store a copy of the input data on disk during the gridding process. By default the temporary directory is set to the Windows system temporary directory; however it may be necessary to map it to an alternative storage location depending on the size of the input dataset.

Note You should always ensure that you have at least the same amount of free temporary storage space as the total size of the input dataset. A good rule of thumb is to set the temp path to a location that has 2x as much storage space as the size of the input dataset.

Gridding Methods

- *Triangulation*
- *Inverse Distance Weighting*
- *Minimum Curvature (Stamped)*
- *Minimum Curvature (Full)*
- *Data Density*

Triangulation

The triangulation gridding method performs a Delaunay triangulation with linear interpolation. The method works by first triangulating the input data into a TIN mesh. It then calculates a value at the centre of each cell in the output grid using linear interpolation from the triangle that overlaps with coordinate of the centre of each grid cell. In order for the software work with huge datasets that could potentially contain billions of triangles on PCs with limited available memory, the input data is scanned, spatially sorted and divided into tiles. Each of the tiles is then triangulated and the resulting TIN mesh is either stored in memory or if there is insufficient memory available it is stored on disk in temporary files. Once the dataset has been triangulated into a TIN mesh the output grid is constructed by interpolating cell values from the triangles.

Because the triangulation process is relatively autonomous it is possible to leave the gridding properties set to their default settings and the software will attempt to automatically compute an appropriate set of parameters for the output grid by examining the field and spatial statistics of the input file(s). If you are unfamiliar with the distribution or range of the input data sets then leaving the gridding properties on their default settings is recommended.

If you are familiar with the input dataset and know in advance what the spatial extents and distribution of the data is like and you have a good understanding of the range of values in the field(s) being gridded then you can manually adjust the gridding properties to suite the input and output data requirements.

Triangulation Properties

The triangulation properties can be used to influence the geometry of the data structures, the number of points and memory that will be used to triangulate each tile of input points.

Maximum triangle side length

This parameter applies to the triangulation phase of gridding and can be used to minimize or eliminate long thin triangles that may be created across large holes or gaps in the data or between widely separated points that lie around the perimeter of the dataset. By default triangles that are created with a length that is greater than half the diagonal length of a tile will be discarded. The size of the tiles used during triangulation are determined automatically by the software after it analyses the spatial statistics of the input points, however you can modify the size of the tiles by applying a *Triangle patch multiplier*.

Distance specified in data units

This parameter is used to control the units of distance that the Maximum triangle side length property is measured in. By default this control is disabled and the maximum distance unit is expressed as a ratio of the tile (or patch) size. If you wish to constrain the Maximum triangle side length to a fixed value that is measured in absolute data units (e.g. 100 m) then enable this control and enter in the appropriate value. If the entered value is large and exceeds the size of an individual tile of data then it may have no effect on the output grid.

Note: If the coordinate system of the input data is Longitude/Latitude then the absolute distance units need to be specified in fractions of a degree (Arc seconds).

Triangle patch multiplier

The triangle patch multiplier can be used to modify the number and size of tiles (or patches) that the software will segment the input data into before sequentially triangulating it. The tile size is automatically determined by the software following a detailed analysis of the spatial statistics of the input data. Under special circumstances the patch size can be modified by applying a Triangle patch multiplier. Increasing the default value of 1 to a higher number will create larger patches and may assist in the infilling of large holes or gaps in the dataset, however it will also increase peak memory usage during the gridding phase. For very large datasets increasing the Triangle patch multiplier will reduce the number (but not the storage requirements) of temporary files that are created during the gridding phase. For most datasets a patch multiplier of 1 or 2 will be sufficient. Increasing the patch multiplier beyond a value of 4 would be rarely necessary.

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 that lie within a specified search radius surrounding the grid cell. 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.

In order for the software work with huge datasets that could potentially contain billions of points on PCs with limited available memory, the input data is scanned, cleansed and stamped into a temporary grid. The grid is then divided into tiles and each tile is then interpolated using the IDW algorithm. The intermediate grids are either stored in memory or if there is insufficient memory available they are stored on disk in temporary files. Once the dataset has been interpolated the output grid is constructed by stitching together the interpolating tiles into a continuous grid.

Before commencing a gridding operation it is important that the IDW Properties are appropriately configured for the input dataset. If you are familiar with the input dataset and know in advance what the spatial extents and distribution of the data is like and you have a good understanding of the range of values in the field(s) being gridded then you can manually adjust the IDW properties to best suite the input and output data requirements.

Inverse Distance Weighting Properties

The IDW properties can be used to influence the geometry and smoothness of the output grid file.

Model

This parameter controls the weighting model that is used to average the data points that are located within the search distance radius. The following four weighting models are available:

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 value. Larger range values produce flatter functions Gaussian functions and a smoother grid. The Nugget, Range and Distance radius values are measured in increments of the output grid cell size.

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). At distances beyond the nugget value the weighting factor will be applied according to the selected model. 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 Distance radius will be assigned an equal weight. The Nugget, Range and Distance radius values are measured in increments of the output grid cell size.

Exponential

Each input point's weight is proportional to its Euclidian 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.

Distance Radius

It is important to set an appropriate size for the search distance radius. 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 transparent in the output grid. Conversely, if the search distance is set to be too large then significant grid smoothing or artefacts may occur. The search distance radius is measured in increments of the output cell size.

To optimise performance, choose a 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 a good rule of thumb is to keep the search distance to a value less than or equal to 5x the output cell size.

Distance Taper

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 NEAR distance the taper function is assigned a constant value of 1 (i.e. no modification is made to the grid node). Between the NEAR and FAR distance the taper function is applied as a linear weighting between the grid node value and the background value. Beyond the FAR distance grid nodes are assigned the background value.

Minimum Curvature (Stamped)

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 percentage change tension parameter. 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

Minimum Curvature (Stamped) Properties

The minimum curvature (Stamped) properties can be used to influence the geometry and smoothness of the output grid file.

Stamping Method

This parameter controls the accumulation rules that are applied when multiple data points fall within a single grid cell. There are several options available:

- **Average all (last in weighted)**: averaged all data points
- **First Only**: the first point value is assigned to the cell. All subsequent points are discarded.

- **Last Only**: the last point value is assigned to the cell.
- **Average All**: averages all coincident point values.
- **Average All (inverse distance weighted)**: averages all coincident points by applying an inverse distance weighting function.

Radius

The radius control defines a distance radius around a grid cell to search for valid input points. Distance is measured in increments of the output grid cell size.

Clipping

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. Options available include

- **None**
  No clipping is applied to the output grid. With none selected the entire output grid will be filled with interpolated values.
- **Near**
  The Near value represents the distance, in cell size increments, that the grid is to be clipped back to from the convex hull of the data points. Areas in the grid which lie beyond the Near distance will be assigned null. Setting a Near only clip value will have the same effect as setting a buffer clip distance.
- **Near and Far**
  The Far distance is the maximum distance for which interpolation will occur between input points. Any area in the grid that has data points greater than the Far distance value will be displayed as null.

Minimum Curvature (Full)

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 percentage change tension parameter. 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

Minimum Curvature (Full) Properties

The minimum curvature (Stamped) properties can be used to influence the geometry and smoothness of the output grid file.

Stamping Method

This parameter controls the accumulation rules that are applied when multiple data points fall within a single grid cell. There are several options available:

- **Average all (last in weighted)**: averaged all data points
- **First Only**: the first point value is assigned to the cell. All subsequent points are discarded.
- **Last Only**: the last point value is assigned to the cell.
- **Average All**: averages all coincident point values.
- **Average All (inverse distance weighted)**: averages all coincident points by applying an inverse distance weighting function.

Radius

The control defines a distance radius around a grid cell to search for valid input points. Distance is measured in increments of the output grid cell size.

Clipping

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. Options available include

None

- **None**
  No clipping is applied to the output grid. With none selected the entire output grid will be filled with interpolated values.

- **Near**
  TThe Near value represents the distance, in cell size increments, that the grid is to be clipped back to from the convex hull of the data points. Areas in the grid which lie beyond the Near distance will be assigned null. Setting a Near only clip value will have the same effect as setting a buffer clip distance.

- **Near and Far**
  The Far distance is the maximum distance for which interpolation will occur between input points. Any area in the grid that has data points greater than the Far distance value will be displayed as null.

Full minimum curvature vs stamped minimum curvature methods

Although the operational phases for these methods appear to be very similar, there are important processing and operational differences between the two techniques. It is important to consider these differences carefully when deciding which gridding method to use. The primary operational difference between the stamped minimum curvature and full minimum curvature methods is that the stamped method is faster and requires less hardware resources (disk space memory and processing power) than the full minimum curvature method. The stamped minimum curvature method is generally faster because it does not spatially sort or retain a full temporary copy of the input data during phase 3. Instead it stamps the input data directly into a temporary grid. In contrast the full minimum curvature method spatially sorts and stores a complete temporary copy of the input data in addition to stamping the points into a temporary grid. The two methods also differ during the final interpolation phase where the full minimum curvature gridding operation loads the spatially sorted temporary data again to complete the interpolation. Loading the data a second time requires additional processing time.

The additional performance cost incurred by the full minimum curvature algorithm is often rewarded in terms of better grid quality and estimation accuracy. The method is able to produce a better estimation for grid cells that contain one or more input data points. In comparison the stamped method may simply shift the data values to the centre of each cell (depending on the data accumulation methodology selected). The full method is able to make a better estimate of the cell value in these cases by taking into account the actual position of the input data value(s) rather than just the cell centre which may improve the estimates for all surrounding grid cells.

Due to the potential improvement that the full minimum curvature method offers in output grid quality it is the recommended method, except where you determine the improvement in the grid quality will not be detectable or significant, or that the additional processing cost will be too high.

The stamped minimum curvature method may be more appropriate in the following situations:

- Re-interpolating existing gridded data onto a finer mesh. If the fine mesh is carefully designed so that the centre of the existing grid cells always fall in the centre of the cells of the new grid, then stamped minimum curvature will return the same result as the full method. However, if the cells are non-aligned then the full method will produce a better result.

- The output grid cell size is very small compared to the input data spacing. As a guideline if the grid cell size is 8x to 10x smaller than the input data spacing. In this case the input data value will be close to the centre of the cell into which it is stamped.

- The output grid cell size is large compared to the input data spacing. As a guideline if the grid cell size is 2x larger than the input data spacing then you are more likely to want the grid cell estimation to represent the average value of the observations within the cell.

- Noisy data. If the input data has a high level of noise then there is little reason to more accurately represent it by using the full minimum curvature method.

- Large data measurement footprint. If the data observations represent an average of a large footprint (e.g. radar) then stamped minimum curvature may be sufficiently accurate. However, where the observations represent an accurate point measurement of a smooth continuous field then full minimum curvature should always be used.

Data Density

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 a kernel estimator function

Data Density Properties Kernel

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 –

$$\hat{f}(x) = \frac{1}{n}\sum_{i=1}^{n} K(\frac{x - x(i)}{h})$$

where n is the number of samples. The following kernel functions are supported:

Kernel	**K(u)**
Uniform	$\frac{1}{2} I(\lvert u \rvert \leq 1)$
Triangle	$(1 - \lvert u \rvert) I(\lvert u \rvert \leq 1)$
Epanechnikov	$\frac{3}{4}(1 - u^2) I(\lvert u \rvert \leq 1)$
Quartic	$\frac{15}{16}(1 - u^2)^2 I(\lvert u \rvert \leq 1)$
Triweight	$\frac{35}{32}(1 - u^2)^3 I(\lvert u \rvert \leq 1)$
Gaussian	$\frac{1}{\sqrt{2\pi}} \exp(-\frac{1}{2} u^2)$
Cosinus	$\frac{\pi}{4} \cos(\frac{\pi}{2}) I(\lvert u \rvert \leq 1)$

Radius

To achieve a good result with the Density gridding 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. The radius distance is measured in increments of the output cell size.

Normalise by area

This control converts the data frequency values into spatial density values by dividing the value of each cell by the area of the search radius.

Bias by input field value

When enabled this control interprets the selected input field(s) as a count of the number of observations at that location. It allows a single point to represent more than one observation.

Data Point Separation

The data point separation gridding method produces a grid which records the distance to the closest input data features to each grid node.

Data Point Separation Properties

The distance radius specifies the maximum distance to search for valid input points. The distance is specified in increments of the output grid cell size.

Output Grid

Data Type

The data type control is used to set the numeric storage type for the interpolated values in the output grid. It is advisable to select the appropriate data type that most efficiently represents the range of data that will be stored in the output grid. For example a signed 2-byte integer is generally suitable for storing typical elevation data at 1m vertical resolution. The table below lists the available options and the valid data range that can be stored by each of the data types.

Data Type	Bytes	Range of Values
Unsigned byte	1	0 to 255
Signed byte	1	-128 to 127
Unsigned short	2	0 to 65,535

Signed short	2	-32,768 to 32,767
Unsigned int	4	0 to 4,294,967,295
Signed int	4	-2,147,483,648 to 2,147,483,647
Float	4	-3.8e-38 to 3.4e38

The automatic option will set the output format to an appropriate data type based on an analysis of the input data range.

File Name and Grid format

The output grid file(s) can be specified as either a single band ERMapper or BIL format grid. Although both of these formats support multiple bands in a single file, the grids created by the Large and Multi-file gridding tool can be extremely large, so we have chosen to output each band (or field) to a separate grid file for convenience. If multiple fields are selected for interpolation then multiple output grid files will be created. The name of each file will be the user specified name (e.g. MY_DEM) with the field name appended to it in square brackets (e.g. MY_DEM_[elevation].ers).

In addition to the ER Mapper (.ERS) or BIL (.BIL) grid header and data files a MapInfo .TAB file will also be generated for each output grid if the Auto open output file(s) option is enabled.

Grid projection

A projection must be assigned to the output grid(s). While it is possible to specify different projections for the input data and output grid it is not recommended for very large datasets as it may degrade the performance of the input processing phase significantly as each point will need to be reprojected into the output grid coordinate system prior to gridding.

Auto open output file(s)

When enabled the output grid file(s) will be automatically open into MapInfo.

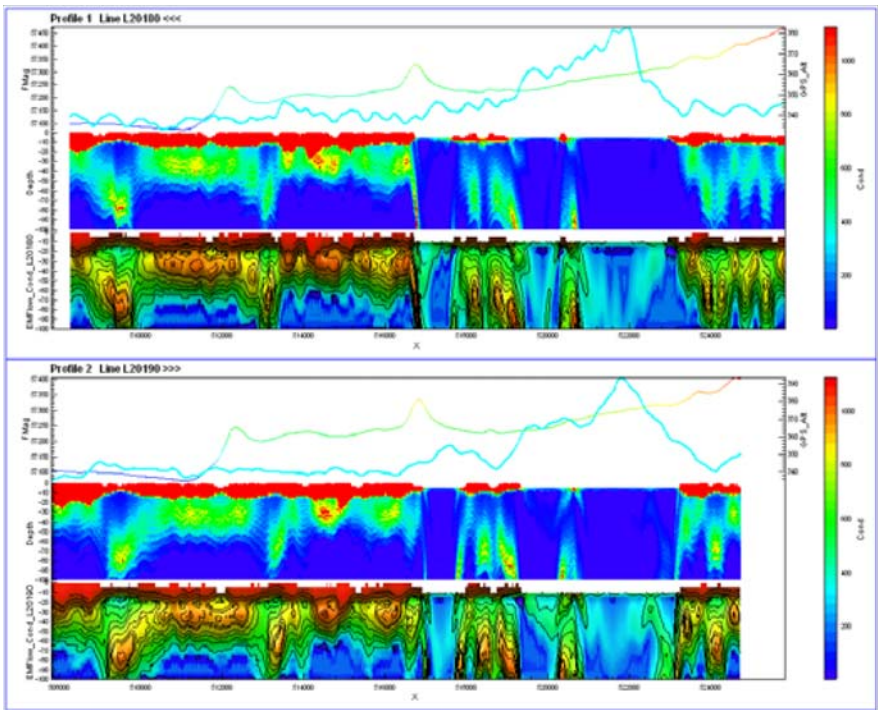
Section Gridding

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

Encom Discover PA can display gridded data in profiles. These items are referred to as Section Grids which are designed to display grids (in ERMapper format) as images. Section Grids can be used with their line association such that the Line Iterator can be used to cycle through survey data.

The Section Grids can be added to documents containing either Profiles or 2D Graphs via the menu command **Add Data>Section Grid** method. An example of a Section Grid is shown below:

Note Although section gridding produces ERS files with registration information, this 3D registration is customized and only recognized by PA. Third-party applications will not display the ERS in the correct 3D geo-location.



Example of Section Gridding Group with conventional section of the same data line.

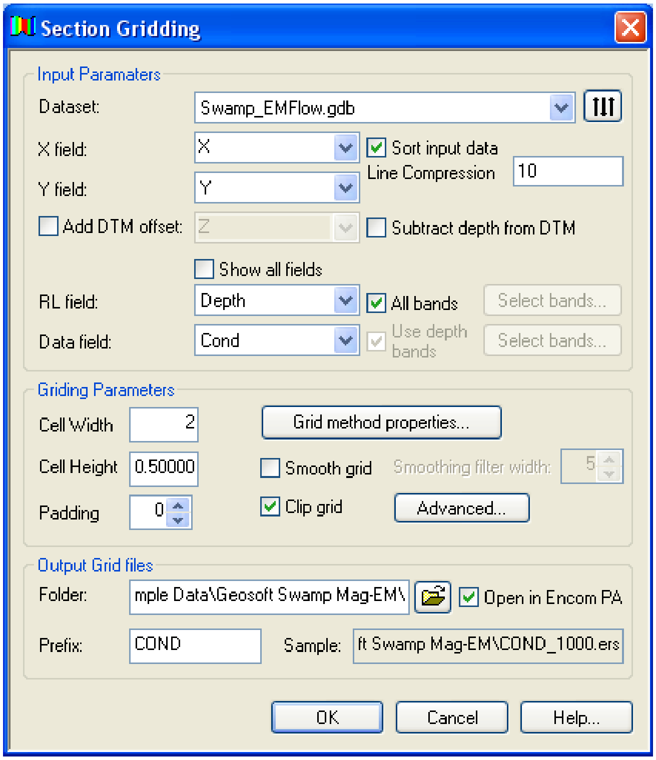
To operate the Section Gridding tool:

1. Choose the source dataset and the lines to grid. One grid file is produced for each line chosen. You can select the lines required from the Line Selection button.

2. Choose the X and Y fields that define the geometry of the line. These fields are used to compute Chordlength that is used as the horizontal field in the output grid.

3. Sort the input data if it lies on an essentially straight line.

4. Compress the line data. This reduces the amount of output line information and is advised for most line data. If your line data is not straight, you can elect to switch this off.



Section Gridding dialog for creating gridded sections

The line data needs to be correctly prepared for gridding. The tool converts the X and Y fields into chordlength that are used as the horizontal grid coordinate. To do this properly the data must be properly sorted so that each station is at least as far or further along the line than the last. The software automatically sorts the input data to ensure this is the case which is appropriate for the majority of datasets that approximate a straight line. If your input data follows a convoluted path the sorting algorithm may fail. In these cases you can disable sorting but you must then ensure that your data is appropriate for gridding.

Usually, the vertical dimension is depth, RL or similar. The vertical field is expected to decrease downwards. If you have depth values that increase downwards you can specify the Subtract depth from DTM option to reverse the sign of the depth, regardless of whether you specify a DTM field. If the depth values are in local coordinates then you can specify a DTM field which is added to the depth field to obtain true RL.

Set the **DTM offset** field if required. If the depth/RL field is increasing downwards, enable the **Subtract depth from DTM** option to reverse the sign.

The Section Gridding tool can be used to grid either multi-banded or single banded data. The prior case is suitable for **EM inversion** data which is generally stored as a multi-banded depth field and a matching multi-banded conductivity field. In this case, ensure that the number of depth bands match the number of conductivity bands. **IP data** is a special case as it usually is single banded with a depth, resistivity and chargeability fields (see *Display of IP Data*). In all cases, the output grid is only single banded so you can only grid one parameter at a time.

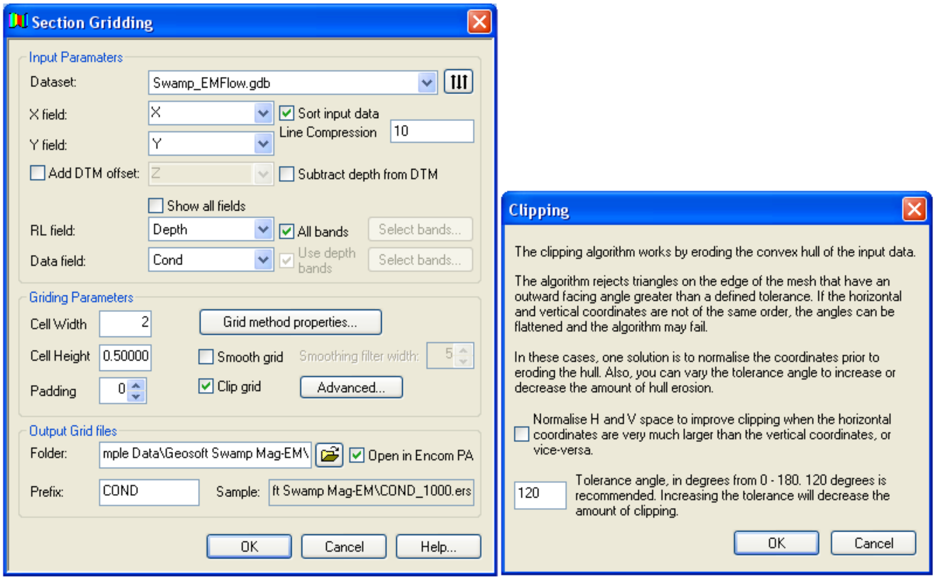
By default the Section Gridding utility only displays the multi-banded fields in the dataset. To display all fields, enable the **Show all fields** box.

Select the depth field and the data field. It is likely you will choose to use all bands from these fields, in which case check the **All bands** box. Otherwise, you can select the bands individually. By checking the **Use depth bands** box you can force the tool to use the same band numbers for data as you have selected for depth.

The tool uses a minimum curvature algorithm to grid the data. You can specify the **grid cell size** plus **smoothing** and **clipping**. Set the grid cell size in the horizontal and vertical dimensions. Note that a suggested cell size is automatically displayed when you change the input data specifications but once you have manually set the cell size it is retained.

Allow some **padding** around the data nodes to ensure the grid is extended beyond the data. The padding distance is specified in units of grid cells. You can smooth the grid prior to output in which case you ought to specify a Gaussian filter width (in units of grid cells).

You can clip the grid prior to output. The clipping algorithm clips tightly about the input data points but takes into account any padding specified. The **Advanced Clipping** dialog allows you to set advanced options.



Section Gridding utility with the Clipping options dialog.

The Section Gridding tool produces an output grid for each input line. To use these grids in a profile display in Encom Discover PA they should all have similar names, varying only by the line number, and they ought to be located in the same disk directory.

If you enable the **Open in Encom Discover PA** option, the first grid generated is opened in a section profile as soon as it is written to disk. To view the section grids in a 3D display, see *Section Profiles in 3D Displays*.

Display of IP Data

Induced Polarisation data can be displayed in Encom Discover PA using the Section Gridding utility. Standard Geosoft IP databases can be used. These datasets routinely have a multi-banded data field for IP, but multiple records for each data location corresponding to a different n-spacing (or receiver dipole pair).

This data storage format means that if you wish to display a conventional psudosection of resistivity or chargeability (using the n-spacing for vertical scaling, the data must first be sorted. The Section Gridding utility sorts the data so that each station is at least as far or further along the survey line than the last point.

Normally the utility automatically sorts the input data to ensure this is the case and for the majority of datasets that approximate a straight line.

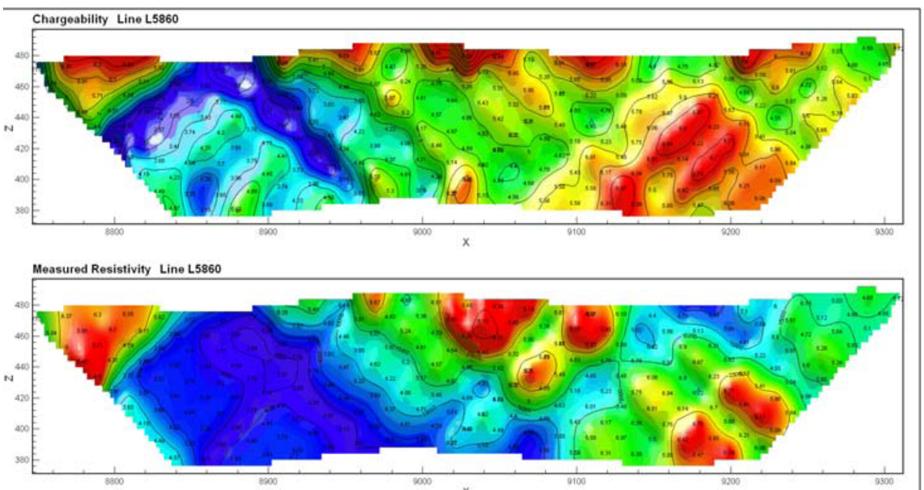
When sorting an IP dataset, the assumption is made that the line is straight, but the sorting uses the n-spacing to order the data. The usual Data fields of Resistivity, Metal Factor or Chargeability could be used and a DTM data field can be used to determine an upper topography surface.

Due to the n-spacing distribution of a pseudosection, clipping is important.

Clipping in the Section Gridding utility generates an outer perimeter of the input data, then erodes this by triangulation from the outside inwards, progressively removing edge triangles whose outward angle exceed a tolerance. Be aware that when the horizontal coordinates are different to the vertical coordinates the triangles can get flattened and the algorithm can perform poorly. For example, if the horizontal coordinates are in metres along line and vertical coordinates are n spacing (e.g. 1-6) then the horizontal range may be 100 times greater than the vertical range or more. In cases like this it is advisable to normalise the data to regularise the triangles. Also, the rejection tolerance can be modified between 0 to 180 degrees. Both of these options are controlled from the Advanced clipping dialog as shown below:

In this dataset, the Resistivity data channel (Res) is being gridded above. Note that topography is being applied as an offset from the DTM field and Smoothing and Clipping are applied. A jagged edge on the outer margins of the grid can result if the horizontal and vertical dimensions are not normalised for clipping.

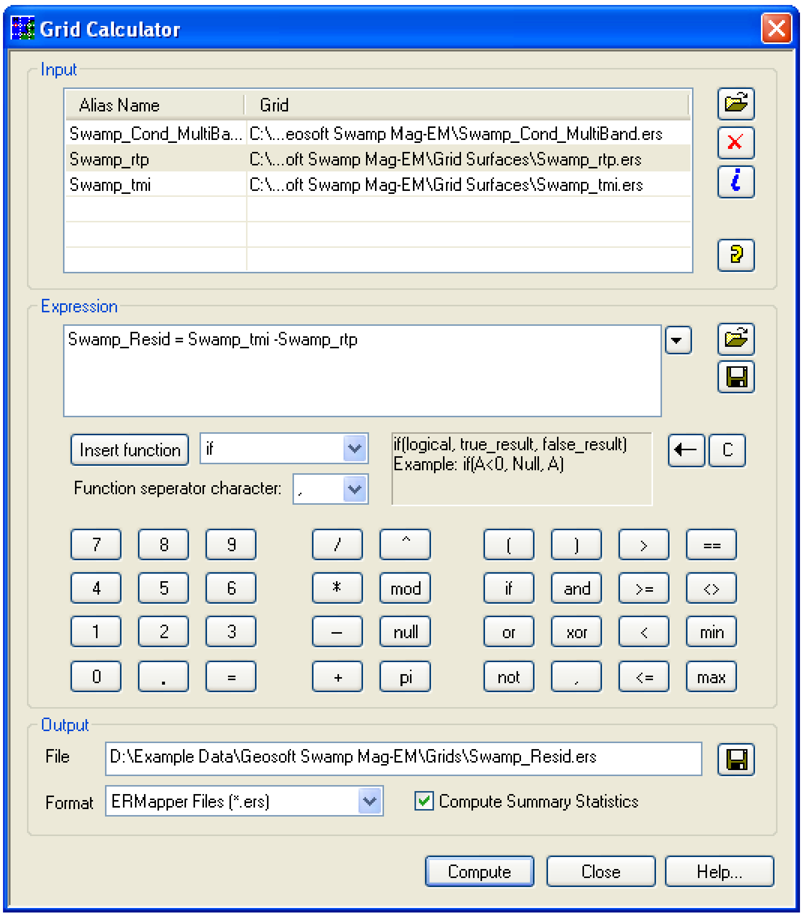
The output gridded and contoured result is shown below.



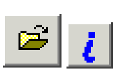
Examples of IP pseudosections with Chargeability and Resistivity

Grid Calculator

The **Grid Calculator** can apply arithmetic operations, logic, comparative and Boolean operations directly to grids or between grids of any supported format. It performs the operations directly writing to the disk file, and does not require the grid to be loaded into memory or opened in Encom Discover PA.



Grid Calculator and specifying a formula



Available grids are displayed in the scrollable list at the top of the dialog. This list is initially populated by all grids currently open. Further grids can be added to the list using the **Load Grid** button, and removed from the list using the **Delete Grid** button. Use the **Information** button to obtain details of a selected grid.

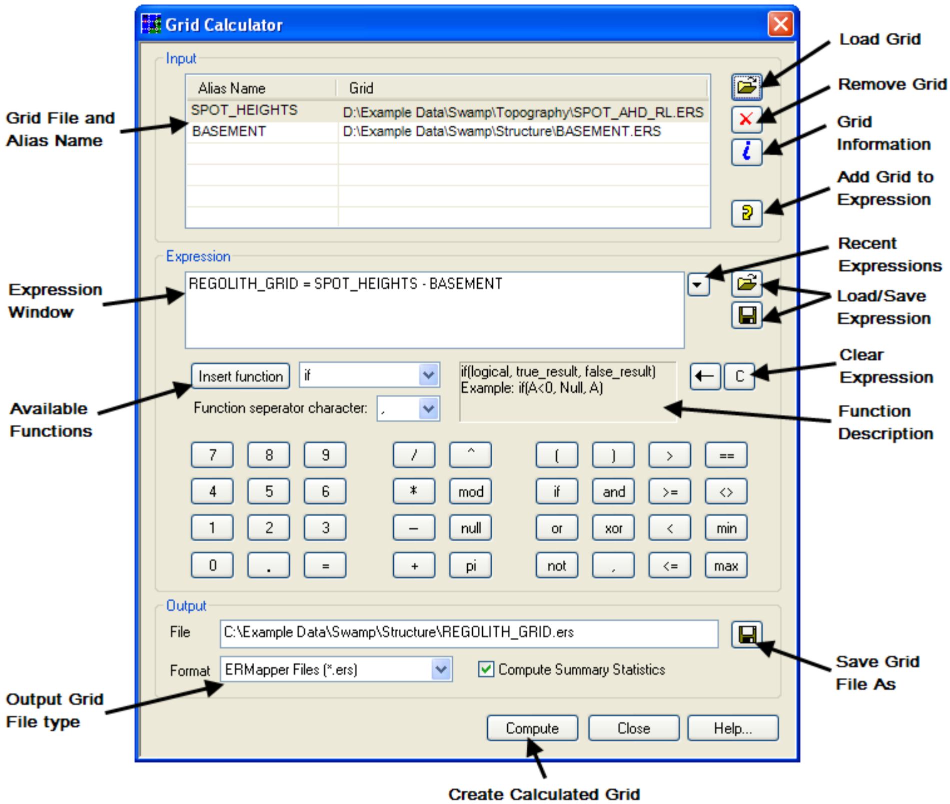
Grids can be assigned an alias name to enable the shortening of complex file names for ease of use in the expression line. For example, the grid file Rockchip_samples_As_210503 can be simplified to the alias name X. This enables the following simple expression output=X/(X*2) to be entered into the expression line rather than full grid name. To create a grid name alias, double click on the relevant cell in the **Alias Name** column and type the desired alias name.

Note In the **Grid Open** dialog, a range of supported grid formats are available. The Grid Calculator can be used to convert grid formats if required. Grid to grid computation can also be performed irrespective of the input formats or requested output format.

The Grid Calculator requires a formula to be specified in the **Expression** line in the middle of the dialog. Complex grid operations such a Boolean and decision functions can also be applied.

If the areas covered by selected grids are different, results are computed only for grids where data overlap occurs. The remaining area of the first grid specified in the command line is filled with nulls. If node points are offset between grids, the data is interpolated prior computation. If grids of different mesh sizes are used, the mesh size of the first specified grid in the command line is output.

The Grid Calculator dialog and expression control is shown below:



Grid Calculator button layout and command line control

To add a grid to the expression line, place the cursor at the desired location in the expression (if not already there), highlight the grid in the grid list and press the **Add Grid** button

An expression must be a ‘correct’ arithmetic formula of the format:

```
output = function(input)
```

A new grid file alias must be entered on the left hand side of the expression. By default this is "output". This alias is automatically used by default for the output grid file name when the output grid is generated.

Click the individual operation buttons to enter the operation into the expression line. Formulas can be entered directly into the expression line by keyboard entry or by selecting the appropriate fields and buttons via the Grid Calculator.

The **Recently Used Expression** button opens a drop down list of functions last used in the Grid Calculator.

The **Save As** button enables an expression to be saved to a text file. Use the **Load** button to re-load the saved expression into the Grid Calculator dialog.

The **Save As** button enables an expression to be saved to a text file. Use the **Load** button to re-load the saved expression into the Grid Calculator dialog.

The **Insert Function** button enables the construction of complex formulae. A wide range of arithmetic, logic and Boolean operations are available from within the Grid Calculator. When a function is selected from the **Insert Function** pulldown list a definition of the function and the syntax required is displayed in the Grid Calculator dialog.

The actual output grid file name can be to be specified when the expression is computed in the **Output** section or using the file **Save As** button. If the output file name is not changed, the alias name is automatically saved. The Output grid file type can also be selected from the **Format** drop down list. The **Compute Summary statistics** option enables the calculations of grid statistics which are used by Encom Discover PA's Grid Handlers to display the grid.

When the expression is completed, press the **Compute** button. If the syntax is correct, a progress window is displayed as the Grid Calculator processes the expression. If there is a syntax error in the expression line a message is displayed:.

Using Multi-banded Grid Inputs

The Grid Calculator also has the ability to reference individual bands from a multi-banded input grid in the formula. Using an example of two grids called GridA and GridB, both with 3 bands, a typical expression to access the second band of GridA and the third band of GridB would be:

```
Output = GridA[2] + GridB[3]*0.5 + GridB[3]*0.25
```

The above example would produce a single banded output grid.

However if one or more inputs were 'all bands' such as the below expression, you would get a multi-banded grid output:

```
Output = GridA + GridB*0.5 + GridB*0.25
```

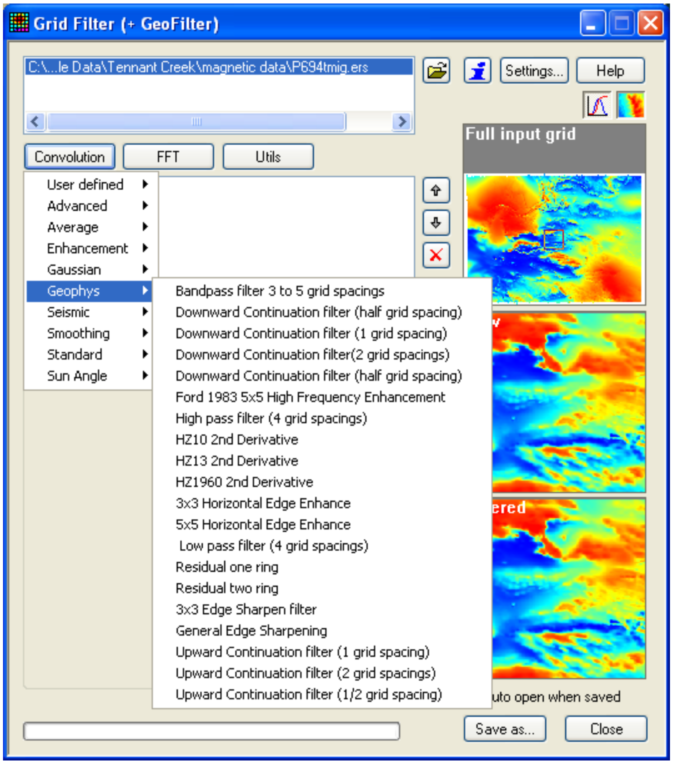
Grid Filter

Note The **FFT>Special>FFT ZS Filters** utility requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

The Grid Filter option is a powerful tool that provides the following functionality:

- Numerous grid operations including padding, filtering and null filling can be undertaken.
- A wide range of convolution grid filters can be applied including average, standard, Gaussian, geophysical, seismic and sun-angle filters.
- FFT filtering using sophisticated operators to apply geophysical filter results.
- Advanced FFT filters specifically designed to provide geophysical processing for continuations, derivatives, reductions and edge and gradient detections.

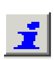
To access the Grid Filter utility select the **Grid>Grid Filter** menu option. The utility dialog appears as shown below:



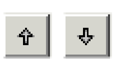
Grid Filter tool with multiple operators applied to an input grid, zoomed and the resultant output grid

The dialog is divided into a number of control and display areas. These are:

- List of specified input grids. Any grids loaded in Encom Discover PA prior to opening the Grid Filter utility will be listed here. Additional grids can be accessed using the **Browse** button. A specified grid can also be removed from the list using the **Delete** button.



- **Information**, **Settings** and **Help** buttons provide information on the specified grid plus display and filtering controls.



- Buttons (**Conv**, **FFT** and **Utils**) provide pull-down lists of available operations. In the area beneath these buttons is an area where selected operations accumulate. The order of the operations is important as processing is incrementally applied from the top to the bottom. The last operation produces the output grid. An operation’s position in the order can be altered by using the **Up** or **Down** arrow buttons after selecting the process to be moved. The **Delete** button can remove a highlighted process.

- Three display windows of the highlighted grid are shown on the right side of the dialog. The top window shows the entire original input grid. The centre and bottom windows may show the same area of the input grid or a zoomed in portion of the original grid. In the upper window, a small rectangle is drawn (for grids larger than 400 x 400 rows and columns) over the original grid. This rectangle indicates the portion that is being processed by the operations. The rectangle can be selected by the cursor and moved to a new part of the input grid if required.

  The lowest preview window indicates the output result of the various operations list. Refer to *Using the Grid Filter Tool* for information on zooming and panning in these windows.

- Some grids may require padding. Padding is used as a pre-processing stage necessary for application of certain filters (such as FFT filters). Refer to *Appendix C: Grid Filters* for additional information on padding.

- The Filter Properties area details any controls that apply to a highlighted operation. The controls may list kernels, required wavelengths and filter specifications or magnetic field controls (for example when using the pseudo-gravity FFT filter).

- The output grid name is listed at the base of the dialog if the Save As button is clicked. The standard Encom Discover PA grid formats are supported (refer to Supported Grid Formats).

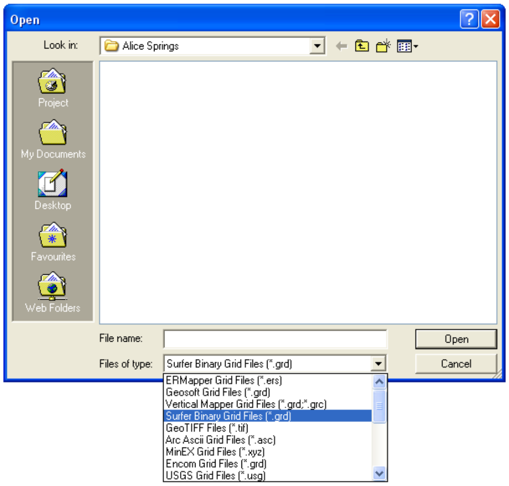
For more information, see:

- *Using the Grid Filter Tool*
- *Preview WIndows*
- *Filter Operations*
- *Order of Operations*
- *Grid Information*
- *Grid Filter Settings*
- *Fill Holes*

A detailed description of the various filters and their application is provided in *Appendix C: Grid Filters* (Convolution Filter Descriptions, FFT Filter Descriptions, Grid Filter Operators, and GeoFilter filters).

Using the Grid Filter Tool

To use the Grid Filter tool, use the **Browse** button to load a grid. You can specify the grid format from the **Files of Type** entry in the Open dialog.

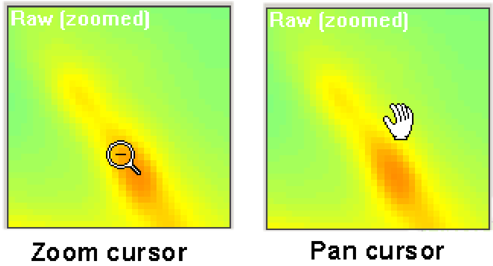


Use the Files of Type option to select a specific file type

Once loaded, the grid content is displayed in the three preview windows. Using the **CONV** (Convolution filters), **FFT** (FFT Filters) or **Utils** (Utilities) buttons and the provided pull-down lists, select the operations to be performed on the grid. Note that as operations are added, they are immediately performed on a portion of the input grid. The area over which the operations are applied is indicated by a square drawn in the top preview window. The content of this square is shown in the second, middle window. The output of the processing steps is shown in the bottom preview window.

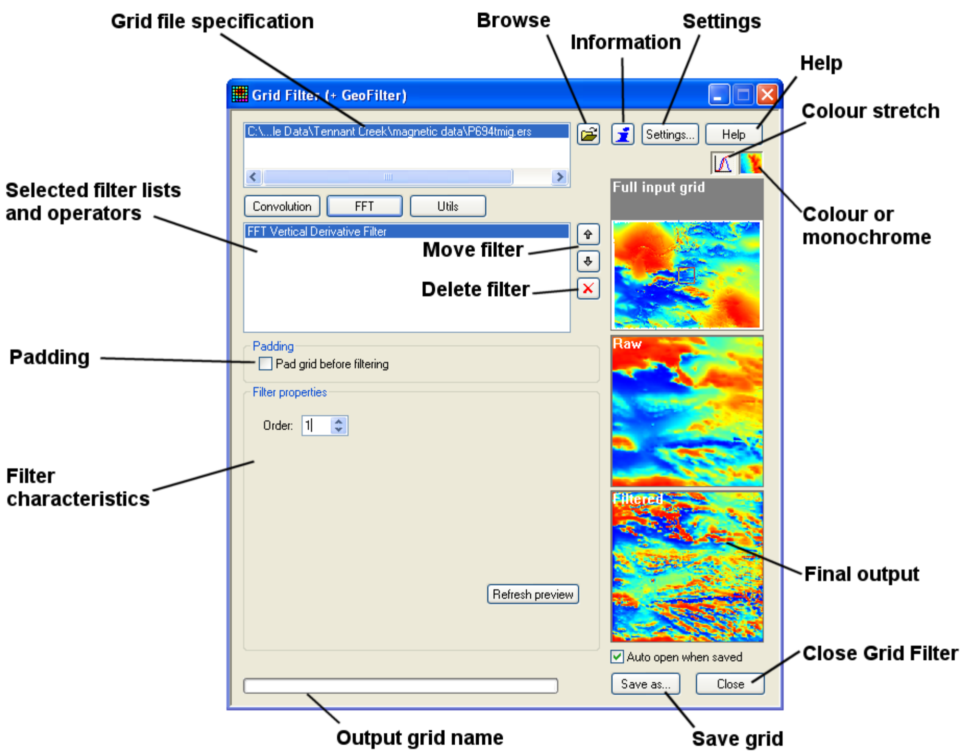
Preview WIndows

If you wish to examine the effects of the processing more closely, you can position the cursor in the middle or lower preview windows and click the left mouse button. When placed in the window initially, the cursor changes to be a magnifying glass. Clicking the left mouse button zooms the view in the middle and output preview windows and allows you to see a portion of the processed area. A subsequent left mouse click returns the view back to the original size. If you wish to pan the zoomed view, click the right button and while the cursor hand is shown, you can drag-and-drop the zoomed image.



Note that the square drawn in the upper window indicates the area of processing. Since processing can take some time for complex operations, only this small (400 x 400 row/column) area is used. If you wish to move the processed area, position the cursor over the square in the upper preview window and with the left mouse pressed, drag-and-drop it to a new location. The processing is then applied to this new data.

When you request that the processed grid be output, using the **Save As** button, the processing is applied to the entire grid and written to the new file specification nominated from the **Save As** button.



The dialog for the Grid Filter with input, raw and output grids

If you wish the gridded output to automatically be displayed in Encom Discover PA after writing to the nominated output file, turn on the **Auto open after save** option.

Filter Operations

The operations provided by the Grid Filter are:

Filter	Type	Option
Convolution	**User Defined**	General mxn Filter
		Average mxm Filter
		Gaussian mxm Filter
	Advanced	amedian
		average
		deviation
		median
		wlocalmadj
	Average	nxn Average Filters
		Diagonal Filter
		Horizontal Filter
		Vertical Ffilter
	Gaussian	Standard Deviation = 0.391
		Standard Deviation = 0.625
		Standard Deviation = 1.0
		Standard Deviation = 1.6
	Geophysical	Bandpass Filter (3 to 5 grid spacings
		High Pass Filter (4 grid spacings)
		Low Pass Filter (4 grid spacings)
		Downward Continuations
		Ford High Frequency Enhancement
		Horizontal Edge Eenhancement
		Residual Ring Filters
		Edge Sharpening Filters
		Upward Continuation Filters
		HZ Second Dderivatives
	Seismic	Easterley Ddip
		Hanning
		Northerly Dip

Filter	Type	Option (Continued)
	Standard	Darn holes
		Default Kernel
		Force no subsampling
		Laplacian Filters
		Horizontal Edge Enhancement Filters
		Vertical Edge Enhancement Filters
		Low Frequency Filter
		Edge Sharpening Filters
		nxn Avg, Threshold = 5 Filters
	Sun Angle	East
		North
		North East
		North West
		South
		South East
		South West
		West
FFT	Component	FFT TMI to Single Magnetic Component (General)
		FFT TMI to XYZ Magnetic Components
		FFT General Directional Transform Filter
	Derivative	FFT Derivative Filter (Advanced)
		FFT Integration Filter (Advanced)
		FFT Vertical Derivative Filter
	General	FFT Analytic Signal
		FFT Band Pass Filter
		FFT Butterworth Filter
		FFT Continuation Filter
		FFT Directional Cosine Filter
		FFT High Pass Filter
		FFT Low Pass Filter
		FFT Directional Pie Slice Filter
	Special	FFT Encom ZS Filters
	Tensor	FFT gz to Gravity Gradient Tensor (GGT) Filter
		FFT TMI to Magnetic Gradient Tensor (MGT) Filter

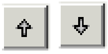
Filter	Type	Option (Continued)
		FFT Tensor (MGT/GGT) to Field Component Filter
	Transform	FFT TMI Pseudo-gravity Transform Filter
		FFT gz Pseudo-magnetic Transform Filter
		FFT TMI Reduction to the Equator Filter
		FFT TMI Reduction to the Pole Filter
		FFT TMI Reduction to the Pole Filter (low latitude)
		FFT TMI General Phase Transformation Filter
Utilities	**Utils**	Fill Holes

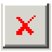
Descriptions of the various filters and operators are provided in *Appendix C: Grid Filters* (Convolution Filter Descriptions and FFT Filter Descriptions). A full description of the GeoFilter filters is also provided.

Note When applying FFT filters, especially such as the Low Latitude filter, to avoid Gibbs Phenomena becoming apparent, it is a good idea to initially apply an Upward Continuation filter of approximately half the grid cell width.

Order of Operations

Filters and processing operations are applied in the order listed. Each operation may require you to specify some controls in the Filter parameters area. The controls and parameters for each operation and filter type are described in the Operation and Filter Descriptions of Appendix B. The required controls vary according to the highlighted operation in the list shown in the dialog.

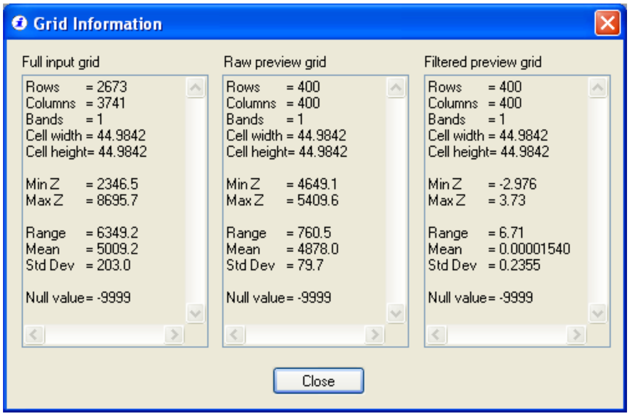




The processing is applied such that the output of one operation is the input of the next. This means that complex processing can be applied cumulatively. If you wish to alter the order of operation, use the **Up** and **Down** button to the right of the operation list. You can also remove an operation by selecting it and then clicking the **Delete** button.

Grid Information

Information relating to the size, rows/columns and data within each of the three preview buttons can be displayed by clicking the **Information** button. When zoomed and processing is applied, each preview window contains different grid data content and so the displayed dialog report indicates this:

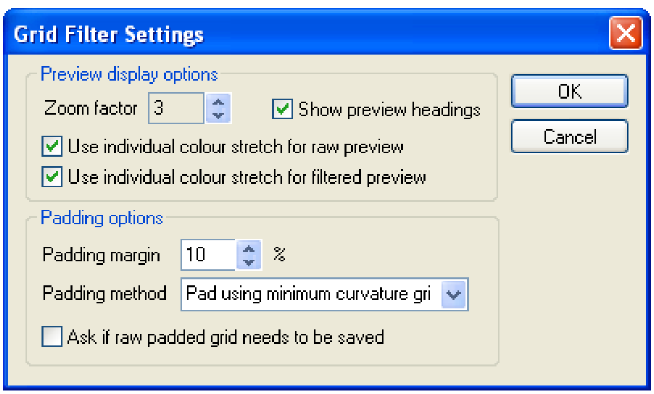


Grid Information for the three preview windows of the Grid Filter

Grid Filter Settings

The Grid Filter allows you to control the zoom and padding of the Grid Filter. Select the **Settings** button to display a control dialog.

The zoom level can be specified as default. The integer indicates the increase of level such that an entry of 5 zooms the area to 5 rows/columns for each one seen in the original, upper preview window. The window titles (**Full**, **Input** and **Filtered**) can also be eliminated if required.



Grid Filter settings for zoom levels and padding

Padding controls permit an area surrounding the input grid to be created to minimise edge and filtering effects when certain filters are applied. Refer to FFT Filter Descriptions for additional information. The area around the original grid to be processed can be set as a percentage of grid area around the perimeter. This is referred to as the Padding margin. Two methods are also provided for the padding. These are:

- **Padding by row and then by column** – This option duplicates the outer rows then columns of the input data grid and extends them outwards to the Padding margin to surround the grid
- **Padding in 2D using gridding** – This method extends the grid margin by extending the grid data and gridding it over the extent of the Padding margin. This method produces a relatively seamless edge to the grid and allows for smooth filter application without edge effects. This method is superior in many cases to the first because it eliminates the horizontal and vertical streaks which padding by row and then by column can introduce.

The Padding Margin as it is appended to the original grid can be saved for examination if required. Enable the **Save padded grid** option to this.

It is recommended that padding is always used when running an FFT filter. If the input grid is not square then FFT filters cannot be used unless padding is selected. Padding squares up the grid.

Fill Holes

The Grid Filter **Utils>Fill holes** option can be used to replace nulls in a grid by extrapolating the surrounding data. There are two options available for this module:

- Fill internal holes only
- Fill internal holes and nulls surrounding grid

If the first option is selected then only null values which are not connected by nulls to the outer edge of the grid are filled that is, internal holes. If the second option is selected, then all nulls in the grid are replaced with extrapolated values.

The method used to fill holes involves progressive extrapolation from the outer rim of the hole towards the centre until the entire hole is filled. This process can be slow for large holes.

Grid Utility

The Grid Utility allows various operations to be performed on one or more specified input grids. These operations include:

- *Classify* – Classify each grid cell into one of a number of ranges.
- *Grid to Grid Clip* – Clip a grid to the non-null area of another grid.

- *Clip* – exclude or include the data from a grid defined by an irregular polygonal region or by a rectangular region. The data can be clipped outside or inside the region.
- *Convert* – input a data grid in one format and save as another to convert the output format as required.
- *Create RGB* – combine separate grids having Red:Blue:Green colour signatures to a single, multi-banded RGB grid file.
- *Curvature* – compute curvature of each cell in a grid.
- *Cut/Fill* – compute the cut and fill differences between two grids.
- *Edit* – point to a grid node and the data values for it and its surrounding cells are displayed. You can edit the data values if required.
- *Fill Holes* – Replace nulls within or around a grid by extrapolating values using the surrounding data.
- *Flip* – the rows or the columns of a grid can be inverted in their location either horizontally or vertically.
- *Merge* - Merge several grids to form a new grid.
- *Outline* - Outline grid bounds or non-null regions as polygons.
- *Overlay* - Modify grid cell values based on polygon boundaries in a specified TAB or MIF vector file.
- *Replace* – allow specific grid values (such as Nulls or nominated values) to be replaced by another data value or Null
- *Reproject* – permits the input grid to be reprojected to an alternative Projection and Spheroid/Datum combination.
- *Resample* – grids can be resampled to a new cell size using any of three available interpolation schemes.
- *Rotate* – a grid can be rotated about its defined origin by a specified angle. Interpolation processing is required for this procedure.
- *Shift* – apply an east or northing offset to the origin of a grid.
- *Slope* – calculate the slope angle or aspect (dip direction) of each cell in a grid.

- *Split* – used for multi-banded grids, this operation outputs separate component grids of the individual single bands.

- *Statistics* – compute statistics for a grid, or the cells contained by one or more overlying vector objects, or by regions in a classified grid.

- *Surface Area* – compute the 3D surface area, planar area, and roughness of cells ina grid.

- *Vectorise* – converts each grid cell or grid outline into a polygon in the output vector file and assigns each polygon a grey-scale value.

- *Viewshed* – compute the view-shed of one or more towers of a specified height above the grid to an observer at a specified height above the grid.

- *Volume* – computes the volume between two grid surfaces or between a grid surface and a defined value.

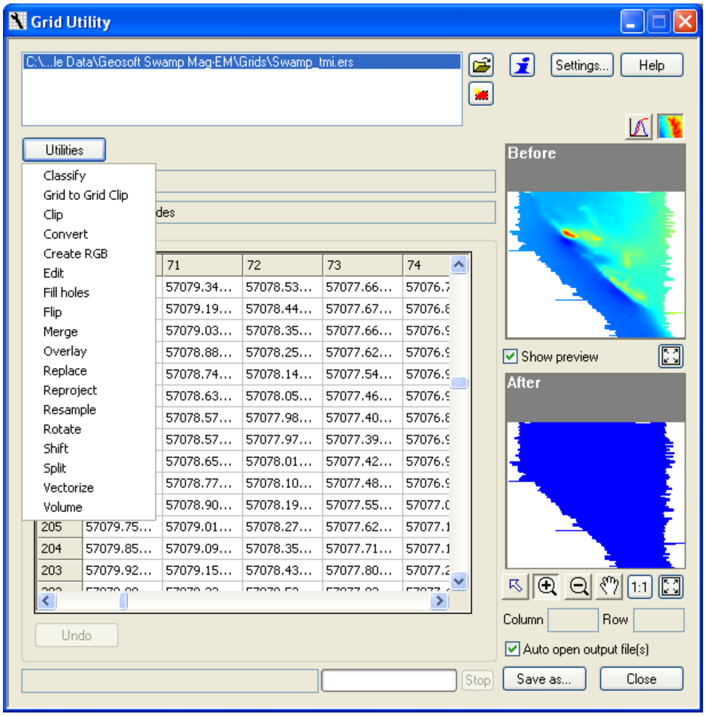
All functions are operated from a common dialog as shown below.


The upper portion of the dialog allows grids to be added to a list (from the **Browse** button). If you wish to apply operations on a created, flat grid, you can use the **Create Grid** button.

By highlighting a grid in the list and clicking the **Information** button, a display of the grid statistics is provided. The current grid projection can also be queried by using this button.

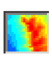
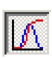
Other controls include a **Settings** button (to control the zoom level in the preview screens) and **Help**.

A **Function** pull-down list allows selection of the required operation and a **Description** for each is displayed when selected. As each Function is individually selected, the control parameters for that operation are displayed.



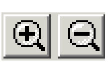
Grid Utility dialog with grid selection and Function parameter displays

In the two Preview panes on the right side of the Grid Utility dialog are **Before** and **After** views of the selected grid. As you change the grid selection or the grid function, so the preview panes redraw and update with the changes. Note that the visual changes occur in memory only and are not permanently saved until you specify and save an output grid using the **Save As** button.

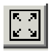
You can control the monochrome or colour appearance of the grids from the **Use Colour Look-Up table** button and a linear or equal area histogram stretch can be applied from the **Histogram Equalisation** button. When the cursor is positioned in either of the Preview panes, its function (and appearance) alters depending on the selected button control at the bottom of the panes.

Pointer mode – Used to select a position or cell value in a grid



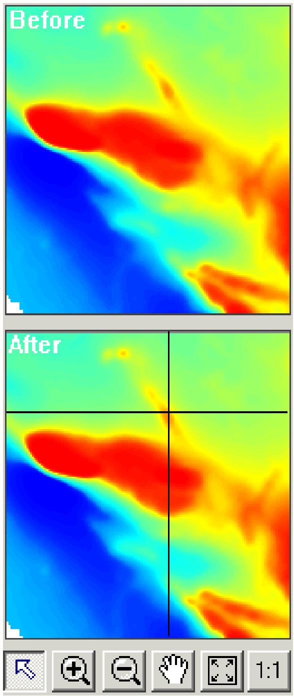
Zoom In or Out – Click the button and move the cursor into the preview pane. Click again and the level of zoom doubles (or halves) each time the mouse is clicked. This zooming occurs in both Before and After panes.

Pan the image content in the preview pane by positioning the cursor, hold the left mouse button down and drag. When released the image is redrawn.



Return a zoomed or panned grid to its **default fitted preview** view.

Return a zoomed grid to a **1:1 zoom ratio.**



The **Settings** button allows a cross-hairs cursor to be displayed in the After pane. A second setting toggles the display of the text After or Before if these are optionally not wanted. Beneath the various button controls (described above), the cursor position indicates the Column and Row location on the displayed grid.

At any time in the processing of a grid, the **Stop** button can be used to halt computation and return to the previous state.

The various grid operation functions are described below.

Classify

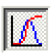
The **Classify** grid utility enables an input grid to have the value of each cell classified into one of a number of ranges. By default the input grid is automatically classified into 5 bins. Each bin represents an equal data range spread between the minimum and maximum grid values. The output cells are assigned a bin value of 1 to 5 based on the input cell value range.

Use the **Auto** button to automatically classify the grid cells. Use the **Auto classify** dialog to modify the **Input classification range** and the **Number of classifications** or bins in the output grid. By default the bin or classification values start at 1 and increase by 1 but these options can be changed by setting a new Output value start and Output value step. The **Output value start** and the **Output value step** are saved in the registry and are used as the defaults next time the Classify utility is opened. The input classification range is re-computed each time a new input grid is loaded.

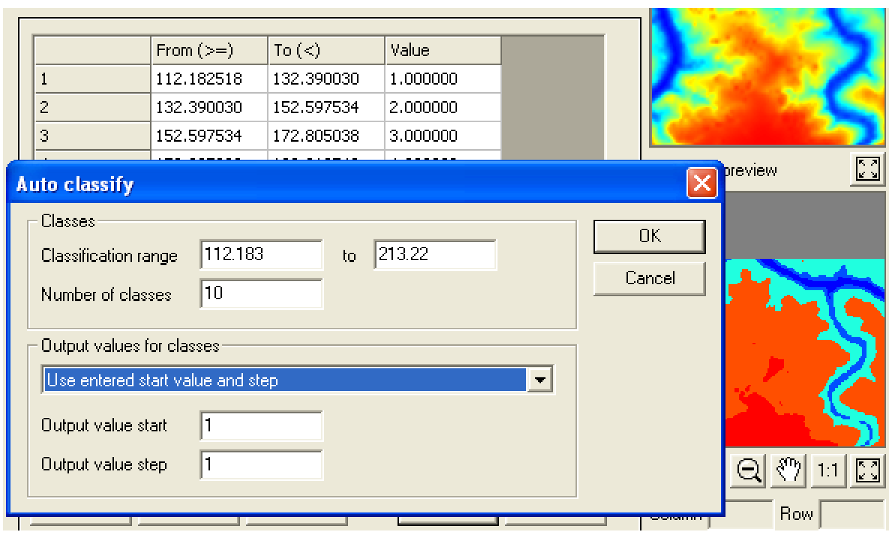
Complete manual control of the classification process is possible by manually editing the bin ranges. In addition use the **Add row** and **Insert row** buttons to append or insert a row and the **Delete rows** button to remove rows from the classification. Hold down the SHIFT or CTRL keys to select multiple rows for deletion.

Note that to assign any classification of the grid ranges to their appropriate bins, you must:

- Turn on the option in the Settings to **Use separate colour stretch after preview**.



- Turn off the **Histogram Equalisation** button for the colour stretch.



Specify the upper and lower range of data to be binned and the number and increment of bins

Saving the output grid stores the binned data ranges with all null values being retained.

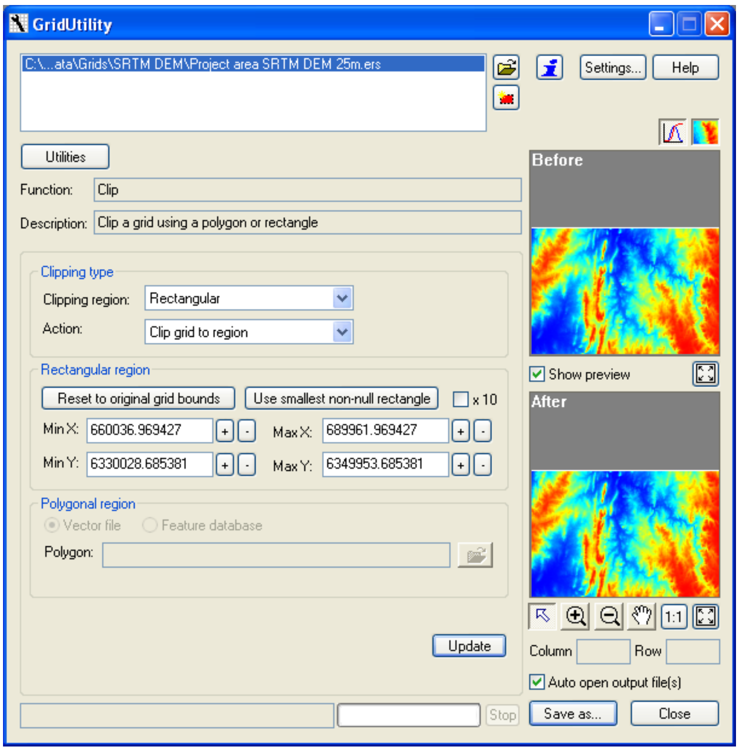
Grid to Grid Clip

The **Grid to Grid Clip** tool allows a source grid to be clipped to the extents of another grid. This requires both grids to be in the same projection (use the *Reproject* utility to convert grid projections), and the grids must overlap.

Select the source grid (i.e. the grid to be clipped) from the listing of available grids at the top of the GridUtility. This will automatically populate the Input grid window. Select the grid to clip against (i.e. the grid whose extents the source grid will be clipped to) in the **Clip to Grid** pull-down list. The output will be presented in the **Preview** window; use **Save As** to generate the output clipped grid.

Clip

By opening a grid and selecting the **Clip** operation from the **Function** pull-down list, the following control parameters are displayed.



Clip parameters in the Grid Utility dialog

The Clipping process can operate in two ways:

- Define a rectangular area (using the Min and Max X and Y corner extents) and clip data as controlled by this region or
- Use a polygonal shape defined by a MapInfo Professional .TAB or .MIF file, ESRI .SHP file or a Encom Discover PA Features database polygon.

In both cases above, you can use the bounding region to clip data inside (using the **Clip grid to region**) or outside the area (using **Blank grid underneath region**).

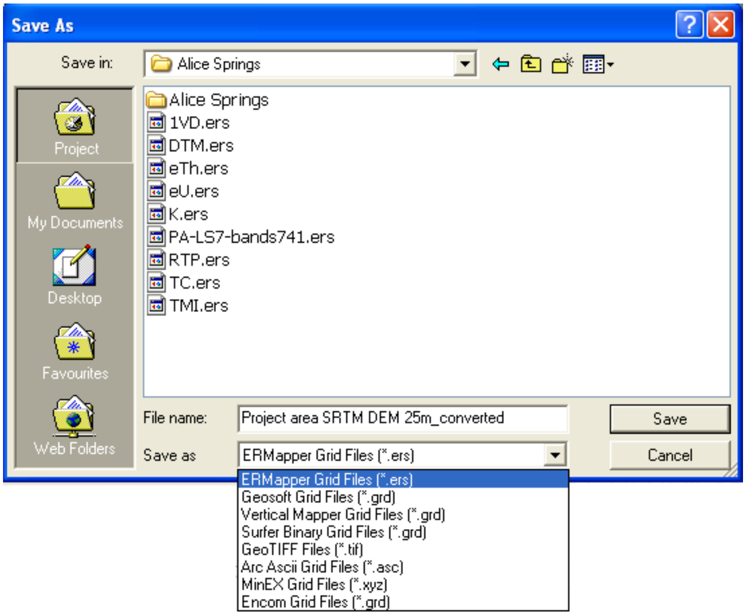
Note that when a grid is clipped, where possible the size of the grid is minimised and nulls and unused whole rows or columns removed. This can substantially reduce the size of a large grid if only a small portion is extracted.

Convert

Selected grids can be converted to an alternate grid format. The formats available include:

- ER Mapper
- Geosoft
- Vertical Mapper
- Surfer
- Encom .GRD
- GeoTiff

No parameters are used for this option. Select the file required to be converted and then click the **Save As** button. From the **Save as type** pull-down list, nominate the output file format required and then give it a location and name. If you do not specify a file extension, the output file has the format's default file extension added (refer to *Appendix E: Supported File Formats*).



Select the required output grid format from the pull-down list

Create RGB

This operation combines single band grids into one multi-banded grid. This operation can be used for combining Red:Blue:Green grids to a single multi-banded RGB grid. It can also be used to combine multi-component data grids such as used in spectrometery (for example, Potassium, Uranium and Thorium).

Curvature

The **Curvature** utility calculates the curvature of a surface at each cell centre. Three types of curvature grids are available:

- **Profile** curvature is estimated along the direction of maximum slope
- **Plan** curvature is estimated across the direction of maximum slope
- **Surface** curvature is computed as the difference between the Profile and the Plan curvatures.

The method used to create the curvature grids is as follows.

Curvature is computed for the centre cell (e0) within a 3x3 kernel such as:

e1	e2	e3
e4	e0	e5
e6	e7	e8

The first step is to estimate the coefficients D through H of a quadratic polynomial equation that fits the 3x3 window.

$$z = Ax^2y^2 + Bx^2y + Cxy^2 + Dx^2 + Ey^2 + Fxy + Gx + Hy + I$$

where

$$D = [(e4 + e5)/2 - e0] / CellSize^2$$
$$E = [(e2 + e7)/2 - e0] / CellSize^2$$
$$F = (-e1 + e3 + e6 - e8) / 4 \times CellSize^2$$
$$G = (-e4 + e5) / 2 \times CellSize$$
$$H = (e2 - e7) / 2 \times CellSize$$

The **Profile Curvature** is estimated along the direction of maximum slope and is computed as:

$$\text{Profile Curvature} = -2 [(DG^2 + EH^2 + FGH) / (G^2 + H^2)]$$

The **Plan Curvature** is estimated across the direction of maximum slope and is computed as:

$$\text{Plan Curvature} = 2 [(DH^2 + EG^2 - FGH) / (G^2 + H^2)]$$

The **Surface Curvature** is the difference between the Profile and Plan and is computed as:

$$\text{Surface Curvature} = -2 (D + E)$$

A positive curvature indicates that the surface is upwardly convex at that point, whilst a negative curvature indicates that the surface is upwardly concave. A value of zero indicates that the surface is flat.

Cut/Fill

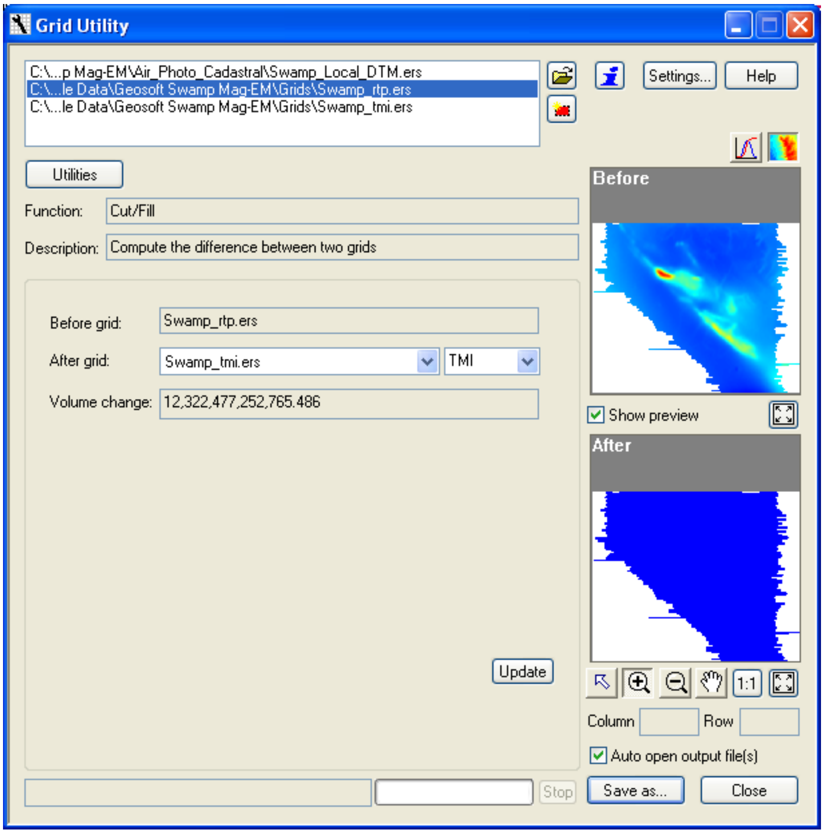
This utility will compute the difference between two grids.

Once two grids have been loaded into the session, open the **Grid Utility** from the **Grid** menu and choose the **Cut/Fill** option from the available list.

In the list of available grid files appearing at the top of the dialog highlight the grid you wish to compute the change from. This grid file will appear in the **Before** grid box further down in the dialog.

Then in the drop down list for the **After** grid choose the grid that you want to subtract values from.

The **Volume change** entry box will automatically update with the computed change in volume between the two grids.



The Cut/Fill grid utility parameters.

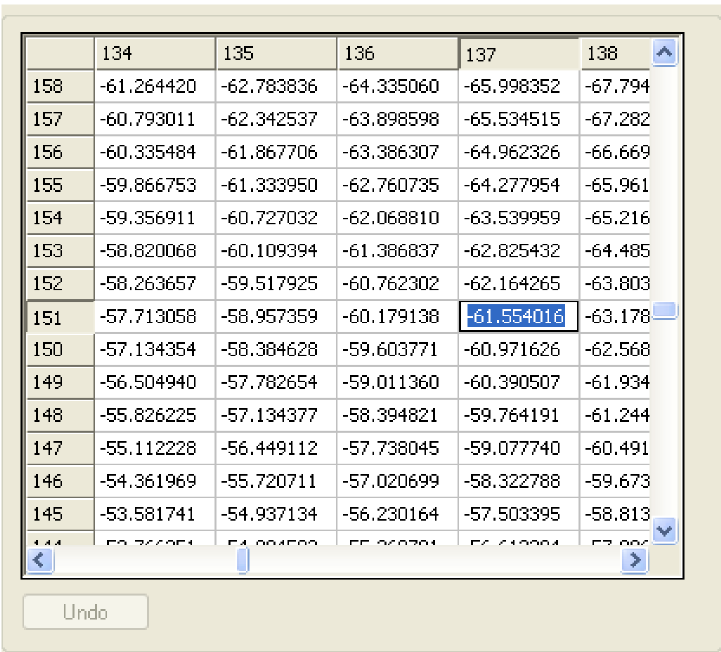
Edit

The data values of grids can be individually selected and modified if required. Select a grid and highlight it in the grid list. The Preview panes show the grid after selection. After choosing **Edit** in the Function pull-down list, the Pointer mode can be used to select individual grid cells for editing. Select the **Pointer** mode button, move the cursor to the Preview pane and select the grid cell. When this is done, the spreadsheet redisplays to indicate the data values of the selected cell and the various nodes around it.

If the level of zoom is inappropriate, select the magnifying glass **zoom in** or **out** buttons and adjust the view.

To more precisely locate the cursor and grid cell, you may also wish to display a line cursor (see diagram below). This is enabled from the **Settings** button.

Once a grid cell has been selected, double clicking inside the spreadsheet cell allows the data value to be edited with a new value typed from the keyboard.



	134	135	136	137	138
158	-61.264420	-62.783836	-64.335060	-65.998352	-67.794
157	-60.793011	-62.342537	-63.898598	-65.534515	-67.282
156	-60.335484	-61.867706	-63.386307	-64.962326	-66.669
155	-59.866753	-61.333950	-62.760735	-64.277954	-65.961
154	-59.356911	-60.727032	-62.068810	-63.539959	-65.216
153	-58.820068	-60.109394	-61.386837	-62.825432	-64.485
152	-58.263657	-59.517925	-60.762302	-62.164265	-63.803
151	-57.713058	-58.957359	-60.179138	-61.554016	-63.178
150	-57.134354	-58.384628	-59.603771	-60.971626	-62.568
149	-56.504940	-57.782654	-59.011360	-60.390507	-61.934
148	-55.826225	-57.134377	-58.394821	-59.764191	-61.244
147	-55.112228	-56.449112	-57.738045	-59.077740	-60.491
146	-54.361969	-55.720711	-57.020699	-58.322788	-59.673
145	-53.581741	-54.937134	-56.230164	-57.503395	-58.813
[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]

Selected grid cell and displayed data value available for editing

Fill Holes

The **Fill Holes** utility is used to replace nulls in a grid by interpolating the surrounding data values. The following Fill Holes options are available:

- Fill internal holes only
- Only internal holes or null grid cells which are not connected by nulls to the outer edge of the grid are given a new interpolated value
- Fill internal holes and nulls surrounding grid
- All null grid cells in the grid are replaced with interpolated values.

The method used to fill holes involves progressive interpolation from the outer rim of the hole towards the centre until the entire hole is filled. This process can be slow for large holes.

Flip

A selected grid can have the order of its rows reversed to produce a vertical flip and columns reversed to produce a horizontal flip. No change is made to the number of rows or columns of a grid in this operation and the extent and origin of the grid in unchanged.

Merge

The **Merge** grid tool allows multiple grids (overlapping or non-overlapping) to be combined into a single output grid. A number of data handling options are provided for overlapping cell values. This tool requires all grids to be in the same projection (use the *Reproject* Grid utility to convert grid projections).

Within the **Merge** grid dialog area, select the source grids from the pull-down list and use the **Add** button to add these to the **Grids to merge** window below. Selecting grids from this pull-down list will not force a refresh of the preview screen each time a grid is selected, which is useful when dealing with large grids. The preview screen can be manually refreshed using the **Update** button. Grids can be removed from the **Grids to merge** window using the **Remove** button.

A primary grid must be highlighted in the **Grids to merge** window; the Merge tool will use this grid’s cell dimensions for the output grid.

A range of **Overlap combining methods** are provided for handling of overlapping grid cells:

Minimum
Maximum
Average
Sum

Outline

The **Outline** utility allows the bounds of multiple grids to be outlined as polygons in a single output vector file. Each bounding polygon is attributed with the source grid file name, and its minimum and maximum X and Y coordinates.

This is a useful way of visualising the extents of multiple regional airborne surveys, such as aeromagnetic and gravity surveys.

Note Each input grid must have the same projection.

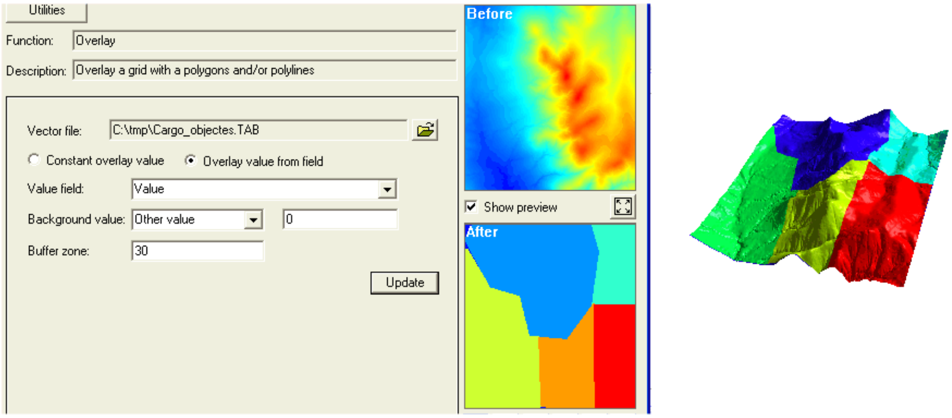
Overlay

The **Overlay** utility enables grid cells to be classified using polygons or polylines from a specified MapInfo Professional TAB or MID/MIF vector file. The dialog is enabled once an appropriate vector file (polyline or polygons) is opened using the **Browse** button

Grid cells may be attributed with a **Constant** overlay value by entering a numeric or text value into the **Overlay value** window. If the polygons or polylines in the vector file contain attributes then choose the **Overlay value from field** option and select the attribute field from the **Value** field pull-down list. All grid cells that are located within a polygon or polyline will have an output value equal to the overlay value of that polygon. Cells which are outside polygons or polylines in the vector file can be assigned a null value, the value of the input grid (unchanged) or a user-defined constant value.

In addition to the use of polygons in the input vector file, polylines and point objects are also supported. A **Buffer** zone may be specified by the user to control how wide these objects appear in the overlay output. The default buffer zone of 0 means that a polyline will create an overlay that is approximately one grid cell wide. For example, a buffer value of 15 will create a line overlay with a width of 30 metres (15 metres each side of the line). Points will result in a circular coverage with a radius equal to the buffer zone.

If the buffer zone value is non-zero then polygons will also have their boundaries extended by the amount specified. An example of an overlay with polygons used to define areas on a DEM surface is shown below:



Use of a polygonal file to assign colour values over a grid surface plus the surface when displayed in a 3D view

Replace

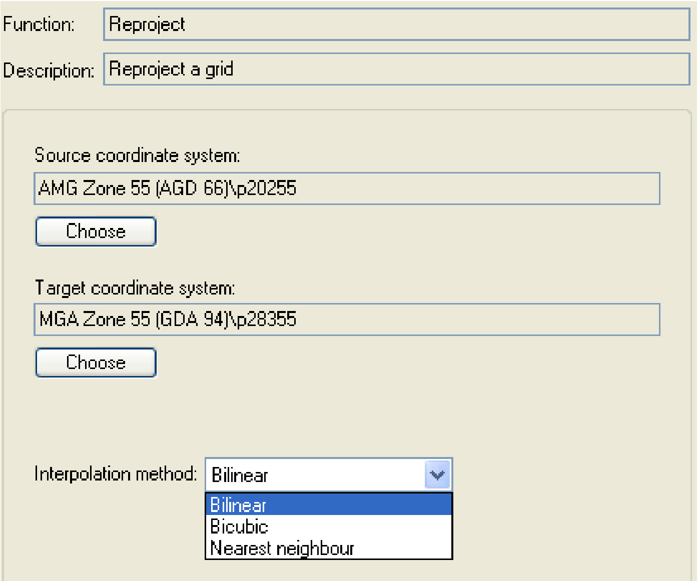
Replace all instances of a specified grid cell value with a new value.

This operation is especially useful when manipulating null values within grids. The value of a **Null** may differ with different grid formats but the Grid Utility compensates for this.

Use **Replace From...To...With** to replace nulls or values, such as negative values.

Reproject

Most grids have a Projection and Spheroid/Datum specification either embedded in their format or in a separate header file (for example, .ERS in the case of ER Mapper files). This information can be re-specified by the **Reproject** option and used when a new output grid is written. Note that this operation relocates the various mesh points in the grid through an interpolation method to match the requested output Projection and Datum combination.



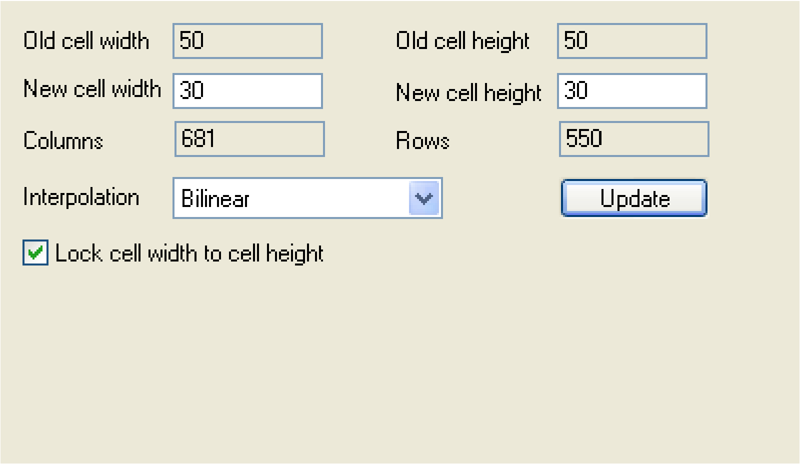
The Reproject dialog and associated controls

The dialog as shown allows the following functions:

- Both X/Y (Easting/Northing) or Latitude/Longitude grids can be reprojected.
- A wide range of Projections, Spheroids and Datums are supported for import and export.
- **False Easting** and **False Northings** can be defined
- The data can be interpolated using either **Bilinear**, **Bicubic** and **Nearest Neighbour** (refer to details in *Resample* a Grid).

Resample

Resampling a grid allows a resizing of the cell size of the grid. Select the **Resample** option in the Function pull-down list. The control parameters appear as below. You can specify a new height or width of the interpolated cells. By default these are set to be the same, but if you want to create rectangular mesh sizes for the grid, deselect the **Lock cell width to cell height** checkbox.



Resample control parameters in the Grid Utility dialog

The resampling is done by interpolation of the specified input grid data values. The methods available include **Bilinear**, **Bicubic** and **Nearest Neighbour**.

These interpolation methods use a transformation (for scaling), where the row/ column values `[u,v]` describe a transformed plane and `[x,y]` the original grid data plane. To determine the samples of the transformed grid `g([m,n])` from the samples of original `f([k,l])` the following steps are applied:

- Calculate `c([k,l])` from the grid values at each `f([k,l])` where `c` = cell reference
- For each `[m,n]` of the transformed grid find corresponding source location `[x,y]` in the input grid
- Compute `f([x,y])` using the spline model, where `g([m,n])=f([x,y])`

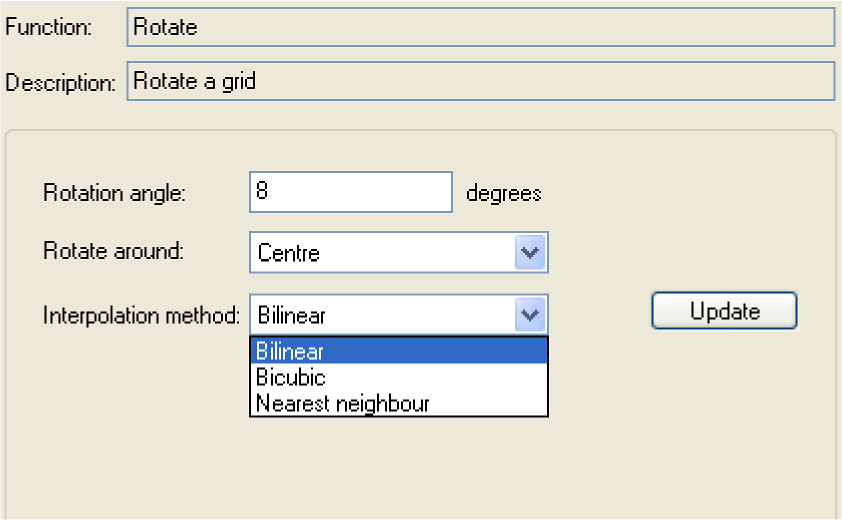
For the resampling of a grid to new cell sizes and comparing the fits using $\varphi^n([m,n])$, the order on the nth term determines the method applied where `n=0` (nearest neighbour), `n=1` (bilinear), `n=3` (bicubic).

Jagged or sharp edges using a nearest neighbour interpolation progressively smooth out in higher order spline-interpolation. Bicubic spline interpolation is popular as cubic splines appear smooth to the human eye.

After a cell size and method have been chosen, you can click the **Update** button to force the processing to proceed and review the result in the preview pane.

Rotate

Similarly to the Resample operation (described above), the interpolation methods of **Bilinear**, **Bicubic** and **Nearest Neighbour** can be used to rotate a grid around a fixed point (by default, its centre or the bottom left corner). The control parameters for this function are shown below:

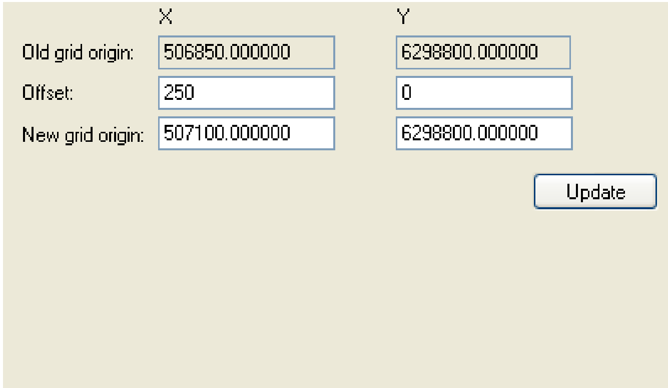


Rotation of a grid and the Before and After results shown in the Preview panes

An angle that is positive causes the rotation to be clockwise and vice versa. Refer to the Resample a Grid operation for a description of the Interpolation methods available.

Shift

The origin of a grid can be transformed horizontally and/or vertically using the **Shift** function of the Grid Utility. The control parameters appear as below. Entries of a **New grid origin** can be made or you can enter a specific **Offset** for either the east (X) or north (Y) directions. Once the shift parameters are entered, clicking the **Update** button causes the grid to redisplay with the adjustment in the After preview pane.



Shifting a grid control parameters

Slope

The **Slope** utility analyses the eight cells surrounding each grid cell and determines the slope angle in degrees or the slope direction for each grid cell. The slope angle or direction is then stored as the cell value for the output grid.

The slope direction is measured in degrees clockwise from the grid Y (north) axis. The slope direction indicates the downwards direction of the maximum gradient (slope) of the cell.

The direction is calculated by

$$(180/pi) \times atan2(dz/dy,-dz/dx)$$

Where dz/dy is the rate of change from above to below the cell (vertically) and dz/dx horizontally across the cell's left and right neighbours.

a	b	c
d	e	f
g	h	i

The rate of change in the x direction for cell 'e' is calculated with the following algorithm:

$$dz/dx = ((c + 2f + i) – (a + 2d + g)) / 8$$

The rate of change in the y direction for cell 'e' is calculated with the following algorithm:

dz/dy = ((g + 2h + i) – (a + 2b + c)) / 8

The slope is the maximum value of slope across the cell, where a percentage value of 100% is equal to 45 degrees gradient. the algorithm calculates this by comparing each of the 8 neighbour cells, and then

(180/pi) x atan(sqrt([dz/dx]^2 + [dz/dy]^2))

To convert the slope direction into a driectional aspect—e.g. N, NE, and so forth—use the **Classify** tool to classify each cell by its slope direction.

Split

The **Split** operation is the converse of the Combine function. The Split outputs the individual bands of a multi-banded grid (such as an RGB grid) and outputs individual, single band grids.

Statistics

This tool calculates a range of standard **Statistics** for either:

- The entire grid.
- Grid cells enclosed or intersected by any objects in a vector file, either totally or partially overlying a grid. The vector file can be composed of a mixture of object styles.
- Grid cells within classified regions in another classified (i.e. integer) grid.

The grid statistics which are computed include:

- Number of cells
- Number of valid (non-null) cells
- Number of null cells
- Minimum cell value
- Maximum cell value
- Mean (average) cell value

- Range (minimum to maximum cell value)
- Variance (average of the square of the difference between the cell value and the mean)
- Standard deviation (a measure of the “spread” of cell values equal to the square root of the variance)
- Sum of all grid cell values
- Percentage of null cells

For large grids, only the preview tile area statistics will be displayed until the **Save As** button is used to process the full grid and save the results.

Any statistical values which can be computed, for example due to no valid cells in the region or negative values, will be written as -9999.

Integer Grid

An "integer grid" is a grid which has been classified into a set of discrete values, i.e. by using the **Classify** tool. Typically these will be integers, but it may also be decimal numbers and can contain null areas.

The regions used for statistics will be area of the integer/classified grid which have the same value (including null value).

One selected output property (statistical value) can be created.

When **Save As** is clicked, or the batch process run, a grid will be created containing cells attributed by the select property.

Note Each input grid must have the same projection, cell size, and should overlap.

Vector File

If a unique identifier field exists for each vector object (e.g. sample number, rock code, etc), this field can be set as the ID Field. The attributes in this field will be used as an identifier in the output file for each vector object., The Objects to process option allows the user to either create statistical output for all object types in the vector file, or one style in particular (points, lines or polygons).

Note The input grid and vector file must have the same projection.

If an object lies partially or entirely outside the grid file extents, this portion of the object will ignored for the statistics. For example, a polygon which is half area outside the grid, will still be 0 or 0% null cells, provided there are no null cells in the grid. Averages etc will be calculated based on the cell count, not the polygon area.

Clicking the **Save As** button will create a new MapInfo vector file named *vectortablename*_stats), with each input object attributed with its calculated statistics (as well as the ID Field if specified). These statistics can also be exported to either a Comma delimited (*.CSV) or Tab delimited (*.TXT) file using the **Export Statistics** button.

The method used to determine if a grid cell is located within a polygon, is whether the centre point of the cell is located within the polygon. Note that some partially covered cells may not be included due to this. To increase the accuracy you can use the **Resample** tool to decrease the cell size.

Surface Area

The **Surface Area** utility calculates the total 3D surface area and 2D planar area for a grid. It also calculates the roughness of the surface, which is the surface area divide by the planar area. A perfectly flat grid (i.e. where every grid cell has the same value) will have a roughness of 1.

The values for the total grid are reported in the dialog in the same units the grid is defined in. Individual grid cell surface area or roughness values are saved to the output grid.

The algorithm calculates the 3D area by diving each cell into 8 triangles, and calculating the side length of the triangles by accounting for the slope of the cell edges. It then calculates the area for each triangle. This is similar to creating a vector TIN and calculating the surface area of this.

Usage examples of this tool include:

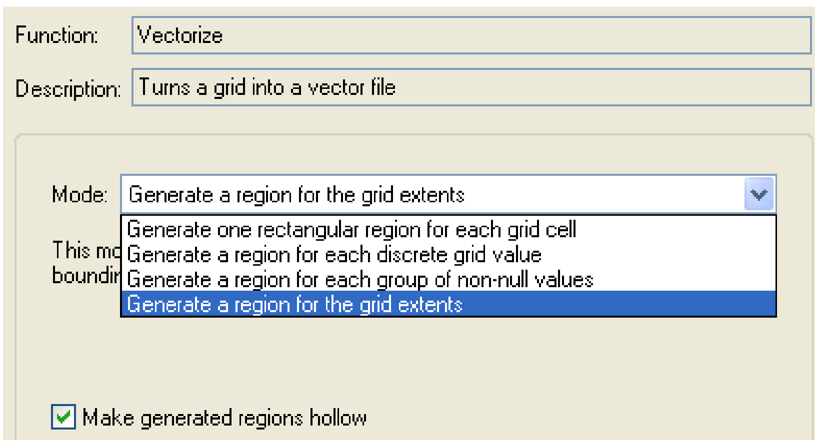
- Comparing the relative roughness of different DEM/DTMs.
- Calculating the actual topographic surface area of a catchment/drainage basin, for 'run-off' studies.
- Analysing the effects of different filters or gridding techniques on the 'topography' of the grids.

Note The preview for either the surface area or roughness will look almost identical with histogram equalization applied, but the value of the grid cells will be different.

Vectorise

The **Vectorize** utility converts a grid into a vector file. Each grid cell of the input grid is converted into a single region (or polygon) in the output MapInfo TAB or MID/MIF file. Each output region cell is assigned a grey-scale colour value based on the input cell value using a simple linear stretch on the data range.

It is recommended that large grids are re-sampled prior to using the vectorize utility to reduce the number of vector polygons.



The Vectorise utility for grid files.

Four options are supplied for vectorising grids in this utility:

- Generate one rectangular region for each grid cell – this produces a region for each grid cell.

  Each grid cell of the input grid is converted into a square polygon in the output MapInfo TAB or MID/MIF file, attributed with the grid cell value. Each output region cell is assigned a grey-scale colour value based on the input cell value using a simple linear stretch on the data range.

- Generate a region for each discrete grid value.

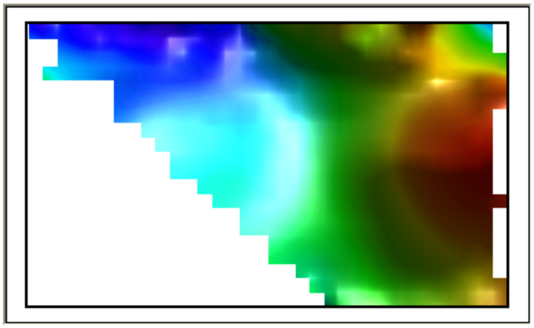
Each unique value in the grid is analysed for contiguous areas and a polygon boundary drawn around these. This will produce a vector map with polygons bounding areas of homogenous grid values, each attributed with the unique region valuer. Each output region is assigned a grey-scale colour value based on the input cell value using a simple linear stretch on the data range.

- Generate a region for each group of non-null values.

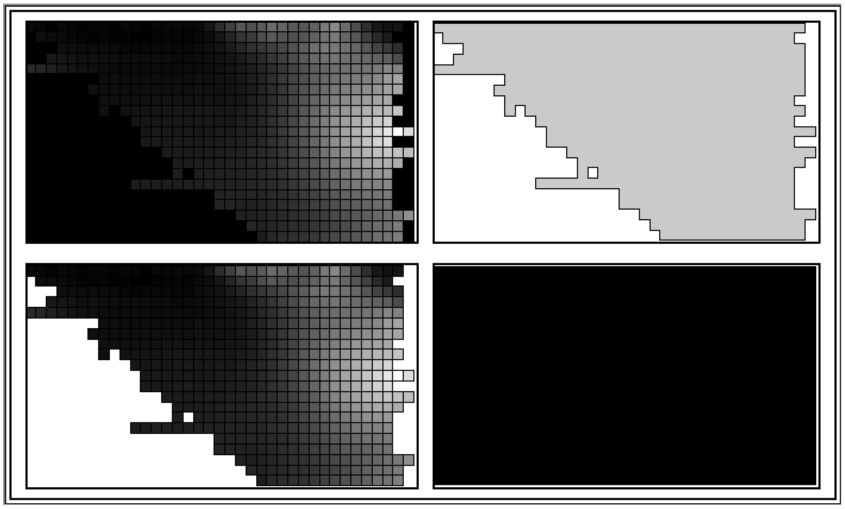
  This produces a separate polygon for each contiguous area of valid (nonnull) grid data. For instance, a series of polygons outlining coastlines could be generated from a DEM by first using the *Classify* Grid Utility to set all grid cells with sea level heights (and lower) as 'Nulls', then Vectorizing the resulting grid using this mode.

- Generate a region for the grid extents.

  Produces a polygon outlining the area covered by the grid cells.



Original grid before being vectorised.



Examples of vectorised grids (clockwise from top left): One rectangular region for each grid cell; One region for each group of non-nulls; One region of the grid extents; One region for each discrete grid value.

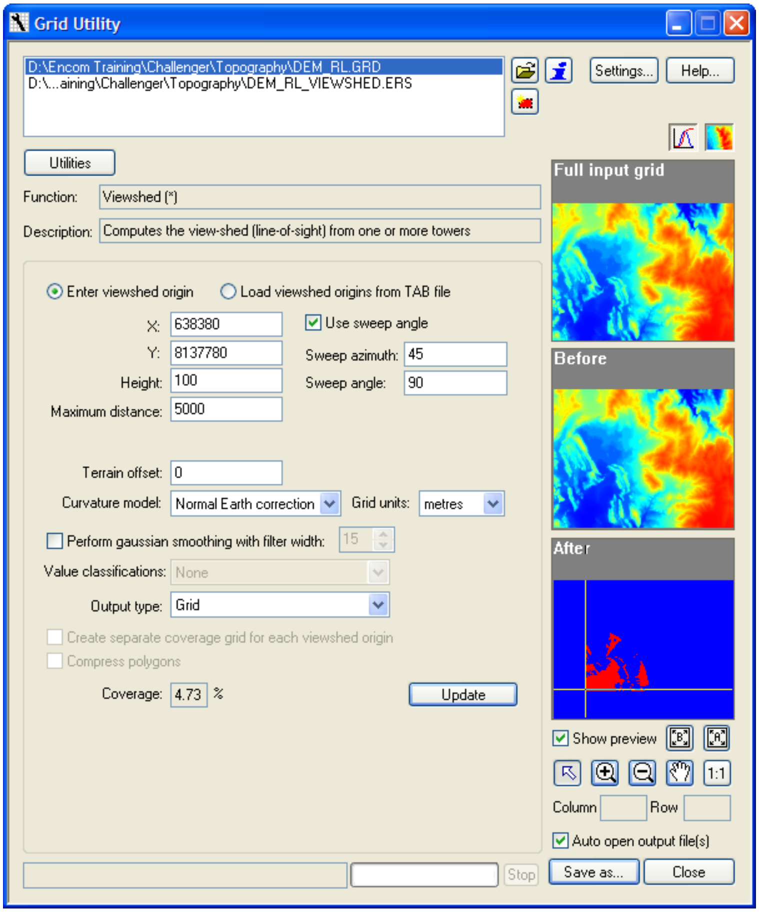
The option **Make generated regions hollow** will produce ‘empty’ vector regions without any fill colour or pattern.

It is recommended that large grids are resampled to a larger cell size (see *Resample*) prior to using the Vectorize utility to reduce the number of vector polygons.

Viewshed

The **Viewshed** utility computes the view-shed from one or more locations of a specified height above the grid to an observer at a specified height (terrain offset) above the grid. The areas of the grid that lie within the line of sight are saved to a new grid. This is a useful tool when considering, for instance, the spatial coverage provided by a series of telecommunication or repeater towers, or the positioning of fire observation platforms.

Note The input grid and vector file must have the same projection.



Grid Utility Dialog showing Viewshed options

The **Viewshed origin** is the location from which the line of sight is to be calculated. The **X** and **Y** position of the **Viewshed origin** can be manually entered or selected by clicking the arrow tool and selecting a location in the “Before” or “After” preview windows. Alternatively, one or more viewshed origins or tower positions can be specified by using a MapInfo TAB or MIF file containing point locations. This TAB/MIF file may optionally include the following fields: sweep azimuth, sweep angle, height, maximum distance and ID field. These optional fields can be specified for the appropriate parameters (below) allowing each viewshed origin to have differing values (e.g. different tower heights and ID’s) instead of a single Constant value.

The **Height** or z-value is the height of the view-point origin above the specified grid; this may represent a structure such as a transmission tower.

The **Maximum distance** is the radial extent around the viewshed origin or tower. The line of sight is calculated for each grid cell within this radius.

Sweep Angle and **Azimuth** controls are available, limiting the view from a Viewshed Origin to the specified bearing range.

Setting a **Terrain offset** adds the specified value to each grid cell (e.g. the height of a receiver or person) and is incorporated into the line of sight calculation.

The curvature of the earth can also be taken into account when calculating the view-shed by specifying a **Curvature Model**. If this option is selected, the grid distance unit needs to be selected to allow the scaling of the earth's radius to the correct units.

A **Gaussian smoothing filter** can be applied to the line-of-sight grid to eliminate spurious lines caused by small irregularities in the input grid. The calculated line-of-sight grid values may also be classified into one of two options:

- **2 (in/out)**
  Denotes whether a cell is visible or not from the viewshed origin.

- **3 (in/fringe/out)**
  Denotes whether a cell is visible, not visible or on the fringe when viewed from the viewshed origin.

Two **Output types** are available:

- **Grid** – if multiple viewshed origins are used (via a .TAB or .MIF file), the tool can **Create a separate coverage grid for each viewshed origin**. The output grids will be suffixed with the **ID Field** (if assigned), otherwise a 1, 2, 3, etc suffix will be used.

- **Polygons** – can only be selected if the **Perform Gaussian Smoothing** option is enabled. This will produce a polygon outlining the extent of the viewshed (or combined viewsheds). The polygon table incorporates a Coverage field, indicating the percentage of the total input grid (null and non-null cells) covered by the polygon. If multiple viewshed origins are used (via a .TAB or .MIF file), the tool can **Create a separate coverage polygon for each viewshed origin**. The output table will incorporate an ID field, which will be populated with either the specified **ID Field** (if assigned), or an integer value (1, 2, 3). The **Compress polygons** option reduces the number of nodes in the output polygons by applying Tomek compression (removing details which are smaller than the cell width). This option is only available for the 2 (In/out) Value Classification.

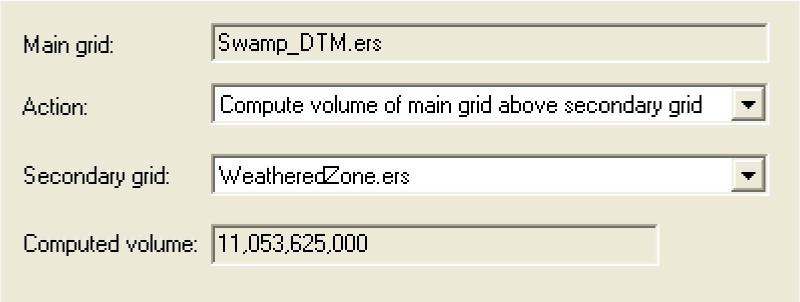
Note For best results with this tool, it is recommended to enable the **Histogram Equalisation** and **Look-Up Table** options.

Volume

The **Volume** utility computes and displays the volume between two grids, or between one grid and a constant Z value. Examples of use include calculating the volume between:

- The water table and a mining surface
- A DEM (top surface) and the same DEM after open-cut mining (modified by overlaying a series of attributed polygons representing mining benches (bottom surface).
- Upper and lower depth surfaces for a heavy mineral layer in mineral sands.
- A DEM and the DEM modified with a new airstrip.

This tool requires both grids to be in the same projection (use the *Reproject* utility to convert grid projections) and have the same cell size (use the *Resample* utility).



Volume calculation parameters for two loaded grids in the Grid Utility

The volume between two grids is calculated by subtracting each bottom grid cell Z value of one grid from the corresponding top grid cell Z value and then multiplying by the first grid cell width (X) and height (Y). Each individual grid cell volume is then added together to compute the final volume.

The following volume calculation options are available:

- **Compute volume of main grid above secondary grid**: only those cells where the main grid lies above the secondary grid are used to calculate the volume
- **Compute volume of secondary grid above main grid**: only those cells where the secondary grid lies above the main grid are used to calculate the volume

- **Compute total volume between the two grids**: all overlapping cells are utilized to calculate the volume, regardless of which grid in is top. The difference between cell values is treated as absolute i.e. all values are positive.
- **Compute volume of main grid above constant Z value**
- **Compute volume of main grid below constant Z value**

Where the grid cell sizes between the two grids are different, a **Nearest Neighbour** interpolation is used on each grid cell in the top grid to determine a value for the bottom grid at that location. This will result in the closest bottom cell grid value to the top grid location being used in the volume calculation for that grid cell.

If the two grids overlap vertically (i.e. portions of the lower grid extend above the upper grid), the calculation will treat the overlap volumes as negative volumes. These will therefore be subtracted from the positive volumes (below the upper grid), resulting in a reduced volume calculation. To treat these overlap volumes as positive volumes, use the **Grid>Grid Calculator**.

Assign Grid Value to Dataset

Grid > Assign grid values to dataset

The Assign grid values to dataset tool will assign values from select grid(s) to a survey Database or Feature database file. It does this by taking the input grids as point datasets, with each points at the centre of a cell. At each of the dataset's XY point, it then interpolates the surrounding grid cell centre points using the select interpolation method, to calculate a value to assign. This can be assigned to a numerical field or to each node's/station's Z value.

Note This interpolation steps (Bicubic, Bilinear or Nearest Neighbour) has the effect of smoothing or blurring the input grid cells resolution and the value assigned will not exactly match the overlying grid cell values.

For Feature baseases, as these contain polygons, polylines, and surface TINs; there are extra options to Inteprolate the Z value.

By default, the Z value of each node will be overwritten and replaced with the value of the grid (**Interpolate to Z - replace**). You may also choose to offset the original features by the grid values with the options -

- **Interpolate to Z - sit** Z = GridZ + (FeatureZ - ObjectsMinimumZvalue)

- **Interpolate to Z - straddle** Z = GridZ + (FeatureZ - ObjectsMidZvalue)
- **Interpolate to Z - hang** Z = GridZ + (FeatureZ - ObjectsMaximumZvalue)

Where FeatureZ is the original Z value of the node, and Z the new node value.

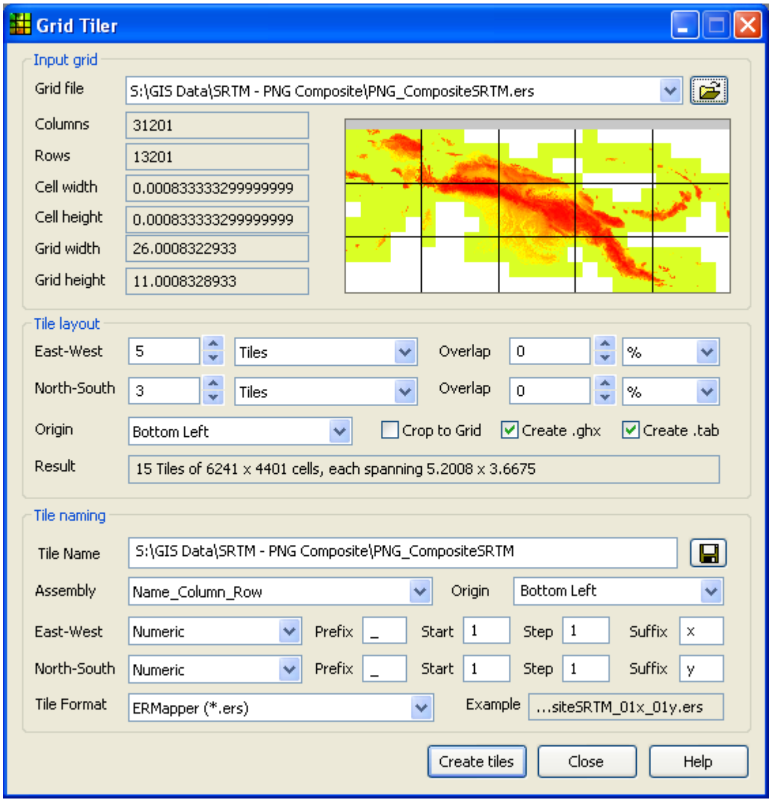
The object's minimum/maximum Z value, is the highest of lowest value of original values in the entire Feature database.

The object's Mid Z value, is the (Minimum Z + Maximum Z)/2.

To assign grid values to a dataset:

1. Ensure the Feature Database or Geosoft Database and grid files are open in PA.
2. Select **Grid>Assign values to Dataset**.
3. Select the **Dataset** to assign to.
4. Check the **Interpolate** setting, to Interpolate to Field or Interpolate to Z (if using a Feature Database)
5. In the **Grids** list, tick or deselect the **Use field** to indicate which grid will be assign to the dataset.

Grid Tiler



Grid Tiler dialog for large grids

The Grid Tiler allows large grids to be sub-divided or tiled into a number of equally sized smaller grids. This is useful when a grid file is either too large to open or import into Encom Discover PA, or too large to modify or interrogate with the Grid Utility or Grid Filter tools.

1. To subdivide a large grid: Open the Grid Tiler (the source grid does not have to be open within Encom Discover PA).

2. Using the **File Open** button at the top right of the dialog, select the source grid file (if the grid is already open within Encom Discover PA, select it from the top pull-down list). A preview of this grid will be shown along with the grid size information.

3. In the **Tile Layout** section, set the number of tiles to divide the source grid into in both the East-West and North-South orientations. See *Tile Layout* below for further information.

4. Ensure that the **Create .GHX** option is enabled to allow the pre-calculation of grid statistics for more efficient grid handling. Enable the **Create .TAB** option if you want to automatically create a .TAB file for output tiled grid to Encom Discover PA.

5. By default the output grid tiles will be created in the same directory as the source grid: this can be modified in the *Tile Naming* section.

6. The output tiles will be named by adding a suffix to the input file name. A number of suffix options exist – see *Tile Naming* below for more information.

7. Select the output **Tile Format** (formats available are ERMapper, BIL, Encom or Surfer grid formats and press Create Tiles. The grid tiles will be automatically registered in the same projection system as the input grid file.

Tile Layout

In addition to setting the number of tiles to create along the X and Y axes, you can instead define the size of the tiles by the distance (width or height) or the number of cells (rows or columns). An **Overlap** can also be defined between adjacent tiles as a percentage of the distance (width or height), or by a number of cells (rows or columns). The overlap interval is applied to both sides of each tile (east and west or north and south).

If the size of tiles does not divide evenly into the input grid dimensions, the edge tiles will extend slightly past the source grid bounds. The **Crop to grid** option will crop these tiles to the extents of the input grid. The Origin can be set to the Top left or Bottom left so that these edge tiles will be located at the Bottom or Top row of the grid. Cropped are always located at the right side of the grid.

Tile Naming

The suffix applied to the output tiles can either be a Name-Column-Row or Name-Row-Column format. For either format, the initial tile **Origin** (i.e. 01x_01y) must be specified as either the bottom left or the top left corner tile.

The E-W or N-S components (i.e. Column and Row suffixes) of the output tile suffix can each independently have either a numeric or character format. Additionally each component can have individual prefixes and/or suffixes. The preview of the entire Tile name suffix is shown in the example box.

Vector File to Grid Utility

The Vector File to Grid plugin utility is a very powerful tool for converting a vector file to a grid file. The utility supports the import of a large range of industry standard vector file formats as well as most industry standard grid file formats for converting the vector file to.

The following input vector and output grid file formats are supported by this utility:

Vector Files:

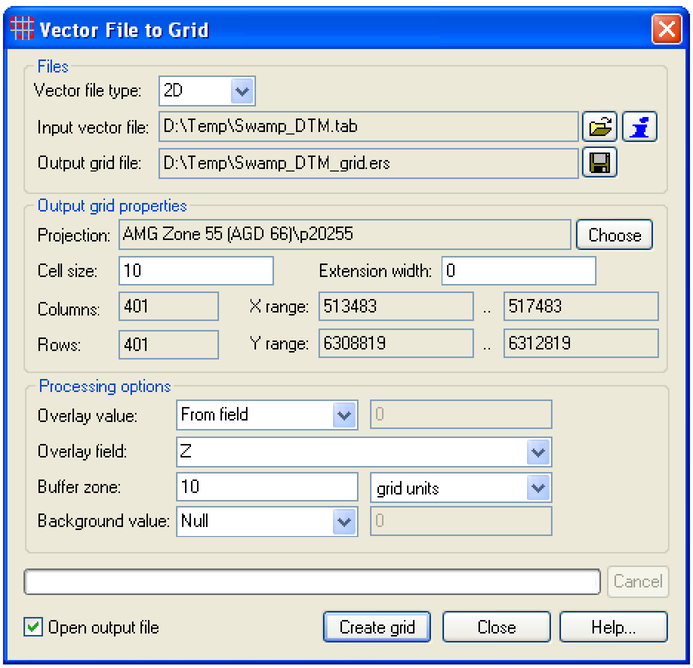
- 3D Studio files (.3DS)
- AutoCAD DXF files (.DXF)
- Datamine wireframe files (.DM)
- ERMapper Vector files (.ERV)
- ESRI TIN files (.ADF)
- ESRI Shape files (.SHP)
- Gemcom BT2 files (.BT2)
- Google KML files (.KML)
- GOCAD vector files (.TS, .PL and .VS)
- GPS eXchange files (.GPX)
- LizardTech MrSID files (.SID)
- MapInfo TAB files (.TAB)
- MapInfo MIF files (.MIF)
- Surpac DTM files (.DTM .STR)
- Vulcan triangulation files (.00T)

Grid Files:

- Arc Ascii grid files (.ASC)
- Arc Gridfloat grid files (.FLT)

- BIL grid files (.BIL)
- Encom grid files (.GRD)
- ERMapper grid files (.ERS)
- Geosoft grid files (.GRD)
- GeoTIFF files (.TIF)
- MapInfo grid files (.MIG)
- Minex grid files (.XYZ)
- Surfer Binary grid files (.GRD)
- Vertical Mapper grid files (.GRD)

Access this utility from the **Grid>Vector File to Grid** menu option and the below dialog will appear.



The Vector File to Grid dialog

Files

Click on the **Browse** button for the Input vector file option to load a vector file from its known file location. Then click on the **Save As** button to specify the intended location of the output file. Once these are specified, the **Data range** section of the dialog will be populated with the minimum and maximum Easting (**X**), Northing (**Y**) and Elevation of other property (**Z**) values. The Output Grid Properties and some of the Processing options will be automatically populated based upon the input vector file data.

Output Grid Properties

Under the **Output grid properties** section the **Cell size** and **Extension width** will be filled with a default (safe) value based on the vector file dimensions. Edit this value to the desired cell size and extension width. The **Rows** and **Columns** boxes will be updated to compute the edited cell size. The **Projection** information will also be automatically populated. Press the **Choose** button to change the coordinate projection of the output grid file.

Processing Options

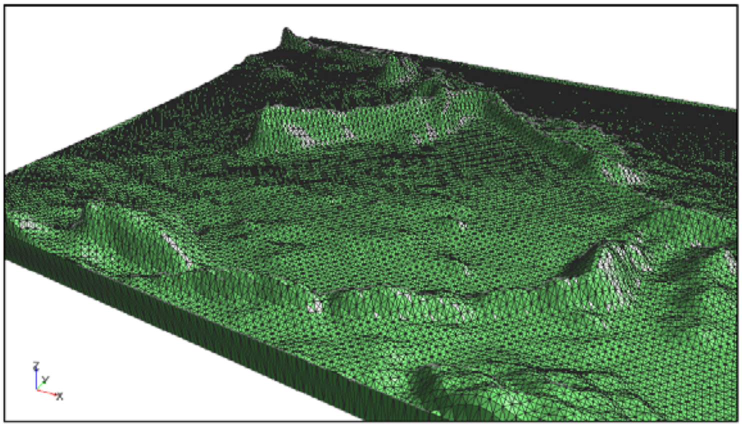
When satisfied with the output grid paramters specified, select the **Create grid** button. The following processing options will become available depending upon whether the input vector file is a 2D vector file or a 3D vector file.

When a 3D vector file is loaded in the utility:

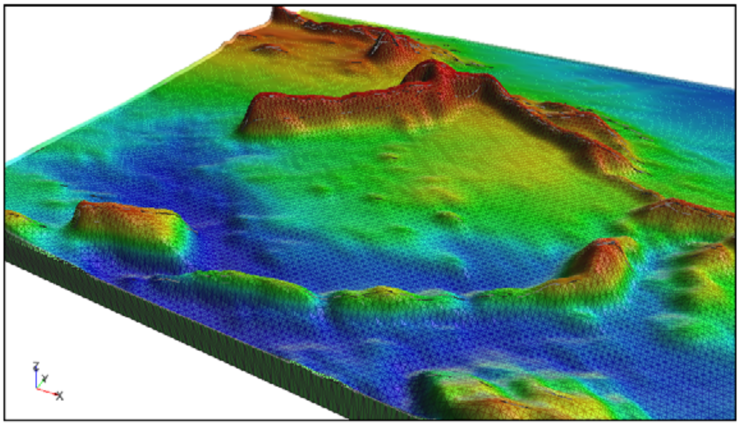
- The **Vector objects to process** option automatically displays as **3D faces**
- The **For cells in the grid where there is no data** option allows the choice between:
    - **Use null values** - populates any cell that doesn't have a Z value with the null value
    - **Use minimum Z value** - populates any cell that doesn't have a Z value with the minimum value
    - **Use maximum Z value** - populates any cell that doesn't have a Z value with the maximum value

When a 2D vector file is loaded in the utility:

- The vector file to grid conversion utility will not automatically open the generated grid into a window display in Encom Discover PA. To view the grid file open a 2D grid map or 3D display window.



An example of a 3D DXF file representing a topographic surface

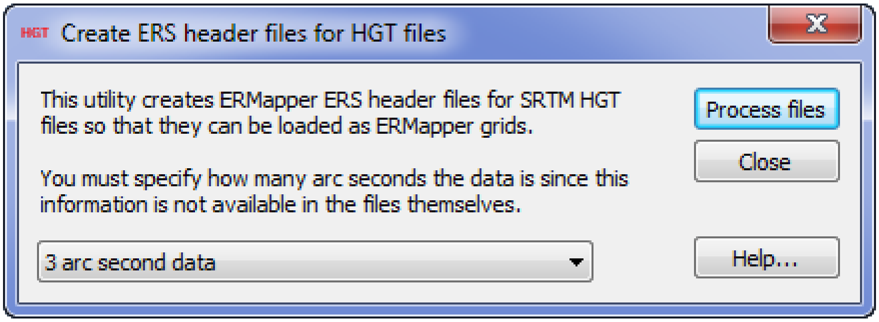


An example of a grid (overlain by the input vector file) created by the Vector File to Grid plugin utility.

SRTM HGT to ERMapper ERS

The SRTM HGT to ERMapper ERS utility is accessed from the **Grid** menu and is a simple tool for converting a SRTM HGT elevation (height) file to a ER Mapper .ERS file.

Each data file covers a one-degree-of-latitude by one-degree-of-longitude block of the earth's surface. The first seven characters indicate the southwest corner of the block, with N, S, E, and W referring to north, south, east, and west. Thus, the "N34W119.hgt" file covers latitudes 34 to 35 north and longitudes 118-119 west (this file includes downtown Los Angeles, California). The filename extension ".HGT" simply stands for the word "height", meaning elevation. It is NOT a format type. These files are in "raw" format (no headers and not compressed), 16-bit signed integers, elevation measured in meters above sea level, in a geographic (latitude and longitude array) projection, with data voids indicated by -32768.



International 3-arc-second files have 1201 columns and 1201 rows of data, with a total file size of 2,884,802 bytes (= 1201 x 1201 x 2).

United States 1-arc-second files have 3601 columns and 3601 rows of data, with a total filesize of 25,934,402 bytes (= 3601 x 3601 x 2).

27 Voxel Utilities

Note All tools in this menu require an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

The **Voxels** menu provides access to the following functionality:

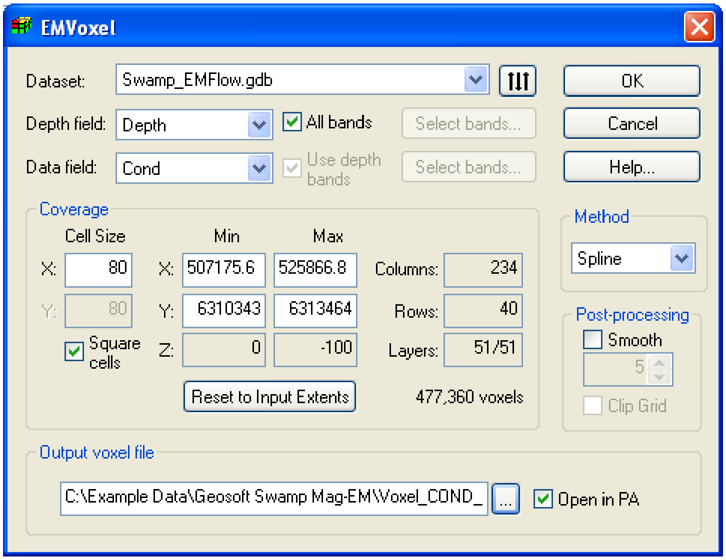
- *Array Data 3D Gridding* – Create a 3D grid (voxel model) of layered data from a selected database containing multi-channel fields, e.g. conductivity-depth fields using minimum curvature.
- *Voxel Toolkit* – Create a voxel model from a loaded dataset or further enhance a voxel model.

Array Data 3D Gridding

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

The **Array Data 3D Gridding** tool is an external utility that creates a 3D voxel model from a database accessed by Encom Discover PA. The database must have two multi-arrayed data fields corresponding to depth and a property (such as magnetic susceptibility or conductivity). Such a database is derived from interpretation software like EM Flow.

The gridding process used by this utility (EM Voxel), creates a 3D voxel model output, but its internal operation is in reality only a 2.5 dimensional gridding process. Each layer of data for the input multi-banded data fields are horizontally gridded using either a spline or a minimum curvature method. Once the grids for each available depth are gridded, they are then combined and output to form the voxel model.



Array Data 3D Gridding (EMVoxel) dialog and control of voxel model creation.

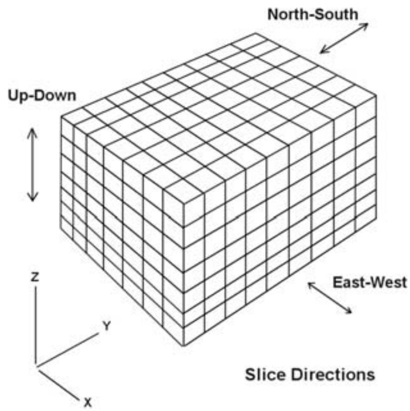
The parameters specified in the EM Voxel gridding tool dialog are:

- **Dataset** – The name of a database accessed by Encom Discover PA. The pull-down list provides the available datasets. Note that the database must contain at least two data channels that contain a multi-arrayed data channel corresponding to depth and a second multi-arrayed data channel corresponding to a property. A property may be magnetic susceptibility, conductivity etc.

- **Depth Field** – The Depth Field specifies the multi-arrayed data channel of the selected database to use for the vertical axis of the gridding. By default the program uses all available array bands, but this can be overridden by turning off the **All Bands** option. With this option turned of, you can select the bands required.

- **Data Field** – This data channel specifies the required property channel. Note that you must have the same number of bands as are defined for depth and the corresponding bands must be selected. For example, you cannot have Bands 1,3,5,7… selected for Depth and Bands 2,4,6,8… selected for Data.

- **Cell Size** – The size (in metres) of voxels horizontally can be specified. The vertical voxel size is determined by the Depth data channel, unless the **Square Cells** option is enabled. In this case, each voxel block is cubic.



- **Min/Max Coverage** – The voxel model created uses the extents of the dataset by default. If these extents are to be altered, you can modify the entry fields. The created number of Columns, Rows and Slices is reported. If a change to the extents is made, the entries can be reverted back to the original data extents by clicking the **Reset to Input Extents** button.

- **Smoothing** – Frequently the data derived from inversion section processing (such as EM Flow) can have noise and processing effects that occur on one data section but not adjacent. Smoothing helps to remove these artifacts. The larger the Smoothing Factor used, the greater the distance in 3 dimensions of smoothing is applied.

- **Gridding Method** – Two methods are provided – Spline or minimum curvature. For most geophysical data, it is recommended that the minimum curvature approach be adopted.

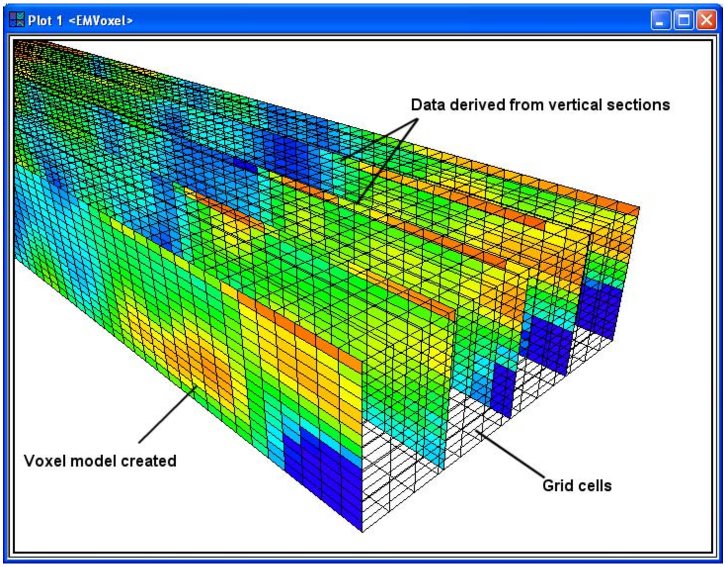
- **Output Dataset** – Specify a path and filename to contain the output voxel model. The Voxel Gridding program creates voxel models using the UBC model format and therefore two files (a header and data file) are created. The data file uses the filename as specified and adds the file extension .DAT.

- **Open in Encom Discover PA** – This option allows a 3D map to automatically display at the end of a 3D Gridding processing and present the created voxel model. The display uses a standard Encom Discover PA template that can be modified if required.

Gridding Operation

The Array Data 3D Gridding utility (called EMVoxel) uses a minimum curvature or spline gridding method and applies it to each depth layer horizontally. Allowance is made for the high difference in data resolution typically seen along data lines (sample intervals), vertically (depth resolution) and widely spaced line separation distances.

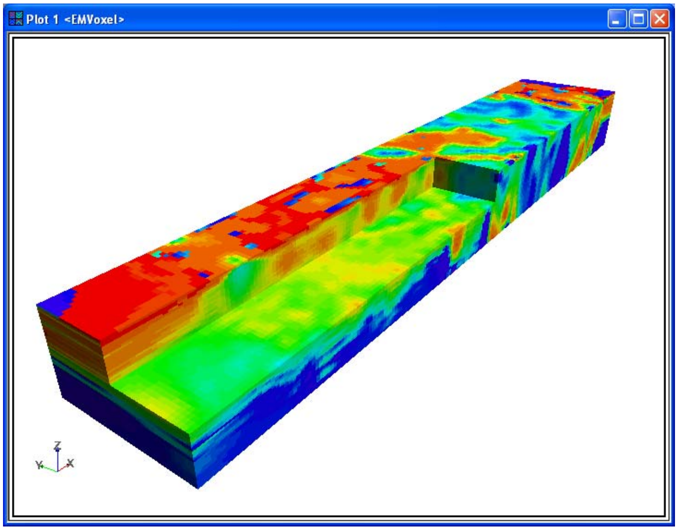
Artifacts observed on some data lines are not present on adjacent lines and are therefore filtered out by the gridding process.



Gridding in each direction horizontally and vertically from properties of each line

Note You may need to correct the depth data for effects such as topography and elevation. If this is required, the appropriate depth and property data channels may require pre-processing using the *Depth Interpolation* option.

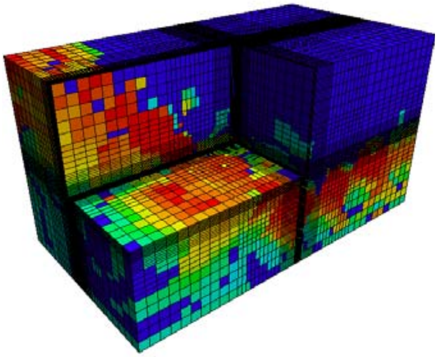
Once you have specified the dataset, data channels and any other required parameters, click the OK button. After process (with the **Show in Encom Discover PA** option enabled), a display window with the created voxel model is displayed. Once displayed, you can use the various Voxel Model settings and controls to clip, threshold and create isosurfaces of the model.



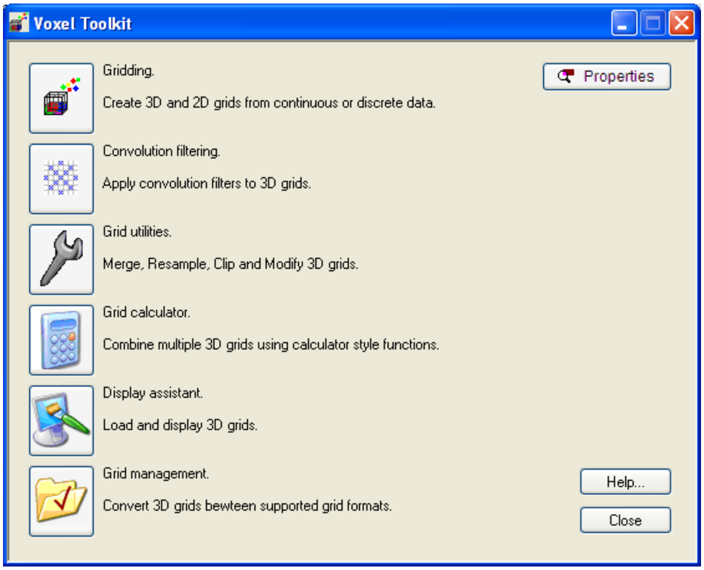
Example voxel model derived from Swamp example dataset

Voxel Toolkit

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.



The Voxel Toolkit is a stand-alone plug-in tool that provides the ability to create, process and export voxel models (often referred to as 3D meshes, block models or 3D grids). 3D 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 The Voxel Toolkit is accessed via the **Gridding** or **Voxels** menu of Encom Discover PA.



Voxel Toolkit dialog with numerous operations available

The main menu page provides the following functionality:

- *Voxel Toolkit Settings* – Click the **Properties** button to access advanced settings for 3D gridding and 3D calulator.
- *Gridding* – Create 2D grids (surfaces) or 3D grids (voxel models) from 3D point or segment data, such as a drillhole database.
- Apply *3D Grid Convolution Filtering* to Voxel Models.
- *Voxel Utilities* – Merge and Resample Voxel Models, Convert Null's and Clip DEM grids.
- *3D Grid Calculator* – conduct mathematical operations on Voxel Models.
- *3D Display Assistant* – load and display Voxel Models in Encom Discover PA.

- *Grid Management* – import and export Voxel Models in a variety of formats.

Using the Voxel Toolkit

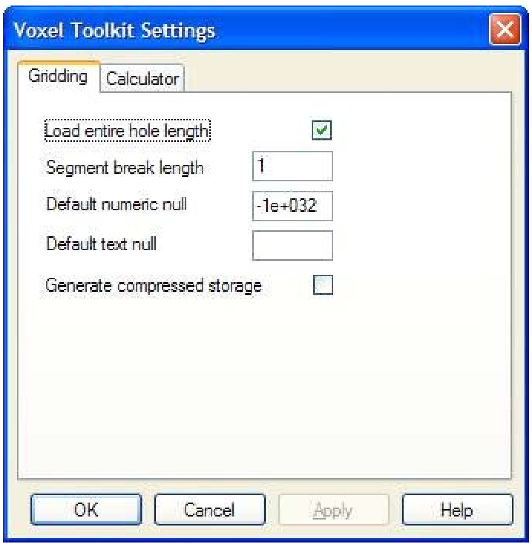
The Voxel Toolkit automatically loads voxel models which are open in Encom Discover PA. The list of grids presented in the various function dialogs will display an icon indicating whether a grid is on-disk, in memory or in memory and unsaved. Grid information can be viewed for any file by selecting the information icon in the list. Grids listed in the Voxel Toolkit are available to all the functions.

Voxel Toolkit Settings

The **Properties** button in the Voxel Toolkit dialog provides advanced settings for the 3D Gridding and 3D Calculator utilities.

- *Gridding Settings*
- *Calculator Settings*

Gridding Settings



Gridding tab options for the Voxel Toolkit Settings dialog.

Load entire hole length

This option modifies the behaviour when drillholes are loaded via Encom Discover PA. If it is turned off, then Encom Discover PA may only supply a part of the hole – namely that part which has some data in it. For example, there may be a 400m long drillhole where the top 200m has no assay information, then there may be some logged assay information for 50m and then for the remaining 150m the drilling should have ceased but didn't, providing no further logged information. So there may only be a portion of data in the hole. If this option is turned off the Encom Discover PA may only supply these stations where assay information was logged.

If the option is enabled then Encom Discover PA will supply the entire hole from top to bottom. This will provide an entire series of nulls. However, if the data conditioning tool (see Conditioning Data Prior to Gridding) is used then the nulls can be converted to a background value (eg zero). Therefore, by loading all stations in the drillhole and converting null values to background some constraints on your gridding procedure can be applied. This prevents assay envelopes, from ballooning out into wasted rock.

Note It is advisable that if you are gridding assay data to enable this option (and to convert NULL to background).

Segment break length

When you import drillhole data each data segment can be broken up into multiple shorter segments of the length entered. This means that data is spread along the hole rather than concentrated at the top of the segment.

Default numeric null

The value entered can be set as the default null value for data conditioning. i.e. it is the value that invalid data is converted to by the data conditioning tool. There is little requirement for this value to be modified.

Default text null

The value entered can be set as the default null value for data conditioning. i.e. it is the value that invalid data is converted to by the data conditioning tool. There is little requirement for this value to be modified.

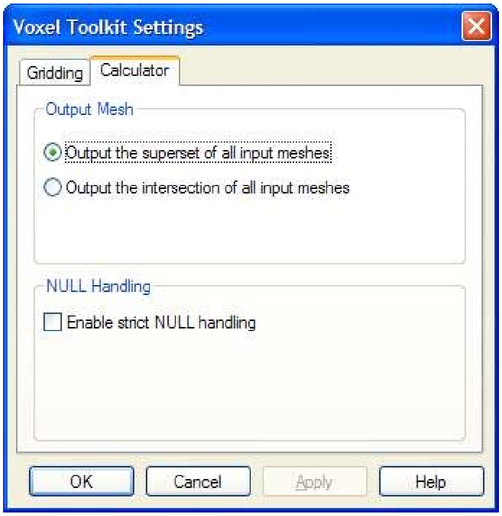
Generate compressed storage

When enabled this option allows the grid to remain compressed in memory and when exported to an Encom model (.e3d) file. If you export to UBC model format however it will be decompressed as it is exported.

This option is useful if you intend to create a very large grid that is very sparsely populated. If the grid is densely populated, then there is no advantage to having it compressed in memory and may actually use more memory. Also, gridding performance is reduced when the model is compressed.

By default this ought to be OFF and only activated by advanced users.

Calculator Settings



Calculator tab options for the Voxel Toolkit Settings dialog.

Output the superset of all input meshes

If the 3D grids (voxel models) loaded into the calculator don't perfectly overlap, this option allows the output of a grid (voxel model) that encompasses all input grids.

Output the intersection of all input meshes

The alternative option for output is to output a 3D grid (voxel model) that only covers the intersection area.

For example, if you add Grid A and Grid B together and they do not overlap, then using the superset method will achieve a result of both grids being merged into a new grid. If the intersection option is used then no result will be achieved.

Note The 3D Grid Calculator uses the intersection method, but 3D Gridding uses the super-set method by default.

Enable strict null handling

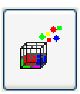
When strict null handling is enabled and if any parameter of the equation is NULL, then the result will be NULL.

When this option is disabled, in some equations you can have a null value as a parameter and the result may be non-null.

For example C = A+B. If A is null, it will return B. This is useful when merging grids etc.

Note The Grid Calculator utility does not have this flexibility – it uses strict null handling always.

Gridding



The gridding operations in the Voxel Toolkit can generate either 2D or 3D grids. 3D gridding interpolates 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 sample. The value of the sampled data is assumed to be constant between the it's end points.

To simplify the process of creating 2D or 3D grids the gridding operations are presented with a wizard interface. The gridding methods available with this tool are discussed under *3D Gridding Methods*.

Creating a voxel model

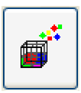
The following steps guides the user through process required to apply 3D gridding to a drillhole dataset. This will apply an interpolation (in this case Inverse Distance Weighting) to a downhole data field (e.g. an assay field), the creating a voxel model. This can then be used to visualise high assay value correlation between holes, using the Voxel Model Properties.

1. Ensure that the drillhole dataset is open in Encom Discover PA (use one of the **File>Open>Downhole** menu options).

2. Open the Voxel Toolkit from the **Grid** or **Voxel** menu.

3. Press the **Gridding** button, and in the following dialog, 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).



5. On the **Conditioning** tab use the **Data Conditioning** button (see *Appendix G: Field Data Conditioning*) 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.

   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.

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.

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.

---

7. 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 one-third the resolution to determine areas where to perform subsequent half and full resolution passes will be calculated.

8. 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.

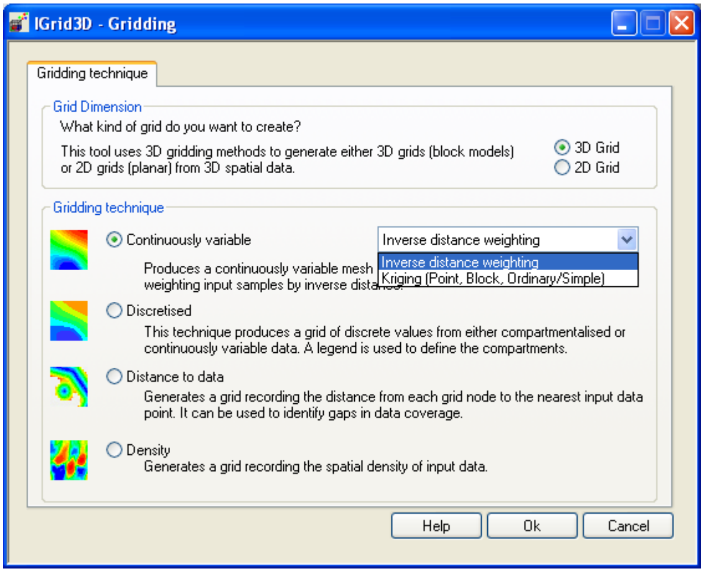
9. The **Sample Selection** criteria can be relaxed by enabling the **Gridding Rule** option.

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 *3D Display Assistant* dialog.

3D Gridding Methods



Grid Toolkit selection of two or three dimensional operation.

The Voxel Toolkit can generate either two or three dimensional grids. Regardless of the dimensionality of the grid, the gridding methodology is identical and three dimensional. 2D grids are generated as horizontal planes at a specified RL level and are exported as ERMapper compatible raster grid files. You can create four different kinds of grids.

- *Continuously Variable*. This method is useful for gridding data like drillhole geochemical assay values. It smoothly interpolates values at the grid nodes based on the surrounding data.

- *Discretised Gridding*. This method allows you to grid discrete or classified data. An example is the geological rock codes recorded for samples down a drillhole. These strings are classified into a legend. The legend can either be automatically generated or the user can specify a legend file used to classify the data. Both string and numeric data can be classified. Once the data is classified it is gridded using similar techniques except that the value at each grid node is assigned to a particular classification.

  The method produces a legend file as one of the products of the gridding process. This legend file can define surface and line colour, style and pattern for each classification. It can be used as a rendering guide when the grid is displayed.

- *Distance Gridding*. 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.

- *Density Gridding*. This method produces an estimation of the 3D point density at the centre of each grid cell.

Continuously Variable

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 Selection

2. Data Conditioning

3. Grid size specification

4. Search parameters

5. Method

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 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 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.

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.

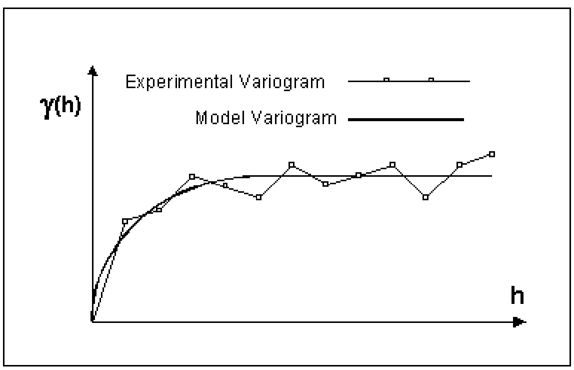
The first step in kriging is to construct a variogram (or semivariogram) from the input data which describes the spatial correlation between the sample points. A variogram generally consists of two parts:

- The experimental (or sample) variogram
- 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 Statistics Explorer Variogram dialog, which is accessed from the Kriging Method dialog.

The gridding wizard dialogs utilised by Kriging are virtually identical to IDW:

1. Input Data Selection

2. Data Conditioning

3. Grid size specification

4. Search parameters

5. Method

The main difference is the final **Method** tab, which provides Kriging-specific options.

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.

Variogram Creation

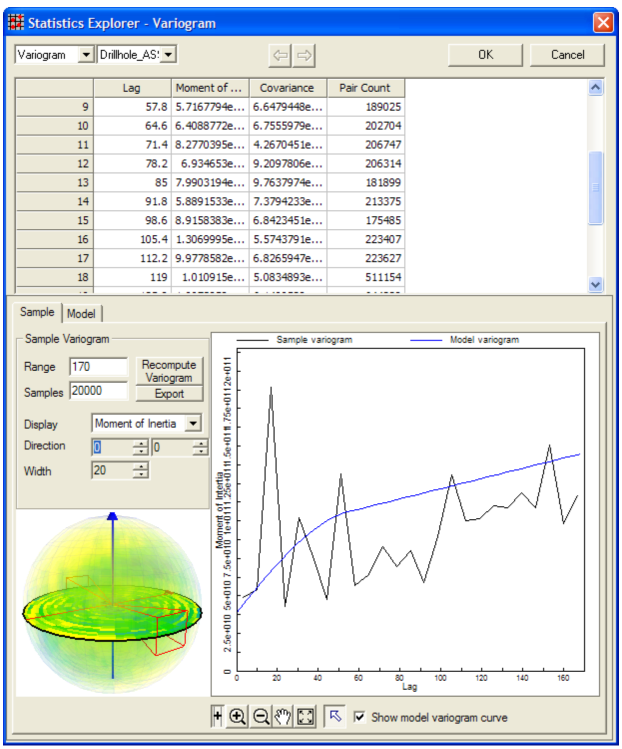
Selecting the **Variogram** button from the Kriging method tab opens the Statistics Explorer Variogram dialog. This utilises the assigned data field in tandem with the spatial coordinates to compute a Sample variogram. A variogram shows the degree of correlation between data in a spatial dataset in different directions and at different distances.

Sample Variogram

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.

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.



Display of the calculated 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.

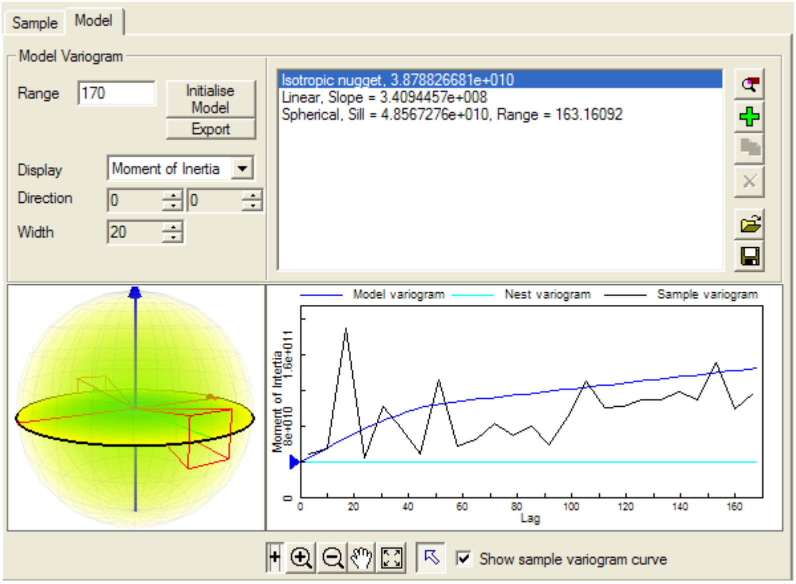
Model Variogram

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 nugget to fit 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.

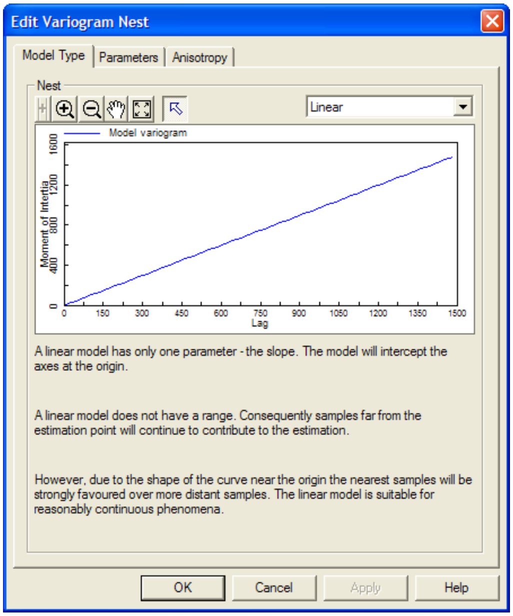
The model variogram is displayed as a grid and also as a directional extraction – just like the sample variogram. The sample variogram curve can be plotted for comparison. In the upper right corner of the Model dialog the model variogram nests are displayed. Each nest corresponds to an individual component of the model. Individual nest 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 model at the top of the dialog and then adjust the parameters by moving the blue edit (triangle) tags on the variogram graph.



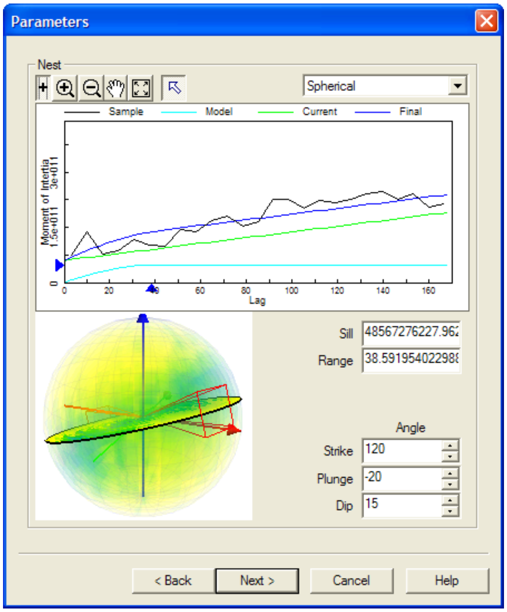
The Model dialog of the Statistics Explorer showing three individual nests of a model variogram. The top portion of the dialog lists the type and parameters of each nest and the corresponding plot at the bottom shows the selected nest graphically.

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

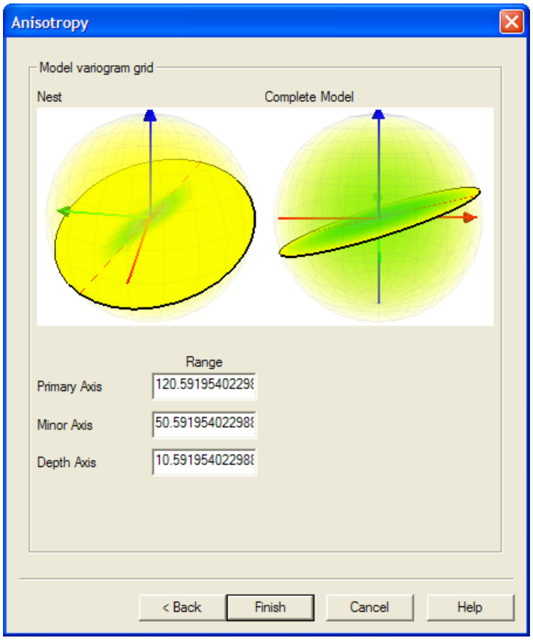
- The **Model Type** page allows you to select an appropriate theoretical model to use as the basis of the nest. A graph is displayed on the Model Type page to show the selected function. The coordinates used in the example graph are theoretical and are not representative of the actual model coordinates.



Controlling the parameters of the variogram

- The **Parameters** page 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 page and cannot be modified here.

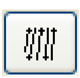


Editing the variogram nest

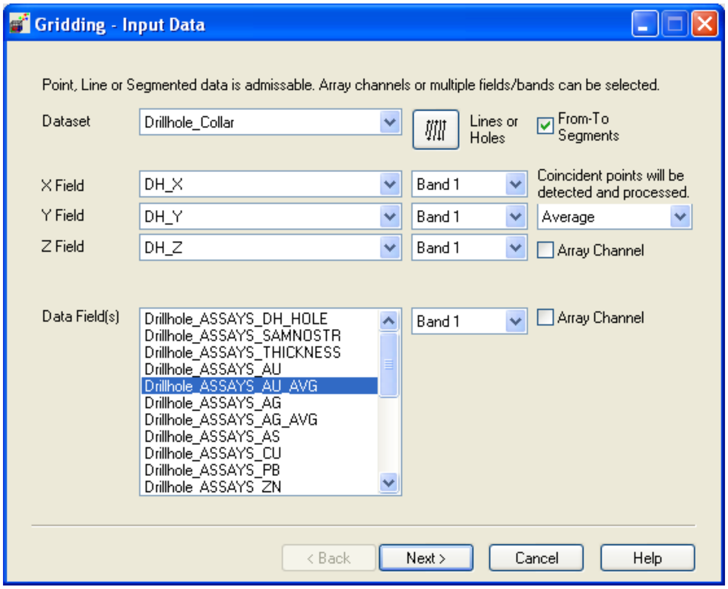
- The **Anisotropy** page controls the range of the model along three axes – the Primary, Minor and Depth axes. A preview of the currently selected nest is shown on the left, whilst the final model preview is shown on the right of the dialog.

  The default Primary Axis range is equal to the range established on the Parameters page. 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.

Gridding Data Specification



Source data can be obtained from any database currently loaded in Encom Discover PA. Both drillhole and survey datasets can be used. A memory efficient direct ASCII file import is also supported where the ASCII file is in a delimited column format. 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. In other words, it has a defined X, Y and Z coordinate and is considered to be a point sample taken at that location. Alternatively, you can specify that the input data are “from-to segment” samples. In this case, each sample is defined by a beginning and end point location. The value of the data over the segment is assumed to be constant. An example of segment data would be drillhole geochemical assays from depth 162.5 to 163 metres. 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.

You are required to specify the X and Y input fields and select a single band from any available for these fields.

There are three ways to specify the Z (or RL) band(s) and data band(s) and each results in a different grid. If a multi-banded grid is generated then multiple independent properties are stored at each grid cell. Note that not all export formats support multi-banded grids.

- **Single Z Band, One or more Data fields** - In this case you can generate a grid band for each data field selected. Select multiple data fields by holding down the CTRL or SHIFT keys whilst selecting fields in the data field list

- **Single Z band, One or more bands from an array Data field**. In this case you can generate a grid band for each band selected from the data array field. Select multiple data bands by holding down the CTRL or SHIFT keys whilst selecting bands in the data field band list.

- **Multiple Z bands and Data bands from Z and Data array fields**. In this situation you can create a single banded grid. The software matches each Z band to a Data band and generates a unique point sample for every station. Note that you should choose an equal number of Z and Data bands. One application of this is the gridding of EM CDI data which is often stored in multi-banded Depth and Conductivity arrays.

Coincident points are detected and processed according to the users specifications defined from a pull-down list (average, minimum etc). If you specify that segment data is being used, the software expects each point to be the first point of the next segment as well as the last point of the previous segment. In this case you are advised to choose an appropriate coincident point handling method carefully – the first point of each segment may be coincident with the last point of the previous segment. If this is the case, you would not want to average or otherwise modify the data at the coincident point – you would elect to leave it unchanged. Otherwise the value of the assay along the entire segment would be modified.

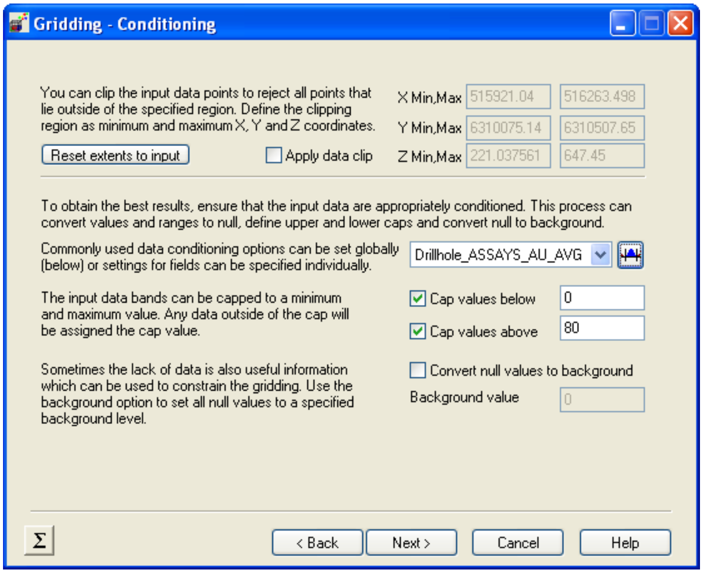
Conditioning Data Prior to Gridding

Source data can be conditioned prior to gridding. This process can be used to eliminate poorly defined data from the input dataset. The main property page presents settings that are common to all the source data bands (or fields). Individual and more detailed control over each band can be obtained via the **Advanced Settings** button.

- *Grid Conditioning*

- *Field Data Conditioning*

Grid Conditioning



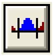
Conditioning of data to be gridded using the Voxel Toolkit

This dialog also provides 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 mineralisation 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.

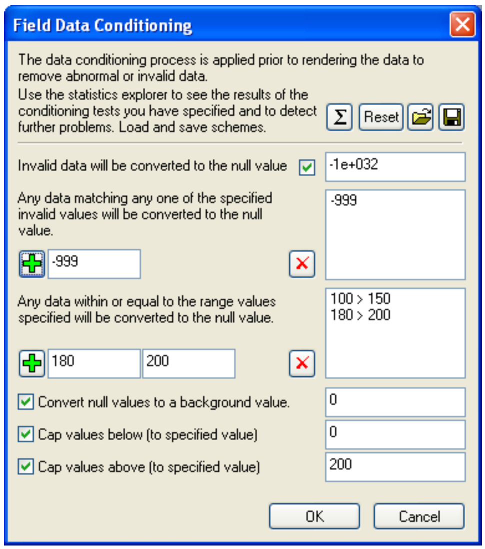
Setting a **Cap values below** or **above** value will cap source data outside the set limit to the limiting value. For example, with Cap values above set to 500, a gold assay on 725ppm will be handled during 3D gridding as a 500ppm value. It is also possible to **Convert null values to a background** 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.

Field Data Conditioning

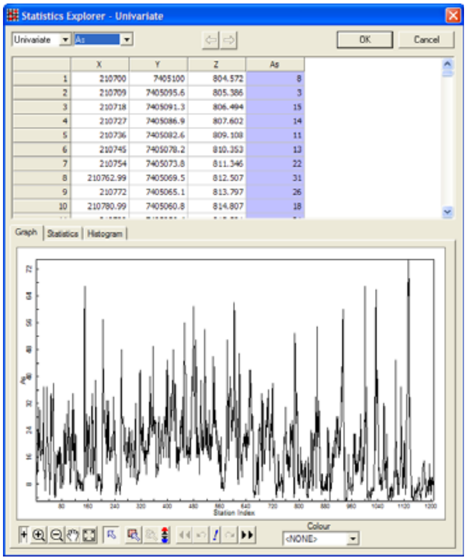


The Field Data Conditioning dialog can be accessed from a number of Encom Discover PA tools using the **Advanced Settings** button (shown left). It allows the user to remove abnormal or invalid data, such as negative values representing missing samples and spurious outliers, as well as providing Null 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.



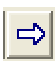
The Field Data Conditioning dialog

The **Statistics** button at the top of the dialog opens the Statistics Explorer. 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 user to check the effect of any conditioning applied to the data range.

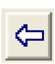


The Graph view of the Statistics Explorer

Within the Graph and Histogram views of the Statistics Explorer, data subsets can be selected using the **Rectangle Selection** tool.

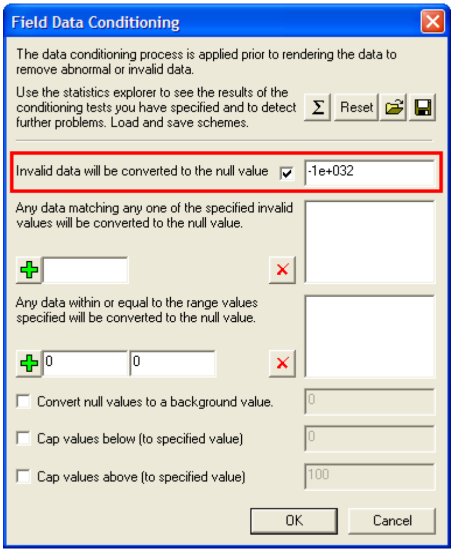


And then viewed using the **View Current Selection** button.



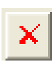
Return to the previous selection (or entire dataset) using the **View Previous Selection** button.

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 datasets default null value: generally -1.0×1032 or -99999. Disabling this option (via the tick box) will disable null handling; invalid data (either user defined or pre-existing null input data) will not be converted to Null values in the output grid, but will be populated and handled as valid data values. A global **Default numeric null** can be set in via the **Properties** button in the main Voxel Toolkit dialog; this value will be applied not only to all data fields, but also the X, Y and Z fields

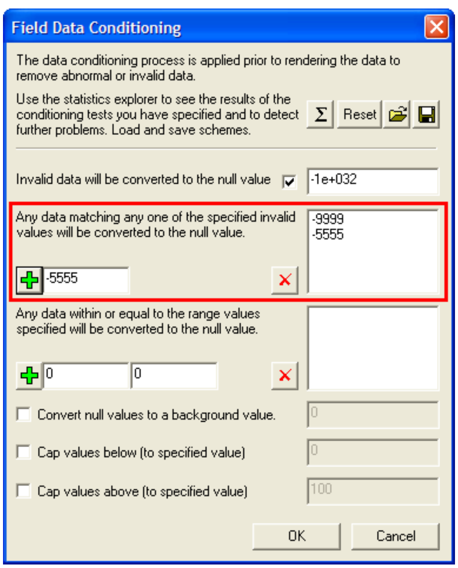


The auto-populating Null value assignment option based on the dataset type

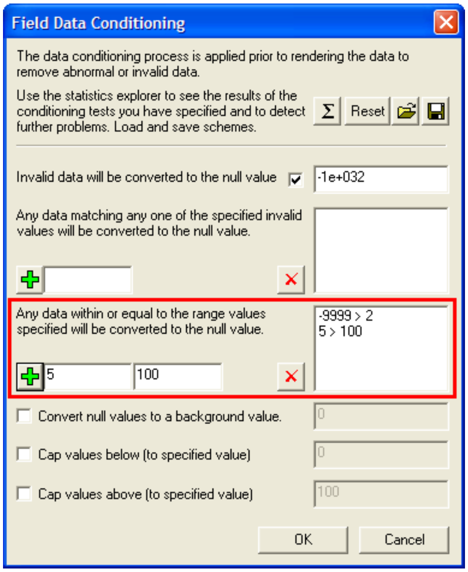
The following options allow specification of a list of user-defined invalid data values and/or ranges specific to the data field. Enter the invalid value or range at the left and then press the **Add** button to add it to the list at the right.



List entries can be removed using the **Delete** button. Some examples of use are presented below:



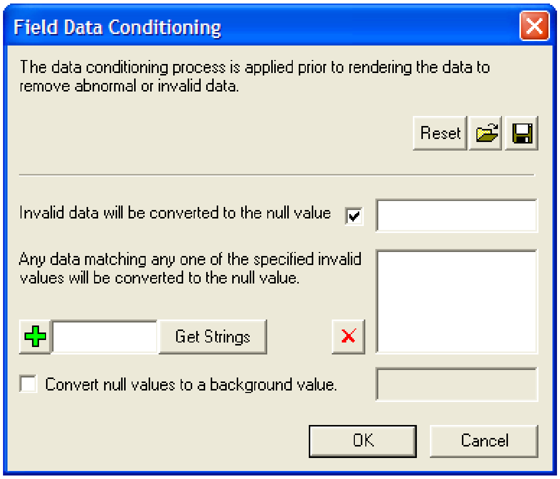
Specifying individual invalid data values. This is appropriate for removing negative values representing BDL (Below Detection Limit), SNR (Sample Not Received), etc.



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.

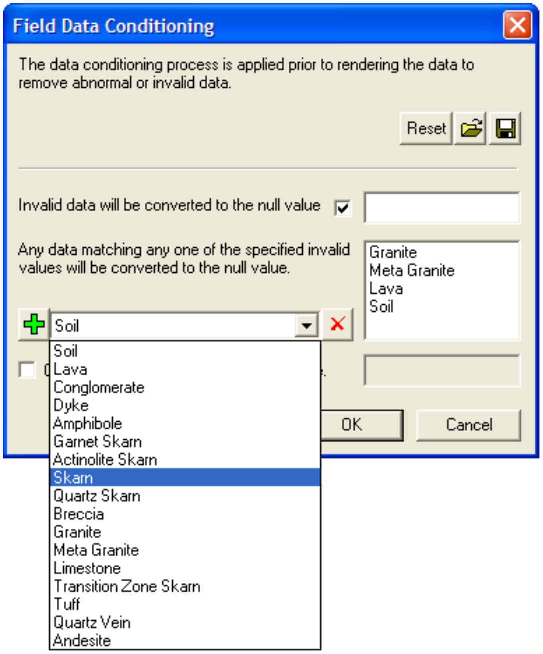
Setting a **Cap values below** or **above** value will cap source data outside the set limit to the limiting value. For example, with **Cap values above set** to 500, a gold assay on 725ppm will be handled during 3D gridding as a 500ppm value. It is also possible to **Convert null values to a background** 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.

If the data field is a **text field** (e.g. lithology codes for use in drillhole colour modulation, or Discretised 3D gridding), the Field Data Conditioning dialog display will be altered accordingly.



The Field Data Conditioning dialog when the source data field is a text field

This allows individual text strings to be specified. The **Get Strings** button will populate a pull-down list with all unique attributes in the text field, allowing a list of invalid text strings to be easily created. A global **Default text null** can also be set in via the Properties button in the main Voxel Toolkit dialog; this null string will be applied to all string data fields.



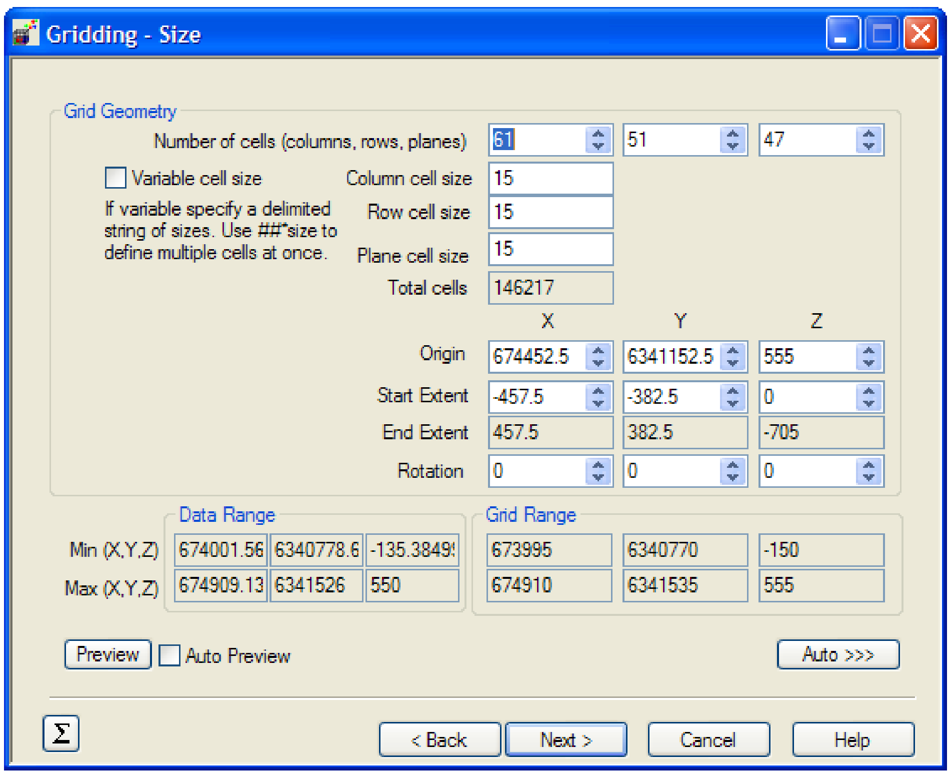
Creating a list of invalid lithological codes (e.g. so that only the valid codes are displayed when colour modulating drillholes)

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.

Specifying the Grid Size

A 3D grid consists of NX*NY*NZ cells in the X, Y and Z dimensions. The gridding process interpolates a value for the centre of each cell from the source data. Thereafter, most visualisation techniques consider the value of each cell to be constant across the cell. Some visualisation techniques require smooth interpolation between cells and to do this they consider the value to be defined at the centre of the cell and to smoothly vary to the values at adjacent cells.

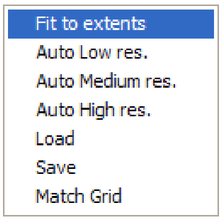
The 3D grid mesh is defined by specifying the bottom left front corner coordinate (minimum) and the sizes and number of cells in the XYZ dimensions. The cell size can be fixed or variable. For example, a larger cell size can be specified on the edges of the mesh and a finer cell size in the centre of the mesh.



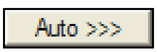
Specify the 3D grid mesh size in each of the three dimensions

You can specify a variable cell size 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.

You can specify a rotation for the grid by adjusting the Rotation about the X, Y or Z axis. Note that this rotation is relative to the origin X, Y, Z. By default the origin is set to the centre point of the Data and Grid range, with cells Extending to either side of this. However in some case, such as when working in a local mine grid, the origin may need to be adjusted to a corner of the grid, or the a local mine grid origin.



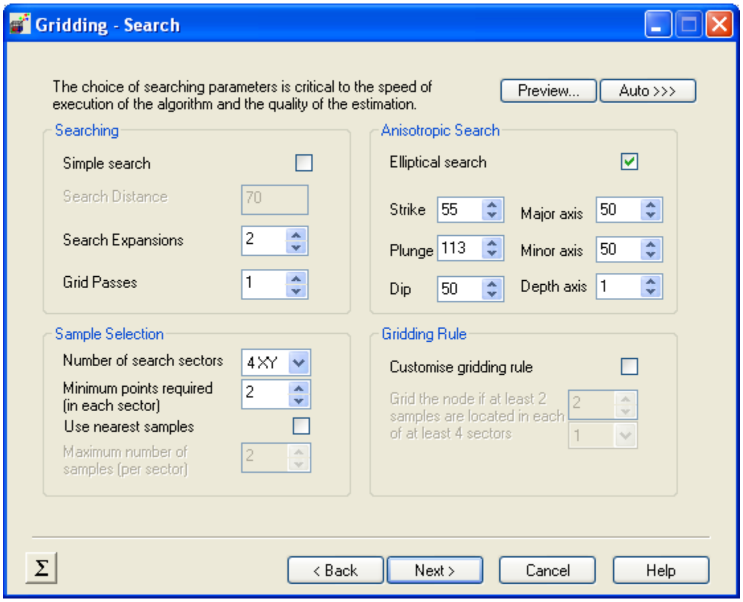
The definition of 2D grids is similar except that variable cell sizes are not supported and only a single plane can be generated.



The software makes an initial grid size suggestion based on the spatial resolution of the source data. However, you almost always want to set the grid size yourself. To make specification of the grid size easier the **Auto** button provides a menu of automatic options which set the grid size. **Fit to extents** sets the origin and cell count based on the current cell size. The **Auto…** options set the origin, cell size and cell count appropriately to cover the source data extents at a variety of useful resolutions. The **Load** and **Save** options retrieve or save settings to a simple ASCII text file with a .E3X file extension. The **Match Grid** option allows you to browse to an existing grid on disk and retrieve the grid size from that.

The wireframe/mesh of the Grid can be previewed at any time by clicking the **Preview** button. This allows alignment of the grid to a feature of trend in a 3D map window.

Defining the Grid Control Parameters



Specifying the Inverse Distance Weighting parameters using a Simple Search

The Voxel Toolkit currently only offers one interpolation method - inverse distance - which is used for both continuously variable and discrete input data.

The inverse distance 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 you are advised to experiment with a low resolution grid first and then generate the final grid at the required resolution once you have determined appropriate gridding parameters.

Search Radius

The first technique designed to improve performance is to introduce a search radius. The algorithm only searches for contributing data points within the specified radius of the grid cell. Data points outside of this distance do not contribute to the solution at that cell.

The first problem this causes is a failure to grid the node as an insufficient number of contributing data points may be found within the search radius to satisfy the algorithm. The 3D gridding algorithm accommodated this by using two additional parameters.

- Firstly, a **Search Expansion** (i.e. increment) ranging from 1 – 5 can be specified. If an insufficient number of data points are found within the search radius, that search radius is increased (by the initial radius) and the search is repeated. This occurs up to “n” times – as specified by the increment. The final search radius is the number of increments multiplied by the initial search radius.

- Secondly, the number of **Grid Passes** can be set to greater than 1. If a number greater than one is applied the grid is generated at a lower resolution first. The nodes gridded in the first pass are added to the input data for the second pass. From 1 to 5 grid passes can be requested. The initial search distance is equal to the number of passes multiplied by the initial search distance. This feature is useful if the input data points are unevenly distributed as it fills in the gaps in coverage more efficiently.

If you use restricted search radii to improve gridding performance you invite the possibility that the contributing data points chosen for each interpolation are not evenly distributed about the grid cell. This can lead to poor solutions. The software therefore uses ellipsoidal searches and sector searches to eliminate this problem.

An **Elliptical Search** radius can therefore be defined by set the major (X), minor (Y) and depth (Z) axis radii. The major axis can be rotated from its default position (aligned east-west).

When gridding data that is sampled sparsely in two dimensions and densely in another, it is advisable to use an ellipsoidal search radius. Optionally, the weighting scheme can take account of the ellipsoidal search radius. This normalises the axes and ensures that data are weighted evenly in all dimensions.

To activate the parameters for this option ensure that the **Elliptical Search** check box is active. The following parameters can be applied:

- Strike of the major horizontal axis in degrees east or north.

- Plunge of the major horizontal axis in degrees positive downwards from the horizontal plane.
- Dip of the minor horizontal axis in degrees positive downwards from the horizontal plane.
- Major horizontal axis search distance
- Minor horizontal axis search distance
- Minor vertical axis search distance.

Glossary:

- **Strike** is the azimuth of the Major Horizontal Axis (Y), degrees clockwise of north. It is a rotation about the Z axis.
- **Plunge** is the plunge of the Major Horizontal Axis (Y), degrees positive downwards from the horizontal, clockwise looking west. It is a rotation about the X axis.
- **Dip** is the inclination of the Minor Horizontal Axis (X), degrees positive downwards from the horizontal, clockwise looking north. This is the rotation of the X axis about the Y axis. It is a rotation about the Y axis.
- **Major Horizontal Axis** is the radius of the Y axis of the ellipse which is positive north when unrotated.
- **Minor Horizontal Axis** is the radius of the X axis of the ellipse which is positive east when unrotated.
- **Minor Vertical Axis** is the radius of the Z axis of the ellipse which is positive up when unrotated.

In all cases: Minor vertical axis ≤ Minor horizontal axis ≤ Major horizontal axis

This rule should never be broken and the property pages will attempt to stop you from doing so. If (for example) you want the X axis longer than the Y, you need to rotate the major axis to achieve it.

These parameters that are available for an elliptical search allow the user to easily introduce a bias into the estimation. In particular, at a new prospect the best information the geologist might have about the directionality trend of mineralization is that it seems to be conformant with the geology. It is likely that the geologist will know the rough strike of the geology and they may also have some idea of the dip. In general they will be drilling along strike and against the dip. This provides good information for estimation bias. The degree of the bias is determined by the ratio of the axis radii to the vertial axis radii. For example, by setting the vertical axis length to 10m, the minor axis length to 20m and the major axis length to 40m you are saying you expect the continuity of the mineralization to be four times as strong along strike and down plunge as it is perpendicular to the strike and dip of the geology. Within the plane of the geology you expect the continuity to be twice as good down plunge as perpendicular to the plunge.

Only data found within the boundary of this 3D search ellipse will be considered in the estimation. Note that with the editing of these parameters the **Search Expansions** and **Grid Passes** specifications incrementally increase those ellipse axis lengths.

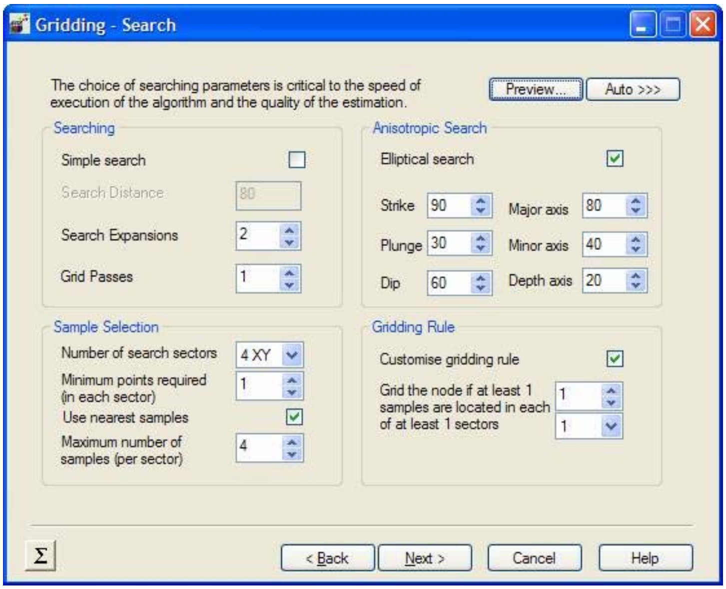
Note that the Sector requirements (in Sample Selection parameters) also refer to this ellipse. In 3D space these sectors will be rotated to match the ellipse axis rotations.

The **Sample Selection** options for 3D gridding improve the final output. Options include:

- Specifying the **Search Sectors** to be used will help ensure that input data is evenly distributed spatially around the grid cell.

- You can specify that the gridding cannot succeed unless a **Minimum number of input points** are found within the search radius within each sector. You can elect to use 1, 2, 4 or 8 sectors by selecting the options: 1, 2X, 2Y, 2Z, 4XY, 4XZ, 4YZ, or 8. For 2 or 4 sectors the axis which separates the sectors is also specified. This option often results in a grid that is not fully defined – that is, many nodes are not gridded as the algorithm requirements cannot be met. You can use this option to improve the solutions, but then override it at the last minute by electing to grid the cell and retain the solution if the algorithm requirements have not been fully met but at least one valid source data point has been found.

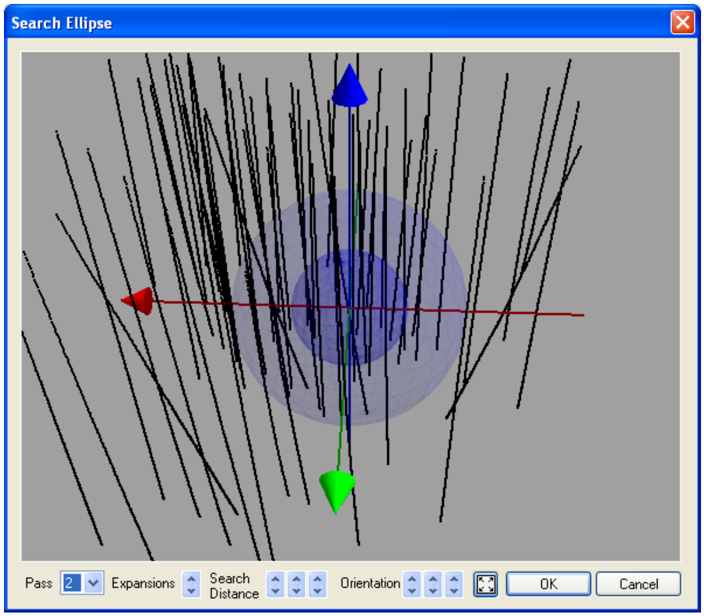
- Within each sector the algorithm may locate many points. You can elect to consider just the closest of those points by entering selecting the **Use nearest samples** check box and then specifying a value (from 1-100) in the **Maximum number of samples (per sector)** box. If you do not restrict the number of input points in this way then many input points may contribute to each solution. This can have unusual effects especially as you begin to move away from input data and the distance to all points becomes approximately equal. Also, the uneven population density of the input data can produce problems. This can be eliminated by applying density corrections which are recommended whenever you allow the algorithm to use all the located points in the search radius as contributing points to the solution.



The Search parameters dialog for 3D Gridding.

Preview Window

The current search spheroid or ellipsoid can be viewed in a simple 3D preview window with the source dataset by pressing the Preview button at the top right of the Search dialog.



The Preview window for the Search parameters of 3D Gridding.

The controls at the base of this dialog link directly back to those in the Search dialog. The user 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.

Search parameters can be saved and reloaded as a grid Parameters file (.e3g) using the Load and Save options under the Auto button.

Weighting of Points

Several weighting schemes can be employed including Linear, Exponential, Power law and Gaussian. The usual choice is Power Law where the contribution from each data point is weighted by the distance to the point raised to the specified power. A higher power ensures that closer points have a higher influence on the solution. The weighting schemes are described below.

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 then 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 (Search Parameters) will be proportionally weighted by their distance from the grid node such that:

`Distance < Range:  Weight =` $Nugget - \frac{Dist}{Range}$

`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: Weight` $= \exp\left(-Dist \times \frac{Power}{Range}\right)$

`Distance>=Range:  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.

$$\texttt{Weight} = \frac{1}{Dist^{Power}}$$

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.

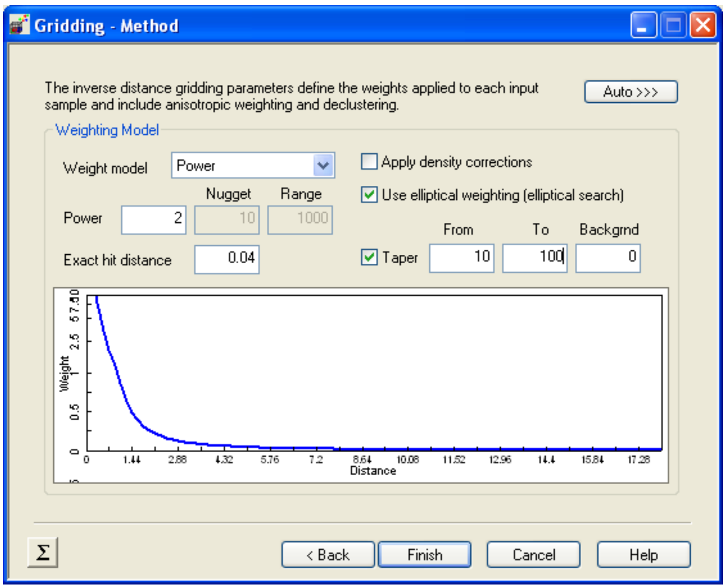
$$\texttt{Weight} = \frac{1}{{}_{2}\left(\frac{Dist}{\left(\frac{Range}{2}\right)^{2}}\right)}$$

The 3D Gridding includes a Simple mode that does not restrict the search radius and allows all input data to contribute to each solution. It may take a long time to run, but the grid generated is often quite smooth and may serve as a guide for further gridding attempts.

For Inverse Distance Weighting and Classified gridding you can activate the check box for **Use elliptical weighting (elliptical search)** when elliptical search is specified in the Seach Parameters dialog. This modifies the weighting assigned to contributing samples by computing the anisotropic distance between the sample and estimation position. When the sample is located in the direction of the major or minor axes this distance will likely be shortened and so the weight assigned to that sample will be increased.

Tapering

As your solution centre moves away from data sources your confidence in the solution may decrease. Statistically, at some point you will exceed the range beyond which there is no correlation between source data. The tapering option enables you to taper the solutions back to background at large distances from source data. The tapering function is linear and applies between the From and To distances. Beyond the To distance all solutions are assigned the background value.



The Method dialog for 3D Gridding which allows the specification of weighting applied to each sample.

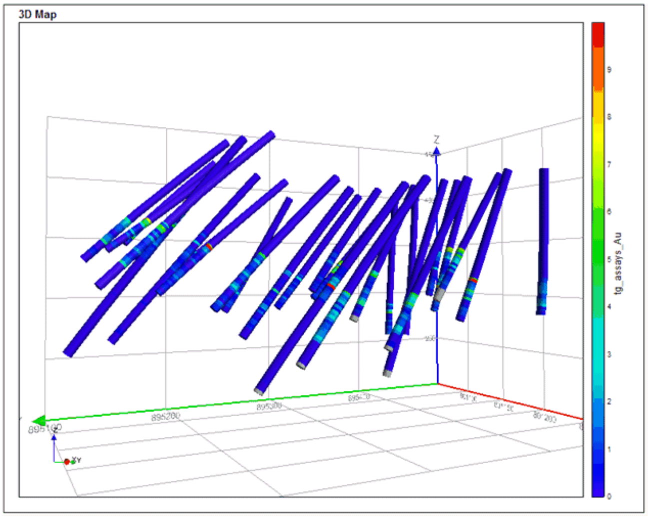
Advanced Properties

Advanced Gridding properties can be accessed via the **Properties** button on the Voxel Toolkit main window.

- **Load entire hole length**: when disabled, Encom Discover PA 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.



Example of drillholes to be gridded using gold assay intervals

Discretised Gridding

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 3D grid. 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 Selection
2. Data Conditioning

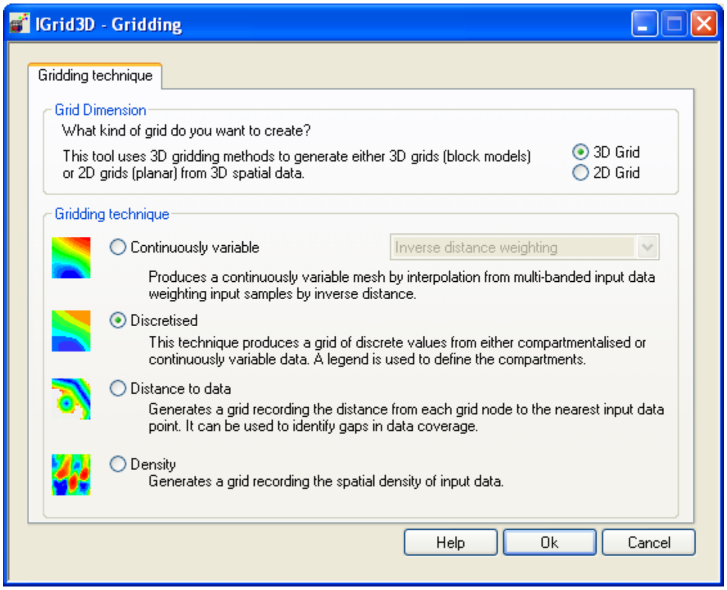
3. Classification

4. Grid size specification

5. Search parameters

6. Method

After selecting the **Discretised** method from the Gridding Technique dialog of the Voxel Toolkit choose the field in the loaded database that is to be gridded from the Input Data dialog. Then click **Next** to proceed to apply Conditioning to the data i.e. converting null values in the data to the background value.

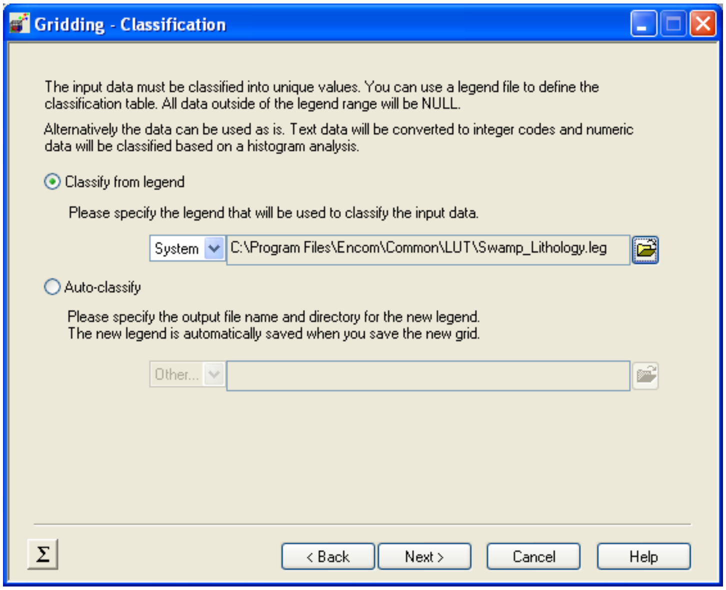


Choose the Discretised gridding technique.

Then click the **Next** button to proceed to the **Classification** dialog which 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. These are normally stored in the C:\Program Files\Encom\Common\LUT directory. Any data which does not match a record contained in the legend file will be treated as a NULL value in the output.

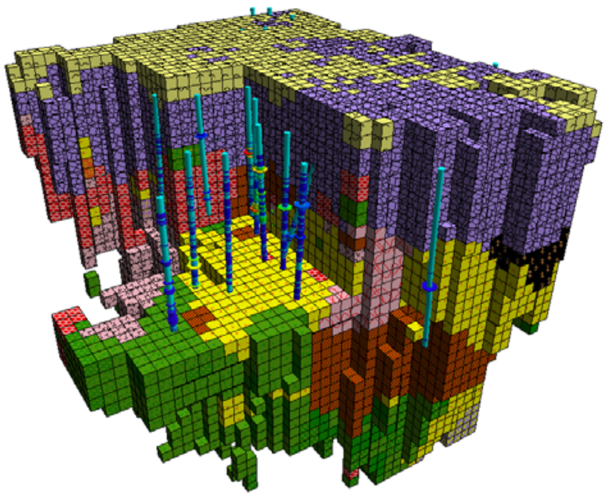
- **Auto-classify**: creates a new legend using the entire input dataset. All input data will be classified unless it has been set as Null in the Field Data Conditioning dialog.



The Classification dialog for Discretised 3D gridding

Proceed to the **Size** parameters dialog to specify the voxel (i.e. cell) sizes and resolution of the gridded model.

The output of this procedure is 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.



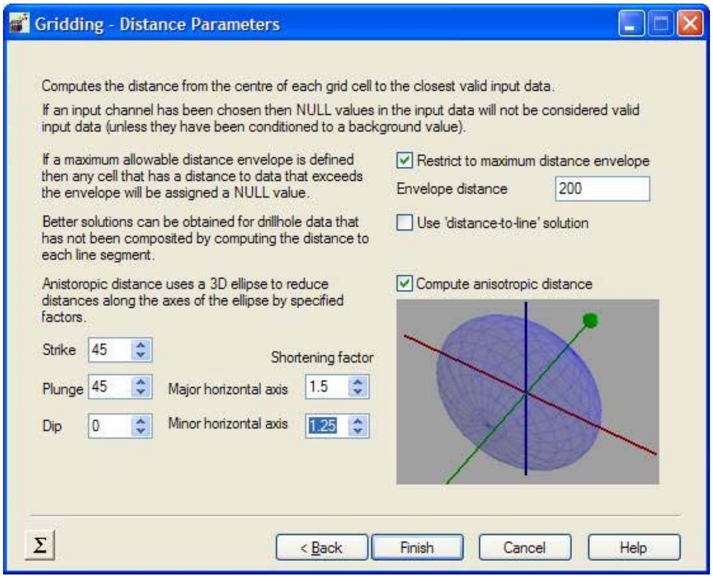
An example of a voxel model showing lithology patterns produced from the discretised grdding technique.

Distance Gridding

Distance gridding utilises the following wizard dialogs:

1. Input Data Selection
2. Data Conditioning
3. Grid size specification

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.



The Distance Parameters dialog

If you have checked **From-To segments** on the Input Data dialog, 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 sufficiently then it is usually sufficient to use just the point data. Note that the performance of the algorithm can be improved considerably by enabling the check box to **Restrict to maximum distance envelope** and then specifying an **Envelope Distance**. This option will restrict the search distance to the maximum envelope distance specified.

The old method computed the isotropic distance from the centre of each grid cell to the closest valid sample point.

There is a check box option available called **Use distance-to-line solution** which when enabled will compute the distance to the input line segments. This is handy for drillhole data that has not been composited but it can degrade the algorithm performance considerably.

The **Compute anisotropic distance** check box option allows the user to define spatial compression factors in two directions in 3D space. The components of a vector in 3D space will be shortened in these directions. For example, in geological terms this means that we can define the distance along the strike and plunge of a geological unit to be shorter than the distance perpendicular to the plane of the geology.

Note The distance envelope remains an isotropic distance regardless of whether anisotropic distance is being computed.

Note changes to the **Distance to data** method (Distance Parameters' property page) shown below.

To compute the anisotropic distance you must specify:

- The **Strike** of the major horizontal axis in degrees east or north
- The **Plunge** of the major horizontal axis in degrees positive downwards from the horizontal plane.
- The **Dip** of the minor horizontal axis in degrees positive downwards from the horizontal plane.
- The **Shortening factor** on the major horizontal axis. The factor on the major horizontal axis must be greater than or equal to the minor horizontal axis factor.
- The **Shortening factor** on the minor horizontal axis. The factor on the minor horizontal axis must be greater than or equal to the minor vertical axis factor.
- The **Shortening factor** on the minor vertical axis is fixed at 1.0 (ie no distortion).

There is a 3D preview graph – click to activate. Use the left mouse button to spin the graph and the right mouse button to zoom in and out.

Density Gridding

The Density gridding method produces a grid which records a measure of the point density at each grid cell.

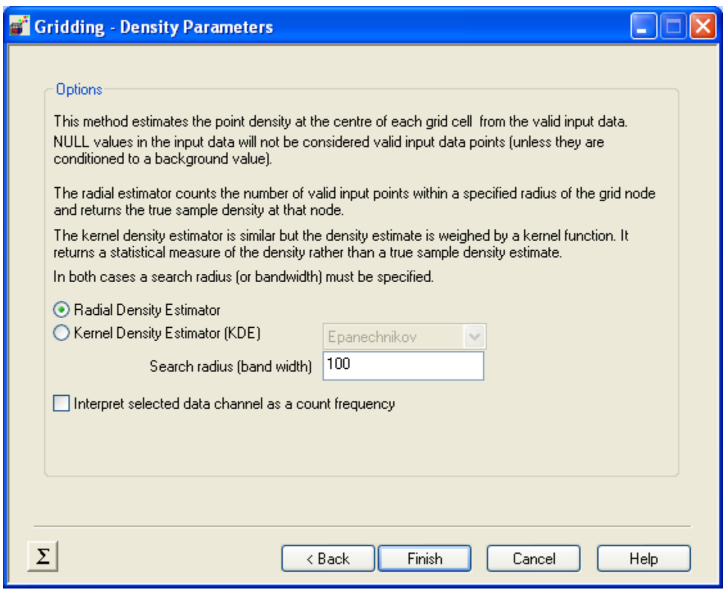
As with Distance gridding, it utilises the following gridding wizard dialogs:

1. Input Data Selection

2. Data Conditioning

3. Grid size specification

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 Density Parameters dialog

- The **Radial density estimator** 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	$\frac{1}{2} I(\lvert u \rvert \le 1)$
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

$$\frac{\pi}{4}\cos(\frac{\pi}{2})I(|u| \leq 1)$$

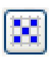
If you have taken multiple samples at each input data location and this information is recorded in the input source data you can use this information to bias the density estimation.

Often a density grid will contain a large number of cells with zero density. These can be converted to NULL values using 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 Grid Convolution Filtering

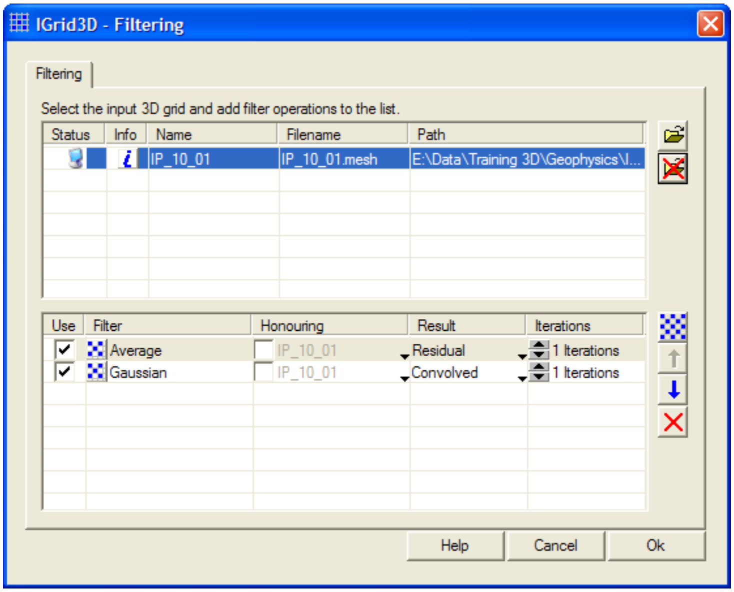
From the Voxel Toolkit wizard, the **Convolution filtering** option enables a filtering tool that allows you to apply one or more 3D filters to a 3D grid.



The tool enables you to load/import and unload grids from a grid manager (click the **Add filter** button) and to select a grid to operate. 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.

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 tool operating over a selected 3D grid

Voxel Utilities

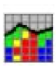
Voxel>Voxel Utilities

Encom Discover PA Professional provides a range of powerful Voxel Utilities, from simple voxel cell size resampling through to advanced analytics and calculations between voxels and polyhedron volumes.

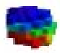
This functionality can aid in determining the average voxel cell value (e.g. gold grade) within a user-modelled polyhedron volume, allowing simple resource calculations; clip a voxel to a mineralisation volume's extents or interpolated unconformity surface; or convert lithological vector volumes into a voxel model of densities.

These utilities are accessible via the **Voxel>Voxel Utilities** menu option in the 3D application:

- *Assign Values to Features* – Attribute multiple feature objects with any coincident voxel cell values.

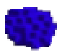


- *Clip to Surface* – Clip a voxel model with a surface, producing two output voxel models.



- *Clip to Vector Volume* – Clip a voxel model to one or more vector volumes, producing two output voxel models.

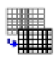
- *Convert Nulls* – This utility converts the NULL value in the input voxel model to a specified value.



- *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 model.

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 Encom Discover PA).
- A feature database open in Encom Discover PA 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 Encom Discover PA.
2. From the **Voxel>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 Encom Discover PA via the browse button at the top right of this dialog.

4. In the **Assign Values to Features** dialog, select the **Feature database** andd **Feature set** 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.

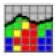
     Point feature datasets can be attributed from either numeric or lithological voxel models.

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 Encom Discover PA).
- A gridded surface or a feature database (preferably open in Encom Discover PA) 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 Encom Discover PA.
2. Open the feature database in Encom Discover PA.
3. From the **Voxel>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 Encom Discover PA 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 Encom Discover PA

2. From the **Voxel>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 Encom Discover PA 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 Encom Discover PA) and
- A feature database volume or volumes open in Encom Discover PA 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 Encom Discover PA.
3. From the **Voxel>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 Encom Discover PA 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.

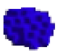
6. Select the appropriate **Test rule** and **Acceptance test** (if appropriate); see *Volume Selection Method* for more information.

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 *Grid 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.

Requirements:

- A feature database with one or more polyhedron volumes open in Encom Discover PA, or
- One or more third-party 3D vector files (eg AutoCAD DXF) containing volumes to be selected by the user, or
- One or more Encom Modelvision TKM files.

To convert vector volumes to voxels:

1. If using a feature volume, open the feature database into Encom Discover PA
2. From the **Voxel>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.
- **Use a ModelVision TKM file** and then use the Browse button to select a TKM file

4. Select the value/s to assign to the resulting output voxel:

   - Select to **Stamp selected fields** to the voxel cells. This allows selection of numeric fields to populate the voxel model from. Overlapping polyhedron volumes will be determined by the **Multiple cell overlaps rule**.
   - **Stamp single value** will apply the same value to all voxel cells within the polyhedron volume.

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**.

Create Mesh

Create a blank voxel mesh, either from an existing voxel model or via user-specified parameters.

Merge

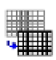
Merge two or more voxel models into a single multi-banded voxel model. All input bands are preserved and all cell boundaries are preserved without interpolation.

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 **Voxel>Voxel Utilities** menu
- Select the voxel models to merge and press **Finish**.

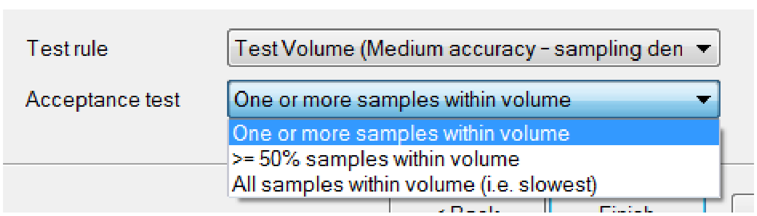
Resample



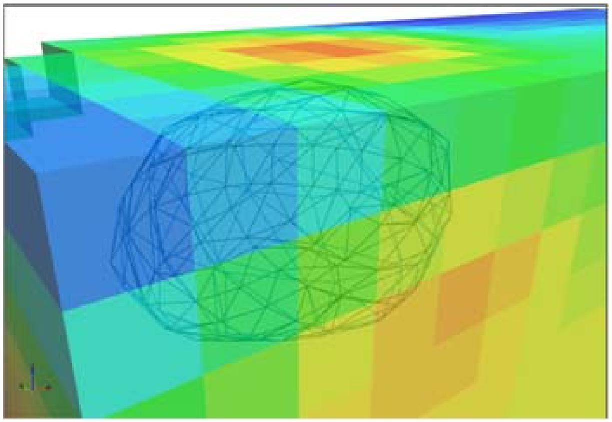
Resample and/or resize an existing voxel model into a new voxel model; a number of interpolation options are available.

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.



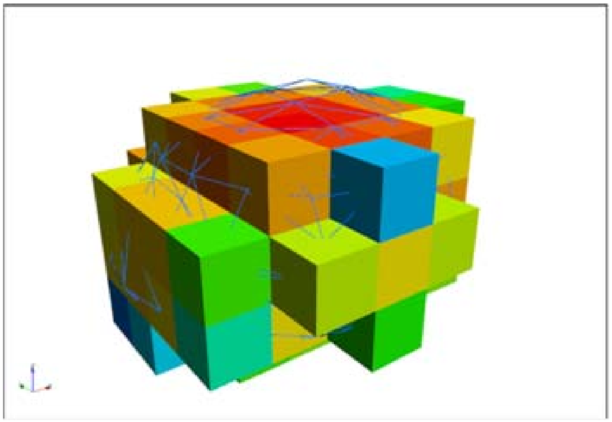
In general, the more accurate the method chosen, the more iterations/tests that need to be performed, and therefore the slower the processing time.



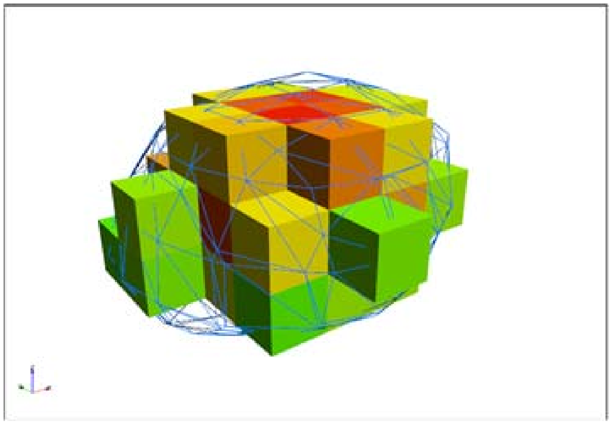
Test Rule

Four levels of density testing are available for determination of whether a voxel cell is within a polyhedron volume:

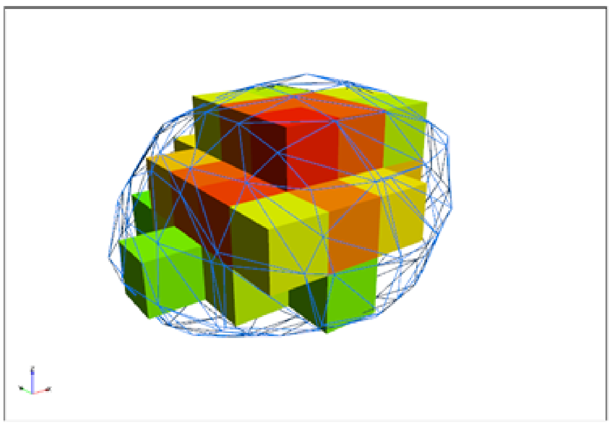
- **Test centre**: Only the voxel cell centre position is evaluated. Fastest but least accurate method.



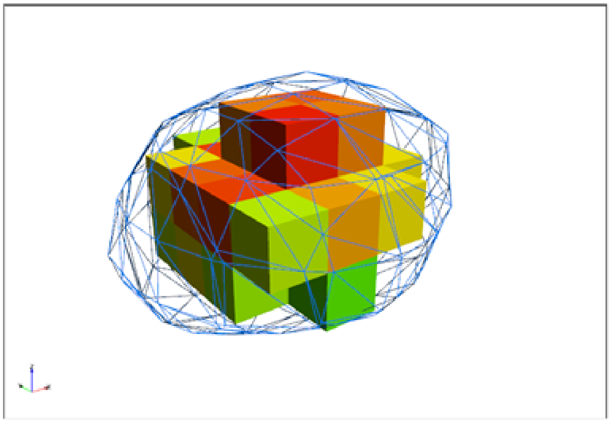
- **Test volume – low accuracy**: Each voxel cell is subdivided into 8 regions (2X x 2Y x 2Z) and the centre position of each of these is evaluated



- **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.



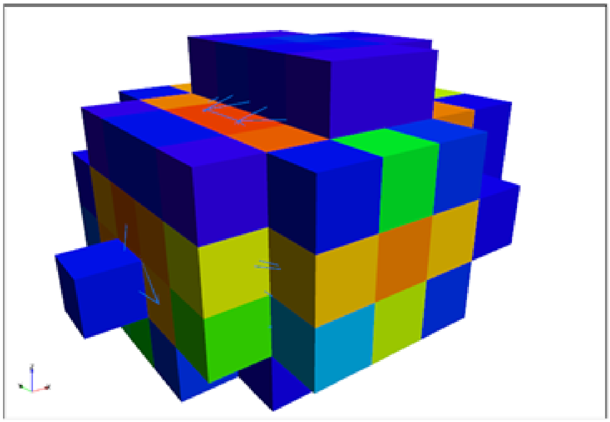
- **Test volume – High Accuracy**: Each voxel cell is subdivided into 64 regions (4X x 4Y x 4Z) and the centre position of each of these is evaluated (i.e. slowest but most accurate method)



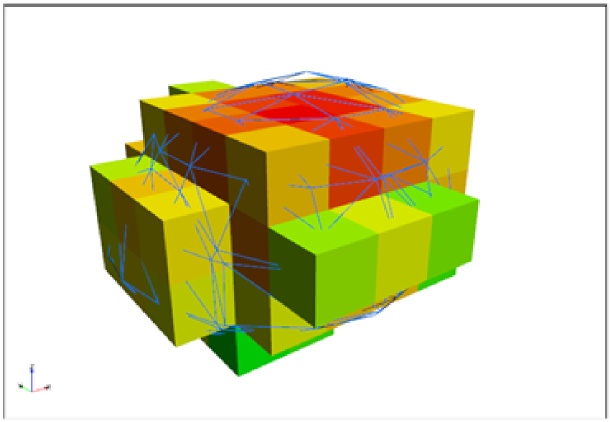
Acceptance Test

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:

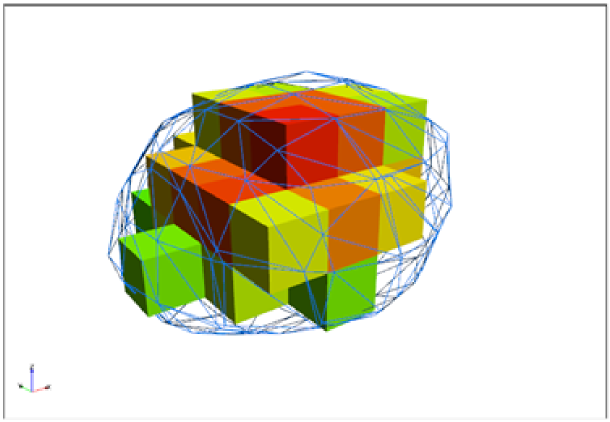
- **One or more samples within the volume**: Fastest but least accurate method.



- **>= 50% of the samples within the volume**: Medium accuracy.



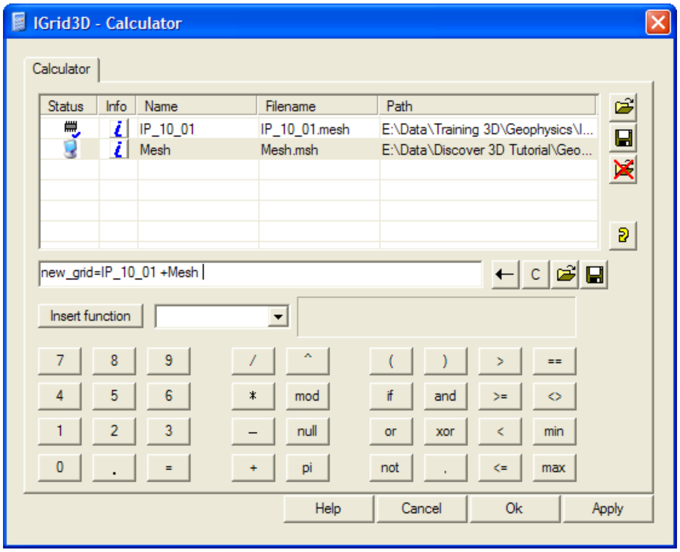
- **All samples within the volume**: Slowest but most accurate method.



3D Grid Calculator

The **3D Grid Calculator** is a powerful tool for modifying 3D grids. To operate the calculator, build an expression using the following:

NewName = exp<Name1>…exp<Name2>



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.

Advanced grid calculator properties are accessed via the properties button on the software main window. This allows you to control the output mesh size. There are two options:

- **A superset 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.

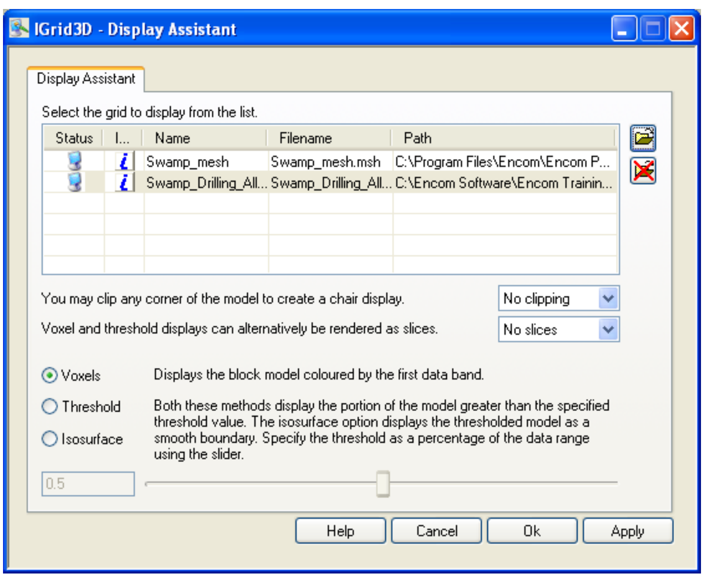
The advanced properties also allow you to control the NULL handling. By default, strict NULL handling is tuned off. This means that the outcome of simple expressions using the +, - , * and / operators will be valid if one of the operands is NULL and it will be equal to the non-NULL operand. If strict null handling is turned on, the outcome of any expression where any operand is NULL will be NULL.

Press **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 from the **Grid Management** wizard.

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 3D grid.

3D Display Assistant

The Display Assistant function of the Voxel Toolkit allows direct access to voxel model displays within Encom Discover PA. Selecting the **Display Assistant** button from the Voxel Toolkit displays the Assistant dialog as shown below:



Display Assistant dialog with two voxel model files selected

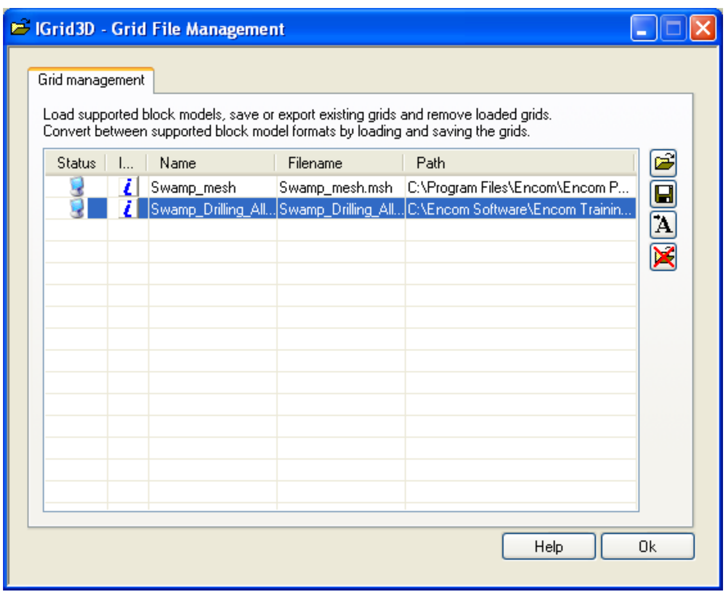
The Display Assistant dialog allows you to do most of the basic display functions of voxels within Encom Discover PA. You can:

- 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)
- Have the voxel model display in Voxel, Threshold or Isosurface format. The Threshold and Isosurface style 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 Voxel Displays and various property descriptions.

Grid Management

The **Grid Management** tool available from the Voxel Toolkit enables you to load, import, save, export and unload 3D grids



Grid management tool to open, load and save 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.

The Voxel Toolkit can load grids in a variety of formats including:

- UBC (University of British Columbia) (.MSH files)
- CEMI (University of Utah) (.CEM files)
- Noddy (Pitney Bowes Software) (.g00 files)
- Encom 3D Grid (Multi-banded) (.e3D files)
- Datamine (*.DM files)
- Surpac (*.MDL files)
- Vulcan (*.BMF files)
- GoCAD (*.VO voxet files)
- Gemcom (.BPR files)
- ASCII Import (.CSV or .DAT files)

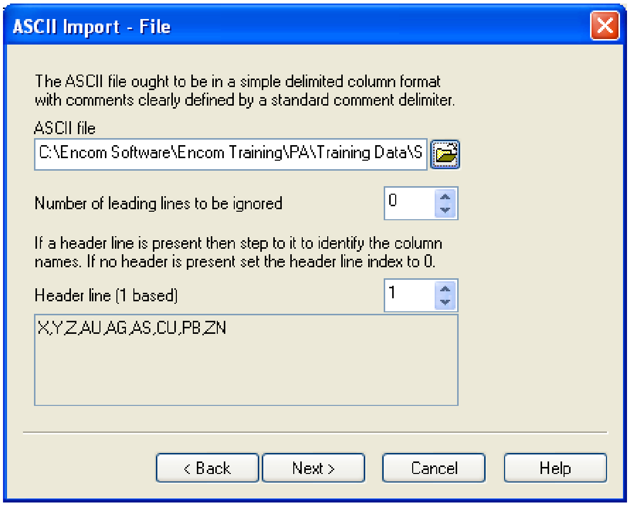
It also includes a powerful ASCII import wizard that loads delimited ASCII data and generate multi-banded 3D grids. This tool and all other grid file imports are accessed from the **Open File** button in the Grid Management tool.

Voxel Toolkit can 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.

ASCII Import of 3D Data

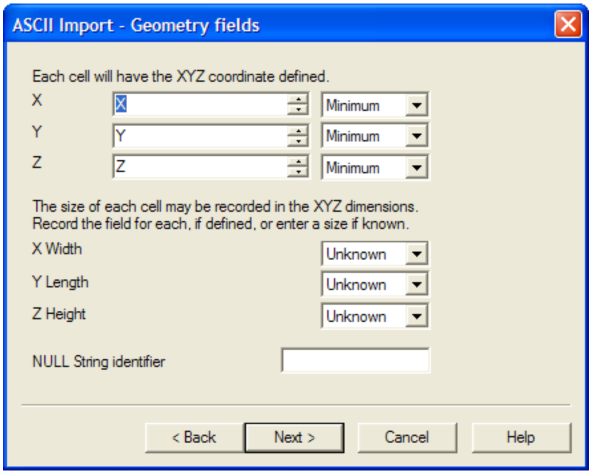
The ASCII import tool loads delimited (comma, tab, space) ASCII data. It may include zero or more header lines. If a header line is present it can be used to assign field names. If no header line is specified (by setting the header line to line 0) then the fields are referred to using column numbers where the first column is column 1.

The tool expects to load an X, Y and Z coordinate for each cell. This may be the minimum, centre or maximum extremity of the cell.



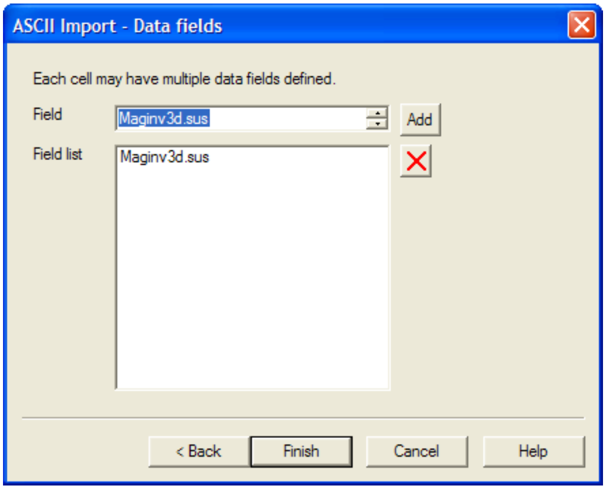
Specify the input data file

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 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.

28 Vector Utilities

- *3D Solid Generator*
- *3D Extrusion Tool*
- *Topology Checker*
- *Transform Vector File*
- *Combine DXF Files*
- *MapInfo TAB to Geosoft GDB*
- *Geosoft GDB to MapInfo TAB*
- *DXF to ModelVision TKM*
- *MapInfo TAB to Vulcan Archive*
- *Vulcan Archive to MapInfo TAB*
- *EM Plate Importer*
- *MapInfo MIF/TAB to ArcView SHP*
- *Gemcom BT2 to MapInfo TAB*

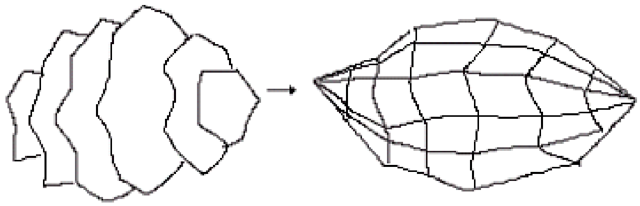
3D Solid Generator

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

The 3D Solid Generator is useful for creating interpreted 3D objects such as orebody, fault or other geological solids (e.g. aquifers), as well as infrastructure such as dams or open-cuts.

This utility interpolates between polygon or polyline objects in a Feature Set (see *Features*) to produce simple, open 3D surfaces (e.g. faults) or closed polyhedral objects (e.g. alteration zones).

The input data for the Solid Generator commonly consists of interpreted geological/infrastructure boundaries that have been digitized onto a number of parallel drillhole cross-sections (or plans) either in 2D or 3D. The utility then processes these polygons/polylines, interpolating the triangular facets between each of the sections/plans to produce a single 3D solid body.



Individual slices of interpreted regions are used to create a solid object DXF

A default output file name is required and the location of this file can be changed using the **Browse** button. Depending on the geometry of the input .DXF file, two output types can be chosen:

- **Simple, open surface** – produces a ribbon-like surface with no closed ends (eg a Fault plane). To generate a simple open surface, the input file must contain at least two open ended (that is, non-closed) lines.

- **Closed, polyhedral object** – produces a fully enclosed 3D solid object. For this output type an **End capping style** can be selected with options ranging from None, Flat, Point or Smooth. To generate a closed 3D solid object the input file must contain at least two closed polylines i.e. polygons.

A .TKM file can be created from this utility which allows the solid object to be imported into Encom ModelVision Pro for modelling and interpretation.

Creating Boundaries

The source Feature Set containing polygons/polylines representing interpreted/ inferred geological/structural/infrastructure boundaries can be created via two methods:

1. 2D – digitizing into section profiles or geological cross-sections using the feature digitising utility (see *Features*).

2. 3D - digitizing directly in the 3D environment into an editable Feature Database and Feature Set using the Cursor Plane functionality. For information on 3D boundaries, see *3D Digitized Boundaries*.

3D Digitized Boundaries

To digitize boundaries 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 (see *Feature Databases*) into which the boundaries are to be digitized is editable (either the cosmetic layer or a new database). Align the Cursor Plane (see *Toolbars* for information on the Cursor Plane Toolbar) as required and use the Feature drawing tools to digitize the appropriate polygon and/or polyline vertices.

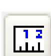
See *Working with Features* for more information about the many tools available for 3D digitizing. In particular, the ability to snap to drillhole intervals is a important function.

Some digitization tips which are controlled via the **Cursor Plane** tab of the Cursor Plane Properties dialog 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 **Azimuth** 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.

  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.



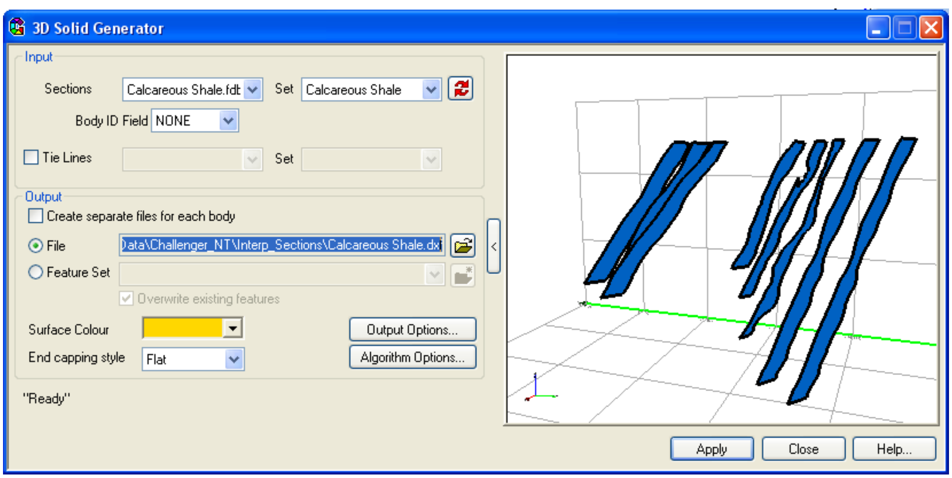
The **Distance/Bearing** tool on the Main Toolbar can be a useful guide to distances,bearings and dip angles between objects on the cursor plane.

- If digitizing onto georeferenced images (e.g. geophysical profiles), use the **Bond to Image** option on the Cursor Plane Toolbar.

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 *Algorithm Options*)

Create 3D Solid

The **3D Solid Generator** is accessed via the **Display Utilities** menu within Encom Discover PA. It requires two or more individual Feature Objects (not in the same plane) in order to create an output DXF solid.

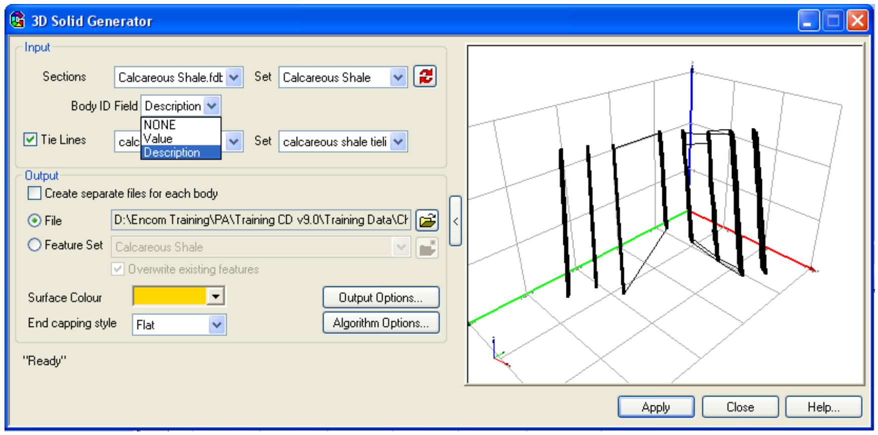


3D Solid Generator dialog

The controls available from this dialog are:

- The top **Feature** pull-down list enables the choice of the source **Feature Database** (if multiple feature datasets are open in Encom Discover PA) and Set containing the source objects. This dialog can be kept open while changes are made to a Feature Database or Set.

- The **Body ID Field** option enables a field in the feature database to be used to separate the output into separate bodies/solids. Within the Solid Generator utility the **Body ID Field** would be set to the **Feature_ID field**, in order to build a separate object for each of the unique attributes in this field. For instance, a series of 2D geological section boundaries are created and attributed using the Feature_ID field with “L1”, “L2” and “L3”, representing different lithology units.



The 3D Solid Generator configured to create separate output bodies (using the Body ID Field selector) for each unique attribute in the Ranking field of a Feature” database.

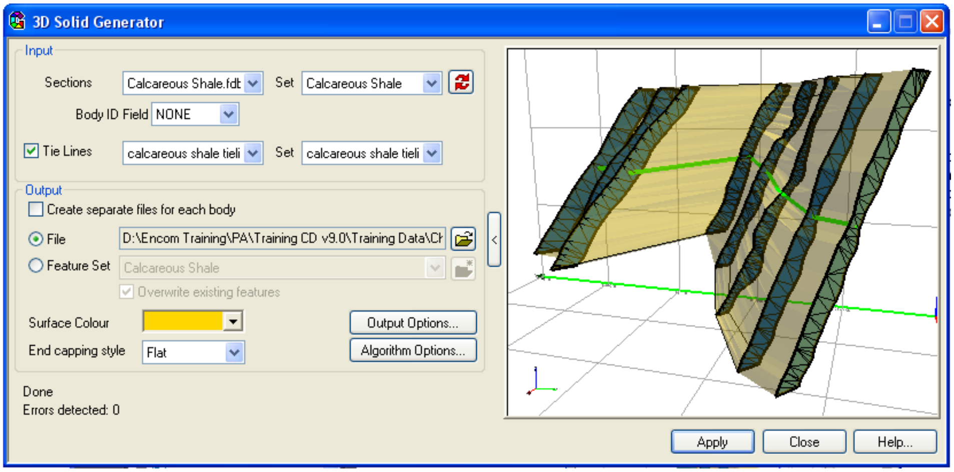
- The **Refresh** button allows any such changes to be updated in preview window of the dataset within the **3D Solid Generator**.

- 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 and set containing the tie lines from the adjacent pulldown list. See *Tie Lines* for further information on creating and using tie lines.

- An **Output** DXF or TKM file name and location can be specified; by default this utilises the same name and location as the source file.

- 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 the output DXF solid/s, as can 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.



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.

For more information, see:

- *Tie Lines*
- *Output Options*
- *Algorithm Options*
- *Structure Analysis*
- *Shape Matching*
- *Compression*

Tie Lines

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.

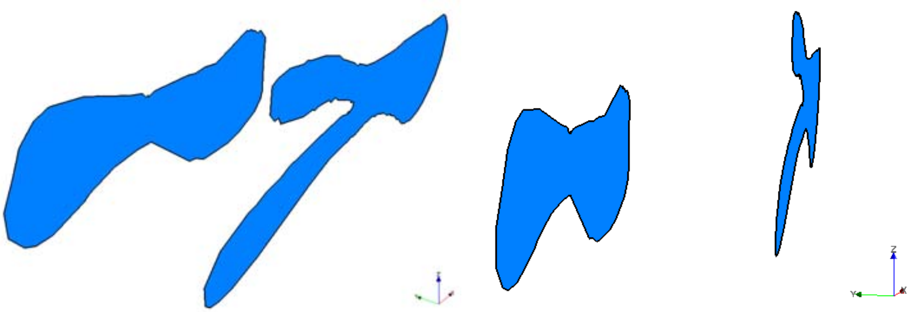


Figure 1: Two adjacent parallel polygons, the left a simple polygon, whilst the right hand shape splays into two lenses

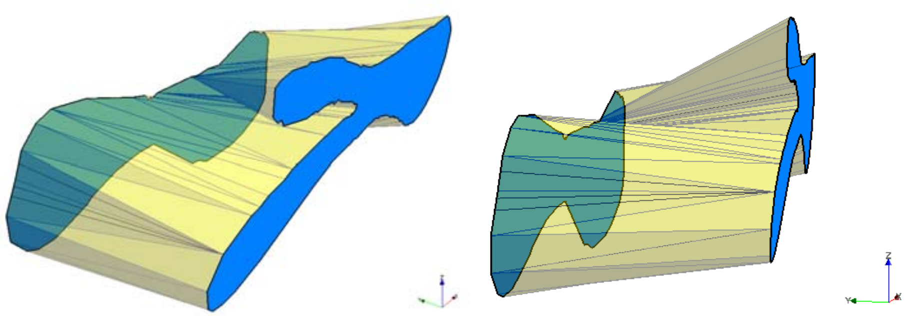


Figure 2: The resulting semi-transparent and wireframed dxf solid generated by the 3D Solid Generator without using tie lines. Whilst the bottom lens of the right-hand polygon has been joined satisfactorily to the left-hand shape, the upper lens on the right hand shape has not.

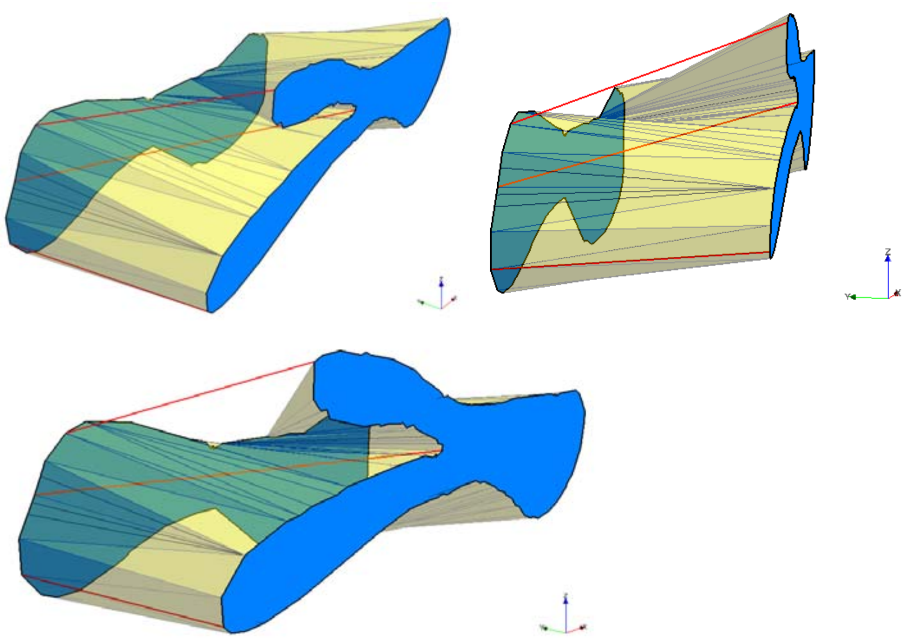


Figure 3: The preferred solid would have both these lenses joined to the near end of the left-hand shape. Red tie lines have been added indicating the user-required vertex-vertex joins between the two objects.

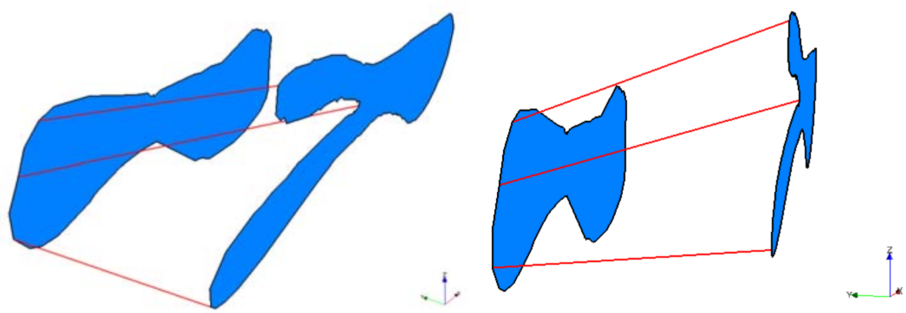


Figure 4: With the original (unsatisfactory) output solid removed, the tie line joins are clearer, linking the most prominent inflexion points (troughs and peaks) on the right-hand shape with the corresponding vertices on the left-hand polygon.

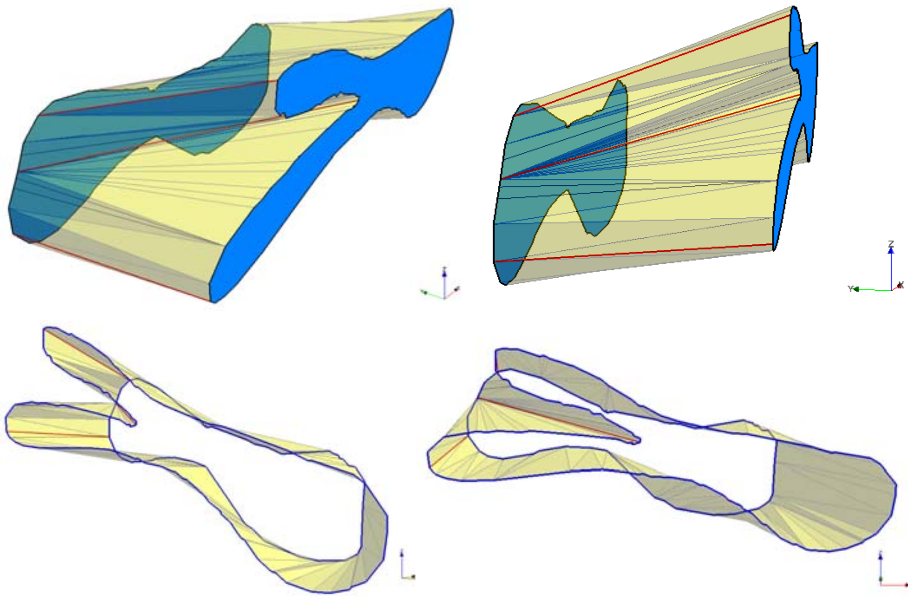


Figure 5: The resulting wireframed and semi-transparent solid incorporating the user-specified tie lines. The twin-lensed shape now clearly joins the left-hand polygon as desired. The red tie-lines are displayed for reference.

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:

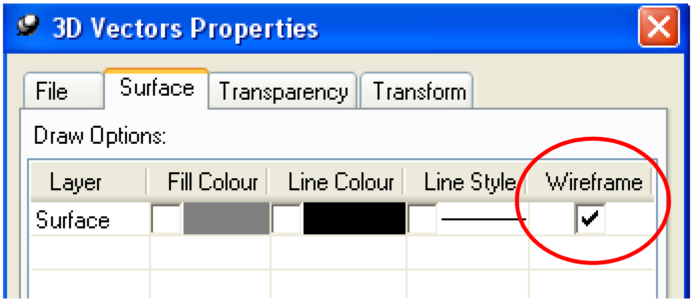
- Tie lines must be placed in a separate feature database to the source polygon/polyline objects being joined (commonly the Cosmetic layer is used for initial tie line creation, and then saved using the **Save Cosmetic Features** command).

- 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.

Recommended procedure for creating tie lines:

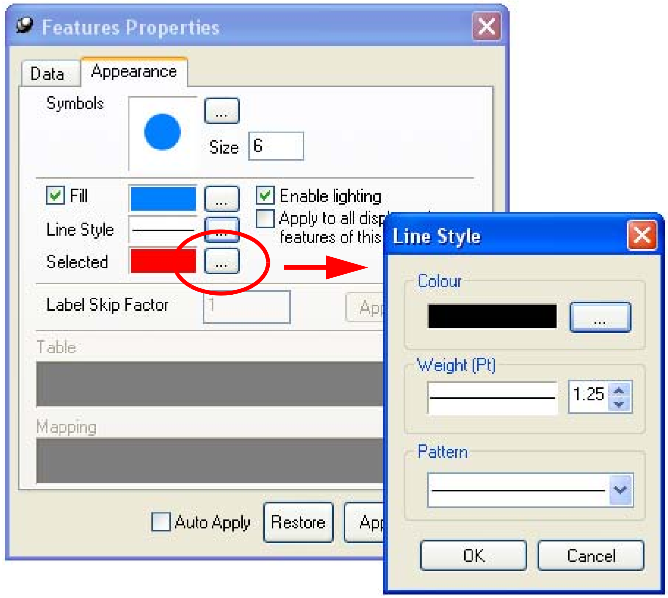
1. Run the source dataset through the 3D Solid Generator (with and without the **Structure Analysis** option enabled) to visualise the solid output. In the 3D window, make this output DXF semi-transparent, and enable the wireframe view.



Enabling the wireframe view of a vector file

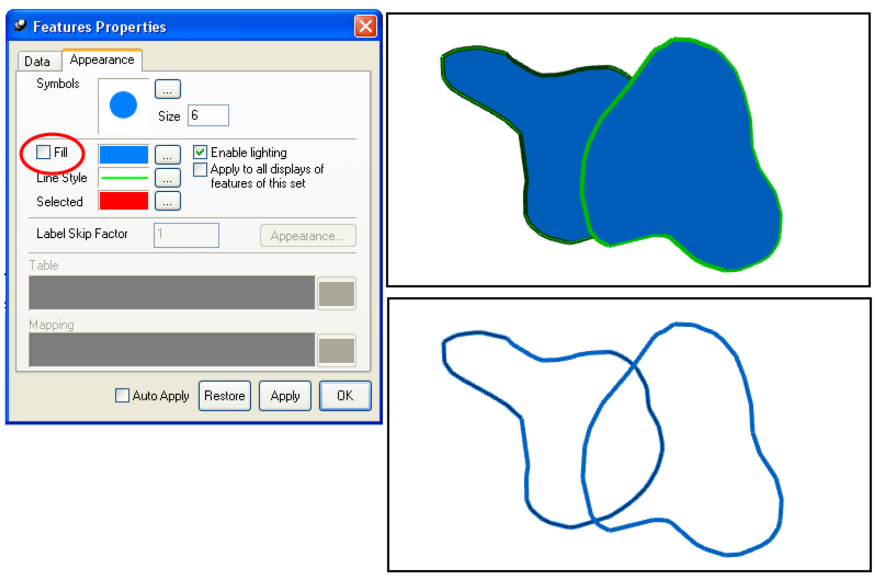
2. Examine in detail via the wireframe view the various vertex-vertex joins created by the 3D Solid Generator algorithm.

3. Make the Cosmetic Layer editable, and set its line colour to red, with a weight of 3 points.

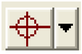


Setting the line style of the tie lines in the Cosmetic Layer to stand out

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.

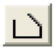


The polygon Fill tick box on the Feature Properties dialog (red circle) allows the fill of polygons to be disabled, allowing their boundaries to be more clearly visualised when a high density of polygons is involved.



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 vertices.

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.

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 manually refresh (at any time) the preview screen within the 3D Solid Generator, click the **Refresh** button.

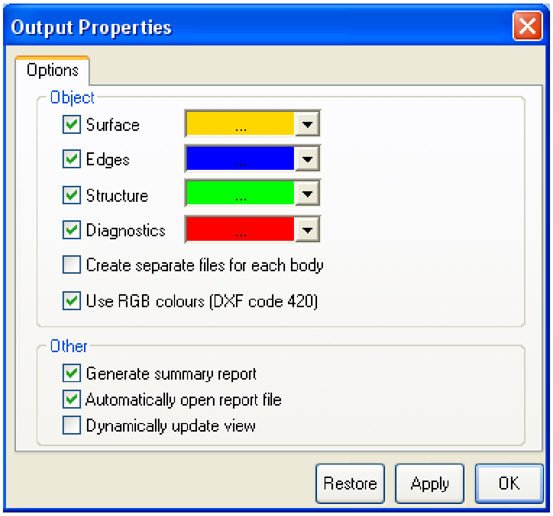
Output Options

The Output Options dialog controls the following components of the generated output solid and their display colour:

- **Surface** - this option is recommended to 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
- **Diagnostics** - contains duplicates of any erroneous/self-intersecting facets

An option to **use RGB colours** in the output DXF file is available. Note however that not all DXF-compatible programs can handle RGB colours within a DXF file.

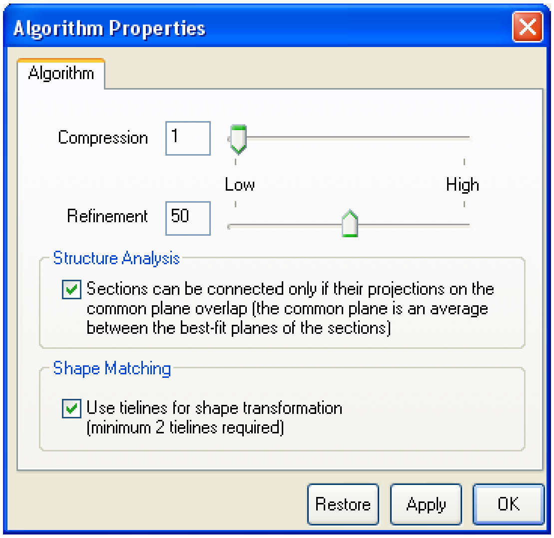
Enable the **Dynamically update view** option to update the 3D view during solid generation (once per facet).



The Output Options dialog for the 3D Solid Generator utility.

Algorithm Options

The Algorithm Options dialog contains a number of advanced functions that can help to improve the quality of the output dxf shapes, particularly when applied in conjunction with *Tie Lines*.

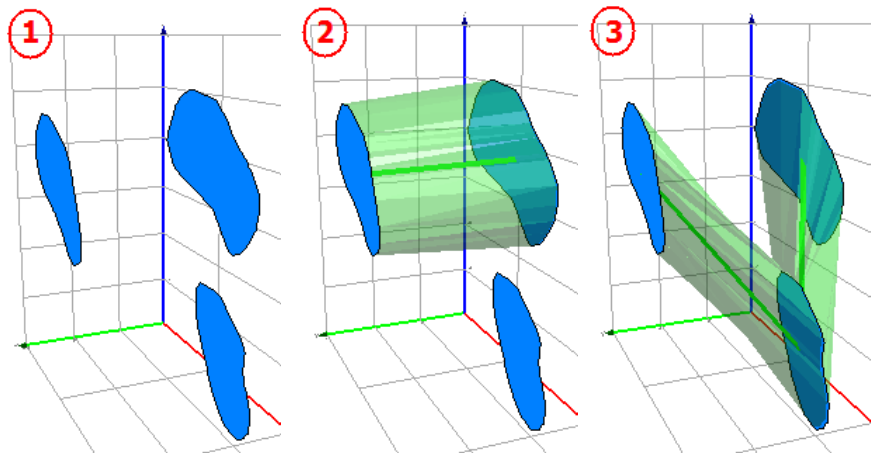


The Algorithm Properties dialog for the 3D Solid generator utility.

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* 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*. 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.

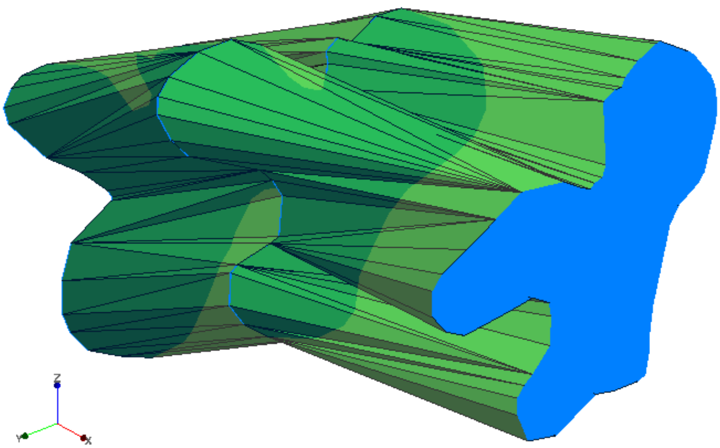


Figure 1: A wire-framed solid dxf (green) created between three polygons without using tie lines. Note the poor correlation between peaks and troughs

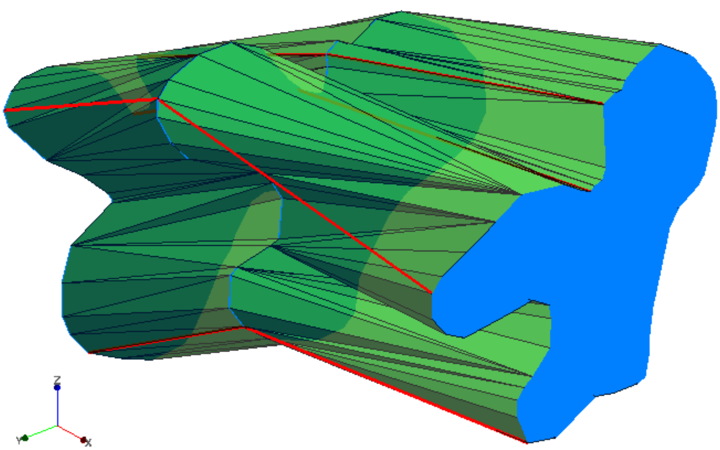


Figure 2: Tie lines (red) created to control the join of major structures

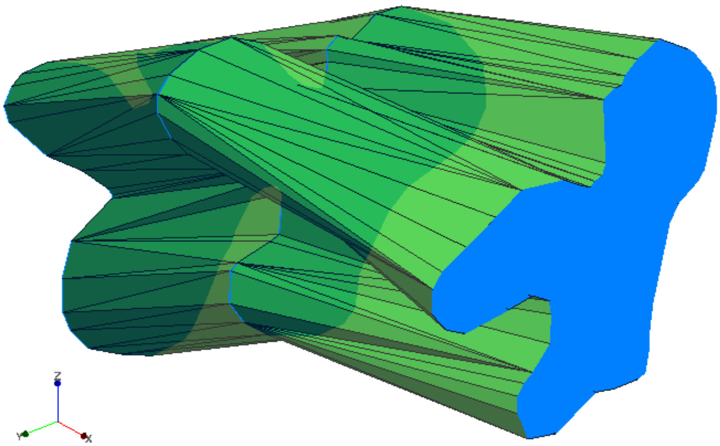


Figure 3: The output solid created using tie lines from the previous step, and Shape Matching disabled. There is a good correlation between peaks and troughs at tie line locations, but it is less robust between these locations.

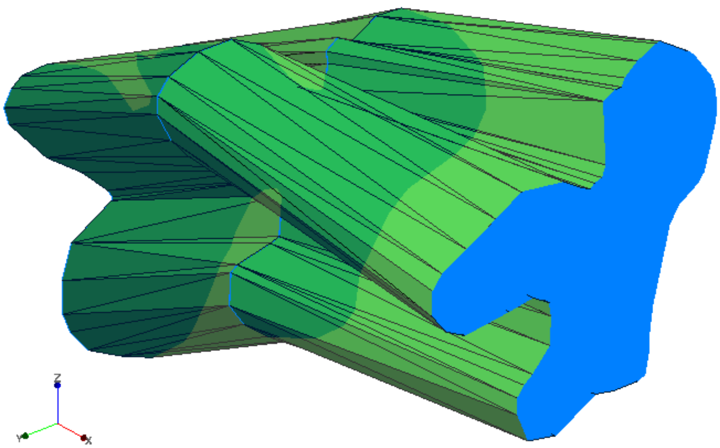


Figure 4: The output solid with Shape Matching enabled and the tie line input from the previous steps. This displays an excellent correlation between peaks and troughs, and realistic surface creation elsewhere.

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 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.

Once the 3D Solid Generator dialog has been configured press the **Apply** button to generate the output DXF 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, tie-lines indicating the drawing order between Feature Objects are displayed and eventually the DXF solid outlines are displayed in the preview screen. A new Vector branch is also displayed in The Workspace Tree populated with the output vector solid. (see *3D Vector Displays*). Simultaneously, the Report screen at the bottom of the 3D Solid Generator dialog details the current processing and upon completion lists the output model centre, extents, dimensions, surface area and volume if appropriate. This report can be cleared at any time using the **Clear Report** button.

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 *Features*), allowing the shape of the input objects or tie lines to be refined prior to reprocessing.

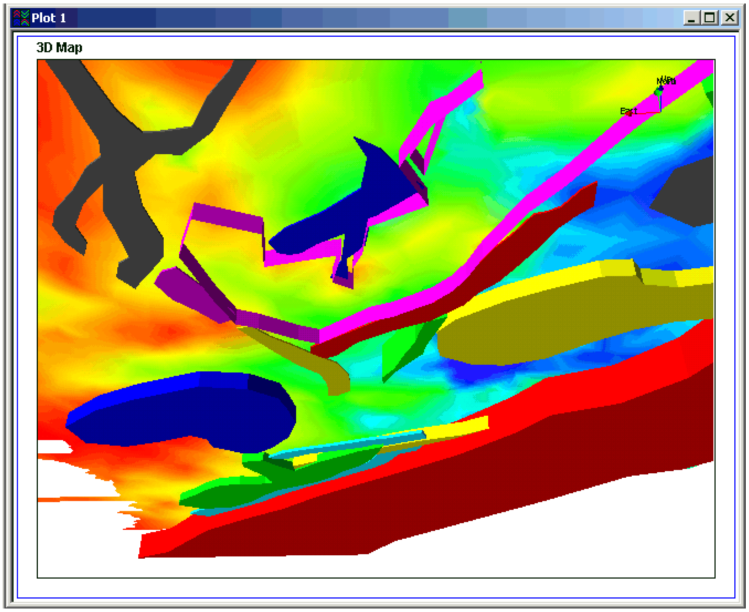
3D Extrusion Tool

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

It is often useful to have two dimensional map polygonal interpretations transfered to meaningful three dimensional displays when the objects can be associated with properties of height or volume. Polygonal objects drawn in a map may have an elevation, but with the addition of size or volume, a 3D display can be much more appealing, allowing visual comparisons between areas. For example, a geological map showing polygons of interpreted geology may be overlain over a topographic surface. However, a 3D representation of geology is more useful if thickness and volume could be displayed What if we could associate the volumes with physical properties such as density or magnetic susceptibility? The volumes could then be geophysically modelled.

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 3D Extrusion Wizard can operate on objects either within a 2D vector file or an open Feature Database. Height information can be specified from fields within the vector file/feature database 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 mapped outline of the 2D object. For more information, see *Using the Extrusion Wizard*.



Example of extruded objects below a terrain surface.

Using the Extrusion Wizard

Step 1 – Load input data

Start the **Extrusion Wizard** from the **Display Utilities** menu.

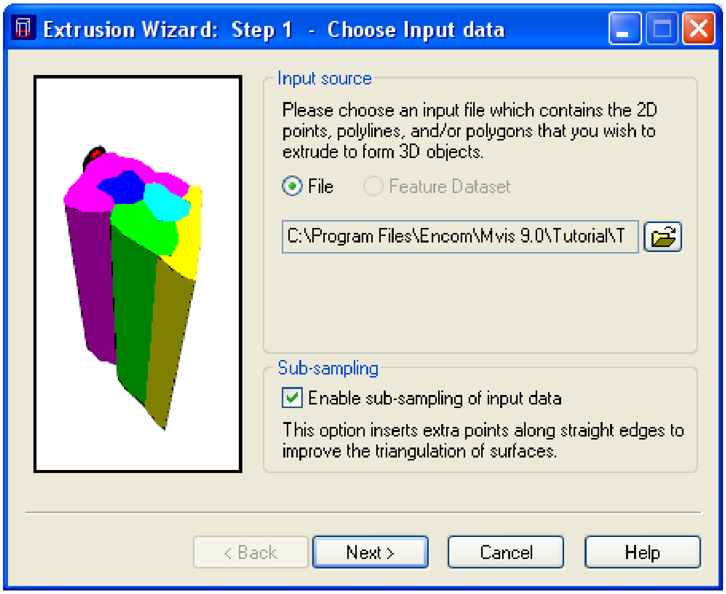
In the first dialog of the Extrusion Wizard that appears select an input file to use that contains polygonal outlines or polylines that are able to be displayed as 3D extruded volumes. To load a **Vector file** select the **File** option and then click on the **Browse** button to locate the file.

To load a **Feature Database** that is already open in the current session of Encom Discover PA as the input data select the **Feature Dataset** option and then select from the drop down lists that appear a Feature Database and a corresponding Feature Set.

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.

Note It is important to select a vector file or feature database that has suitable map object outlines such as polygons, closed polylines (eg. Contours) or regular objects such as rectangles, squares or circles. If the vector file of feature database contains point objects the **3D Extrusion Wizard** can create three dimensional objects such as curtains or vertical bars using point data.

Click **Next** to proceed to the next dialog of the Wizard.



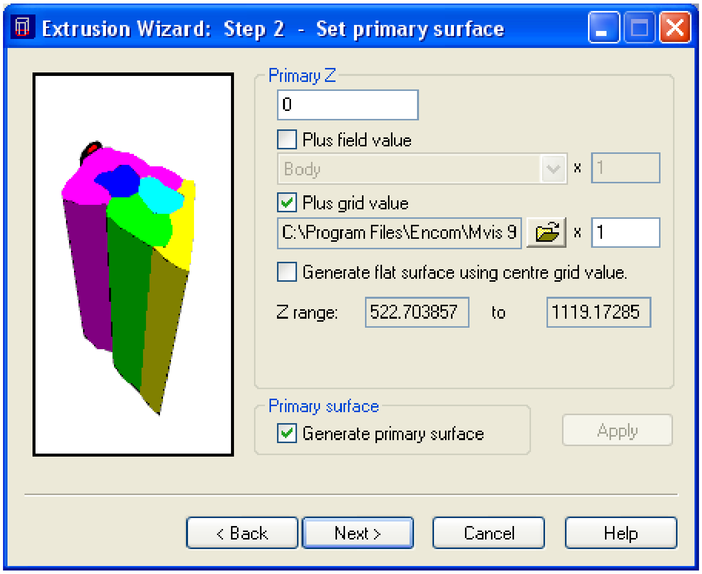
Select the File or Feature dataset with 2D outlines of items to be extruded

Step 2 – Set Primary Surface Properties

The Base Z value of the objects to be extruded can be specified in one of 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 vector file or feature database for each map object to be extruded. For example, if geological units were to be used, these could be stored in the selected file with a data column called Vert_base or Thickness. The values are in metres and are added to any entry in the Primary Z field.

- **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 that these values are added to any Primary Z and Field Value entries.



3D Extrusion Wizard Step 2 defines the base of the extruded object

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 is turned on by default. This option creates a base for the extruded object. If this option is left unchecked the extruded object is left open at the base. Click Next to continue.

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.

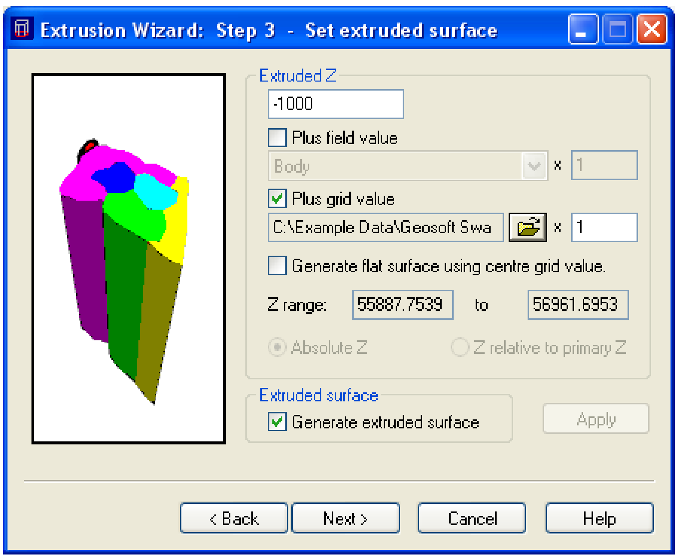
- **Plus Field Value**: use an extruded Z value from a field contained within the MapInfo Professional table for each map object to be extruded. For example, if a geological depth below surface were to be used, these could be stored in the selected table with a data column called Dpth_to_Top or Depth. The values are in metres.

- **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 or depth of the base or top 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** (ie. 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.

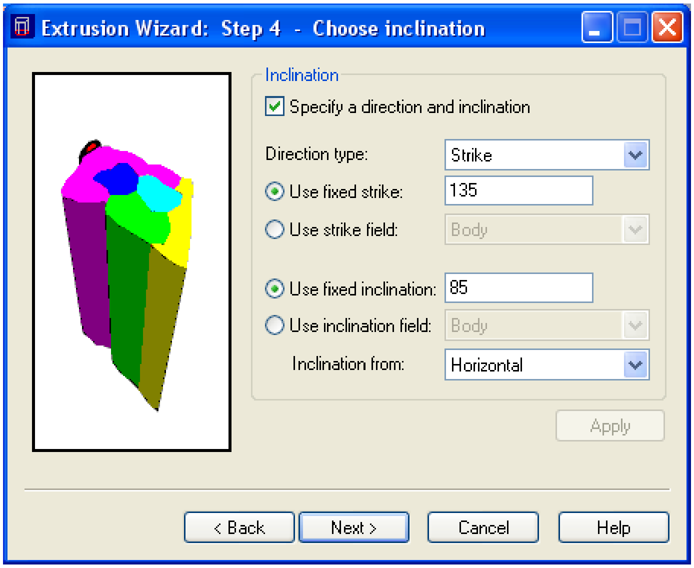


Step 3 - Define the upper surface of the extruded objects

Step 4 – Set Azimuth and Dip

Ticking the **Specify an azimuth and di**p 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.

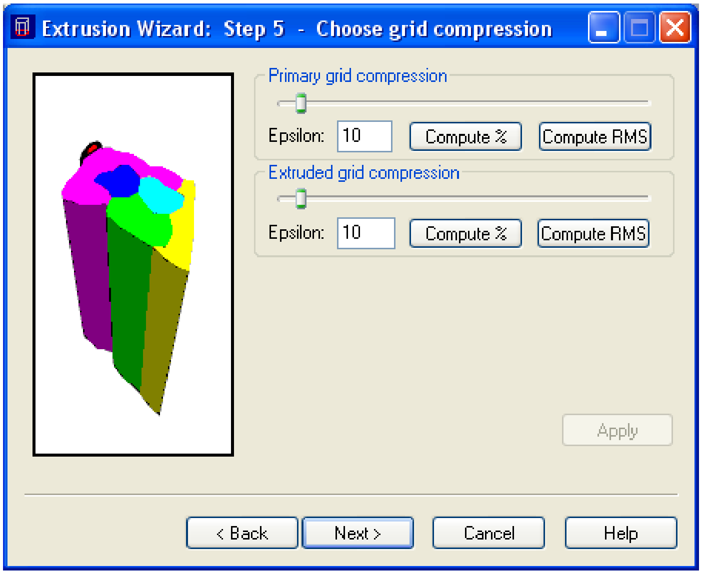
Bearing and **Inclination** values can be specified either as fixed/constant values, or from appropriate fields in the object table.



Specify the azimuth and dip angles for volumes to be tilted

This feature is useful for fault definitions or block offsets of interpreted objects. By default the dip angle is 90 degrees (vertical). An angle of 0 degrees is horizontal.

Step 5 – Set Grid Compression

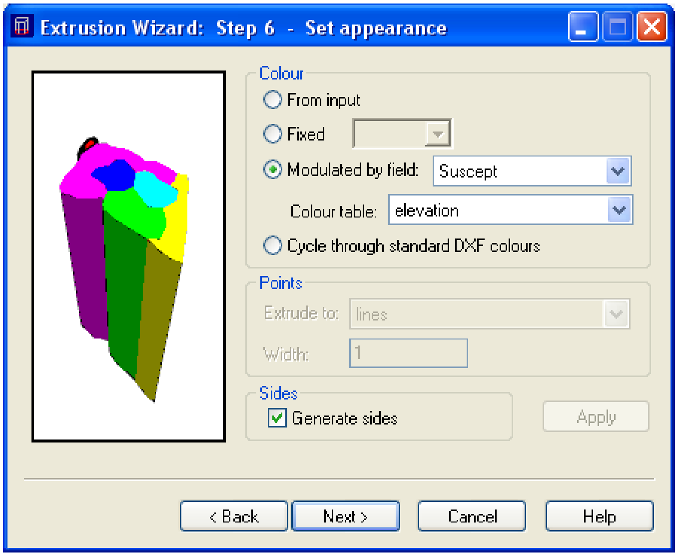


Adjust the primary and extruded grid compressions in Step 5

Use the Grid Compression options available in the dialog for Step 5 of the extrusion wizard if a grid has been used in defining either the primary or extruded surfaces (in Steps 2 and 3). This will reduce the size of the output 3D model vector file.

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

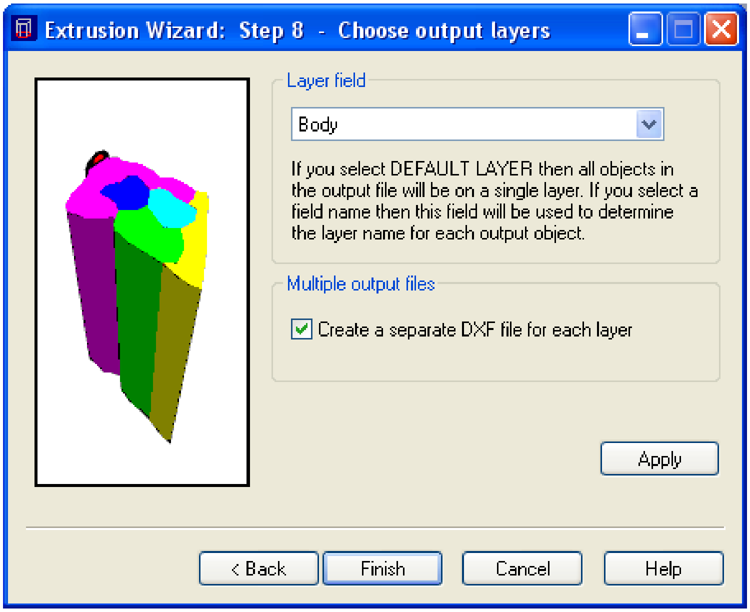
Four options are provided to allow the colour of the objects to be controlled:

- **From input** - the individual objects obtain their colour from the default list of standard colours
- **Fixed** - all the created objects are of the same colour. Select the required colour from the pull-down palette.
- **Cycle through standard DXF colours** - apply colours from the 16 standard colours provided for .DXF support
- **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 is provided.

If the original data specified for extrusion is points rather than polygons, the point data can be extruded as Lines, Tubes, Square or Triangular prisms.

By default the **Generate Sides** option is ticked to create an enclosed object.

Step 7 – Default Layer



Specify a layer name association with the output data field

In the final screen, 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.
- If the **Finish** button is clicked, the extruded objects are created in the defined DXF file(s) and added to the Discover PA 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 PA 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 a 3D map window.

Step 8 – Choose Output File

The Output File screen allows the extruded shapes to be saved as a DXF vector file or feature dataset, and to generate a summary report:

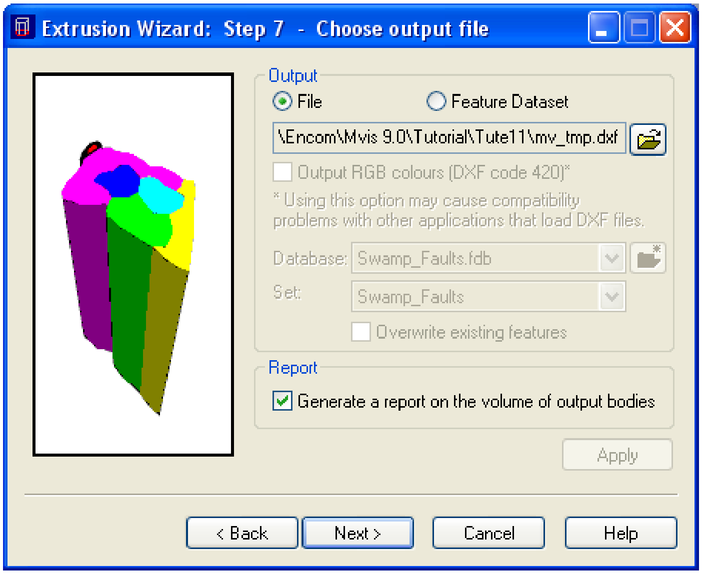
- *DXF/TKM Vector File Output*
- *Feature Dataset Output*
- *Generating a Report*

DXF/TKM Vector File Output

Enabling the **File** option in Step 1 will create an output .DXF vector file utilising the same name and directory as the input file by default. 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.

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). Pressing the Report button in this report 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).



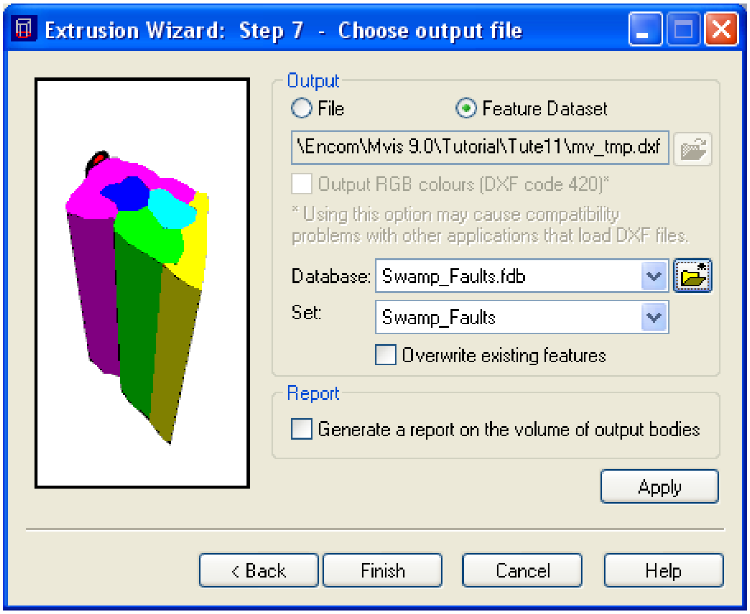
The Output File screen configured to generate a .DXF Vector file.

Feature Dataset Output

Selecting the **Feature Dataset** option in Step 1 allows the extruded objects to be added to either a new or existing Feature Database. These can then be edited in 3D using the feature editing tools (see *Working with 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.

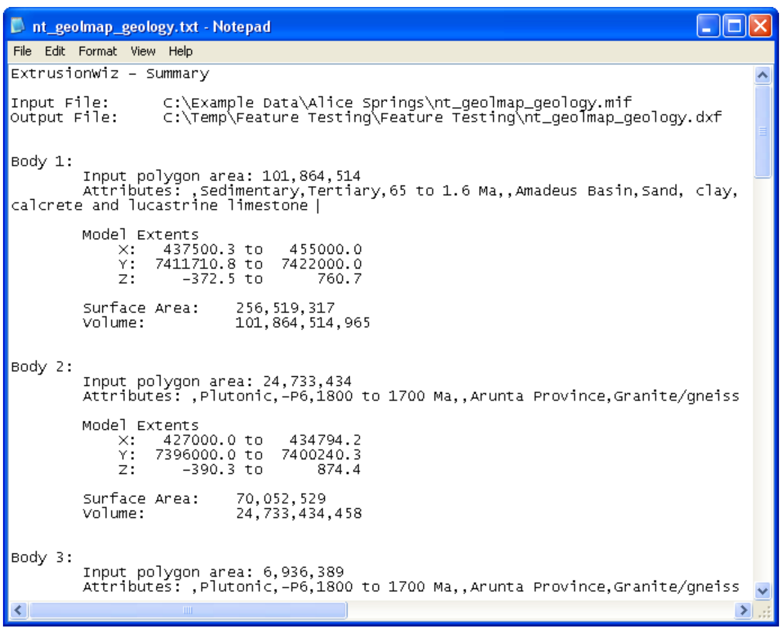
If the **Finish** button is clicked, the extruded objects are created in the defined feature database and added to the 3D map window. The Extrusion Wizard is then exited.

If the **Apply** button is clicked, the wizard creates the feature database and automatically displays it in the 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 3D map window

Generating a Report

If the **Generate a report on the volume of output bodies** option is enabled a summary of the 3D model generated by the extrusion process will be created in a text file and automatically displayed in Notepad. The report summary provides information such as:

- The number of bodies
- Input polygon area of an extruded body
- Attributes from the input file, such as lithology, age, etc.
- 3D Model extents for each body (XYZ)
- Surface area of extruded body
- Volume of extruded body




```
ExtrusionWiz - Summary

Input File:      C:\Example Data\Alice Springs\nt_geolmap_geology.mif
Output File:     C:\Temp\Feature Testing\Feature Testing\nt_geolmap_geology.dxf


Body 1:
        Input polygon area: 101,864,514
        Attributes: ,Sedimentary,Tertiary,65 to 1.6 Ma,,Amadeus Basin,Sand, clay,
calcrete and lucastrine limestone |

        Model Extents
            X:   437500.3 to   455000.0
            Y:  7411710.8 to  7422000.0
            Z:     -372.5 to      760.7

        Surface Area:    256,519,317
        Volume:          101,864,514,965


Body 2:
        Input polygon area: 24,733,434
        Attributes: ,Plutonic,-P6,1800 to 1700 Ma,,Arunta Province,Granite/gneiss

        Model Extents
            X:   427000.0 to   434794.2
            Y:  7396000.0 to  7400240.3
            Z:     -390.3 to      874.4

        Surface Area:    70,052,529
        Volume:          24,733,434,458


Body 3:
        Input polygon area: 6,936,389
        Attributes: ,Plutonic,-P6,1800 to 1700 Ma,,Arunta Province,Granite/gneiss
```

Example report generated from the Extrusion Wizard utility

Topology Checker

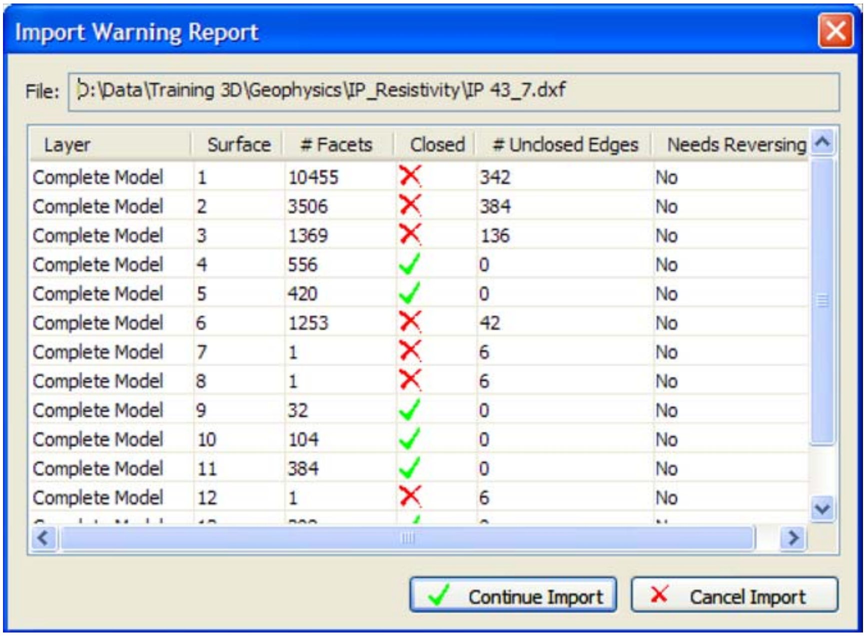
Encom Discover PA can check the integrity of models imported from 3D vector files prior to import into a feature database or for modelling purposes in an external software package, such as Encom ModelVision. This is important if advanced feature object editing functionality is to be used on the model, such as the Combine, Intersect and Cut tools). These tools require a consistent method of ordering the vertices of every face and that all surfaces were closed. This is an essential requirement of any modelling software and any gaps or incorrectly defined faces in a model will lead to errors in the model computations.

The Topology Check utility collates all faces and organises them into closed surfaces reporting any that break the rules. It can then correct some errors and export the intact surfaces as a new DXF file, import into a feature database or as individual polyhedron bodies in a TKM (Encom ModelVision format) file that can be imported into Encom ModelVision. If any surfaces are reported as not closed, the model must be closed in the software application that it originated from before using it in the Topology Checker. 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.

- *Using the Topology Checker*
- *Topology Checker Controls*
- *Surface Options*
- *Global Options*
- *Rendering Options*
- *3D View Controls*

Using the Topology Checker

Generally 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 or when attempting to model in Encom ModelVision. Errors in each surface will be indicated by the following Import Warning Report dialog, detailing which layers are unclosed, and which need their vertices reversed.



Errors reported in the Import Warning Report dialog.

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 **Display 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 Unclose**d 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.

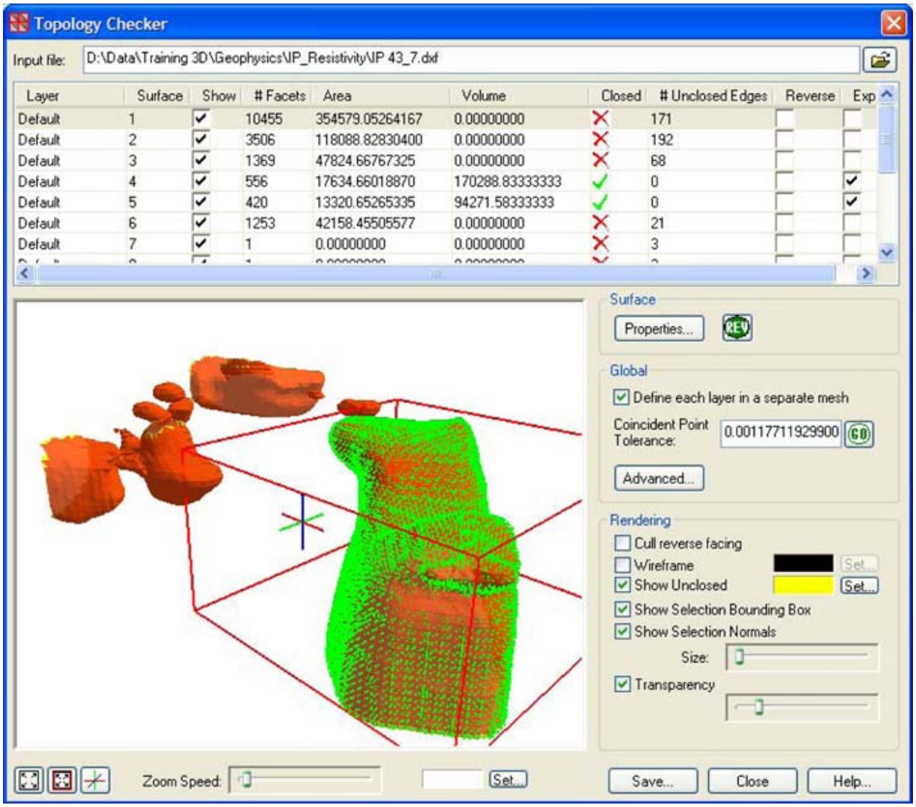
5. Press the **Save** button to export the selected layers to a new vector file.

6. Rerun the *Feature Database Creation Wizard* using this new vector file as the source file.

Topology Checker Controls

The Topology Check option is accessed from the Display Utilities menu. The Topology Check dialog that appears consists of the following properties:

- **Input File** - Selects the model file to load into the Topology Check 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.
    - **Layer** - Name of the layer in the input file of one is provided.
    - **Surface** - The surface number.
    - **Show** - Select/Unselect the tick box to show/hide the surface.
    - # **Facets** - The number of facets in the mesh for the 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 (displayed on the left) button to execute the reverse process for all enabled layers. Any layers requiring vertex reordering to facilitate use in Encom Discover PA will automatically have their vertex order reversed when the Save button is pressed.
    - **Export** – Indicates whether the mesh for this surface should be added into the output file when saving.



The Topology Check utility dialog

Surface Options

- **Properties** - 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** - The Advanced button will open the Advanced Options dialog. This dialog allows the user to specify the Body Density and Body Susceptibility values for the model. These are used when saving the model to a .TKM file for use in Encom ModelVision Pro.

Rendering Options

- **Cull reverse facing** - Turn on/off display of any back facing triangles.

- **Wireframe** - Turn on/off display of the wireframe mesh for the model. The colour of the wireframe mesh can be chosen by using the appropriate Set… button.

- **Show unclosed** - If this option is turned on then the colour of any triangle faces that are not closed will be overrided 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.

- **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** - Fits the view so that the entire model can be seen.

- **Fit view to bounding box of selected surface** - Fits 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:

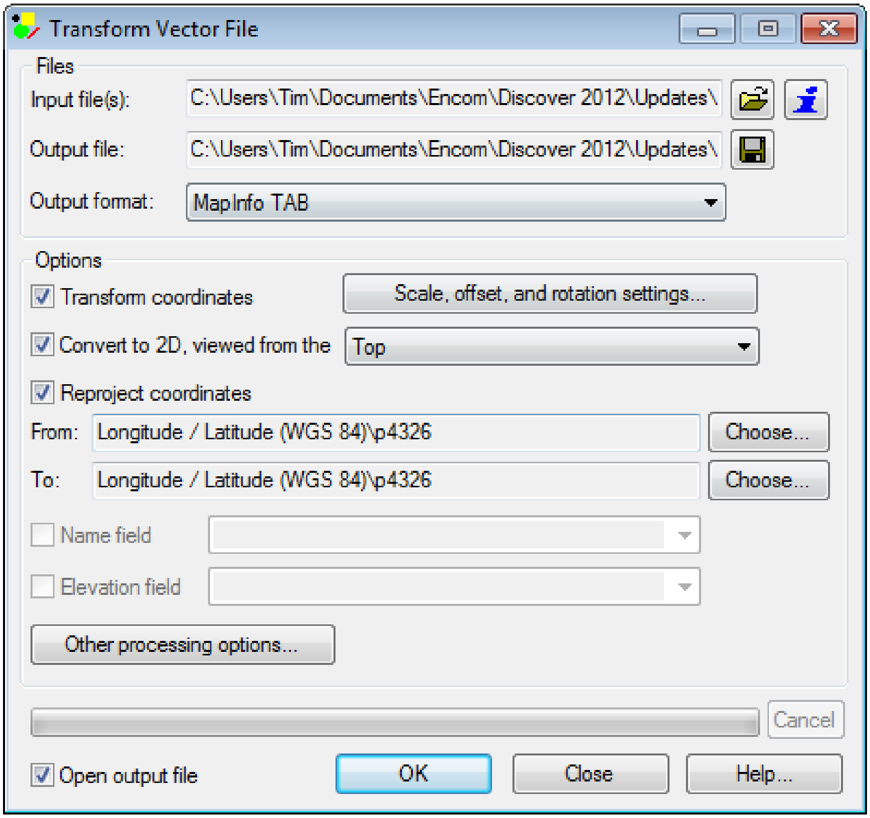
3D Studio .3DS
AutoCAD .DXF
Datamine wireframe (point and triangle) .DM
ER Mapper vector .ERV
ESRI shapefiles .SHP
ESRI TIN .ADF
Gemcom .BT2
GOCAD vector .TS, .PL, .VS
Google Keyhole Markup Language .KML
GPS exchange .GPX
LizardTech MrSID .SID
MapInfo Professional .MIF
MapInfo Professional .TAB
Surpac 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 coordinate**s – 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 fro**m – 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.

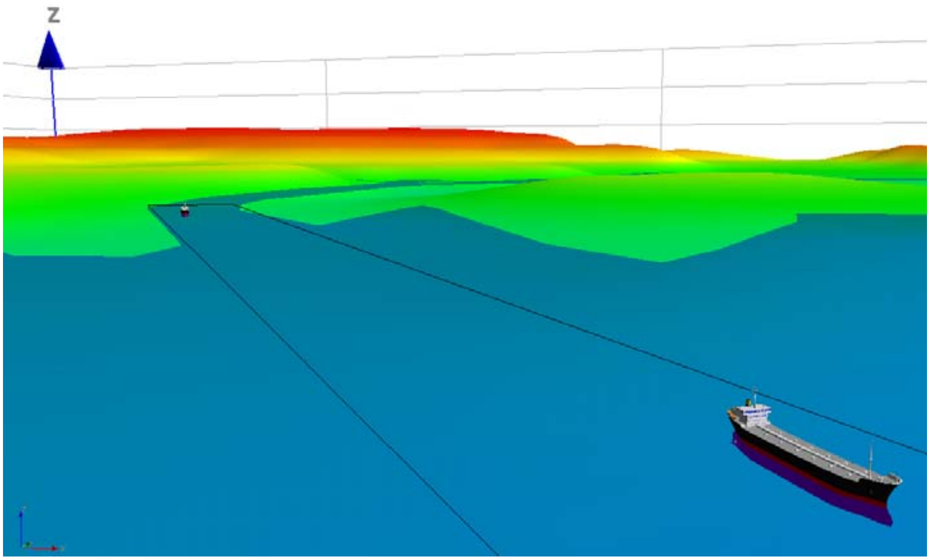
- **Compress polylines and polygons** – removes any nodes with a spacing less than the defined distance.

- **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 Vector Import utility is stored in a separate file to the `MapInfow.prj` file. If you wish to reproject 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).

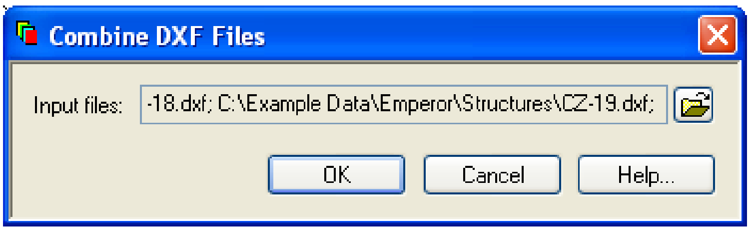
5. To reproject the input file, enable the **Reproject coordinates** option, and specify the input (From) and output (To) projections. See *Coordinate Reprojection* for more information.

6. A range of additional advanced transformation options (described above) are also available via the **Other processing options** button.

Combine DXF Files

The **Combine DXF Files** utility enables multiple 3D .DXF files to be combined to create a single 3D .DXF file.

If there are a number of 3D .DXF files that are generally opened together in Encom Discover PA, to save having to open each of them separately, a single 3D .DXF can be created. The 3D .DXF files must be in the same directory prior to combining. In the **Combine DXF** dialog click on the **Browse** button and browse to the directory containing the 3D DXF files to be combined. Select all the DXF files to combine using the CTRL and SHIFT keys depending on whether they are consecutive in the file list.



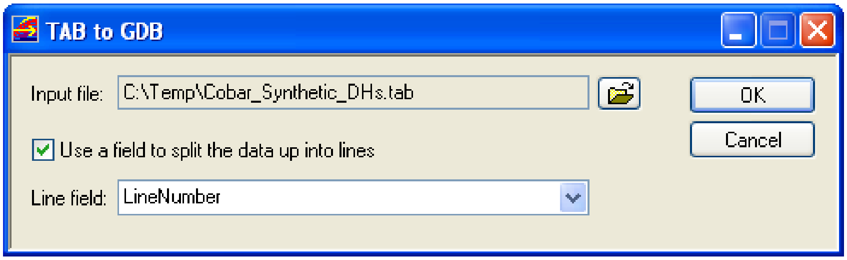
Combine DXF dialog

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 Encom Discover PA map window. Use the **Add Data>3D Vector** (right click on the **3D Map** branch in the Workspace Tree) menu option to display the combined .DXF data.

MapInfo TAB to Geosoft GDB

MapInfo Professional .TAB file formats can be imported directly into Encom Discover PA but only if they contain data columns that record the location (Easting and Northing data fields). These two data fields are mandatory. Elevation data is optionally required if the imported data is to be displayed in 3D.

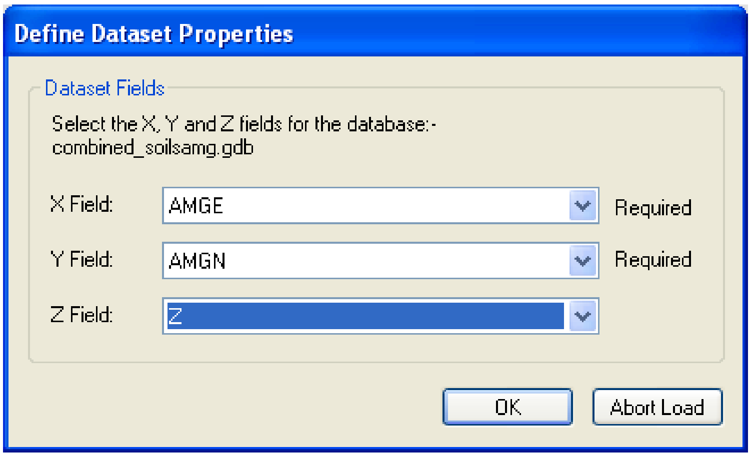
Line information is also optional but required if the data are to be navigated using the line navigation facilities. The menu option **File>Open>MapInfo TAB** allows import of .TAB files.



Conversion of a .TAB file into a Geosoft .GDB format database

An **Open Folder** dialog is displayed from the **Browse** button. Navigate the path to locate the required folder containing one or more .TAB files.

Once Encom Discover PA accesses the file, it displays a dialog to permit you to specify the Easting, Northing (mandatory) and optionally, the Z (elevation) and optionally a Line data field.



Dialog to specify the required and optional .TAB data fields as required

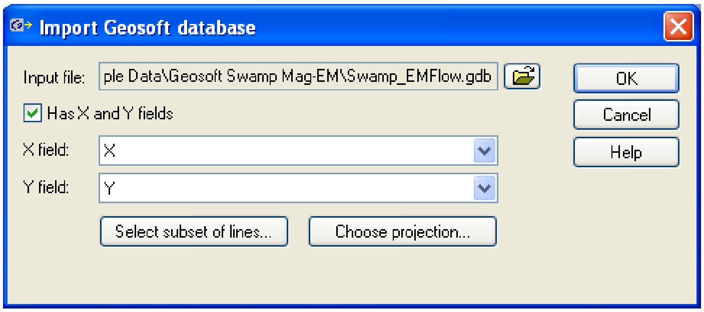
With a MapInfo Professional .TAB file accessed, Encom Discover PA can display the data using its available display objects that support the data available. For example, a .TAB file that does not have a Line definition could not be displayed in a profile display. However, if the .TAB data had located (X, Y) readings representing say a geochemical or gravity survey, they could be displayed as a Point display (refer to *Points*).

The imported data is stored in a native Geosoft .GDB database file. All data converted is as a single data channel in the .GDB database. If the data to be imported has no line identifier it is available as a points database. If however the data in the .TAB file has a Line data channel, select the **Use Line** option and specify the Line data channel.

Geosoft GDB to MapInfo TAB

The Geosoft GDB to MapInfo TAB conversion utility enables you to convert data from a Geosoft Oasis montaj database to a MapInfo Professional table.

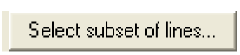
The converted data is stored in native MapInfo Professional .TAB file format. When importing databases which contain multi-channel array data (such as Geotem or EM data) into a .TAB file, each array channel is imported as a single column. Since MapInfo Professional has a limit of 255 columns in a table, any Geosoft databases which contain more than 255 channels are truncated during import. In this situation you are warned that some data loss will occur.



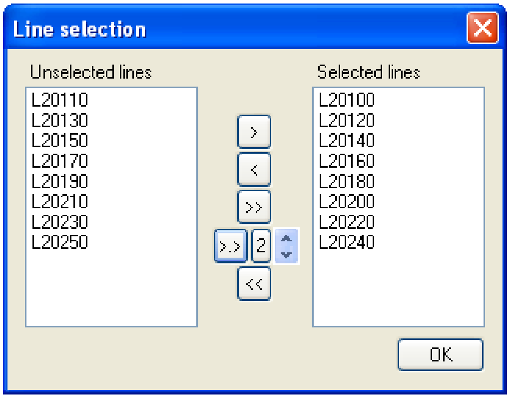
Importation of a Geosoft database to a MapInfo Professional .TAB file

To run the conversion:

1. Click the **Open** button and browse for the database file to import. Geosoft databases have a .GDB file extension.



2. By default Encom Discover PA imports all lines in the Geosoft database; however you can opt to import only a subset of the lines from the database by clicking on the **Select subset of lines** button and selecting the required lines in the selection dialog.



Select lines from Geosoft Database to import

3. Assign an appropriate MapInfo Professional projection for the imported data using the **Choose Projection** button.

4. Select the **Open in map** box to create points for the data and view the data in a map window or clear the box to import the data into a browser only.

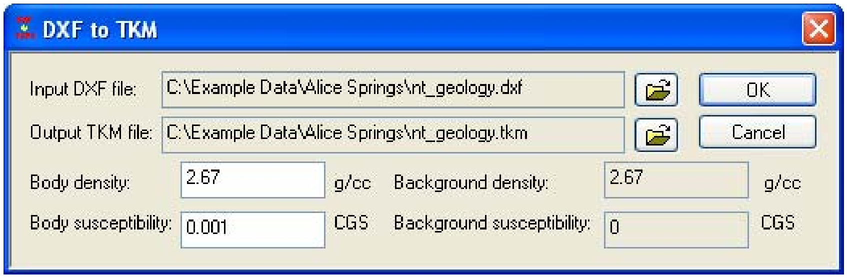
5. Click **OK** to start the import process.

DXF to ModelVision TKM

A conversion utility to convert an AutoCAD .DXF to Encom ModelVision model file of type .TKM is available. For use and specification of the .TKM model file, refer to the ModelVision Reference Manual. This conversion is especially useful when using a created .DXF CAD file and requiring its geophysical response be computed within ModelVision .

To run the conversion:

1. The conversion utility requires only AutoCAD 3D solids to satisfactorily create an output file. The dialog appears as below:



DXF conversion to ModelVision TKM model file

2. Specify the **Input .DXF file**. Use the **Open** button to navigate and specify the required file.

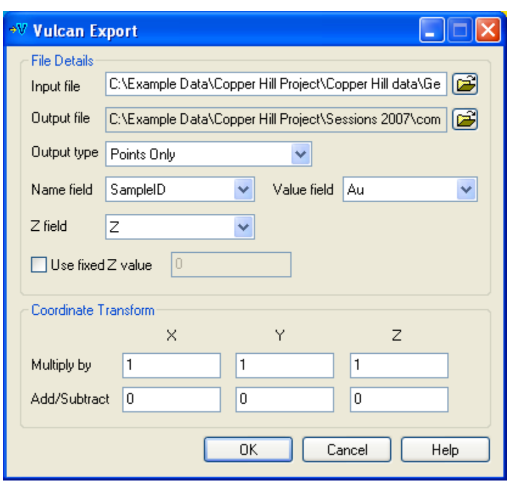
3. Specify the **Output .TKM file** to contain the ModelVision model file.

4. The ModelVision file requires physical properties to be assigned. These can be defined in the **Body density** (in g/cc) and **Body susceptibility** (in CGS units) entry fields. The values entered are relative to the Background properties specified.

5. Click the OK button to create the output .TKM file.

MapInfo TAB to Vulcan Archive

MapInfo Professional files can converted into Vulcan Archive ASCII format so they can be read directly into Vulcan.



MapInfo TAB to Vulcan utility

1. Select the MapInfo Professional table to convert from the **Input Table** pull-down list.

2. Use **Open** button to browse to a directory and assign a name for the **Output file.**

3. Choose the **Output type** from the three available options: **Labels Only**, **Points Only** or **Auto Select**.

4. Select two attribute fields using the **Name field** and **Value field** pull-down lists.

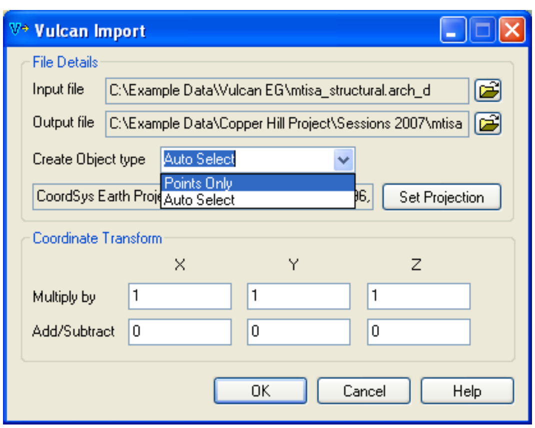
5. Select a column to use for elevation from the **Z field** pull-down list. Alternatively, check the **Use fixed Z value** for all objects in the table or leave the “0” as default.

6. **Coordinate Transform** parameters can be entered for X, Y and Z values. This is particularly useful for converting data from world coordinates to local mine grid coordinates. Enter values to add, multiply or subtract from the existing coordinate values.

7. An ASCII file in Vulcan Archive format is then produced that can be read directly into Vulcan.

Vulcan Archive to MapInfo TAB

The Vulcan Archive Import tool enables you to convert Vulcan Archive ASCII files into MapInfo Professional tables. The data can either be imported as points representing each vertex in the string file, strings (polylines) or as closed strings (polygons).



Vulcan to MapInfo TAB utility

1. Select the **Input Vulcan file** by clicking on the **Open** button.

2. Select the output .TAB file name by clicking on the **Open** button and selecting an appropriate location. By default the program will save the output file with the same name and in the same location as the source file.

3. Select the required object import method from the **Create Object type** pull down list. Two options are available:

   - **Points Only** - will create points for each vertex in the string file. Use this option if you wish to import the string vertices only and preserve the original string file X, Y, Z coordinate information.

   - **Auto Select** - creates either point or polyline objects depending on whether there are single or multiple line entries per block of data.

Note The X, Y coordinate values cannot be imported into the .TAB and the Z value will be an average of the vertex Z values for each string in the original file as MapInfo Professional does not support multiple vertex records per object in a browser.

4. Select an appropriate MapInfo projection for the data.

5. Set the **Coordinate Transform** options to apply a transform to the data during import if required. The transform options allow you to **Add/ Subtract** or **Multiply** the X, Y coordinates in the Vulcan Archive file by constant values during import. This is especially useful if you need to convert the data from local mine grid coordinates to world coordinates.

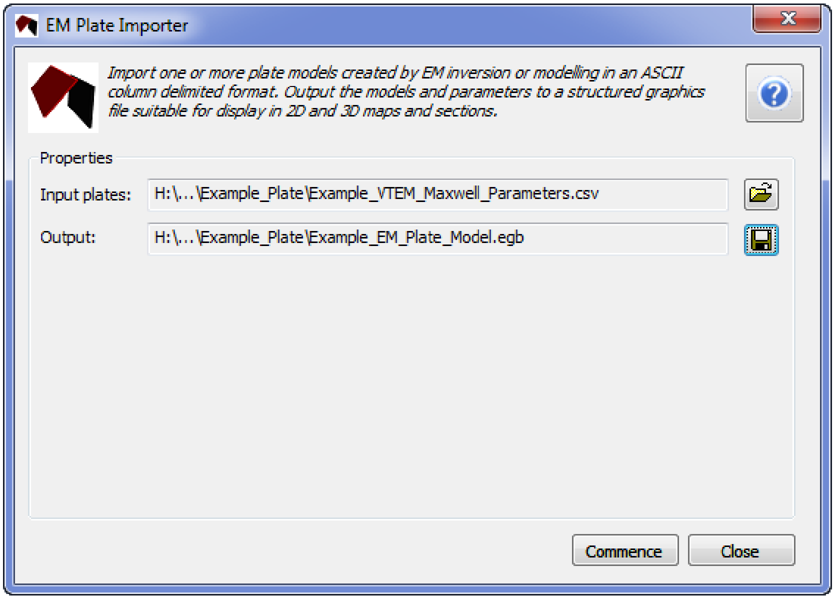
6. Click OK to commence the import process.

EM Plate Importer

Note This tool requires an Encom Discover PA Professional licence. If you wish to purchase or evaluate this module, please contact Pitney Bowes Software.

Encom Discover PA provides a mechanism for importing the output from the Maxwell^{TM} software, developed by Electromagnetic Imaging Technology (EMIT). Maxwell is a desktop application designed for the processing and interpretation of all forms of EM geophysical data: time-domain, frequency-domain, ground-based, airborne, dB/dt and B field. The output from this software is a Maxwell EM modelled plate body in either ASCII table format or DXF format.

To display the table format in Encom Discover PA the EM Plate Importer utility must be utilised to convert the EM plate model to a format supported by Encom Discover PA. To import the EM plate data use the **EM Plate Importer** utility, this is accessed from the Vector menu.



The EM Plate Importer utility dialog.

Input Plates

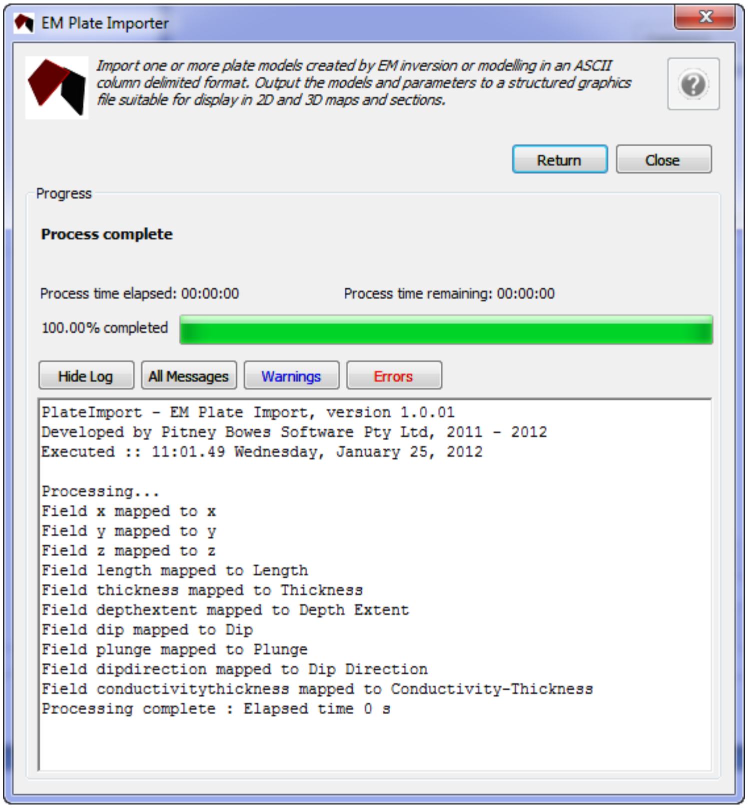
The EM Plate table must be in comma delimited (.CSV) ASCII file format.

Output

The output from the EM plate Importer utility is a Structured Graphics file (.EGB) which is native to Encom Discover PA. If the output location and file name is to be different to the input file press the Save As button and specify a new name and location.

To convert the plate model file press the **Commence** button.

The file is not automatically loaded into Encom Discover PA after it is generated so once the importing process has completed and the utility dialog appears similar to that shown below, press the **Close** button.



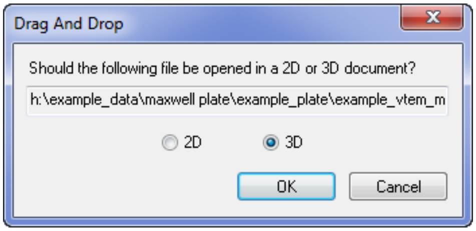
The Log view for the EM Plate Importer utility.

Then proceed to open the structured graphics .EGB file in Encom Discover PA in one of the following methods.

Drag and Drop

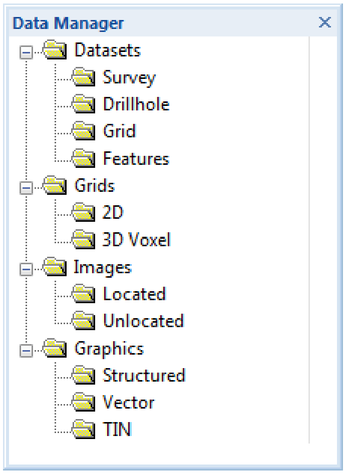
In Windows Explorer or the *Explorer Window* in Encom Discover PA, navigate to the file location as specified in the **Output** field of the EM Plate Importer utility.

Select the .EGB File and holding the left mouse button down drag the file into the display area of Encom Discover PA and release the mouse button. A prompt dialog similar to below will appear asking if you would like to display the .EGB file in a 2D Map or 3D display document.



The drag and drop prompt dialog in Encom Discover PA.

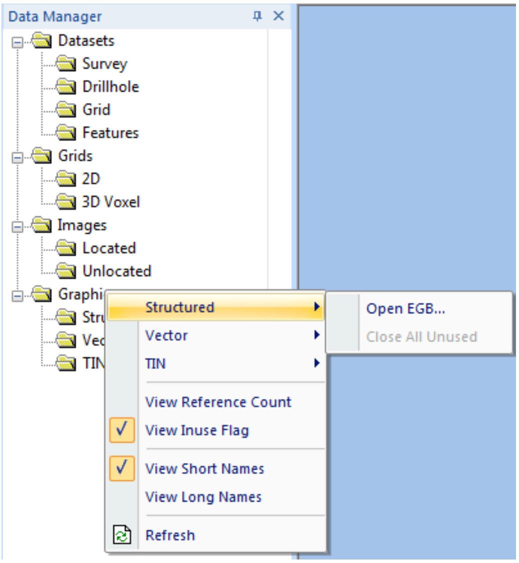
Data Manager Window



An empty Data Manager window.

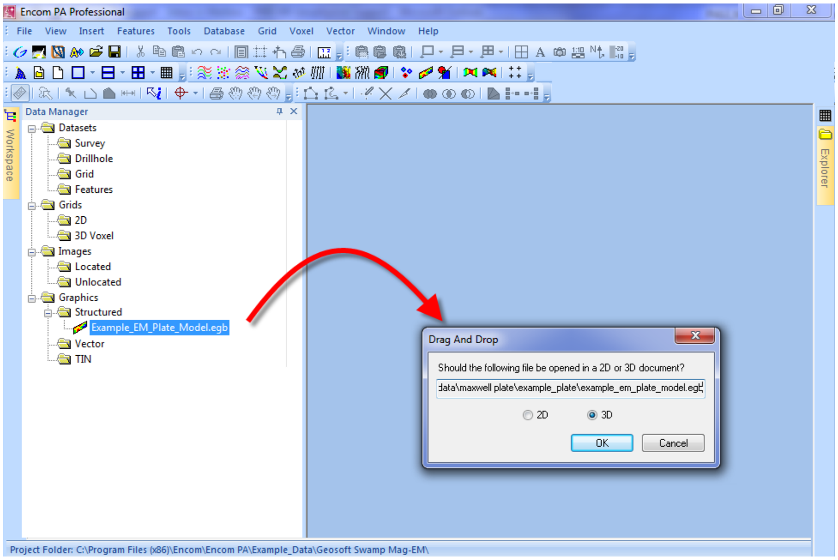
If not already visible in Encom Discover PA, open the Data Manager window by selecting **Data Manager** from the **View** menu.

Position the mouse cursor over the **Graphics** branch and press the right mouse button to view the pop-up menu and select the **Structured>Open .EGB...** option. The .EGB file will then appear in the Data Manager under the Structured Graphics branch.



Opening a structured graphics file in the Data Manager window.

To open this in a 2D Map or 3D display document select the .EGB file in the Data Manager and, while holding the left mouse button, down drag the file into the display area of Encom Discover PA and release the mouse button. A prompt dialog similar to below will appear asking if you would like to display the .EGB file in a 2D Map or 3D display document.

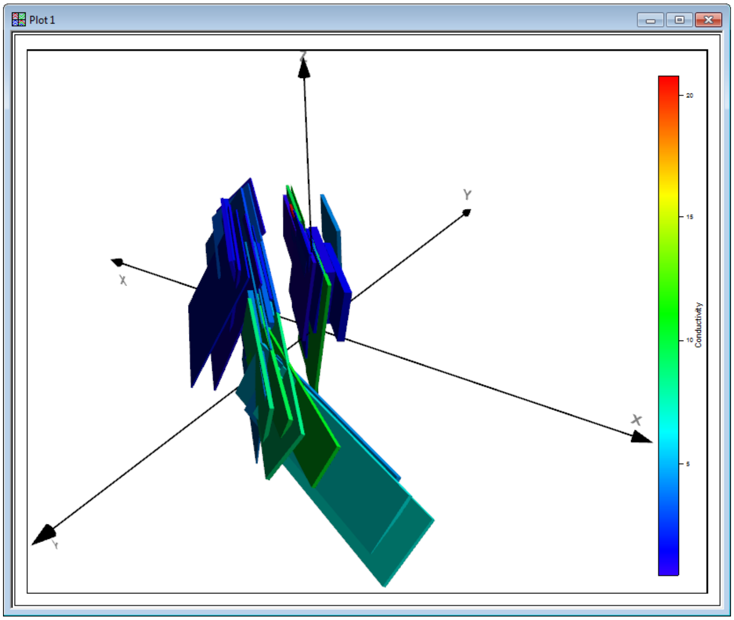


The drag and drop prompt dialog in Encom Discover PA.

To open the EM plate model in a section profile display refer to *Section Profile EM Plate Model Display*.

3D EM Plate Model Display

If the **3D** option in the **Drag and Drop** dialog is selected, the plate model will appear in a new 3D display window, similar to the example shown below.

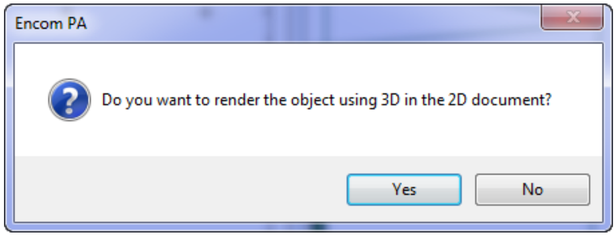


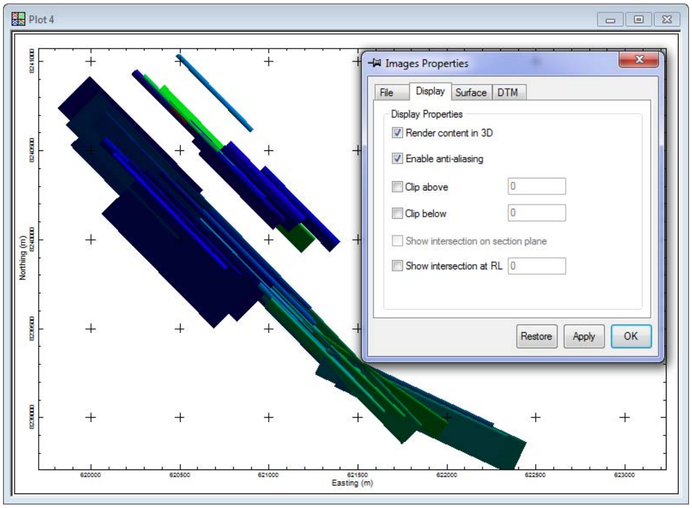
Example Maxwell plate model displayed in 3D as a structured graphic (.EGB).

For more information on editing the appearance of this structured graphic (.EGB) file refer to *Located Images*.

2D Map EM Plate Model Display

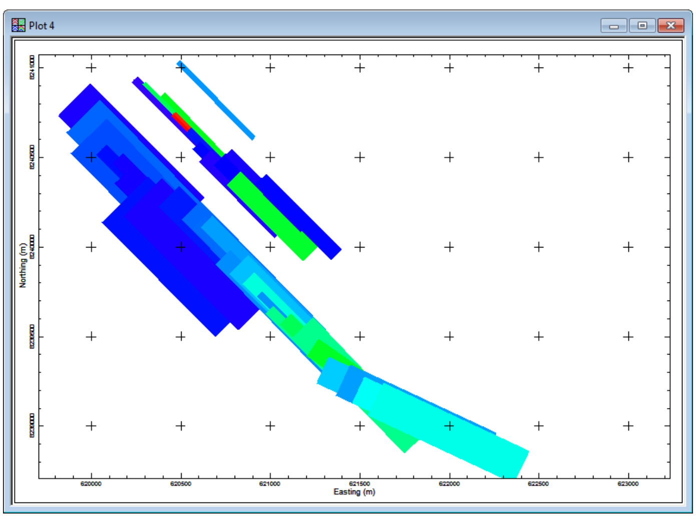
If the **2D** option in the **Drag and Drop** dialog is selected a prompt dialog asking if you want to render the object using 3D in the 2D document will appear. If **Yes** selected, this option will apply lighting to the plate model as if being displayed in a 3D display.





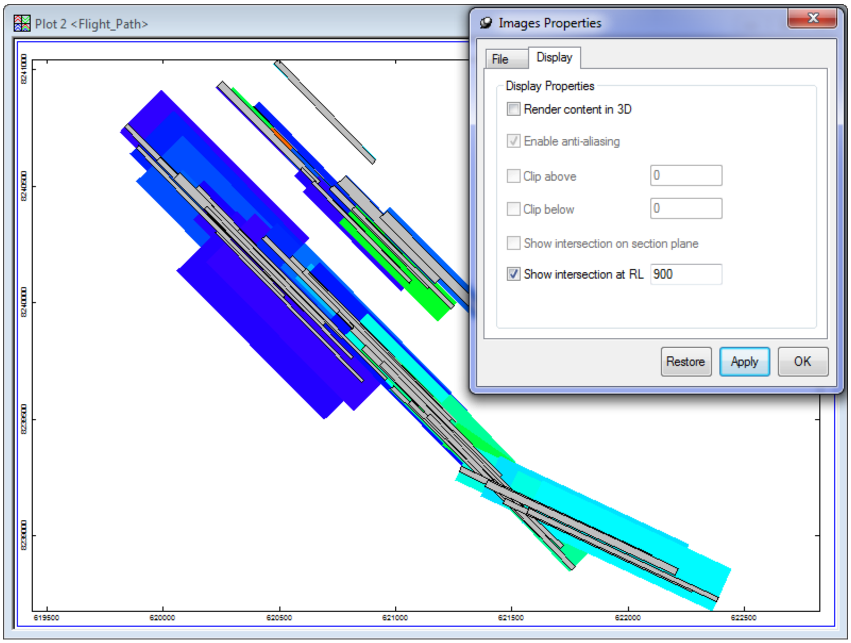
Example Maxwell plate model displayed in 2D with 3D rendering applied.*

If **No** was selected for rendering the model in 3D, the plate model will appear in a new 2D Map window, similar to the example shown below.



Example Maxwell plate model displayed in 2D without 3D rendering applied.

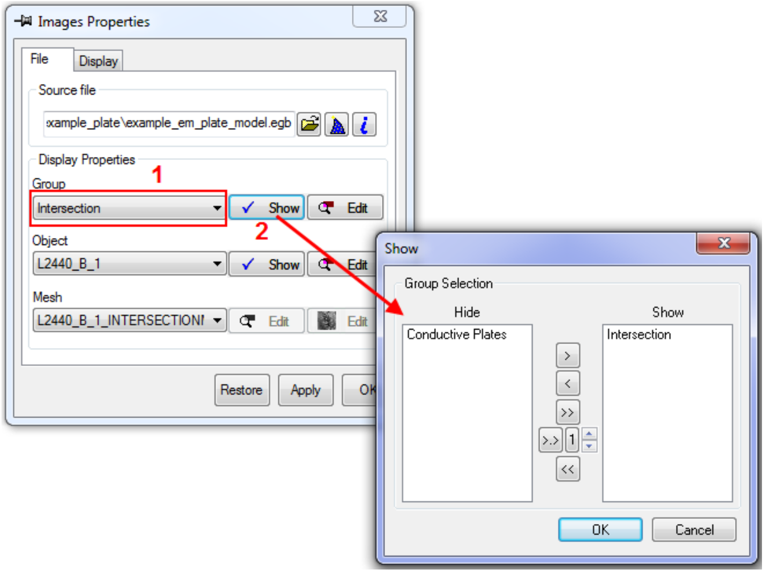
From the images shown above it will be obvious that the full plate models are displayed in 3D, including their dip and depth extent. To show only the top surface of the model at a particular elevation the **Show intersection at RL "n"** option must be enabled in the **Display** tab of the Images Properties dialog.



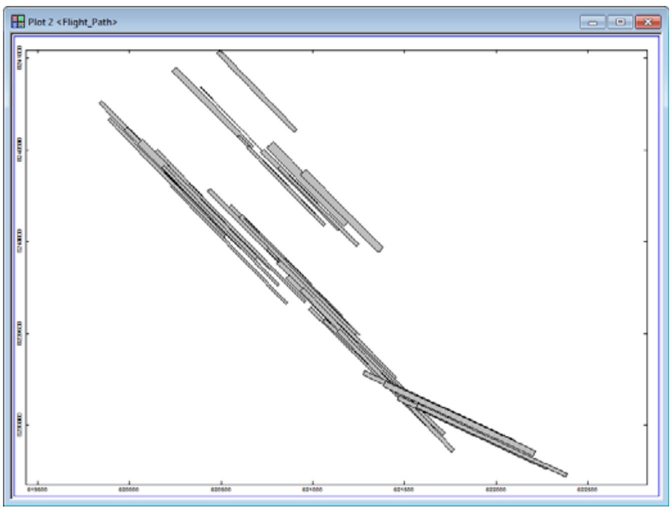
Example Maxwell plate model displayed in 2D with both full plates and Intersection at 900m RL applied.

The result of enabling this option will be that additional grey-coloured rectangles will appear in the map display representing the intersection of the plate bodies at 900m elevation. Select **OK** to close the Properties dialog.

To display only the intersections of the model at the specified elevation the full plates will need to be removed from the display. To do this, view the Properties dialog again and in the **File** tab first select the **Intersection** option from the drop-down list for the **Group** and then select the **Show** button for the **Group** and move the **Conductive Plates** option to the **Hide** list so that only the Intersection group is in the **Show** list. Press **OK** on both dialogs.

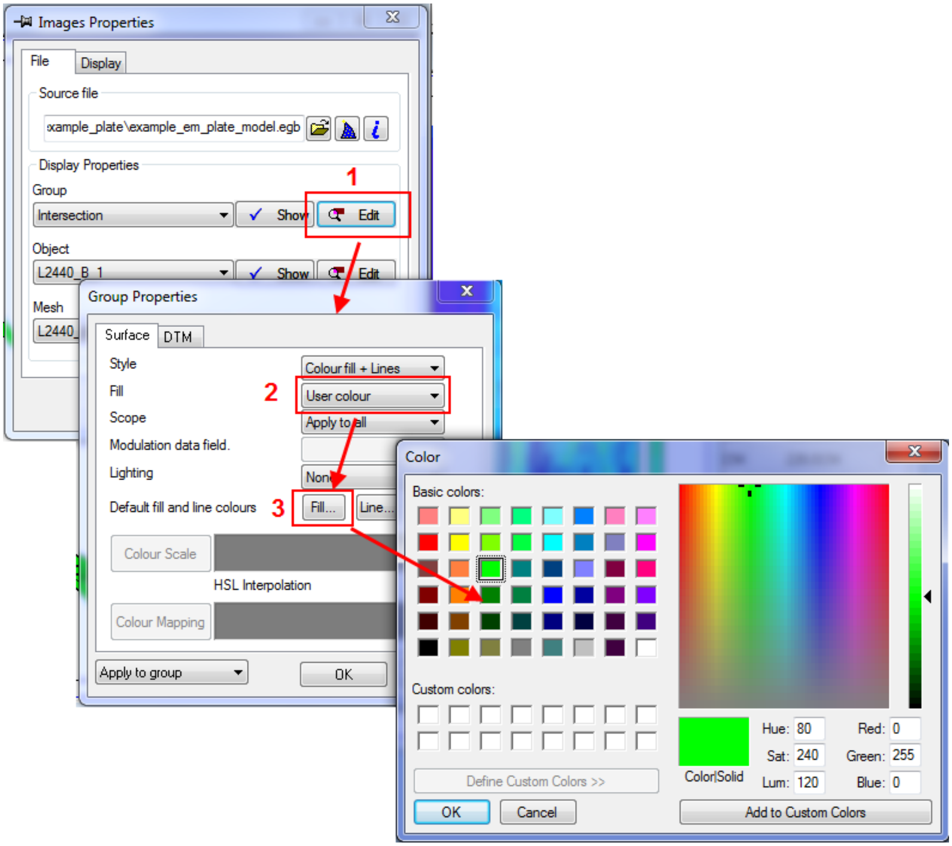


Specifying only Intersection group to be displayed for a 2D map document.



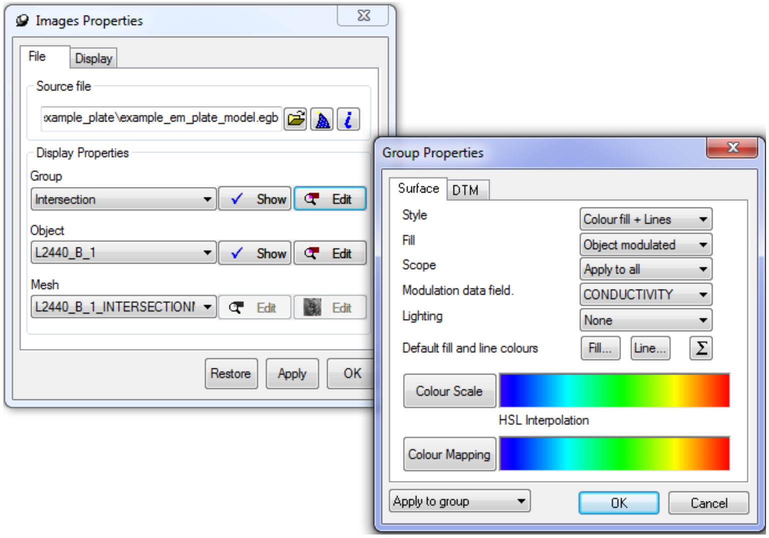
Example Maxwell plate model intersection at 900m displayed in 2D document.

To adjust the colour fill of the plate model intersection (by default this is set as grey) press the **Edit** button for the Group object in the **File** tab dialog, which will display the **Surface** tab of the **Group Properties** dialog. Ensure that “User Colour” is selected for the **Fill**, then press the **Fill** (and **Line**) button next to **Default fill and line colours** to specify a fill colour. Press **OK** three times to apply this change to the map document.

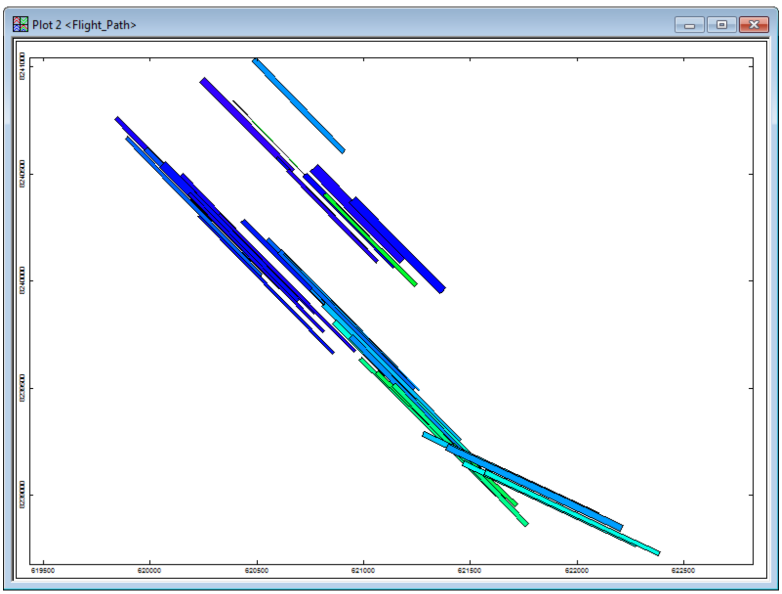


Specifying a fill and line colour for the plate intersection appearance.

To colour modulate the plate intersections by a field within the plate model file, e.g. Conductivity, view the **Group Properties** dialog again (press **Edit** button for Group) and then select **Object modulated** for the **Fill** option in the **Surface** tab dialog and select the desired field from the drop-down list for **Modulation data field**. Press **OK** twice to return to the map document.



Specifying a field to colour modulate the plate intersections by.



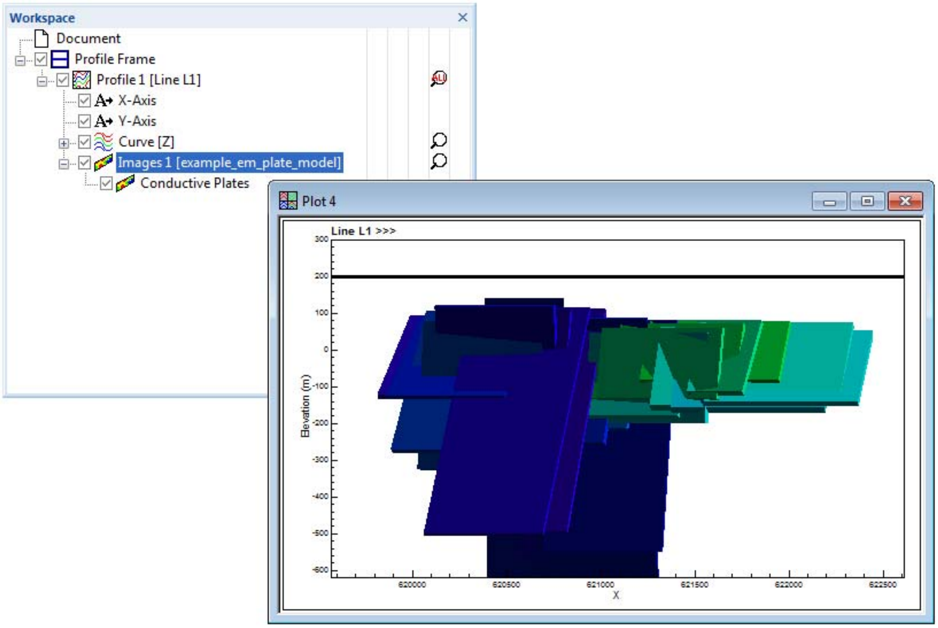
A 2D map display of plate intersections which are colour modulated by the Conductivity field.

Section Profile EM Plate Model Display

To display a EM Plate model in a Profile display a Section Profile or Curve Profile display must be created first and must have a database associated with it for the purpose of displaying lines within the database.

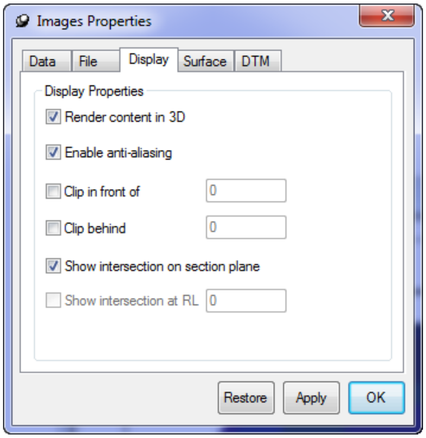
Once a Profile document is displayed to add the EM plate model as a .EGB file to this either "drag and drop" the EM plate model .EGB file from the **Data Manager** to the Profile display window or else right mouse click on the **Profile 1** branch in the workspace tree and select **Add Data>Located Image** from the pop-up menu that appears.

An example Profile display and associated workspace tree is displayed below.



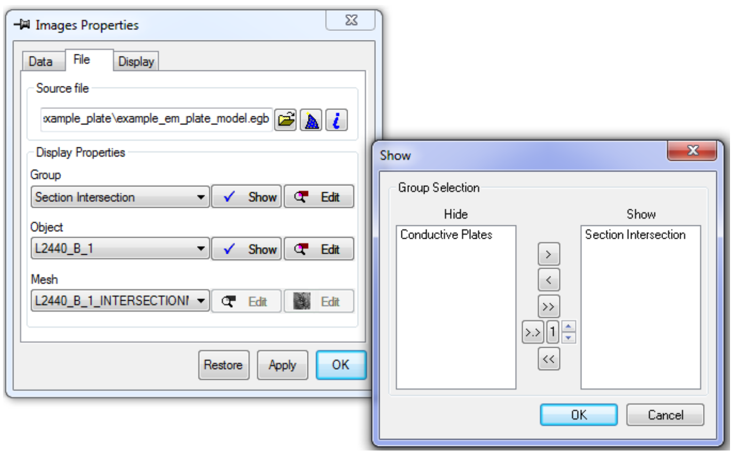
An example Profile display showing a curve (representing elevation) and the conductive plates of an EM plate model .EGB file.

The above image is a representation of the full plate models. To display only the point of intersection for the plates with the current line in the database view the **Display** tab of the **Located Image Properties** dialog and enable the **Show intersection on section plane** option. Press **OK**.



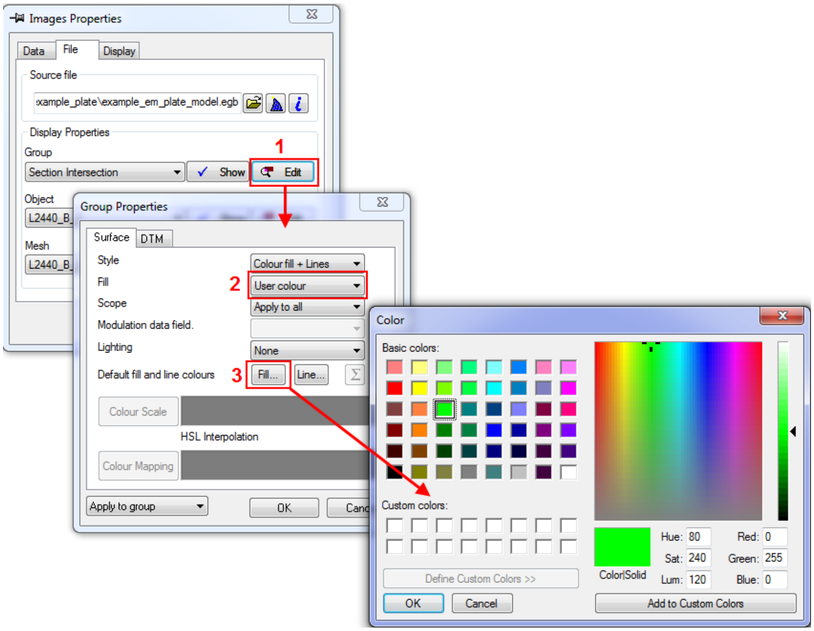
Display Properties for showing the intersection on section plane for a Profile display.

Then open the **Located Image Properties** dialog again and view the **File** tab. Ensure that the **Section Intersection** option is selected in the **Group** drop-down list and then press the **Show** button for the **Group** and move the **Conductive Plates** option to the **Hide** list so that only the **Section Intersection** group is in the **Show** list. Press **OK** on both dialogs.

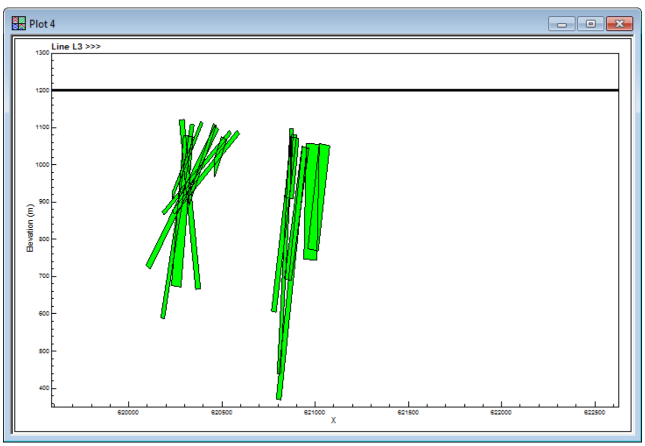


Specifying only Intersection group to be displayed for a profile display document.

To adjust the colour fill of the plate model intersection (by default this is set as grey) press the **Edit** button for the **Group** object in the **File** tab dialog, which will display the **Surface** tab of the **Group Properties** dialog. Ensure that **User Colour** is selected for the **Fill**, then press the **Fill** (and **Line**) button next to **Default fill and line colours** to specify a fill colour. Press **OK** three times to apply this change to the profile display document.

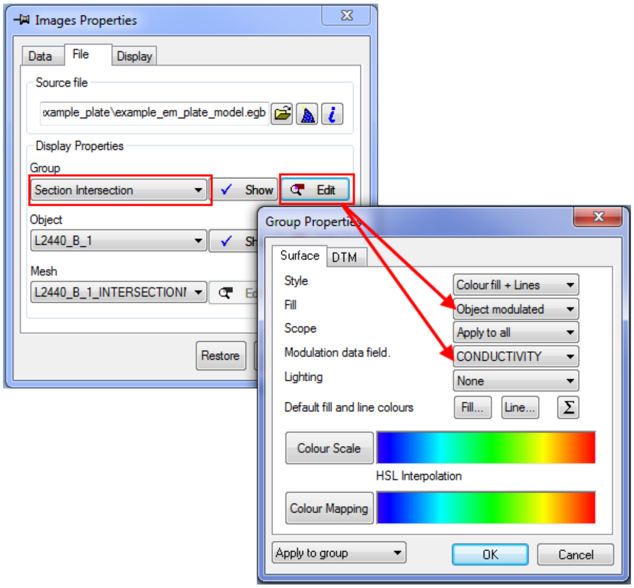


Specifying a fill and line colour for the plate intersection appearance for a Profile display.

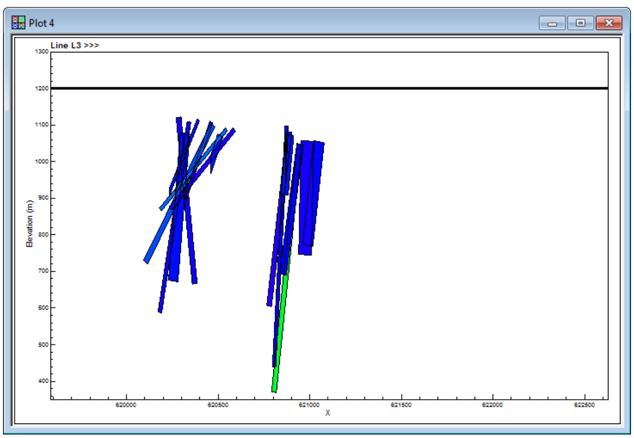


A profile display of plate intersections for a single survey line with a user specified line and fill colour.

To colour modulate the plate intersections by a field within the plate model file, e.g. Conductivity, view the **Group Properties** dialog again (press **Edit** button for Group) and then select **Object modulated** for the **Fill** option in the **Surface** tab dialog and select the desired field from the drop-down list for **Modulation data field**. Press **OK** twice to return to the profile display document.



Specifying a field to colour modulate the plate section intersections by.



A profile display of plate intersections which are colour modulated by the Conductivity field.

To open the EM plate model in a 3D Display display refer to *3D EM Plate Model Display*.

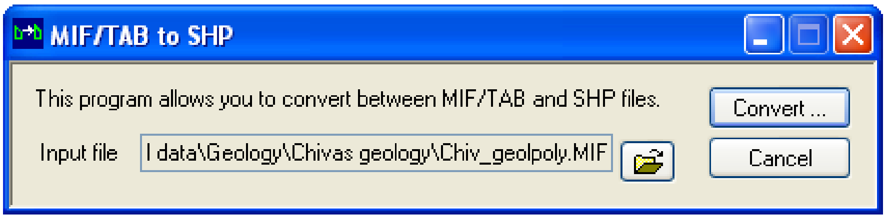
Or to open the EM plate model in a 2D Map display refer to *2D Map EM Plate Model Display*.

MapInfo MIF/TAB to ArcView SHP

MapInfo Professional .MIF or .TAB files can be converted to ArcView .SHP files using this utility. The .SHP file format contains only polygon, polyline or point data and is not by itself capable of retaining attribute data (such as line colouring, linestyles etc).

To run the conversion:

1. The dialog to convert a MapInfo Professional .MIF or .TAB file to an ArcView .SHP file is as below:



The MIF/TAB to .SHP conversion utility dialog.

2. Specify the Input file. This file can be either a MapInfo Professional .MIF, .TAB or an ArcView .SHP file. Use the **Open** button to navigate and specify the file.

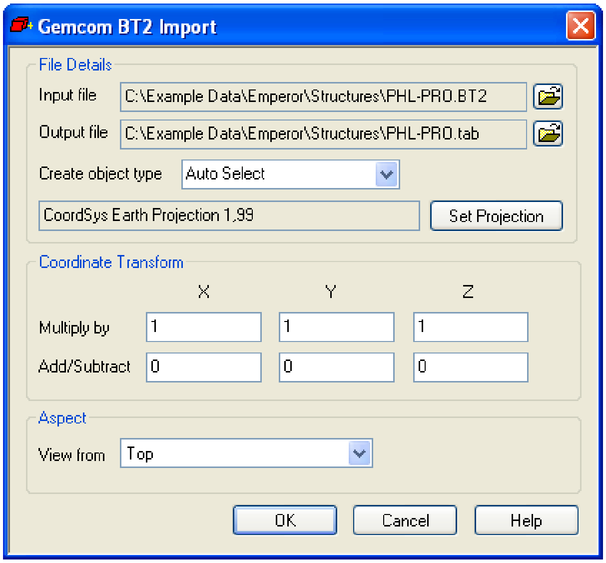
3. Click the **Convert** button and nominate the file type to be output from the File Type pull-down list. Depending on the input type selected, you can convert the output to be either an ArcView .SHP or a MapInfo Professional .MIF or .TAB file.

4. The output data file is created once the OK button of the Open File dialog is clicked.

Gemcom BT2 to MapInfo TAB

The Gemcom .BT2 format is a 3D file format that can be directly loaded and displayed in Encom Discover PA using a 3D Vector object in the Workspace tree. There are also situations that require a .BT2 file to be displayed in a 2D Map and for this, it needs to be imported and seen as a 2D object. For this, a conversion to a Mapinfo Professional .TAB or .MIF file is performed.

Conversion of a .BT2 file to a .TAB or .MIF format is available from the BT2Import utility. The following dialog is displayed:



Gemcom BT2 conversion utility to a Mapinfo Professional .TAB or .MIF file

To run the conversion:

1. Select the **Input file** by clicking on the **Open** button

2. Select the output .TAB or .MIF file name by clicking on the **Open** button and selecting an appropriate location and type. By default the conversion utility saves the **Output file** with the same name and in the same location as the source file.

3. Select the required object import method from the **Create Object type** pull down list. Two options are available:

    - **Points Only** - create points for each vertex in the string file. Use this option to import the string vertices only and preserve the original string file X, Y, Z coordinate information.

    - **Auto Select** - create either point or polyline objects depending on whether there are single or multiple line entries per block of data.

Note The Z value will be an average of the vertex Z values for each string in the original file as MapInfo Professional .TAB format does not support multiple vertex records per object.

4. Select an appropriate projection for the data

5. Set the **Coordinate Transform** options to apply a transform to the data during import if required. The transform options allow you to **Add/ Subtract** or **Multiply** the X, Y coordinates in the .DXF file by constant values during import. This is especially useful if you need to convert the data from local mine grid coordinates to world coordinates.

6. Specify the **Aspect** between a Top, South or North view. Note that only the Top selection is appropriate for a 2D Map display.

7. Click OK to commence the import process.

29 Customising Encom Discover PA

Encom Discover PA allows the customisation of the user interface including:

- Toolbars
- Menu and toolbar command descriptions
- Keyboard shortcuts
- External application tools (using the Edit Menu Options tool).

To open the Customise dialog, select **Tools>Customise** or click on the down arrow located at the end of each toolbar.

The Customise dialog can be accessed by clicking on the arrow at the end of each toolbar

- *Create and Customise Toolbars*
- *Keyboard Shortcuts*

Create and Customise Toolbars

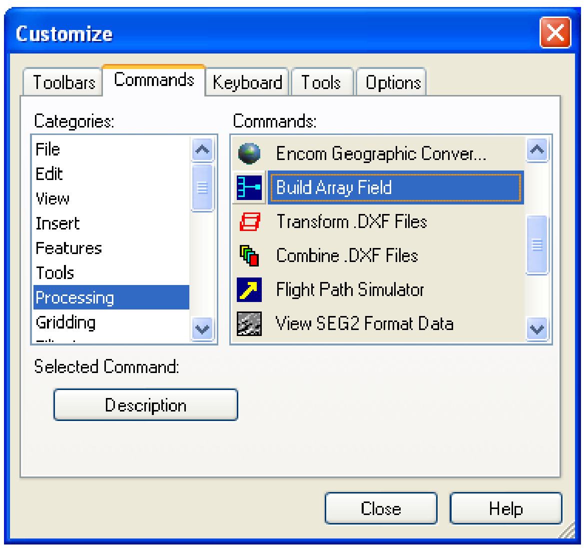
The **Toolbars** tab of the Customise dialog allows the creation of a new toolbar or customisation of an existing toolbar.

To create a new toolbar click the **New** button, enter a name for the toolbar and click OK. This creates a blank floating toolbar on the screen which can be docked in the menu area by clicking and dragging the toolbar. To add commands to a toolbar, highlight the toolbar in the **Toolbar** tab of the Customise dialog and, after selecting the **Category** that contains the command you wish to add, select the command from the **Commands** list and drag it to the toolbar where you wish to add it.

An example of a customised toolbar is shown below:

A toolbar can be customised to display frequently used commands

To remove a command from a toolbar, display the Customise dialog with the specific toolbar name highlighted in the **Toolbars** tab, and click and drag the command away from the toolbar and release it in the display window area.



To add a command to a toolbar, select the command to add and drag it to the desired toolbar

Keyboard Shortcuts

The Customise dialog also allows a user to link existing Encom Discover PA 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 Customise dialog.

To add a keyboard shortcut:

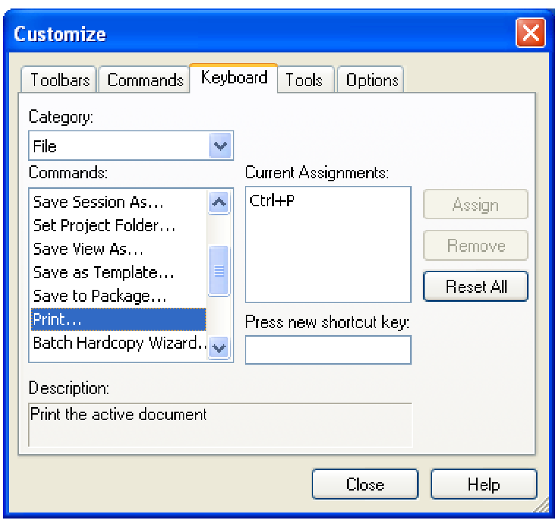
1. Open the Customise dialog and select the **Keyboard** tab
2. Select a **Category** from the pull-down list and select a command from the list displayed.
3. Click the mouse cursor in the **Click New Shortcut Key** box and select the keyboard shortcut (use CTRL and SHIFT keys if desired).
4. Click the **Assign** button to complete the shortcut creation.

To remove a keyboard shortcut:

1. Select if from the **Current Assignments** box
2. Click the **Remove** button.

To remove all custom shortcuts:

- Click **Reset All**.



Keyboards shortcuts and setting the key strokes.

Appendices

A Encom Discover Integration

In this section:

Integrating with Encom Discover

Encom Discover PA can import and display data from the GIS software application Encom Discover

Note Encom Discover is a desktop GIS designed especially for the geosciences, providing tools to effectively compile, visualise, analyse and map spatial geoscience data. Encom Discover runs as an extension to MapInfo Professional, the most widely-used desktop mapping system in the geoscience industry.

Encom Discover can be used to:

- Build geological datasets.
- Compile drillhole cross-sections and plans.
- Produce high-quality scaled maps, with geological symbols and linework.
- Create graphs with maintained spatial link.
- Create, manipulate, contour and profile gridded surfaces.
- Maintain and manage metadata for spatial databases.

Maps, sections and especially drillholes can be displayed in Encom Discover PA in 2D and 3D windows. Encom Discover can be used to create geological maps, geochemistry and present geoscience data in a variety of ways. In particular, Discover can import and manage drillhole data and using this, create interpretive sections. Map data, drillhole projects plus sections can be directly loaded into Encom Discover PA from Discover to provide an integrated approach to geology maps, geophysics and geochemical data. When presented in 3D displays, the integration of various data types plus the powerful 3 dimensional navigation capability of Encom Discover PA provides a tool for understanding your data better than with conventional 2D maps.

2D Map Displays

2D Maps can be displayed in PA by dragging and dropping the TAB files into the PA window. Repeat this for each layer required.

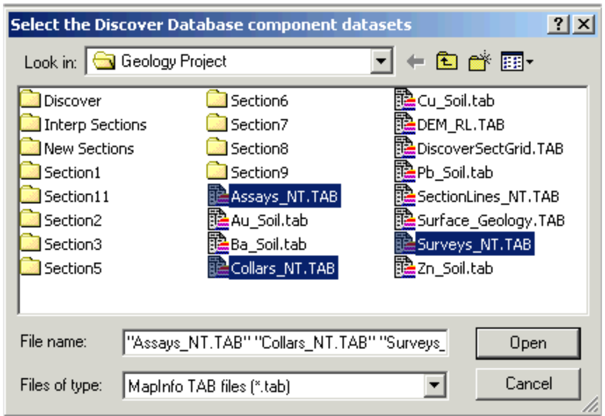
Alternatively if Discover is installed, you can create a draped 3D view of the map winodw. Simply right click in a Map Window in Mapinfo-Discover, and select **View in 3D**. Create a file name by ticking **Save Permanently**. This will automatically open in the Discover3D window, and you will also be able to drag and drop the saved EGB file into PA.

Encom Discover Drillhole Data

Encom Discover has a fully featured drillhole display facility for managing drilling projects, displaying sections and maps and analyzing drillhole data. Refer to the Discover documentation for information on the import, use and display of these data types. For use in Encom Discover PA, a standard Encom Discover project must have three file types:

- **Collar table** – With hole ID, location, elevation, azimuth, dip and total drillhole depth.
- **Survey table** – Drillhole survey data with hole ID, survey position distance down the axis of the hole, azimuth, dip and computed, actual XYZ location.
- **Data table** – Data recorded in the drillhole. The hole ID, sample location and data channels are present. The data location can be in either Point reading or From-To sampling.

To access these in Encom Discover PA, select the **File>Open>Downhole>Discover Downhole** menu option. Navigate to the folder containing the Encom Discover project and open the three files corresponding to the Collar, Survey and Data types. Use the CTRL key to multiply select the required TAB files.



Opening a Collar, Survey and Data .TAB files

The minimal required files are a Collar and Survey TAB file. You can open a number of data TAB files related to the drillholes defined by the Collar and Survey TAB files if required. Once read by Encom Discover PA, the Spreadsheet can be used to view the contents of the files (refer to *Spreadsheets*) for the various drillholes.

To assist in constructing an Encom Discover Drillhole Project and importing .CSV or Microsoft Excel data drillhole data files, refer to *Encom Discover Drillhole Data*.

With the drillholes and data available, both 2D and 3D maps can be used to display them. For information on adding drillholes to 2D and 3D maps and presenting them with data, refer to Adding Drillholes to a Display, Display of Drillholes in Maps, Drillhole Display Properties.

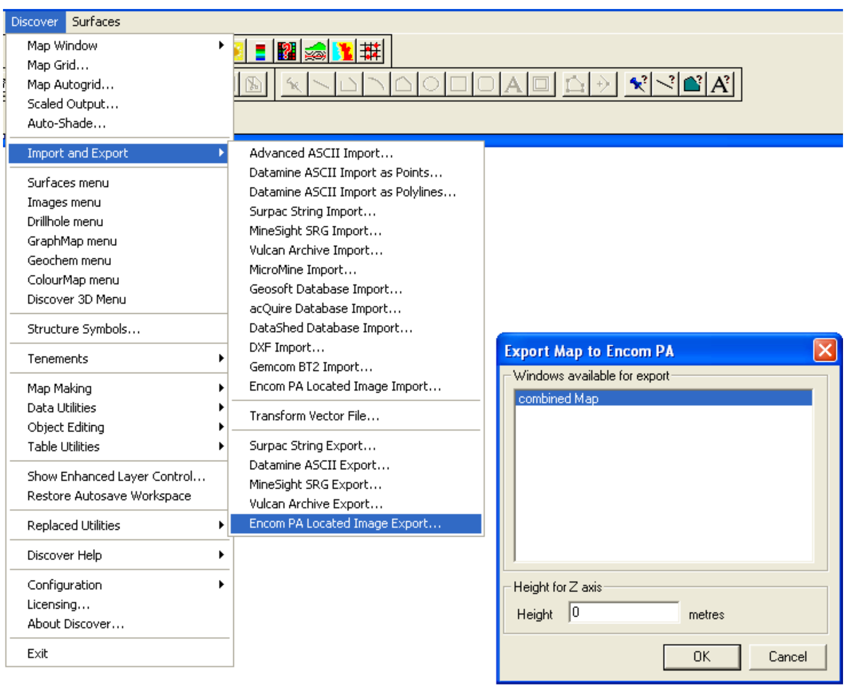
Displaying Sections From Discover in Encom

Discover PA

Encom Discover provides the ability to create interpreted geological sections. These sections often form part of Discover drillhole projects. Refer to the Discover documentation for details on creating sections.

Displayed Discover sections can be transferred to Encom Discover PA for display using Located Images. The method used for this transfer is by Discover copying the section window contents to a Located Images plus a second file called an Encom Georeferenced Bitmap file (.EGB). The Located Image format is described in *Appendix B: Data File Specifications*. The content of the .EGB file allows Encom Discover PA to accurately position the four corners of the captured bitmap.

To capture and locate an image section (or map) displayed with Discover, select the **Drillholes>Import/Export>Export Located Image to Encom Discover PA** menu option. When chosen, a dialog is presented to nominate the windows for export.



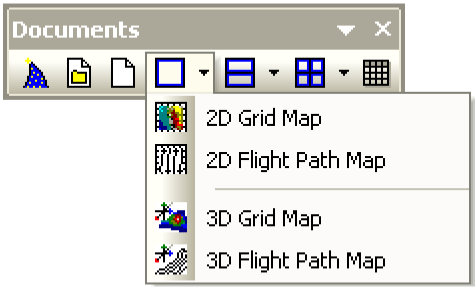
Discover menu item to capture a section for display in Encom Discover PA

You can choose to capture all the available section windows, only the selected section window and you can include a drillhole collar map to locate the holes. When accepted, you are prompted to specify a folder and filename for the .EGB file to be created.

Follow these steps to display the located images in Encom Discover PA:

Step 1

Create a 3D Flight Path Map or 3D Grid Map using the **File>New>Map>3D Grid Map** or **3D Flight Path Map** menu option (or from the appropriate buttons).



Step 2

With a Map plot created, select the Map branch of the Workspace tree and click the right mouse button. From the **Add Data** menu item, select **Add Grid**. This option creates a 3D map display as described in the *About Three-Dimensional Displays*. Manipulate the surface as required with suitable vertical exaggeration and offset. Note that the data values of the displayed grid determine the vertical position of your display and this is important later when displaying bitmaps with a different vertical location.

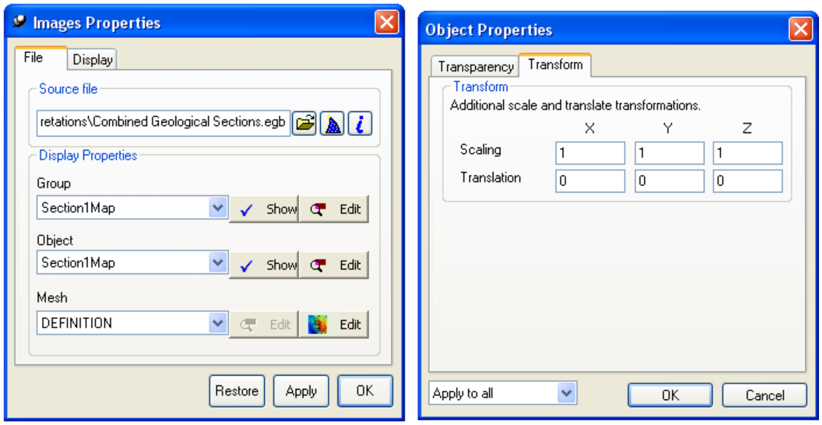
For example, if you wish to display a magnetic grid (with average data values of say, 55,000nT) combined with geological sections (with vertical reference of say 1-200 metres) as shown above, then an offset is required to bring the two data objects into a similar vertical reference frame. Typically therefore, you would need to reduce the vertical location of the magnetic grid down by about 55,000. The steps following describe this procedure.

Step 3

Select the 3D Map branch of the Workspace tree and click right. Select the **Add Data>Located Image** option. An Images branch is added to the tree associated with the 3D Map.

Step 4

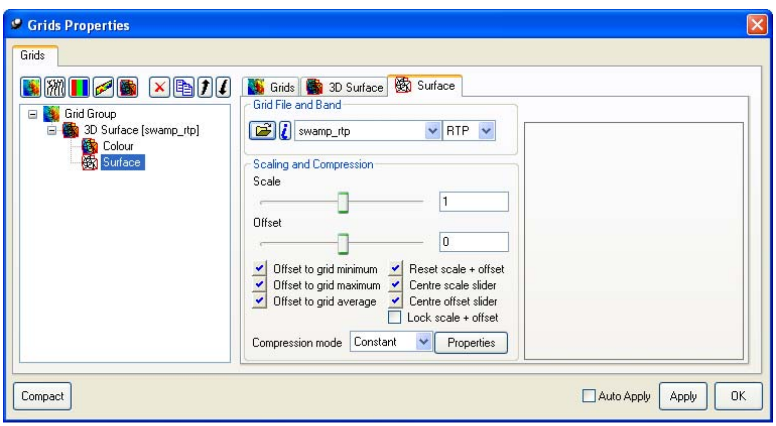
Display the properties dialog of the added Images branch. Initially specify the path and Located Image created from Encom Discover. You can use the Browse button to navigate to the required .EGB file.



Located Image properties dialog to specify the EGB file and display options

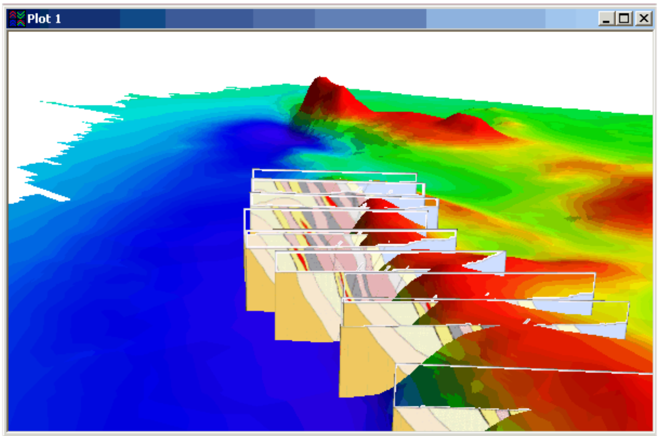
Step 5

If the displayed map surface and image are widely separated, their vertical offset may require adjustment. This can be done either by editing the vertical location in the .EGB file (to offset the located image), or by interactively setting an offset amount in the surface display control. To do this, display the Surfaces Properties dialog and highlight the Surface branch of the Grid Group tree. At the base of the dialog are controls to scale and offset the grid surface. Set this to an appropriate value by using the slider and increment entries.



Using the Surface dialog to control the Vertical scaling and offset of a grid image

If additional bitmap sections are to be displayed in the one 3D view, repeat steps 3-4. An example of multiple located images displayed with a geophysical surface is shown below:



Magnetics surface with multiple Located Image depth sections in a 3D display

B Data File Specifications

In this section:

To view a full list of file types supported by Encom Discover PA, see *Appendix E: Supported File Formats*.

ASEG-GDF2 Format Data

This ASCII data format uses a self-defining external data definition file (usually with filename extension of .DFN) to describe a separate ASCII data file. A .DFN appears as below:

```
DEFN   ST=RECD,RT=COMM;RT:A4;COMMENTS:A76
DEFN 0 ST=RECD,RT=;FLTLINE:A10:NAME=Line
DEFN 1 ST=RECD,RT=;X:F13.1:NULL=-1E30:NAME=X
DEFN 2 ST=RECD,RT=;Y:F13.1:NULL=-1E30:NAME=Y
DEFN 3 ST=RECD,RT=;MAG:F13.1:NULL=-1E30:NAME=MAG
DEFN 4 ST=RECD,RT=;GRAV:F13.5:NULL=-1E30:NAME=GRAV
```

The above .DFN file specifies data in its accompanying ASCII data file as follows:

```
1600     200.0    53800.0      56481.0     -0.01000
1600     220.0    53810.0      56479.5        -1E30
1600     240.0    53820.0      56478.4     -0.00300
1600     260.0    53830.0      56477.5     -0.00100
1600     280.0    53840.0      56477.0      0.00000
1600     300.0    53850.0      56476.8      0.00000
1600     320.0    53860.0      56476.7     -0.00100
1600     340.0    53870.0      56476.7     -0.00300
1600     360.0    53880.0      56476.7     -0.00500
```

A full format specification of the ASEG-GDF2 is available from *www.aseg.org.au*.

ASCII XYZ Voxel 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. cells 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.

An example is repesented below:

X Y Z susceptibility

```
-287.5 -287.5 -387.5 0.00012721899972
-262.5 -287.5 -387.5 0.00010271179969
-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 2.435779397e-005
 -87.5 -287.5 -387.5 2.702029997e-005
 -62.5 -287.5 -387.5 2.904669964e-005
```

Datamine Block Model File

The Datamine block model file format uses a standard definition as specified in the table below.

Block Model File

Field	Numeric/ Character	Implicit/ Explicit	Description
XMORIG YMORIG ZMORIG	N	I	(X,Y,Z) Origin of the model. DATAMINE sets the origin with respect to the corner of the first parent cell and NOT its centroid.
XINC YINC ZINC	N	I/ E	XYZ cell dimensions. If the model is not to contain any subcells then these three fields can be implicit (not stored on every record). This will reduce the storage space required by the model.
NX NY NZ	N	I	Number of model parent cells in XYZ. DATAMINE allows a value of one for modelling seams. The number of cells, in combination with the cell dimensions, defines the extent of the model.
XC YC ZC	N	E	(X,Y,Z) coordinates of the cell centre.

Field	Numeric/ Character	Implicit/ Explicit	Description
IJK	N	E	Code generated and used by DATAMINE to uniquely identify each parent cell position within the model. Subcells that lie within the same parent cell will have the same IJK value.

In addition to the above, Block Model Files normally contain one or more value fields. Block Model files are normally sorted on increasing IJK values.

CEMI Model Format

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 the 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 2DQA 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.

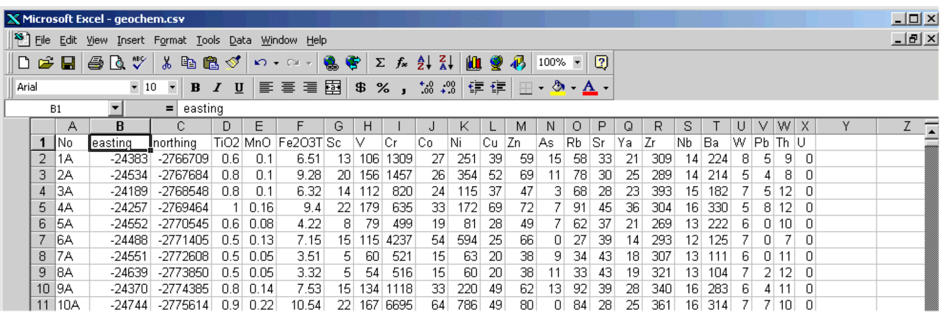
CSV (Comma Separated Variable) Files

CSV files are primarily used as a transfer format for feature information. CSV format files can be imported and exported from Encom Discover PA and used with external programs such as Microsoft Access and Excel. The format has an ASCII structure with each data value being separated by a comma in variable width columns along individual records. Within the first record is a single data line that describes the column names assigned to the following data records.

An example of a geochemistry feature CSV file follows:

```
No,easting,northing,TiO2,MnO,Fe2O3T,Sc,V,Cr,Co,Ni,Cu,Zn,As,Rb,Sr,Ya,
Zr,Nb,Ba,W,Pb,Th,U
1A,-24383,-2766709,0.64,0.1,6.51,13,106,1309,27,251,39,59,15,58,33,2
1,309,14,224,8,5,9,0
2A,-24534,-2767684,0.77,0.1,9.28,20,156,1457,26,354,52,69,11,78,30,2
5,289,14,214,5,4,8,0
3A,-24189,-2768548,0.75,0.1,6.32,14,112,820,24,115,37,47,3,68,28,23,
393,15,182,7,5,12,0
4A,-24257,-2769464,1,0.16,9.4,22,179,635,33,172,69,72,7,91,45,36,304
,16,330,5,8,12,0
5A,-24552,-2770545,0.56,0.08,4.22,8,79,499,19,81,28,49,7,62,37,21,26
9,13,222,6,0,10,0
6A,-24488,-2771405,0.54,0.13,7.15,15,115,4237,54,594,25,66,0,27,39,1
4,293,12,125,7,0,7,0
7A,-24551,-2772608,0.53,0.05,3.51,5,60,521,15,63,20,38,9,34,43,18,30
7,13,111,6,0,11,0
8A,-24639,-2773850,0.52,0.05,3.32,5,54,516,15,60,20,38,11,33,43,19,3
21,13,104,7,2,12,0
9A,-24370,-2774385,0.8,0.14,7.53,15,134,1118,33,220,49,62,13,92,39,2
8,340,16,283,6,4,11,0
10A,-24744,-2775614,0.9,0.22,10.54,22,167,6695,64,786,49,80,0,84,28,
25,361,16,314,7,7,10,0
```

Within Excel, the same data appears as:



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X
1	No	easting	northing	TiO2	MnO	Fe2O3T	Sc	V	Cr	Co	Ni	Cu	Zn	As	Rb	Sr	Ya	Zr	Nb	Ba	W	Pb	Th	U
2	1A	-24383	-2766709	0.6	0.1	6.51	13	106	1309	27	251	39	59	15	58	33	21	309	14	224	8	5	9	0
3	2A	-24534	-2767684	0.8	0.1	9.28	20	156	1457	26	354	52	69	11	78	30	25	289	14	214	5	4	8	0
4	3A	-24189	-2768548	0.8	0.1	6.32	14	112	820	24	115	37	47	3	68	28	23	393	15	182	7	5	12	0
5	4A	-24257	-2769464	1	0.16	9.4	22	179	635	33	172	69	72	7	91	45	36	304	16	330	5	8	12	0
6	5A	-24552	-2770545	0.6	0.08	4.22	8	79	499	19	81	28	49	7	62	37	21	269	13	222	6	0	10	0
7	6A	-24488	-2771405	0.5	0.13	7.15	15	115	4237	54	594	25	66	0	27	39	14	293	12	125	7	0	7	0
8	7A	-24551	-2772608	0.5	0.05	3.51	5	60	521	15	63	20	38	9	34	43	18	307	13	111	6	0	11	0
9	8A	-24639	-2773850	0.5	0.05	3.32	5	54	516	15	60	20	38	11	33	43	19	321	13	104	7	2	12	0
10	9A	-24370	-2774385	0.8	0.14	7.53	15	134	1118	33	220	49	62	13	92	39	28	340	16	283	6	4	11	0
11	10A	-24744	-2775614	0.9	0.22	10.54	22	167	6695	64	786	49	80	0	84	28	25	361	16	314	7	7	10	0

Example of CSV file imported directly into Excel

For drillhole .CSV files, refer to *Importing Drillhole Data*.

Decay Curve Ancilliary Data Files

Data used in Encom Discover PA that is derived from banded databases (such as Intrepid and Oasis montajTM), uses an external ASCII data file (usually called ATEM.TXT). This file is used to define the EM system geometry, transmitted waveforms and units used. Each set of entries is arranged relative to flights and provides for different calibration or waveforms associated with each.

An example of an ancilliary data file is provided here.

```
--------------------------------
Begin survey

XChannelCenter[18] = {78.0625, 103.3875, 128.7125, 154.0375,
228.7499, 341.450, 454.15, 566.85, 735.90, 1017.76, 1412.10, 1919.25,
2539.1, 3271.65, 4116.9, 5074.85, 6145.5, 7328.85} # microSeconds.
End survey
--------------------------------

Begin survey

ZChannelCenter[20] = {468.7500, 781.2500,1093.7500, 1406.2500,
4609.3750,
4765.6250, 4921.8750, 5156.2500, 5468.7500,
5859.3750, 6328.1250, 6875.0000, 7578.1250,
8437.5000, 9453.1250, 10625.0000, 12031.2500,
13750.0000, 15937.5000, 18593.7500} # microSeconds.

ZChannelWidth[20] = {312.5000, 312.5000, 312.5000, 312.5000,
156.2500, 156.2500,
156.2500, 312.5000, 312.5000, 468.7500,
468.7500, 625.0000, 781.2500, 937.5000,
1093.7500, 1250.0000, 1562.5000, 1875.0000,
2500.0000, 2812.5000} # microSeconds.

XWaveform[128] = {27560, 1208839, 1748987,
1696193, 1612784, 1507101, 1380841, 1236006,
1075005, 900210, 714267, 520125, 320526,
118576, -82920, -280937, -472693, -655550,
-826972, -984723, -1126681, -1251043,
-1356134, -1440744, -1503772, -1544610,
-1561507, -804875, 31103, -13322, -11322,
-9755, -8565, -7617, -6873, -6172, -5558,
-5154, -4683, -4323, -3976, -3722, -3444,
```

```
-3193, -2988, -2799, -2626, -2450, -2344,
-2171, -2098, -1995, -1871, -1759, -1691,
-1663, -1551, -1464, -1372, -1372, -1277,
-1249, -1145, -1123, -1084, -1075, -994,
-967, -901, -896, -830, -856, -766, -766,
-775, -735, -701, -679, -688, -687, -649,
-617, -590, -594, -592, -590, -571, -534,
-546, -492, -520, -511, -485, -464, -444,
-448, -427, -413, -444, -428, -411, -417,
-399, -382, -369, -344, -383, -371, -344,
-369, -335, -325, -327, -334, -318, -302,
-305, -344, -307, -304, -278, -279, -279,
-291, -277, -287, -252, -261} # microVolts

ZWaveform[128] = {27627, 1079321, 1603609,
1568275, 1492445, 1395440, 1279329, 1145986,
997693, 836572, 665098, 485927, 301680,
115151, -71116, -254151, -431492, -600741,
-759477, -905659, -1037269, -1152628,
-1250328, -1329052, -1387937, -1426296,
-1441915, -763195, 6733, -12169, -10708,
-9269, -8131, -7180, -6431, -5780, -5211,
-4748, -4364, -3972, -3676, -3390, -3122,
-2898, -2719, -2528, -2363, -2210, -2062,
-1947, -1862, -1719, -1609, -1556, -1481,
-1357, -1368, -1261, -1174, -1162, -1091,
-1045, -1021, -983, -921, -904, -848, -842,
-815, -762, -765, -702, -693, -683, -632,
-650, -610, -581, -603, -576, -539, -545,
-552, -496, -505, -504, -506, -447, -489,
-444, -436, -447, -398, -403, -396, -386,
-378, -376, -375, -355, -349, -364, -327,
-319, -334, -317, -351, -295, -277, -309,
-297, -276, -275, -286, -274, -269, -258,
-290, -232, -279, -264, -250, -253, -251,
-239, -241, -215, -255} # microVolts

End survey
-------------------------------------------
```

EGB Files – Located Image Descriptor Files

These ASCII files are used to define the extent, location and geographical projection of Located Images to be used for display within Encom Discover PA. The image formats available for display include BMPs, TIFs, PNGs, JPGs. The files can be derived from any other application and may be created directly by the application or by a screen capture program. Any colour depth and resolution is suitable.

EGB Located Image Header File

Information is defined at various levels of scope. The highest level of scope is file scope where information tags are defined outside of any block structures.

Comments can be defined at file scope. A comment must be preceded by one or more of the following characters `~!#*/\`.

If a GeoreferenceImage block is defined outside of a GeoreferenceImageGroup block then a group will be created to house the object.

The Include command allows you to include other EGB files and so access the definitions in those files. For example, to build a forest of trees you may have 10 different tree species each defined in separate EGB files. These can be included in the EGB file that defines the forest group and objects. Note that the image ID and object ID tags in all included files have to be unique. If not an error will be generated.

The ImageDefinition block can contain any of the following information tags.

```
ID                = STRING // New format
Image             = STRING
ImageFormat       = <Identifier>
TransparentColour = <Identifier>,INT
ImageStretch      = <Identifier>,<Identifier>
ImageDimension    = <Identifier>,REAL,REAL
```

Note that the ID cannot be specified in the old format. Also, the Image tag must be ought to be specified prior to all other image tags in the old format. If you want to include any of the image processing tags in the old format then either the Image or ImageFormat tags must be specified first.

The ID is a unique identifier that you will use to refer to this image in object definitions. If it is not specified then a unique identifier will be generated. If the ID is not unique the system will report an error.

The Image is the filename of the image. The image file name can be a local, relative or complete path name. If it is relative it will be relative to the location of the EGB file. If no image file is specified one will be constructed from the EGB file path and name and the image format. The following examples are all valid –

```
C:\TEMP\TEST.EGB
""                             (c:\temp\test.bmp)
Picture.bmp                    (c:\temp\picture.bmp)
\IP\Picture.bmp                (c:\temp\ip\picture.bmp)
\IP\                           (c:\temp\ip\temp.bmp)
c:\program files\ip\picture.bmp  (c:\program files\ip\picture.bmp)
```

The ImageFormat specifies the file format of the image. The following image formats are supported. Use the text enumeration to identify the format.

```
Windows Bitmap             BMP or BITMAP
Jpeg                       JPG or JPEG
Portable Network Graphics  PNG
TIFF                       TIFF or TIF
GIF                        GIF
GeoTiff (located)          GEOTIFF or GEOTIF
TAB (raster located)       TAB
```

The TransparentColour defines a colour (or colour range) that will be considered fully transparent. When the image is rendered any pixels with a matching colour will not be displayed. The first variable – Defined/Undefined - activates the transparent colour. The second variable specifies the colour. You can use the identifiers WHITE and BLACK or specify a TBGR colour as a 4 byte integer.

In 2D and in 3D the image needs to be stretched onto the target geometry. In 2D this requires only one operation. In 3D it requires two operations – stretch then interpolation. The ImageStretch command specifies the 2D/3D stretch mode and the 3D interpolation mode. The stretch mode can be either –

```
BLACKONWHITE or PRESERVE_BLACK
WHITEONBLACK or PRESERVE_WHITE
COLORONCOLOR or PRESERVE_COLOUR
HALFTONE or HIGH_QUALITY
```

In 2D, HALFTONE is usually the best choice as it interpolates smoothly.

The interpolation mode (3D only) can be either:

```
NONE or COPY
LINEAR or SMOOTH
MIPMAP or HIGH_QUALITY
```

Which mode is best varies widely and may require some experimentation. In general, the quality of the image improves from NONE to MIPMAP. But that rule is not hard and fast – in some cases it degrades the image appearance. Also, graphics card drivers still contain bugs of a very basic nature. A computer's graphics card may not render the image correctly in one or more of the stretch modes. Often this is related to the size of the image as well.

The ImageDimension command can be used to specify a size for the image. For example, if the image is a tillable texture of some bricks you can specify the size of the image coverage in meters. This information is used to compute texture coordinates and tile textures at an appropriate resolution. The following identifiers are valid:

```
UNDEFINED
LOCAL
WORLD
```

If LOCAL or WORLD are defined, they ought to be followed by the width and height of the image in either local coordinates or world coordinates. Note that only local coordinates are currently supported.

The ObjectDefinition block can contain any of the following tags:

```
ID                     = STRING
ImageID                = STRING
Geometry               = <Identifier>
DimensionX             = <Identifer>
DimensionY             = <Identifer>
ImageWrap              = <Identifier>
ClipToImage            = <Identifier>
SurfaceFragmentStyle   = <Identifer>
SurfaceColourMode      = <Identifer>,<Identifer>
SurfaceColourField     = STRING
```

It can also contain the following block definitions:

```
Registration Begin   // Identical in new and old format
Registration End
SubObject Begin      // New format only
SubObject End
```

The ID is a unique identifier that you will use to refer to this definition in other object definitions and objects. If it is not specified, a unique identifier is generated. If the ID is not unique the system reports an error.

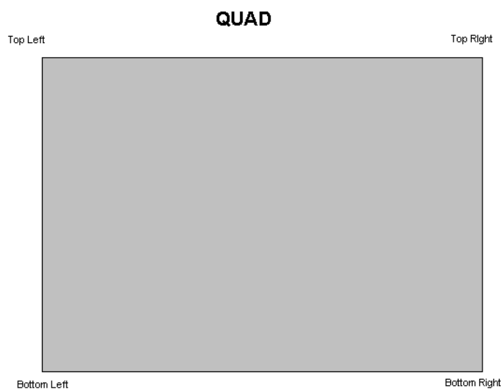
Each ObjectDefinition comes in one of two types – a mesh definition or a root object definition. A mesh definition is an end-node and includes a Registration block to define the mesh geometry. A root node contains one or more SubObject blocks which contain references to other definitions. It does not contain a Registration block, Geometry, ImageID, ImageWrap, SurfaceSource or DimensionX/Y tags.

The ImageID refers to an ID from an ImageDefinition block. It specifies the image is textured onto the mesh geometry.

The Geometry specifies what type of mesh is defined. It has the following types –

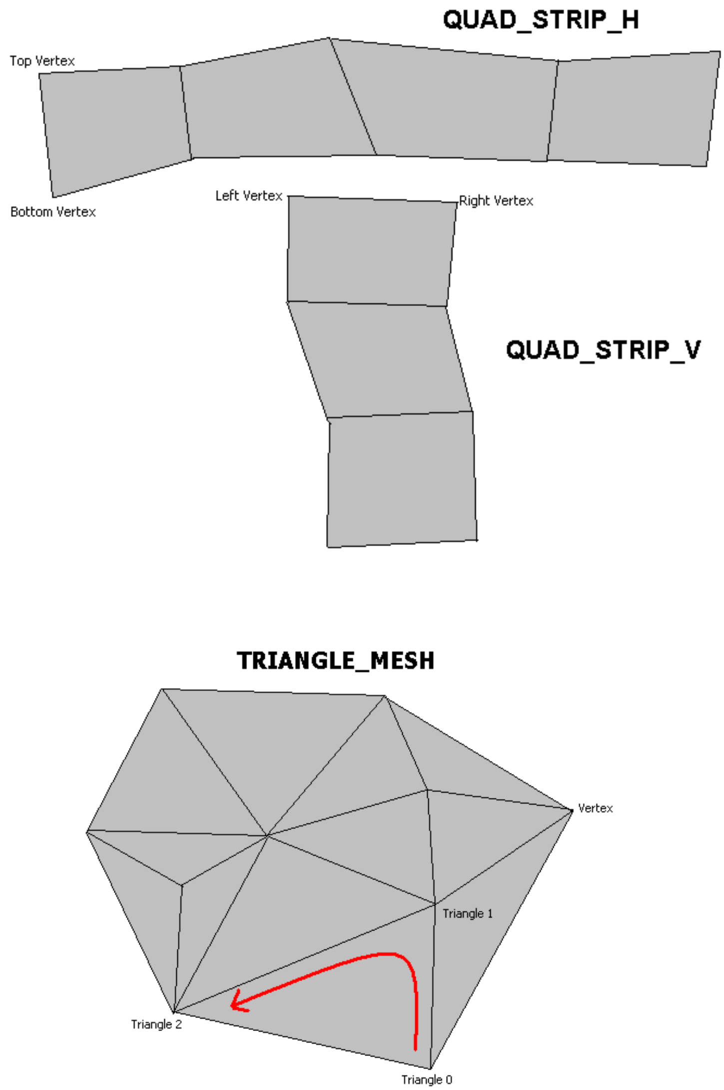
```
QUAD
QUAD_STRIP_H
QUAD_STRIP_V
TRIANGLE_MESH
```

A QUAD is defined by four coordinates in space located at the corners of the surface. It need not be planar.



A QUAD_STRIP is a strip of QUADs and it can either be defined in a horizontal or vertical orientation. This orientation is not implicit in the geometry – it is actually a rotation of the image.

A triangle mesh is a collection of vertices in space from which a surface is defined by connecting vertices into many triangles. Generally, no two triangles should overlap. The surface described can have holes and islands and can be either closed or open or comprised of multiple patches.



Mesh definition diagrams

The DimensionX and DimensionY commands are used to specify the default horizontal and vertical dimensions of the mesh. Valid settings for these commands are X, Y or Z. If these settings are not defined then a default setting will be computed from the input registration points.

These settings are overridden in many cases. In all 3D graphs and in 2D maps and 2D profiles and sections they are always reset to X/Y. If a triangulation is constructed from a grid then X/Y is always used.

In a 2D graph these settings are used to control which dimension is horizontal coordinates and which is vertical.

The ImageWrap command controls texture tiling. The usual mode is CLAMP which means only texture coordinates between 0-1 are valid. It can also be REPEAT which tiles the texture and in this case any texture coordinate is valid.

The Surface… commands have the same syntax as the same commands defined for an object group. The only exception is that at this level of scope you do not specify any override options. These settings are the default surface colour settings that will be applied for the mesh if it is not overridden by the group properties. These surface settings need not be defined and appropriate defaults are assigned.

The ClipToImage option can take the values NONE, CLIP, FORCED_CLIP. If clipping is defined then the surface is clipped to fit the image.

The Registration block is where the coordinates and structure of the mesh for the object definition are specified. All commands in the Registration block are common to both the old format and the new. Within this block you need to define the mesh vertex coordinates, the mesh structure and the image extents which will be converted to texture coordinates. There is plethora of ways to do this.

The texture coordinates of the image range from 0 to 1 in X and Y and the origin (0,0) is in the bottom left corner of the image. The process of rendering the image onto the surface is called texture mapping. Generally, the image is texture mapped onto the surface without distortion and so it is sufficient to supply the coordinates in space of the corners of the image. In this scenario the texture coordinates will be automatically generated. Alternatively you can specify all the texture coordinates. This will be necessary if the surface is closed or if you wish to warp the image across the vertices.

Firstly the extents of the image in world coordinates can be specified using the following commands. X and Y coordinates are required, Z coordinates are optional. If specified, these coordinates are all that is required to correctly locate an image on a surface. If these parameters are not supplied the EGB texture coordinates will be generated either by reading them explicitly from the EGB file or by assuming the texture fits perfectly to the extents of the surface. This is usually a valid assumption for QUAD and QUAD_STRIP… geometries but for TRIANGLE_MESH geometries it will usually be necessary to specify the Bitmap extents as described above.

```
BitmapOrigin       = x,y,(z)
BitmapExtremity    = x,y,(z)
```

To define a QUAD surface specify the four corner coordinates as XYZ with the following information tags. Note that if one optional parameter is supplied then all must be supplied. However, if you leave and empty data tag it will be ignored – eg ,,,,.The texture coordinates (tx,ty) generally range from 0-1 but they can exceed this. In this case the texture may be clamped or repeated. The colour is recorded for each vertex and is specified as a TTBBGGRR value. Alternatively you can specify a colour for the entire face using the Colour command.

```
TopLeft        = x,y,(z),(tx),(ty),(colour)
TopRight       = x,y,(z),(tx),(ty),(colour)
BottomLeft     = x,y,(z),(tx),(ty),(colour)
BottomRight    = x,y,(z),(tx),(ty),(colour)
Colour         = colour
```

To define a QUAD_STRIP_H surface specify the upper and lower coordinates of the strip using the following information tags. Note that the number of upper and lower vertices must match.

```
TopVertex      = x,y,(z),(tx),(ty),(colour)
BottomVertex   = x,y,(z),(tx),(ty),(colour)
Colour         = colour
```

To define a QUAD_STRIP_V surface specify the left and right coordinates of the strip using the following information tags. Note that the number of left and right vertices must match.

```
LeftVertex     = x,y,(z),(tx),(ty),(colour)
RightVertex    = x,y,(z),(tx),(ty),(colour)
Colour         = colour
```

To define a TRIANGLE_MESH surface specify the vertices of the mesh and then specify the optional triangle definitions in an anticlockwise fashion. Each triangle is defined by three vertex indices. The vertices are indexed from zero in the order in which they are defined in the file. If no triangle definitions are supplied then the vertices will be triangulated automatically. Again, colour can be defined for each vertex and for each face.

```
Vertex         = x,y,(z),(tx),(ty),(colour)
Triangle       = vertex0,vertex1,vertex2,(colour)
```

The old format defined a GeometrySource block within the Registration block within which you could specify additional source information for triangle meshes. This block is no longer supported but it will be read and parsed without error.

```
GeometrySource Begin
 …
GeometrySource End
```

The following commands are recognised.

```
GeometryFormat    = <Identifier>
GeometryFile      = STRING
GridBand          = INT
ClipTriangulation = <Identifier>,REAL,(<Identifier>),(<Identifer>)
GridCompressionMode     = <Identifier>
GridCompressionConstant = <Identifier>,INT,INT
GridCompressionVariable = <Identifer>
GridVertexTableBands    = INT,…
```

The following commands are also recognised for backward compatibility but are now obsolete.

```
GridName         = STRING
GridCompression  = INT (0 - 100 variable compression value)
GridClipOption   = INT (0/1/2 = Complete/Partial/None)
```

The current options for GeometryFormat are:

```
GRID
ARC_TIN or ESRI_TIN
DXF
GOCAD
GEMCOM
TKM or MODELVISION
```

These specify that the triangle mesh will be constructed from a raster grid file, an ESRI TIN file(s), a 3D DXF file, a GoCAD TSurf or GEMCOM BT2 file. In all cases you need to specify the source file name using the GeometryFile command. Specify the grid band (if the input grid is multi-banded) as a zero based integer.

The ClipTriangulation command is only valid if you have defined a set of mesh vertices but have not specified any triangulation. In this case a convex hull delauney triangulation of the vertices will be generated. The clipping option removes long thin triangles which may have been generated on the edges and in holes. Use the identifier Defined or Undefined to turn the option on or off. If turned on, then specify the length to height ratio of clipped triangles. Optionally specify EDGEONLY to only clip triangles on the edge and/or RECURSIVE to continue clipping until all thin triangles have been removed.

The grid compression options allow you to decimate or compress the input grid to reduce the size of the triangulation. This may be necessary to improve rendering performance or reduce memory consumption. GridCompressionMode can have the following values.

```
NONE
CONSTANT
VARIABLE
```

CONSTANT will decimate the grid by a factor specified with the GridCompressionConstant command. VARIABLE will apply a variable compression technique that attempts to minimise the size of the triangulation whilst retaining surface detail.

GridCompressionConstant is used to specify decimation parameters. It has the value of SYSTEM, USER or STEP. SYSTEM and USER choose and X and Y decimation factor so that the grid is no larger than a specified size in rows and columns. In the case of first the size is specified by the system and no further parameters are required. In the latter case you can specify the maximum size and the columns and rows ought to be included. The final option, STEP, decimates the grid using the X and Y step supplied. A factor of 1 indicates every node is used.

GridCompressionVariable is used to specify the variable compression parameters. Specify the compression factor from 0 – 100 (low to high compression). A minimum of 30 is recommended. Then specify one of three optional clipping options – COMPLETE, PARTIAL or NONE. If you specify COMPLETE then you get a well clipped grid that matches the input grid. However, this algorithm does fail on some grids. If so, use the PARTIAL option. The grid should still be well clipped. If you use NONE then all holes will be filled and the grid will be a convex hull of the input nodes.

The grid can be used as a source of field data for the vertex table and any number of bands can be loaded using the GridVertexTableBands command. Specify one or more zero based band numbers.

The block can also contain ancillary data tables at four levels of scope - Mesh, Surface, Face and Vertex (a fifth data table can be declared for Objects at group scope). These all have the same structure and are declared in the following way –

```
MeshTable Begin
  FieldDefinition Begin
    Field = STRING,<Identifier>,<NULL>,REAL
  FieldDefinition End
  DataDefinition Begin
    Data = <Data>,…
  DataDefinition End
MeshTable End
SurfaceTable Begin/End
FaceTable Begin/End,
VertexTable Begin/End
```

You can declare an unlimited number of fields for each table. Each field definition must include the field name. It may also include the field data type, a NULL value and a unit conversion value (which converts from field to SI units). The field data type can have any of the following values.

```
BYTE USINT1       (= Unsigned 1 byte integer )
CHAR INT1         (= Signed 1 byte integer )
USINT2            (= Unsigned 2 byte integer )
SHORT INT2        (= Signed 2 byte integer )
USINT4 DWORD      (= Unsigned 4 byte integer )
LONG INT INT4     (= Signed 4 byte integer )
FLOAT REAL REAL4  (= 4 byte real   )
DOUBLE REAL8      (= 8 byte real   )
STRING MEMO       (= String   )
```

The data is defined in one or more Data lines. A value is expected for each declared field. A data line is expected for each vertex/face/surface/mesh/object.

The SubObject block will generally only be defined in the object definition if it is not defining a mesh. In this case the object definition is a container for one or more other object definitions each of which is stored in a SubObject block. The only other information in the block is the transformation parameters for the object definition referred to.

```
ObjectID   = STRING
Translate  = REAL,REAL,REAL
Scale      = REAL,REAL,REAL
Rotate = <Identifier>,<REAL,REAL,REAL>/REAL,<Identifer>,<Identifer>
Transform = REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,
REAL,REAL,REAL,REAL,REAL
```

The ObjectID is a predefined object definition ID.

The transformation that is computed for the object is defined by zero or more Translate, Scale or Rotate commands. The order in which these are specified in the file is important. They are applied in last to first order. The Transform command data can be specified on a single line or broken up into multiple lines.

The Translate and Scale commands require X, Y and Z translation and scale factors. The Transform command requires the specification of a complete 4x4 matrix in row then column order.

The rotate command first requires the rotation axis to be specified. This can be defined as X, Y, Z or GENERAL. In the latter case you should then supply a vector for the rotation axis. You should then supply the rotation angle. By default the angle is specified in radians. If you append a DEGREES or DEG or RADIANS or RAD identifier after the angle you can specify the exact units of the angle. You can also specify the angle is CLOCKWISE or ANTICLOCKWISE.

The ObjectGroup block can contain any of the following commands:

```
Name                  = STRING
ReferenceDTM          = <Identifier>,STRING,INT,REAL,<Identifer>,…
Lighting              = <Identifier>
SurfaceFragmentStyle  = <Identifer>
SurfaceColourMode     = <Identifer>,<Identifer>,<Identifier>
SurfaceColourField    = STRING
DisplayList           = <Identifier>
ColourMapping         = <Identifier>,<Identifier>,REAL,REAL
```

Also supported are the following image definition commands are also valid in groups:

```
TransparentColour     = <Identifier>,INT
ImageStretch          = <Identifier>,<Identifier>
```

The ReferenceDTM command is used to specify a DEM grid which can be used to shift the Z coordinates of all objects in the group. For example, a forest group may define a large number of trees in a landscape. The Z coordinates of the trees can then be automatically determined by the system by reference to the DEM grid. Specify DEFINED or UNDEFINED to turn the feature on or off. Specify the grid file name and the zero based band number. Also specify a scaling factor where a value of 1 indicates no scaling. You can use this to scale a DEM without scaling the objects in the DEM. You can then specify a number of flags.

```
PER_OBJECT
PER_VERTEX
```

This is the rate at which the DEM offset is applied. It can be applied to every vertex individually or it can be applied to the centre of the object.

```
HORIZONTAL
VERTICAL
MULTI_COMPONENT
```

This is one of the flags which affect the way the offset is applied to the Z coordinate. If the object is HORIZONTAL the Z coordinate is replaced by the DEM Z coordinate. If the object is VERTICAL the Z coordinate is added to the DEM Z coordinate. If it is MULTI_COMPONENT there may be a mix of different mesh types but you are indicating that you want the integrity of the structure of the object to remain.

```
SIT
CENTRE
HANG
```

These flags position the object either above the DEM, on the DEM or below the DEM. For example a tree or a house will sit on the DEM, a road will straddle the DEM and a CDI section will hang from the DEM.

Lighting is a surface property and not all lighting modes are compatible will all surface properties. You can specify NONE, PER_FACE or PER_VERTEX. Generally you ought to ensure that colour modulation of the surface is at the same scope as the lighting.

The SurfaceFragmentStyle command controls the appearance of the triangle mesh. You can specify FILL, FILL_LINES or LINES.

The SurfaceColourMode command specifies the scope for both the fill and the lines (if displayed) and also the mesh override settings. Fill and line variables have one of the following values.

```
IMAGE   - texture map the loaded image
USER    - use the user specified fill/line colour
OBJECT  - modulate by an object table field
MESH    - modulate by a mesh table field
SURFACE - modulate by a surface table field
FACE    - modulate by a face table field
VERTEX  - modulate by a vertex table field
```

The override setting has one of the following values –

```
ALL     - all meshs will use the group settings
DEFAULT - meshs with no defined surface source use the group settings
```

If any of the tables are used then the SurfaceColourField command is used to specify the field name.

The DisplayList command specifies how display lists will be generated within a group. This is an advanced command. You can define granularity of PER_OBJECT or PER_GROUP. If the displays lists are generated for each object then the transform can be INTERNAL or EXTERNAL to the list. If you use the Facing command for objects in a group then you need to have display list granularity set to PER_OBJECT. If you are using animation you will require external transforms.

The ColourMapping command stretches any colour assignment. Valid values include NONE, LINEAR, LINEAR_DECREASING, LOG, LOG_DECREASING and HISTOGRAM. Further control is available from the series property pages. You can also specify a band pass as either a percentage (0-100) minimum and maximum or as absolute values. Specify BANDPASS or ABS_BANDPASS and then the minimum and maximum values.

The following blocks can also be specified in groups, although the GeoreferenceImage block is only valid in an old style group.

```
ObjectTable Begin
:
ObjectTable End
CoordinateSpace Begin
:
CoordinateSpace End
Object Begin
:
Object End
GeoreferenceImage Begin
:
GeoreferenceImage End
```

The ObjectTable is just like the mesh/surface/face/vertex tables previously discussed and has exactly the same command structure. It stores data for each object in the group.

The block declares the coordinate system information for the group – hereafter referred to as the source coordinate system. If the graph in which the EGB file is being plotted has a known coordinate system (hereafter referred to as the destination coordinate system) then the vertices in all meshes will be converted from the source to destination coordinates prior to display. Note that this transformation is the last operation applied and is computed on the final coordinates of the mesh after all other transformations have been applied.

```
Projection        = STRING
Datum             = STRING
Units             = STRING
CoordinateSystem  = STRING
TransformRate     = <Identifier>
```

The Projection, Datum and Units strings are old format strings and currently are not used. The CoordinateSystem string is a MapInfo Professional compatible string which defines the complete coordinate system.

The TransformRate command can be either PER_VERTEX or PER_OBJECT. If it is PER_VERTEX then each vertex in the mesh is individually transformed. If it is PER_OBJECT then the centre of the object will be transformed. This will result in a translation and a scaling.

The Object block can contain any of the following tags -

```
Name         = STRING
ObjectID     = STRING
Line         = STRING
Colour       = INT
Transparency = INT,<Identifier>
Translate    = REAL,REAL,REAL,<Identifer>
Scale        = REAL,REAL,REAL,<Identifer>
Rotate       = <Identifier>,<REAL,REAL,REAL>/REAL,<Identifer>,
<Identifer>,<Identifer>
Facing       = <Identifier>,<Identifier>,<REAL,REAL,REAL>
Transform    = REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,
REAL,REAL,REAL,REAL,REAL,REAL,REAL,<Identifier>
Name         = STRING
ObjectID     = STRING
Line         = STRING
Colour       = INT
Transparency = INT,<Identifier>
Translate    = REAL,REAL,REAL,<Identifer>
Scale        = REAL,REAL,REAL,<Identifer>
Rotate       = <Identifier>,<REAL,REAL,REAL>/REAL,<Identifer>,
<Identifer>,<Identifer>
Facing       = <Identifier>,<Identifier>,<REAL,REAL,REAL>
Transform    = REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,REAL,
REAL,REAL,REAL, REAL,REAL,REAL,REAL,<Identifier>
```

The name is cosmetic and is used to identify the object definition in property pages. The ObjectID is a reference to an object definition ID. It is the object that will be rendered.

The line identifier can be used in applications to automatically control which objects are displayed in a 2D or 3D graph. It is utilized when EGB objects are used as drill hole curtains and when CDI sections and other data are displayed in 2D sections and profiles.

The colour of the object, if specified, will be recorded in the object table in the "_Colour" field.

The transparency of the object can range from 0 (opaque) to 100 (invisible). You can optionally declare NO_SELF_SEE_THROUGH to prevent the user seeing the surface through itself.

The translate, scale, rotate and transform commands have the same structure as for an object definition with an important exception – you must add a identifier to specify whether the transform is applied before the DEM transform (PRE_DTM) or after the DEM transform (POST_DTM). This is important only if a DEM transform is being applied (by the group). If so you will want geneally want to scale and translate the object to place it in its true XY location, then apply the DEM offset, and then apply any further transforms.

An additional transformation command for objects is the Facing command. If specified, this transformation is a rotation applied after all other transformations. The effect of the command is to always face the object towards the eye position. Turn it on by specifying DEFINED. You can restrict this rotation so that the object can only be rotated through a specific axis. Define this as X, Y, Z or GENERAL. In the latter case you must then specify the axis of rotation as an XYZ vector. Note that this transformation must be recomputed at every render cycle so it may reduce rendering performance. It will force the object to use an individual display list and this may also reduce performance. Generally, you would use this for every object in a group and quarantine the objects that use this option in that way. Also, the coordinates of these objects are always computed as unrotated and this may affect selection.

The old format GeoreferenceImage block contains a mixture of commands found elsewhere in the new format. It also contains Comments and Version strings that are no longer read and are obsolete. If declared outside of a group it may contain some group commands as it will be assigned its own group automatically. As it defines a single object definition, image definition and object it may contain image definition commands and object definition commands. A summary of valid commands is shown in the example below.

```
GeoreferenceImage Begin
  Name = "Obsolete"
  Comments = "This string is obsolete"
  Version = 1.0 // This version number is obsolete and never used
  Image = "air_photo.jpg"
  ImageFormat = JPG
  TransparentColour = DEFINED,WHITE
  ImageStretch = HALFTONE,MIPMAP
  ImageDimension = UNDEFINED
  ImageWrap = CLAMP
  ClipToImage = CLIP
  Transparency = 0
  Geometry = QUAD
  Line = L1001
  DimensionX = X
  DimensionY = Y
  CoordinateSpace Begin
// All commands in this block are valid
  CoordinateSpace Begin
  Registration Begin
// All commands in this block are valid
  Registration End
GeoreferenceImage End
```

ER Mapper Look-Up Files

Encom Discover PA uses standard ER Mapper look-up tables for controlling the colour of images, displays and contours. A full description of these ASCII files is provided in the ER Mapper Reference Manual Version 5.x or 6.0.

An example of an ER Mapper look-up file is shown below:

```
LookUpTable Begin
 Version       = "1.0"
 Name          = "Azimuth"
 Description   = "LUT"
 NrEntries     = 256
 LUT           = {
  0   0   0   0
  1  900  860   0
  2 1800 1720   0
  3 2700 2580   0
  4 3600 3440   0
  5 4500 4300   0
  6 5400 5160   0
  7 6300 6020   0
```

```
  8 7200 6880   0
  9 8100 7740   0
  10 9000 8600   0
  11 9900 9460   0
  12 10800 10320   0
  13 11700 11180   0
  14 12600 12040   0
  15 13500 12900   0
  16 14400 13760   0
  17 15300 14620   0
   :  :   :   :
 242 7560 11536 13160
 243 7020 10712 12220
 244 6480 9888 11280
 245 5940 9064 10340
 246 5400 8240 9400
 247 4860 7416 8460
 248 4320 6592 7520
 249 3780 5768 6580
 250 3240 4944 5640
 251 2700 4120 4700
 252 2160 3296 3760
 253 1620 2472 2820
 254 1080 1648 1880
 255  540  824  940
 }
LookUpTable End
```

ETS Encom Tube Shape File Formats

Refer to the Tube Shape Manager and the Display of Curve Data for a description of the use of the Tube Shape files. Tube Shape files are stored normally in the Program Files\Encom\Common\TubeShape folder. An example of a Tube Shape file is:

```
// Comments…
Shape Begin
  Version = "1.0"
  Comments = "Star tube"
  PointFormat = AngleRadius
  AngleFormat = Degrees
  Name = "4-point Star"
  PointData Begin
    Point = 0,1
    Point = 45,0.5
```

```
        Point = 90,1
        Point = 135,0.5
        Point = 180,1
        Point = 225,0.5
        Point = 270,1
        Point = 315,0.5
        Point = 0,1
      PointData End
    Shape End
```

Each cross-section is defined by a Shape Begin and Shape End block. The version is defined as a numeric string. Comments may be added using standard comment leading delimiters or after a Comments tag. The shape is identified by the Name string.

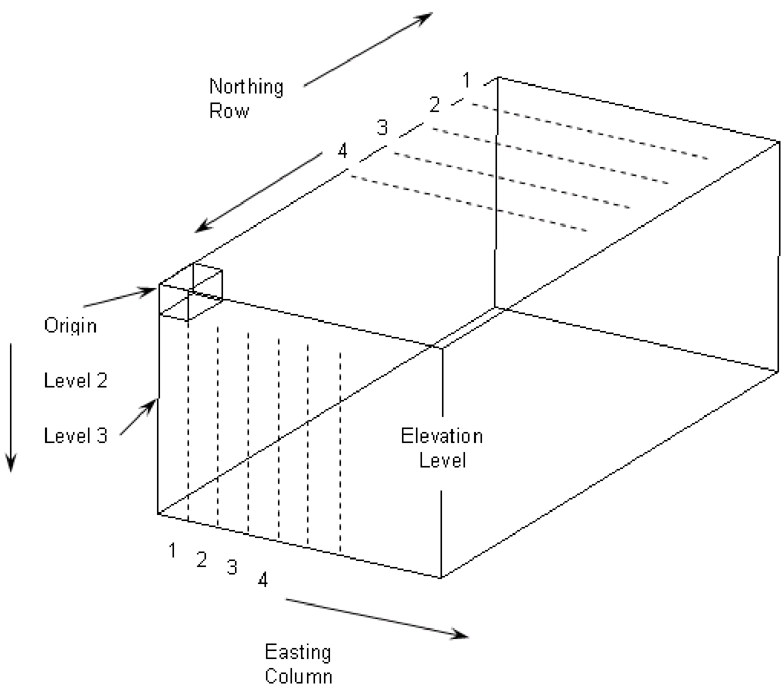
The PointFormat tag specifies the format of the point data. Valid tags are AngleRadius, Point or ScaledPoint. If AngleRadius is specified then each point ought to be specified as an angle (see notes below) and a radius. The angle ought to be monotonically increasing. If Point is specified then each point is specified as an XY coordinate pair. In this case a centre point ought to be specified and the angle/radius information is computed relative to the centre point. If ScaledPoint is specified then each point is specified as an XY coordinate pair. In this case a centre point ought to be specified and the angle/radius information will be computed relative to the centre point. In this case the radius information is scaled from 0 to 1.

The AngleFormat tag specifies the units of the angle data. Valid tags are Radians or Degrees. Radians are defined anticlockwise from East and Degrees are specified clockwise from North.

The point data is listed between PointData Begin and PointData End tags. Points are specified as Point = Angle,Radius or Point = X,Y depending on the point format. If Point or ScaledPoint is used a centre point ought to be specified using Centre = X,Y. Currently it scales the data by finding the centre point (from the input point data range) and scaling the shape into +/-1 space.

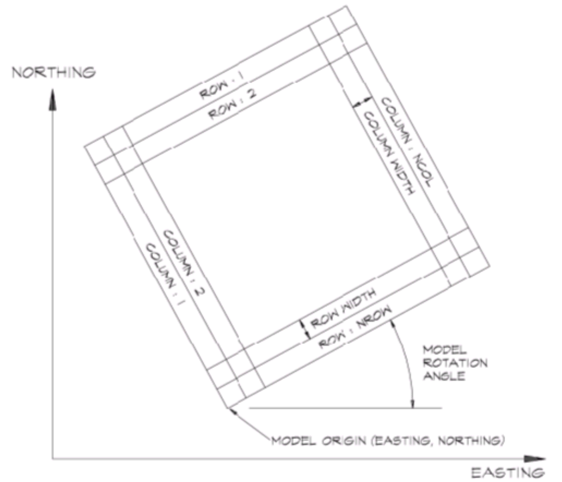
Gemcom Block Model

The Gemcom file format of a standard block model has the file extension of .TB2 and is as illustrated below



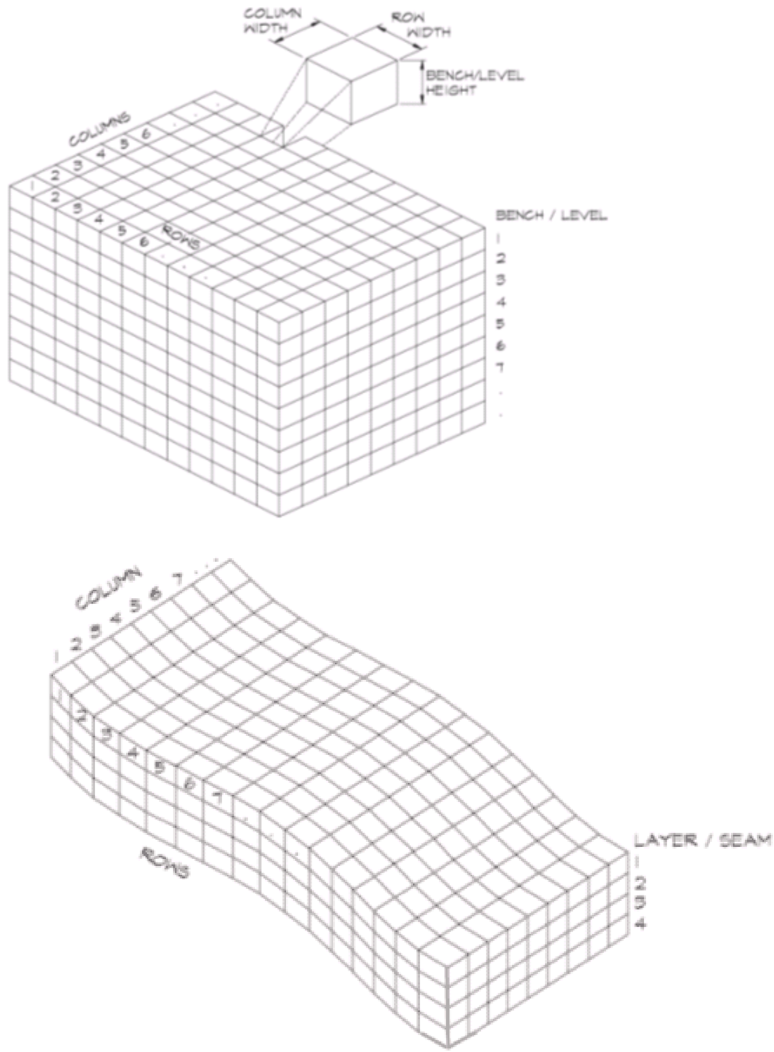
Gemcom block model definition

In the diagram above the block size is 10x5x5, the block model origin is at the top left corner, the upper surface of the block is the reference origin for the levels which increase vertically down. Attributes within the block model table relate to values calculated at the centroid of the block. Note the block levels increase downward and not upward. The following plan and isometric views present the geometric layout of the Gemcom block models.



Plan view of block layout

The origin is relative to the numbering sequence for the rows. Below are 3D representations of the row/column and layer sequence of the Gemcom model file.



Typical block model cell distributions

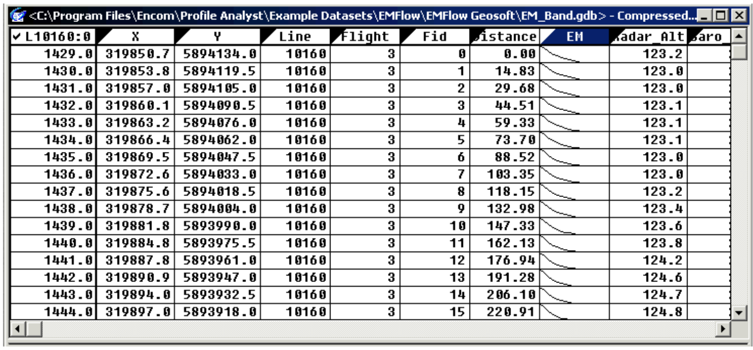
The file format is binary and has a file extension of .BMF.

Geosoft Oasis montaj Input Template

An example of a template used in Oasis montaj for input of single and multi-banded (termed array) data is shown below.

```
/ Wizard Generated Template for .\EM_raw_data_and_CDI_Results.asc
[IMPORT ARCHIVE]
FILEHEADER  3
RECORDFORM  DELIM," \t","","","\""
SKIPSTRING "/"
DATA  0,6,NORMAL,,,,
CHAN  Line,DOUBLE,NORMAL,8,0,LABEL=Line
DATA  1,2,NORMAL,,,,
CHAN  Flight,DOUBLE,NORMAL,8,0,LABEL=Flight
DATA  2,4,NORMAL,,,,
CHAN  Fid,DOUBLE,NORMAL,8,0,LABEL=Fid
DATA  3,9,NORMAL,,,,
CHAN  X,DOUBLE,NORMAL,9,1,LABEL=X
DATA  4,10,NORMAL,,,,
CHAN  Y,DOUBLE,NORMAL,10,1,LABEL=Y
DATA  5,8,NORMAL,,,,
CHAN  Distance,DOUBLE,NORMAL,8,2,LABEL=Distance
DATA  6,6,NORMAL,,,,
CHAN  EM{18},DOUBLE,NORMAL,8,0
DATA  24,6,NORMAL,,,,
CHAN  Radar_Alt,DOUBLE,NORMAL,9,1,LABEL=Radar_Alt
DATA  25,6,NORMAL,,,,
CHAN  Baro_Final,DOUBLE,NORMAL,10,1,LABEL=Baro_Final
DATA  26,7,NORMAL,,,,
CHAN  DEM,DOUBLE,NORMAL,8,2,LABEL=DEM
DATA  27,9,NORMAL,,,,
CHAN  Mag_Final,DOUBLE,NORMAL,9,2,LABEL=Mag_Final
DATA  28,7,NORMAL,,,,
CHAN  TimeCon1,DOUBLE,NORMAL,8,4,LABEL=TimeCon1
DATA  29,8,NORMAL,,,,
CHAN  TimeCon2,DOUBLE,NORMAL,8,4,LABEL=TimeCon2
DATA  30,7,NORMAL,,,,
CHAN  Altitude,DOUBLE,NORMAL,8,2,LABEL=Altitude
DATA  31,8,NORMAL,,,,
CHAN  Distance,DOUBLE,NORMAL,8,2,LABEL=Distance
DATA  32,15,NORMAL,,,,
CHAN  Est_Pene,DOUBLE,NORMAL,15,11,LABEL=Est_Penetration
DATA  33,15,NORMAL,,,,
CHAN  DC{31},DOUBLE,NORMAL,15,11
```

In the example template above, the 6th input data channel (labelled EM) has 18 individual data columns in the input ASCII data file. The 18 data columns are specified by enclosing brackets {18}. When displayed in a spreadsheet within Oasis, individual data channels of the 18 columns are not able to be numerically displayed. Instead they are shown as a decay of data (see the EM data field below).


<C:\Program Files\Encom\Profile Analyst\Example Datasets\EMFlow\EMFlow Geosoft\EM_Band.gdb> - Compressed...

L10160:0	X	Y	Line	Flight	Fid	Distance	EM	Radar_Alt	Baro
1429.0	319850.7	5894134.0	10160	3	0	0.00		123.2	
1430.0	319853.8	5894119.5	10160	3	1	14.83		123.0	
1431.0	319857.0	5894105.0	10160	3	2	29.68		123.0	
1432.0	319860.1	5894090.5	10160	3	3	44.51		123.1	
1433.0	319863.2	5894076.0	10160	3	4	59.33		123.1	
1434.0	319866.4	5894062.0	10160	3	5	73.70		123.1	
1435.0	319869.5	5894047.5	10160	3	6	88.52		123.0	
1436.0	319872.6	5894033.0	10160	3	7	103.35		123.0	
1437.0	319875.6	5894018.5	10160	3	8	118.15		123.2	
1438.0	319878.7	5894004.0	10160	3	9	132.98		123.4	
1439.0	319881.8	5893990.0	10160	3	10	147.33		123.6	
1440.0	319884.8	5893975.5	10160	3	11	162.13		123.8	
1441.0	319887.8	5893961.0	10160	3	12	176.94		124.2	
1442.0	319890.9	5893947.0	10160	3	13	191.28		124.6	
1443.0	319894.0	5893932.5	10160	3	14	206.10		124.7	
1444.0	319897.0	5893918.0	10160	3	15	220.91		124.8	

Example of Geosoft Oasis montajTM spreadsheet with multi-banded data

GoCad TSURF Surfaces Definition

For a full description of the GoCad TSurf surface, refer to the GoCad Reference Documentation. The data file of a GoCad TSurf surface include the inherited file elements of an Object and of an Atomic (see below). The Atoms (Vertices) of the TSurf are defined before their joining triangles.

- **GoCad Objects**: The Object file elements define the Object style and the Object coordinate space and Units

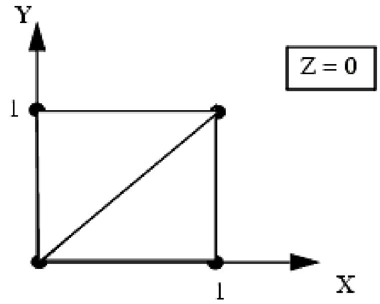
- **Atomics**: The Atomic data format defines the points, locations and Property values, of an Atomic (currently, this includes PointsSet, Curve, Surfaces, Solids and GShapes). The Atomic inherits all of the Object file format elements, plus Atomic data.

Triangle Definition

Triangles definition must occur after vertices definitions. Each Triangle is defined by its three Vertices (Atoms) in the following format:

```
TRGL id1 id2 id3
```

where ids are the IDs of the already-defined Atoms (Vertices). For example, the ASCII file corresponding to a TSurf named SQUARE, containing four vertices, and two triangles is shown below.




```
GOCAD TSurf 1
HEADER{
name:SQUARE
}
GEOLOGICAL_TYPE top
TFACE
VRTX 1   0.   0.   0.
VRTX 2   1.   0.   0.
VRTX 3   1.   1.   0.
VRTX 4   0.   1.   0.
TRGL 1  2  3
TRGL 1  3  4

END
```

An example of a TSURF and its data file

In addition to the VRTX or PVRTX point/vertex descriptor find in the Atomic format definition, on can find the additional line inside a TSurf ASCII file:

```
ATOM id1 id2
```

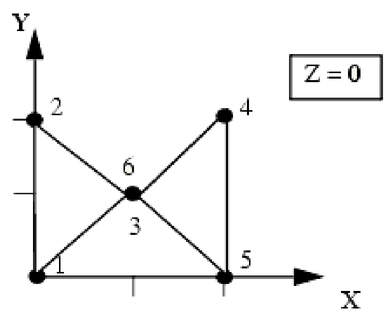
where id1 > id2

where id1 is the index of the new ATOM and id2 is the index of the already existing VRTX node which XTZ this Atom shares.

The goal of the ATOM keyword is to create a new ATOM but which shares an already existing vertex. An Atom node has its one Property values but it is spatially linked to the existing VRTX node. In other word, an ATOM and its referenced VRTX are co-located, but not connected. Triangles construction will use the vertex id or the atom id.

An example of the use of such Atom record is given below:

```
GOCAD TSurf 1
HDR name:s1
VRTX 1 0 0 0
VRTX 2 0 2 0
VRTX 3 1 1 0
VRTX 4 2 2 0
VRTX 5 2 0 0
TRGL 1 2 3
TRGL 3 4 5
END
```




```
GOCAD TSurf 1
HDR name:s1
TFACE
VRTX 1 0 0 0
VRTX 2 0 2 0
VRTX 3 1 1 0
TRGL 1 2 3
TFACE
VRTX 4 2 2 0
VRTX 5 2 0 0
ATOM 6 3
TRGL 4 5 6
END
```

An example of a shared vertex TSurf

The input Tsurf file on the left creates a Surface shown at the centre. The Surface has two TFaces, because two Triangles need to share an edge to be considered connected. When the Surface is saved, GOCAD outputs the file on the right. It recognises that the two Triangles both have one vertex at an identical location, but topologically, they cannot be the same point. So, GOCAD creates a new point, ATOM 6, that shares the identical location of VRTX 3 but is an independent point in the sense that it has its own Property values. Had there been two more triangles in the input file, TRGL 2 4 3 and TRGL 6 5 1, the ATOM line would not have been created because the four triangles are then connected through direct or indirect edge-sharing.

Intrepid DDF Format

The Intrepid Data Definition File format is an ASCII data file used to define data for input to the Intrepid database software system (refer to Encom Discover PA Data Links).

The data is banded such that a series of like data channels are combined to a single channel. For example, instead of having EM1, EM2, EM3… all the similar EM channels can be banded together into a single data channel such as EM Data.

For additional information on the DDF file format, refer to the Intrepid Import and Export Reference Manual. An example of a DDF format input file is provided here.

```
TYPE(LINE)
 LINE  1-5 INTEGER*2
 EASTING 7-15  REAL*4 PROJ(TMAMG55,AGD66) IsX
 NORTHING 17-26 REAL*4 PROJ(TMAMG55,AGD66) IsY
 DAY 121-122 INTEGER*2
 MONTH 119-120 INTEGER*2
 YEAR  117-118 INTEGER*2
 FLIGHT 99-100 INTEGER*2
 LINETYPE 52-53  INTEGER*2
 FID    128-137 REAL*4
 MAGNETIC 184-192 REAL*4
 ALTITUDE   152-159 REAL*4
 GPSZ 163-170 REAL*4
 BARO 174-180 REAL*4
 X_CHANNEL[1] 194-203 REAL*4
 X_CHANNEL[2] 205-214 REAL*4
 X_CHANNEL[3] 216-225 REAL*4
 X_CHANNEL[4] 227-236 REAL*4
 X_CHANNEL[5] 238-247 REAL*4
 X_CHANNEL[6] 249-258 REAL*4
 X_CHANNEL[7] 260-269 REAL*4
 X_CHANNEL[8] 271-280 REAL*4
```

```
  X_CHANNEL[9] 282-291 REAL*4
  X_CHANNEL[10] 293-302 REAL*4
  X_CHANNEL[11] 304-313 REAL*4
  X_CHANNEL[12] 315-324 REAL*4
  X_CHANNEL[13] 326-335 REAL*4
  X_CHANNEL[14] 337-346 REAL*4
  X_CHANNEL[15] 348-357 REAL*4
  Y_CHANNEL[1] 359-368 REAL*4
  Y_CHANNEL[2] 370-379 REAL*4
  Y_CHANNEL[3] 381-390 REAL*4
  Y_CHANNEL[4] 392-401 REAL*4
  Y_CHANNEL[5] 403-412 REAL*4
  Y_CHANNEL[6] 414-423 REAL*4
  Y_CHANNEL[7] 425-434 REAL*4
  Y_CHANNEL[8] 436-445 REAL*4
  Y_CHANNEL[9] 447-456 REAL*4
  Y_CHANNEL[10] 458-467 REAL*4
  Y_CHANNEL[11] 469-478 REAL*4
  Y_CHANNEL[12] 480-489 REAL*4
  Y_CHANNEL[13] 491-500 REAL*4
  Y_CHANNEL[14] 502-511 REAL*4
  Y_CHANNEL[15] 513-522 REAL*4
  Z_CHANNEL[1] 524-533 REAL*4
  Z_CHANNEL[2] 535-544 REAL*4
  Z_CHANNEL[3] 546-555 REAL*4
  Z_CHANNEL[4] 557-566 REAL*4
  Z_CHANNEL[5] 568-577 REAL*4
  Z_CHANNEL[6] 579-588 REAL*4
  Z_CHANNEL[7] 590-598 REAL*4
  Z_CHANNEL[8] 600-609 REAL*4
  Z_CHANNEL[9] 611-620 REAL*4
  Z_CHANNEL[10] 622-631 REAL*4
  Z_CHANNEL[11] 633-643 REAL*4
  Z_CHANNEL[12] 645-654 REAL*4
  Z_CHANNEL[13] 656-665 REAL*4
  Z_CHANNEL[14] 667-676 REAL*4
  Z_CHANNEL[15] 678-687 REAL*4
GROUP BY LINE
```

Intrepid SuveyInfo File

This file is used to create aliases that are used by Encom Discover PA during database import. Encom Discover PA must have certain data fields available when importing Intrepid databases. These fields include:

```
X (Easting)
Y (Northing)
LineNumber
Fiducial
FlightNumber
```

If any of these fields exist in a database but as a different name (such as Easting instead of X), the SurveyInfo file can be used to rename the channels during import without having to physically alter the data structure.

An example of the SurveyInfo file follows:

```
#### Audit Stamp Audit Stamp V3.2- 8/12/2000
Alias Begin
  Comments = "Audit Stamp V3.2- 8/12/2000"
  Bearing = CHORDDIR
  ChordFirstX = CHORDX
  ChordFirstY = CHORDY
  ChordLength = CHORDLEN
  Clearance = RAD_ALT
  Fiducial = FIDUCIAL
  FlightNumber = FLIGHTNUMBER
  LineNumber = LINENUMBER
  X = EASTING
  Y = NORTHING
Alias End
```

Note in this example that Encom Discover PA also creates auxiliary data channels (ChordFirstX, ChordLength etc). These data channels are automatically appended by Encom Discover PA and you do not have to add them. In addition, data channels such as Clearance (equating to data field RAD_ALT) are available.

Micromine Block Model File

The Micromine definition is more completely specified in the Micromine Reference Manual. The block model file is an ASCII file having a Comma Delimited format of a CSV file. The file has with a simple record structure and with the first record defining:

The origin in local coordinates
Row (X), column (Y) and layer (Z) increment sizes
Colour definition
Cell property (eg assay or grade)
Index to a specific block in the model
Code relating to data source
Various items that may relate to ore grade, properties (eg Specific Gravity) and error andf statistical information (eg Standard Deviation).

An example of a Micromine block model follows:

```
X,Y,Z,Xinc,Yinc,Zinc,COLOUR,AU,BLOCKINDX,CODE,CUT15,ORECOD,POINTS,SG
,STD_DEV
10835.0,5185.0,432.5,5.0,2.5,2.5,,,,,,,,,
10840.0,5185.0,432.5,5.0,2.5,2.5,0,0.84,002900150014,M37,0.84,,3.0,2
.64,1.057
10845.0,5185.0,432.5,5.0,2.5,2.5,0,0.85,003000150014,M37,0.85,,3.0,2
.64,1.057
10850.0,5185.0,432.5,5.0,2.5,2.5,0,1.13,003100150014,M37,1.13,,2.0,2
.62,1.294
10825.0,5182.5,432.5,5.0,2.5,2.5,0,1.11,002600140014,M37,1.11,,2.0,2
.62,1.294
10830.0,5182.5,432.5,5.0,2.5,2.5,0,0.86,002700140014,M37,0.86,,3.0,2
.64,1.057
10835.0,5182.5,432.5,5.0,2.5,2.5,0,0.89,002800140014,M37,0.89,,3.0,2
.64,1.057
10840.0,5182.5,432.5,5.0,2.5,2.5,0,0.9,002900140014,M37,0.9,,3.0,2.6
4,1.057
10845.0,5182.5,432.5,5.0,2.5,2.5,0,1.16,003000140014,M37,1.16,,2.0,2
.62,1.294
10820.0,5180.0,432.5,5.0,2.5,2.5,0,1.98,002500130014,M37,1.98,,1.0,2
.55,0.0
10825.0,5180.0,432.5,5.0,2.5,2.5,0,1.15,002600130014,M37,1.15,,2.0,2
.62,1.294
10830.0,5180.0,432.5,5.0,2.5,2.5,0,1.17,002700130014,M37,1.17,,2.0,2
.62,1.294
10835.0,5180.0,432.5,5.0,2.5,2.5,0,1.17,002800130014,M37,1.17,,2.0,2
.62,1.294
```

Noddy Model Format

Noddy models can be viewed in the Voxel Modelling facility. The Noddy software was originally developed in 1997 at Monash University by Dr Mark Jessell. 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).

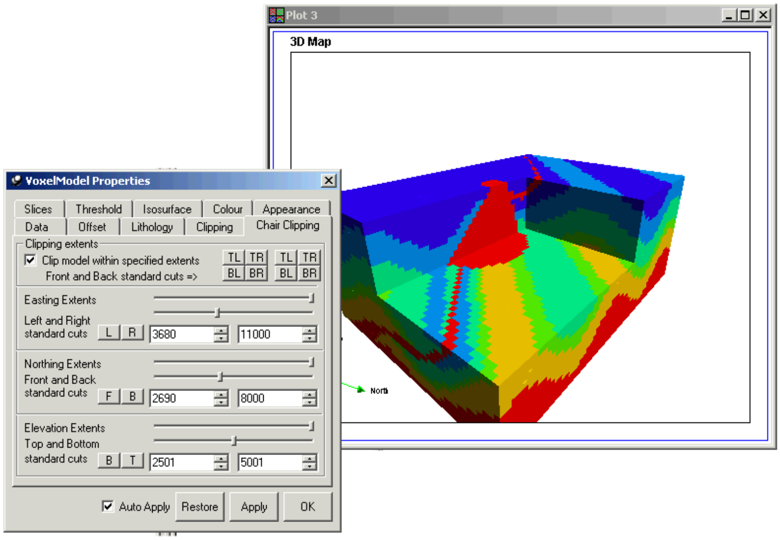
An example of a .G00 file is shown below:

```
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.
```

Within the Noddy application, the models are derived from the **Tools>Export Block>Export Noddy Model** option.



Noddy model with folds, dyking and intrusive

SEG-2 Data Format

This data file is a SEG (www.seg.org) specified seismic and Ground Penetrating Radar format and approved by their Standards Committee. The data files consist of a File Descriptor Block, one or more Trace Descriptor Blocks and one or more Data Blocks.

Pointers are used to indicate locations of blocks with respect to the beginning of the file. The pointers are always unsigned long integers (32 bits).

FILE DESCRIPTOR BLOCK
TRACE DESCRIPTOR BLOCK 1
DATA BLOCK 1
TRACE DESCRIPTOR BLOCK 2
DATA BLOCK 2
...
TRACE DESCRIPTOR BLOCK N
DATA BLOCK N

For a full definition of the SEG-2 file format, refer to the specification published by the Society of Exploration Geophysicists (http://www.seg.org).

Surfer Look-Up Tables

Encom Discover PA can use standard Surfer look-up tables for controlling the colour of images, displays and contours. A full description of these ASCII files is provided in the Surfer Reference Manual Version 6.x, 7.x or 8.0.

An example of an Surfer look-up file is shown below:

```
ColorMap  1 1
  0  153  102  255
 20    0    0  255
 40    0  255 0
 60  255  255 0
 80  255  102 0
100  255 0 0
```

Surpac Block Model File Format

The Surpac three dimensional Block Model is simple to use and understand yet is fast in its creation. Modelling parameters can be added and modified at any time.

The Surpac Block Model is a form of database. This means that its structure not only allows the storage and manipulation of data, but also the retrieval of information derived from that data. It differs from a more traditional database, in that data stored are likely to be interpolated values, rather than true measurements. Another major difference is that these values may be spatially referenced as well as being spatially related. A third important difference is that the Block Model is recalled in its entirety into memory which makes dynamic operations such as colouring of attributes possible but imposes significant memory overheads.

For example, consider a Geological database. Records have spatial attributes which relate them to a spatial position. However, the converse does not necessarily hold as spatial positions are not necessarily related to a record in the database.

The Block Model apportions space into a set of blocks, each being related to a record. The records may be spatially referenced, so information may be retrieved for any point in space, not just for points that have been explicitly measured. This spatial referencing allows the addition of a number of operators to the querying capabilities of the database manipulation scheme, namely spatial operators such as INSIDE and ABOVE, which may operate on solids and surfaces. Outside and below may be built using the NOT logical operations of NOT INSIDE or NOT ABOVE.

The Block Model comprises of a number of components:

- **Model Space**: The model space is a cuboid volume outside of which nothing exists in terms of the Block Model.

- **Attributes**: The properties of the model space that are to be modelled are termed attributes. These attributes may be nominal, ordinal, interval or ratio measurements expressed as numeric or character data. Attributes may also be calculated from the values in other attibute fields, for reporting and visualising.

- **Constraints**: Constraints are the logical combinations of spatial operators and objects that may be used to control the selection of blocks from which information may be retrieved and/or into which interpolations may be made. Constraints may be saved and have file extensions of .CON.

  The model itself is a binary image constructed in the model space and defined by the existence or non-existence of blocks. Model files have file extensions of .MDL.

  The Block Model may be applied to any situation where properties of a volume of space are to be modelled in terms of the distribution of values through that space.

Block Model Concepts

The following terms are used in a Surpac model definition:

- **Origin** - The origin of the model is the lower, front, left hand corner (ie. the minimum Y, X and Z coordinates) of the model expressed in Y, X, Z Cartesian coordinates. The origin is the anchoring point from which rotations involving the Bearing, Dip and Plunge are to be performed.

- **Extent** - The extent of model is the dimensions of the model in the Y, X and Z directions. For example, if a model was to cover the following area:

  3000 m N to 3650 m N
  1500 m E to 2100 m E
  120 m El to 270 m El

  The origin is: Y=3000 X=1500 Z=120

  The extent of the model will be: Y=650 X=600 Z=150

- **Bearing** - The bearing of the model is the horizontal angle in degrees of the direction of the major axis of the model. A bearing of zero indicates a non-rotated model where the major axis of the model is in a north-south orientation.

- **Dip** - The dip of the model is the vertical angle of the blocks in degrees from the horizontal in a direction perpendicular to the bearing of the model. A negative dip is an angle below the horizontal to the right when looking along the bearing of the model. A dip of zero indicates horizontal blocks normal to the bearing of the model.

- **Plunge** - The plunge of the model is the vertical angle of the blocks in degrees from the horizontal along the bearing of the model. This can also be referred to as the tilt of the model. A negative plunge is an angle below the horizontal when looking along the bearing of the model. A plunge of zero indicates horizontal blocks along the bearing of the model.

- **User Block Size** - The block size in the Y, X and Z directions. The user block size is used as the reporting unit for the Block Model. The user block is also the block size upon which interpolation is performed.

  The user block size depends on the Model purpose (ie. Grade Control, Resource Calculation, Pit Optimization) with reference to the data spacing.

  For example, what block size is appropriate for a prospect drilled on a 100m x 100m pattern, which is to have a resource estimate completed? It would not be appropriate to set this model up with a block size of 5x5x5, as the small blocks won't give a better estimate of the resource, as the original data is widely spaced. Perhaps, 25x25x10 may be more realistic (that is, one-third to one-quarter of the sample spacing).

- **Maximum sub-blocks per side** - The maximum number of blocks along each side of the model. This number must always be 2 to the power of an integer. (eg 2, 4, 8, 16, 32, 64, 128, 256, 512)

  This value will need to satisfy a base resolution. For the example used previously: extents Y=650 X=600 Z=150 user block size 25x25x10

  The number of blocks along each side will be 26x24x15(extent divided by user block size). This means that the base resolution will be 32 (the number greater than the maximum number of blocks and is 2 to the power of something). If we wish to allow sub-blocking (the sub-dividing of blocks), the resolution needs to be greater than base resolution.

For this example:
if maximum sub-blocks per side = 64
smallest sub-block = 12.5x12.5x5
if maximum sub-blocks per side =128
smallest sub-block = 6.25x6.25x2.5

In this way we find it possible to fill a model with interpolated values calculated at a User Block Size, that is user block size 25x25x10 and still constrain the data within geological envelopes that are able to be sub-blocked to smaller sizes that is 6.25x6.25x2.5. This becomes important when considering the size of the model and the number of calculations to be performed to fill the model.

The model definition for the file Block_Model .MDL is:

Type	**Y**	**X**	**Z**
Min Coordinates	7100	1270	-10
Max Coordinates	770	2030	250
User Block Size	20	20	20
Min Block Size	5	5	5
Rotation	0	0	0

Attribute Name	**Type**	**Decimals**	**Background**	**Description**
Gold	Real	2	-99	Gold grade estimated from drill hole samples in database
sg	Real	2	-99	Specific Gravity of material

UBC Model File Formats

The UBC model requires two separate ASCII files. These are:

- *Mesh File* – 3D mesh defining the discretisation of the 3D model region
- *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
DE1   DE2   …….   DENE
DN1   DN2   …….   DNNN
DV1   DV2   …….   DVNV
```

where:

NE NN NV = Number of cells in the East, North, and vertical direction.

EO NO VO = Coordinates, in meters, 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).

DEn = Cell widths in the easting direction (from West to East).
DNn = Cell widths in the northing direction (from South to North).
DVn = 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 horizontal direction. The cell width in this area is usually uniform. The maximum depth of the mesh used for inversion should be large enough so that the no magnetic material below that depth would produce a noticeable anomaly with the length scale covered by the data area. A rule of thumb is that the maximum depth should be at least half of the longest side of the data area. Based upon the user’s knowledge of the survey area, one may adjust the maximum depth as necessary. The cell thickness in vertical direction usually increases slightly with depth. In the shallow region, the ratio of thickness to width of about 0.5 is good, especially when surface topography is present.

At depth, a cell thickness close to the cell width is advisable. Once this core mesh is designed, it can be extended laterally by padding with a few cells, possibly of variable width. This padding is necessary when the extracted anomalies are close to the boundary of the core mesh or if there are in influences from anomalies outside the area which cannot be easily removed.

The vertical position of the mesh is specified in elevation. This is to accommodate the inversion of a data set acquired over a topographic surface. When there is strong topographic relief and one wishes to incorporate it into the inversion, special care should be taken to design the mesh.

An example of a mesh file where a 10 10 5 mesh where each cell is 50m by 50m by 50m.

```
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 Suceptibility, abut 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.

C Grid Filters

Filtering of gridded surface data can be done in the frequency domain (using FFTs and the Grid Filter option) or in the spatial domain using convolution filter methods in the Grid Filter tool. A broad suite of smoothing (low-pass) filters, contrast enhancement filters, edge detection filters, geophysical and general high-pass filters, are provided. Several user-defined filters also enable you to create and apply your own designs.

In this section:

The grids supported by Encom Discover PA are listed in *Appendix E: Supported File Formats*.

The Convolution Filtering Process

Grids are comprised of equi-spaced data values located along rows and columns. The intersection of these rows and columns is called a mesh point or node. When filtering a grid, each grid node of the output grid is calculated as a function of the corresponding node and its neighbors. The size of the neighbourhood used in the filtering process is defined by the size and shape of the filter (or kernel). Filters are generally defined as a rectangular sub-array of nodes, which are assigned a set of filter weights. Because the filter neighborhood needs to be centered on a grid node during the filtering process, filters are generally defined by an odd number of rows and columns. For example, if the width and the height of the filter neighborhood are both three, then the neighborhood of the output grid node centred at (Row 40, Column 32) is the following rectangular sub-array is:

(39,33) (40,33) (41,32)
(39,32) (**40,32**) (41,32)
(39,31) (40,31) (41,31)

Shifting the filter neighbourhood across the grid and computing a new value for each grid node produces the filtered output of the image. If the heights of the neighborhood nodes are represented by H and the width by W, the number of nodes in the neighborhood equals H × W. Therefore, any nodes in the neighborhood can be defined as:

Filter Weight (`Row+i,Col+j`) where:

$$i = -\left[\frac{H}{2}\right], \ldots\ldots, \left[\frac{H}{2}\right] \quad \text{and} \quad j = -\left[\frac{W}{2}\right], \ldots\ldots, \left[\frac{W}{2}\right] \tag{1}$$

where each array is the largest integer less than or equal to its neighbour array.

When using the Grid Filter module, the Rows and Columns in the dialog specify the neighbourhood size of the selected filter. The weights for each grid node in the neighbourhood are displayed below. Each element of the matrix is used to weight the grid node that lies below it. The products are computed and then summed, normalised, and assigned to the value below the centre node. The filter is then shifted to the next node and the process is repeated until all nodes of the input grid have been processed.

Convolution Filter Descriptions

The filters supplied with the Grid Filter utility are text files (with file extension of .KER) and are located in the `\FILTER` folder of the main Grid Filter directory. The kernel files used are identical in format to those specified for use by ER Mapper.

Below is a list of the provided filter types and a brief summary of their application:

- *Smoothing Filters*
- *Enhancement Filters*

Smoothing Filters

- *Averaging Filters*
- *Gaussian Filters*

Averaging Filters

Averaging filters preserve the low frequency components in an image by reduce the amount of intensity variation between one grid cell value and the next. They achieve this by simply replacing each grid cell value in an image with the average (mean) value of its neighbours, including itself. Averaging filters have the effect of eliminating grid cell values, which are unrepresentative of their surroundings, thereby removing noise and smoothing its appearance. A 3×3 square kernel will generally provide sufficient smoothing for most datasets, although larger kernels (e.g. 5×5 or 7x7) can be used for more severe smoothing. In some situations applying a small (3x3) averaging filter to a grid several times may produce a slightly better result than filtering once with larger kernel.

Averaging filters can also be used to enhance the sharpness of an image or improve the appearance of edges (high frequency components). This technique is commonly known as the “Unsharp” filtering method. To sharpen an image using a smoothing filter, first apply the smoothing filter to the grid. Then subtract the smoothed grid from the original grid to produce the Unsharp image. This technique can be used for crispening the appearance of images prior to printing.

The averaging filters supplied with Grid Filter include:

3x3 Average
3x3 Diagonal
5x5 Average
7x7 Average
9x9 Average

Gaussian Filters

Gaussian convolution filters are smoothing filters that can be used to blur images, thereby removing high frequency detail and noise. The degree of smoothing produced by a Gaussian filter is largely determined by the standard deviation of the filter kernel. The Gaussian filters output a “weighted average” value for each grid cell's neighbourhood, with the average weighted more towards the value of the central grid cells. This is in contrast to an averaging filter, which uses a uniform weighting for all cells. Because of this property, a Gaussian filter provides gentler smoothing and preserves edges better than a similarly sized averaging filter. Because Gaussian smoothing filters remove high spatial frequency components from an image they are often used as a pre-processing step to edge enhancement filters such as the Laplacian and Sobel filters. In most situations a Gaussian filter will provide the best smoothing for grids with high frequency noise such as geochemistry or magnetics.

The Gaussian filters supplied with Grid Filter are:

3x3 Gaussian, Standard Deviation = 0.391
5x5 Gaussian, Standard Deviation = 0.625
5x5 Gaussian, Standard Deviation = 1.0
9x9 Gaussian, Standard Deviation = 1.0
11x11 Gaussian, Standard Deviation = 1.6

Enhancement Filters

- *Sharpening Filters*
- *Line and Edge Enhance Filters*
- *Laplacian Filter*
- *Laplacian of Gaussian*
- *Sobel Filters*
- *Sun Angle Filters*
- *Custom User-Defined Filters*

Sharpening Filters

Sharpening filters enhance areas of high spatial frequency or contrasting gradients in an image by removing the low frequency components. These filters can be useful for enhancing edges in an image as well as “sharpening” the overall appearance. Applying a sharpening filter to a digital terrain image prior to performing a lineament analysis for example, may help emphasise structural discontinuities.

Four general sharpening filters are provided with Grid Filter:

5x5 Horizontal Edge Enhance
5x5 Vertical Edge Enhance
3x3 Edge Sharpen
5x5 Edge Sharpen

Line and Edge Enhance Filters

Line and edge enhance filters are designed to selectively enhance image features with specific directional components (gradients). The filters output an approximation of the first derivative and therefore enhance edges in an image. The sum of the directional filter kernel elements is zero, so areas within an image with uniform intensity (or grid cell values) will compute to zero in the output grid. Areas of variable intensity or contrast will be amplified and appear as bright edges.

Grid Filter provides the following directional filters:

3x3 Horizontal Edge Enhance
3x3 Vertical Edge Enhance
3x3 Horizontal Line Enhance
3x3 Vertical Line Enhance

Laplacian Filter

Laplacian filters approximate a 2D isotropic measure of the second spatial derivative of an image. Therefore they highlight regions of rapid intensity change, which are often associated with edges. Laplacian filters emphasise maximum values within an image by using a kernel with a high central value, surrounded by negative weighted values with lower weights. Laplacian filters can be very sensitive to noise (high spatial frequency data) so it is often best to apply a Gaussian smoothing filter prior to convolving the image with the Laplacian filter. Laplacian filters are normally applied to scaled graylevel images.

Grid Filter provides the following Laplacian filters:

3x3 Laplacian
9x9 Laplacian

Laplacian of Gaussian

A Laplacian of Gaussian filter (LoG) is similar to a Laplacian filter, with the only exception being that it has already been convolved with a Gaussian filter. The advantage of using a LoG filter over the standard Laplacian filter is that you not required to smooth the grid prior to applying the LoG as the whole process is achieved in a single pass. This property makes the LoG filter faster to compute on large datasets. LoG filters calculate an approximation of the second spatial derivative of an image. Therefore areas in an image that have a constant intensity (or a gradient of zero) will produce values of zero in the Laplacian image, while areas of high intensity variation (or gradient) will produce positive or negative values. Adding the LoG filtered grid back to the original grid will have the effect of enhancing contrast in the original image and making edges appear much sharper.

Grid Filter provides the following LoG filter:

9x9 Laplacian of Gaussian (Standard Deviation = 1.4)

Sobel Filters

Like the Laplacian filter, the Sobel filter approximates a 2D spatial gradient measurement on an image and therefore emphasises regions of high spatial frequency. This type of filter is typically used to find the approximate absolute gradient magnitude at each point in an input grayscale image. Two Sobel filters are provided in Grid Filter. These are a pair of 3x3 filters, where one filter is simply the other rotated by 90°. The filters provided are designed to respond maximally to edges running vertically and horizontally relative to the grid cell orientation. One filter is supplied for each of the two perpendicular orientations (horizontal & vertical). The filters can be applied separately to the input grid, to produce separate measurements of the gradient component in each orientation or they can be combined together to find the absolute magnitude and orientation of the gradient at each point. The Sobel filter will often reduce edges in an input image to lines in the output image. This property makes the filter useful for assisting with lineament interpretations and structural mapping.

Grid Filter provides two Sobel filters:

Sobel Horizontal
Sobel Vertical

Sun Angle Filters

These filters provide directional enhancement to grid surfaces. The filter kernels are designed to amplify gradients perpendicular to the direction of perceived ambient lighting. For example, an East-West sun angle filter enhances frequency content for artificial illumination from the north or south. Supplied filters include:

North
North East
East
South East
South
South West
West
North West

Custom User-Defined Filters

The following three user-defined filters are supplied with Grid Filter:

General user-defined (mxn) filter
Average mxn filter
Gaussian mxn filter

These filters can be used as templates for designing and creating your own custom filters and offer a convenient way of experimenting with the properties of digital filtering. When you have designed a filter you wish to save click the Save As button and assign it a name. The filter is stored in a `\CUSTOM` sub-folder in the Filters directory.

Once a filter has been saved it will be displayed using the assigned name in the Available filters list and can be reapplied during a subsequent filtering session

One Dimensional FFT Line Filter Descriptions

- *The General Linear Filter*
- *The 1D FFT Vertical Continuation Filter*
- *General Expression for the 1D FFT Directional Derivative Filter*
- *The 1D FFT Vertical Derivative Filters*
- *The 1D FFT Horizontal Derivative Filters*
- *The General 1D Phase Transformation Filter*
- *The BT(k) to BH(k) Transformation Filter*
- *The BT(k) to Bz(k) Transformation Filter*
- *The 1D FFT Reduction to the North Magnetic Pole Filter*
- *The 1D FFT Reduction to the Magnetic Equator Filter*
- *The 1D FFT Pseudogravity Filter*
- *The 1D FFT Pseudomagnetic Filter*
- *The 1D FFT Low-Pass and High-Pass Butterworth Filters*

The General Linear Filter

A one dimensional line profile of scalar potential field data representing a harmonic scalar potential field function $F_1(k, z_0)$ may be transformed to another scalar potential field function $F_2(k, z_0)$ in the one dimensional wave-number domain via use of the convolution theorem, namely, if the transfer function is $G(k)$ then:

$$F_2(k;z_0) = G(k)\ F_1(k, z_0) \tag{2}$$

Here it is noted that $F_1(k, z_0)$ is the one-dimensional Fourier transform of the input scalar potential field function $f_1(x, z_0)$, namely,

$$F_1(k;z_0) = F_1(k_x, 0;z_0) = \int_{-\infty}^{\infty} f_1(x;z_0)\ \exp[-ikx]\, dx \tag{3}$$

where $k = k_x$ and $k_y \equiv 0$.

For measurements of a scalar potential field function $f1(x_i, 0; z=z_0)$ defined on a one-dimensional profile line $1 \leq i \leq N_x$, the one-dimensional discrete Fourier transform is defined as a complex Hermitian function $F_1(k_1, z_0)$ such that

$$F_1(k_1;z_0) = \sum_{i=1}^{N_x} f_1(x_i, z_0)\ \exp[-2\pi i k_1 x_i / N_x \Delta x] \tag{4}$$

where $x_i = (i - 1)\Delta x$ and the discrete kx wave-number is given by the relation:

$$k = k_x = 2\pi(k_1 - 1)/N_x\Delta x;\ 1 \leq k_1 \leq N_x \tag{5}$$

The 1D FFT Vertical Continuation Filter

The equivalent expression for the vertical continuation operator $G_{UDC}(k)$ for line profile potential field data, i.e. at a point $k = k_x$ in the 1D wave number domain $(k, 0, |k|)$, is given by the following relation

$$G_{UDC}(k) = \exp\{|k|\delta z\};\ \delta z < 0 \text{ for upward continuation to } z_1 \tag{6a}$$

$$G_{UDC}(k) = \exp\{|k|\delta z\};\ \delta z > 0 \text{ for downward continuation to } z_1 \tag{6b}$$

where $\delta z = (z_1 - z_0)$ is the change in vertical datum from z_0 to z_1 for the 1D line profile and $|k|$ is the modulus of the 1D wave number $k = k_x$

General Expression for the 1D FFT Directional Derivative Filter

The equivalent expressions for calculating the p th order ($p \geq 0$) directional derivative of a 2D harmonic scalar potential field function $F(x, y=0, z=z_0)$ in the 1D wave number domain are derived by setting $k = kx$ and $ky \equiv 0$ in equations (37) and (38) below. Hence for $-i$ forward Fourier transform, the expression for the complex gradient vector g_{1D} in the 1D wave number domain is

$$g_{1D} = [ikx, iky, |k|] = [ik, 0, |k|] \tag{7}$$

where $|k|$ is the modulus of the 1D wave number $k = k_x$. Also since the direction of measurement q is within the vertical plane containing the line profile then the direction cosines are now $q_{1D} = (q_x, 0, q_z)$. Whence the equivalent expression for the transfer function $G_{qdd}(k, z_0)$ for computing the p th order $(p \geq 0)$ directional derivative at a point $k = (kx, 0, |k|)$ in the 1D wave number domain is now given by the following vector dot (scalar) product relation:

$$G_{qdd}(k) = (g_{1D} \cdot q_{2D})^p \tag{8}$$

It is noted that the order p of the directional or partial derivative may be a whole or fractional non-negative number, i.e. $p \geq 0$.

The 1D FFT Vertical Derivative Filters

The transfer function $G_{zdd}(k)$ for the vertical derivative operator of order $p>0$ acting upon a grid of scalar potential field data in the 2D wave number domain is derived from equation (8) by putting the direction cosine vector $q_{1D} = (0, 0, 1)$. Hence:

$$G_{zdd}(k) = |k|p \text{ ; for } p > 0 \tag{9}$$

where $|k|$ is the modulus of the 1D wave number vector k. Whence the expression for the first vertical derivative operator (i.e. for $p = 1$) is:

$$G_{1vd}(k) = |k| \tag{10}$$

The 1D FFT Horizontal Derivative Filters

The transfer function $G(k)$ for the horizontal directional derivative operator of order $p > 0$ acting upon a line of scalar potential field data in the 1D wave number domain is derived from equation (8) by putting the direction cosine vector $q_{1D} = (1,0,0)$. Hence from equations (7) and (8),

$$G_{xdd}(k) = [ikx]p = [ik]p \qquad (11)$$

The General 1D Phase Transformation Filter

The equivalent expressions for transforming a 1D line profile of total magnetic field intensity (TMI) measurements $B_T(k;z_0)$ to a line profile of scalar magnetic field measurements $B_q(k;z_0)$ along a newly specified directional component q_{1D} with direction cosines $(q_x,0,q_z)$ in the 1D wave number domain are derived by setting $k = kx$ and $ky \equiv 0$ in equations (16) to (17) below.

$$G_{T2q}(k) = (g_{1D}\cdot q_{2D})/(g_{1D}\cdot f_{2D}) \text{ for } B_T(k;z_0) \rightarrow B_q(k;z_0) \qquad (12)$$

or in terms of the transformed 1D Hilbert operator $H_{1D} = g_{1D}/|k| = (H,0,1)$ where

$$H = H_x(k) = ik/|k| = i \text{ sgn } k \qquad (13)$$

Then,

$$G_{T2q}(k) = (H_{1D}\cdot q_{2D})/(H_{1D}\cdot f_{2D}) \text{ for } B_T(k;z_0) \rightarrow B_q(k;z_0) \qquad (14)$$

or in terms of Θ'_f and Θ'_q where $\Theta'_f = H_{1D}\cdot f_{2D}$ and $\Theta'_q = H_{1D}\cdot q_{2D}$, then

$$G_{T2q}(k) = \Theta'_q/\Theta'_f \text{ for } B_T(k;z_0) \rightarrow B_q(k;z_0) \qquad (15)$$

The BT(k) to BH(k) Transformation Filter

From the general 1D relations in equations (12) to (15) and after noting that the vector of direction cosines q_{2D} is $(1,0,0)$ for the in-line horizontal field component along the x axis, then for the transformation to $B_x(k)$ then:

$$G_{T2H}(k) = ik_x/(g_{1D}\cdot f_{2D}) = H_x/(H_{1D}\cdot f_{2D}) = H/\Theta'_f \qquad (16)$$

The BT(k) to Bz(k) Transformation Filter

Similarly, after noting that the vector of direction cosines q_{2D} is $(0,0,1)$ for the vertically down or Z_d field component, then for the transformation to $B_z(k)$ then

$$G_{T2z}(k) = |k|/(g_{1D}\cdot f_{2D}) = 1/(H_{1D}\cdot f_{2D}) = 1/\Theta'_f \tag{17}$$

The 1D FFT Reduction to the North Magnetic Pole Filter

The equivalent expressions for calculating the reduction to the pole (RTP) transfer filter for a line of total magnetic field intensity measurements in the 1D wave number domain are derived by setting $k = k_x$ and $ky \equiv 0$ in equations (18) to (21) below. The following relation applies:

$$G_{rtp}(k) = |k|^2/[((g_{1D}\cdot f_{2D})((g_{1D}\cdot m_{2D})] \tag{18}$$

where

> $g_{1D} = (ik,0,|k|)$ is the complex gradient vector in the 1D wave number domain
>
> $f2D = (f'_x,0,f'_z)$ is the vector of direction cosines for the local geomagnetic field vector expressed within the $[x',0,z]$ coordinate system of the line profile
>
> $m_{2D} = (m'_x,0,m'_z)$ is the vector of direction cosines for the resultant magnetization vector expressed within the $[x',0,z]$ coordinate system of the line profile.

The one-dimensional RTP transfer function may also be expressed in terms of the transformed 1D Hilbert operator $H_{1D} = g_{1D}/|k| = (H_x,0,1)$ as defined in equations 5.2a,b above:

$$G_{rtp}(k) = [(H_{1D}\cdot f_{2D})/(H_{1D}\cdot m_{2D})]^{-1} \tag{19}$$

or in terms of Θ'_f and Θ'_m where $\Theta'_f = H_{1D}\cdot f_{2D}$ and $\Theta'_m = H_{1D}\cdot m_{2D}$, then:

$$G_{T2q}(k) = 1/[\ \Theta'_f\ \Theta'_m] \tag{20}$$

And for the induced magnetization case in which the direction cosines m_{1D} of the resultant magnetization vector are identical to the direction cosines f_{1D} of the ambient geomagnetic field, then equation (47) becomes:

$$G_{rtp}(k) = 1/\Theta'^{2}_f \tag{21}$$

The 1D FFT Reduction to the Magnetic Equator Filter

The equivalent expressions for calculating the reduction to the pole (RTP) transfer filter for a line of total magnetic field intensity measurements in the 1D wave number domain are derived by setting $k = kx$ and $ky \equiv 0$ in equations (22) to (25) below. The following relationship applies:

$$G_{rte}(k) = (g_{H1D} \cdot n_{2D})^2 / [((g_{1D} \cdot f_{2D})((g_{1D} \cdot m_{2D})] \quad (22)$$

where

$g_{H1D} = (ik, 0, 0)$ is the horizontal complex gradient vector in the 1D wave number domain

$g_{1D} = (ik, 0, |k|)$ is the complex gradient vector in the 1D wave number domain

$f_{2D} = (f'_x, 0, f'_z)$ is the vector of direction cosines for the local geomagnetic field vector expressed within the $[x', o, z]$ coordinate system of the line profile

$m_{2D} = (m'_x, 0, m'_z)$ is the vector of direction cosines for the resultant magnetization vector expressed within the $[x', o, z]$ coordinate system of the line profile

$n_{2D} = (n'_x, 0, 0)$ is the vector of direction cosines for the specified true north or magnetic north direction within the $[x', o, z]$ coordinate system of the line profile

The one-dimensional RTE transfer function may also be expressed in terms of the transformed 1D Hilbert operator $H_{1D} = g_{1D}/|k| = (H, 0, 1)$ as defined in equation (22) above:

$$G_{rte}(k) = (H_{1D} \cdot n_{2D})^2 / [(H_{1D} \cdot f_{2D})(H_{1D} \cdot m_{2D})]^{-1} \quad (23)$$

or in terms of Θ'_f and Θ'_m where $\Theta'_f = H_{1D} \cdot f_{2D}$ and $\Theta'_m = H_{1D} \cdot m_{2D}$, then

$$G_{rte}(k) = \Theta'^2_n / [\Theta'_f \; \Theta'_m] \quad (24)$$

For the induced magnetization case in which the direction cosines m2D of the resultant magnetization vector are identical to the direction cosines f2D of the ambient geomagnetic field, then the expression for the 1D RTE operator becomes:

$$G_{rte}(k) = \Theta'^2_n / \Theta'^2_f \quad (25)$$

The 1D FFT Pseudogravity Filter

The equivalent expressions for calculating the 1D pseudogravity transfer filter for a line of total magnetic field intensity measurements in the 1D wave number domain are derived by setting $k = kx ; ky \equiv 0$ and $|k| = |k|$ in equations (52) and (53) above. The following relationship applies:

$$G_{psg}(k) = C_{psg}/(\Theta'_f \; \Theta'_m \; |k|) \tag{26}$$

where $\Theta'_f = H_{1D} \cdot f_{2D} ; Q'_m = H_{1D} \cdot m_{2D}$ and C_{psg} are the same quantities as those defined above. And for the induced magnetization case in which the direction cosines m_{2D} of the effective resultant magnetization vector are identical to the direction cosines f_{2D} of the ambient geomagnetic field, then equation (26) becomes

$$G_{psg}(k) = C_{psg}/(\Theta'^2_f \; |k|) \tag{27}$$

The 1D FFT Pseudomagnetic Filter

The equivalent expressions for calculating the 1D pseudomagnetic transfer filter for a line of total magnetic field intensity measurements in the 1D wave number domain are derived by setting $k = k_x ; k_y \equiv 0$ and $|k| = |k|$ in equations (55) and (56) below. The following relationship applies:

$$G_{psm}(k) = C_{psm} \; |k|/(\Theta'_f \; \Theta'_m) \tag{28}$$

where $Q'_f = H_{1D} \cdot f_{2D} ; Q'_m = H_{1D} \cdot m_{2D}$ and C_{psg} are the same quantities as those defined above. And for the induced magnetization case in which the direction cosines m_{2D} of the effective resultant magnetization vector are identical to the direction cosines f_{2D} of the ambient geomagnetic field, then equation 28 becomes:

$$G_{psm}(k) = C_{psm}|k|/\Theta'^2_f \tag{29}$$

The 1D FFT Low-Pass and High-Pass Butterworth Filters

The equivalent expressions for calculating the 1D low-pass and high-pass Butterworth filters are derived by setting $k = k_x ; k_y \equiv 0$ and $|k| = |k|$ in equations (69) and (70) below. The expression for the low-pass Butterworth filter of degree m and central wave number k_0 at a point $k = k_x$ in the 1D wave number domain is given by:

$$G_{lp}(k) = 1/[1+(|k|/k_0)^m] \tag{30}$$

where $k_0 = 2\pi/\lambda_0$ and λ_0 is the cutoff wavelength (metres). The expression for the high-pass Butterworth filter of degree m and central wave number k_0 at a point $k = k_x$ in the 1D wave number domain is given by

$$G_{hp}(k) = 1-G_{lp}(k) = (|k|/k_0)^m/[1+(|k|/k_0)^m] \qquad (31)$$

Two Dimensional FFT Filter Descriptions

Frequencies of filters are specified in terms of wavelengths (distance in metres). Cut-off rates determine the sharpness of the filter and the tapering of the energy spectrum. A high value of the cut-off has the effect of removing high frequencies, but causes ringing on the edges of large amplitude changes.

A theoretical description of the filters available in the Grid Filter option is provided below.

A 2D grid of scalar potential field data representing a harmonic scalar potential field function $F_1(k,z_0)$ may be transformed to another scalar potential field function $F_2(k,z_0)$ in the 2D wave-number domain via use of the convolution theorem, namely, if the transfer function is $G(k)$ then:

$$F_2(k;z_0) = G(k)F_1(k,z_0) \qquad (32)$$

Here it is noted that $F_1(k,z_0)$ is the two-dimensional Fourier transform of the input scalar potential field function, namely,

$$F_1(k,z_0) = F_1(k_x,k_y;z_0) = \iint_{-\infty}^{\infty} f_1(x,y,z_0)\exp[-i(k_x x + k_y y)]dxdy \qquad (33)$$

Whence for measurements of a scalar potential field function f1(xi,yj;z=z0) defined over the 2D grid 1 £ i £ Nx , 1 £ j £ Ny , the 2D discrete Fourier transform is defined as a complex Hermitian function F1(k1,k2;z0) defined over the same size (Nx*Ny) grid, namely,

$$F_1(k_1,k_2;z_0)=\sum_{i=1}^{N_x}\sum_{j=1}^{N_y} f_1(x_i,y_j,z_0)\exp[-2\pi i k_1 x_i/N_x\Delta x]\exp[-2\pi i k_2 x_j/N_y\Delta y]$$

$$F_1(k_1,k_2;z_0)=\sum_{i=1}^{N_x}\sum_{j=1}^{N_y} f_1(x_i,y_j,z_0)\exp[-2\pi i k_1 x_i/N_x\Delta x]\exp[-2\pi i k_2 x_j/N_y\Delta y]$$

where $x_i = (i - 1)\Delta x$ and $y_j = (j - 1)\Delta y$ and the discrete k_x, k_y wave-numbers are given by the relations:

$k_x = 2\pi(k_1-1)/N_x\Delta x$; $1 \le k_1 \le N_x$ and

$k_y = 2\pi(k_2-1)/N_y\Delta y$; $1 \le k_2 \le N_y$

The FFT Derivative Filters

- *Derivative Filter of nth Order (Advanced)*
- *The General Expression for FFT Directional Derivative Filters*
- *The FFT Vertical Derivative Filters*
- *The FFT Horizontal Derivative Filter*
- *The FFT TMI Reduction to the Pole Filter*
- *The FFT TMI Reduction to the Equator Filter*
- *The FFT Pseudo-gravity Transform Filter*
- *The 2D FFT Pseudo-magnetic Transform Filter*

Derivative Filter of nth Order (Advanced)

Derivative filters can be applied in any direction with options of East, North, Vertical or Any defined orientation. If any direction is selected, an Azimuth and Inclination need to be specified.

The horizontal derivative (East or North) can be used for creating shaded images. The operator used for this computation is simply:

$$L(f) = (i/f)n$$

where:

n = order of differentiation
f = the frequency component in the specified horizontal direction
i = square root of -1 (the imaginary component).

The vertical derivative is commonly applied to potential field data to enhance shallow geological sources. As with other filters that enhance the high frequency components of the spectrum, also apply low-pass filters to remove high frequency noise.

The General Expression for FFT Directional Derivative Filters

The transfer function $G(k, z_0)$ for computing the p^{th} order ($p \geq 0$) directional derivative of a harmonic scalar potential field function along a specified direction $q = (q_x, q_y, q_z)$ .at a point $k = (k_x, k_y)$ in the 2D wave number domain is given by the following vector dot (scalar) product relation (see for example, Blakely, 1995) :

$$G_{qdd}(k) = (g \cdot q)^p \tag{36}$$

where g is the complex gradient vector in the 2D wave number domain. For a –i forward Fourier transform (see above) the gradient operator g is defined as the complex vector:

$$g = [ik_x, ik_y, |k|] \tag{37}$$

where $|k|$ is the modulus of the 2D wave number vector k, namely,

$$|k| = [k_x^2 + k_y^2]^{1/2} \tag{38}$$

It is further noted that the order p of the directional or partial derivative may be a whole or fractional non-negative number, i.e. $p \geq 0$.

The FFT Vertical Derivative Filters

The transfer function $G_{zdd}(k)$ for the vertical derivative operator of order $p > 0$ acting upon a grid of scalar potential field data in the 2D wave number domain is derived from equation (36) by putting the direction cosine vector $q = (0, 0, 1)$. Hence:

$$G_{zdd}(k) = |k|^p \text{ ; for } p > 0 \tag{39}$$

where $|k|$ is the modulus of the 2D wave number vector k. Whence the expression for the first vertical derivative operator is

$$G_{1vd}(k) = |k| \tag{40}$$

The FFT Horizontal Derivative Filter

The transfer function $G(k)$ for the East (IGRF-y) or North (IGRF-x) horizontal directional derivative operator of order $p > 0$ acting upon a grid of scalar potential field data in the 2D wave number domain is derived from equation (36) by puting the direction cosine vector $q=(1,0,0)$ for the x-North direction or $q=(0,1,0)$ for the y-East direction. Hence

$$G_{xdd}(k) = [ik_x]^p \quad \text{the } x\text{-}N \text{ directional derivative of order } p > 0 \qquad (41)$$

and

$$G_{ydd}(k) = [ik_y]^p \quad \text{the } y\text{-}E \text{ directional derivative of order } p > 0 \qquad (42)$$

The FFT TMI Reduction to the Pole Filter

The reduction to the pole (RTP) filter is used to transform a grid of total magnetic field intensity measurements to a grid of magnetic intensity measurements that would be observed at the north magnetic pole. The expression for the RTP transform operator $G_{rtp}(k)$ at a point $k = (kx,ky)$ in the 2D in the wave number domain is given by Gibert and Guillamin (1985) and Blakely (1995) as follows :

$$G_{rtp}(k) = |k|^2/[(g \cdot f)(g \cdot m)] \qquad (43)$$

where

$g = (ik_x, ik_y, |k|)$ is the complex gradient vector in the 2D wave number domain

$f = (f_x, f_y, f_z)$ is the unit vector of direction cosines for the local geomagnetic field vector

$m = (m_x, m_y, m_z)$ is the unit vector of direction cosines for the resultant magnetization vector.

Here it is noted that the direction cosines are measured using the International Geomagnetic Reference Field (IGRF) coordinate system, namely, X-North; Y-East, Z – vertically down. Furthermore, it is possible to express the RTP transfer function in terms of the transformed 2D Hilbert operators $H_x(k), H_y(k)$ (Nabighian, 1983) in the $k = (k_x, k_y)$ wave number domain, namely,

$$H_x(k) = ik_x / |k| \; ; \; H_y(k) = ik_y / |k| \qquad (44)$$

After puting $H = g/|k| = H_x, H_y, 1)$, then:

$$G_{rtp}(k) = [(H \cdot f)(H \cdot m)]^{-1} \tag{45}$$

Or, after putting $\Theta_f = H \cdot f$ and $\Theta_m = H \cdot m$, in the notation of Blakely (1995, Ch 12), then:

$$G_{rtp}(k) = 1/[\Theta_f \ \Theta_m] \tag{46}$$

Whence for the induced magnetization case in which the direction cosines m of the resultant magnetization vector is identical to the direction cosines f of the ambient geomagnetic field, then equation (46) becomes:

$$G_{rtp}(k) = 1/\Theta_f^{\,2} \tag{47}$$

The FFT TMI Reduction to the Equator Filter

The reduction to the equator (RTE) filter is used to transform a grid of total magnetic field intensity measurements to a grid of magnetic intensity measurements that would be observed at the magnetic equator, i.e. where the inclination of the geomagnetic field is zero degrees. The expression for the RTE transform operator $G_{rte}(k)$ in the wave number domain is given by Gibert and Guillamin (1985) and Blakely (1995) as follows :

$$G_{rte}(k) = (g_H \cdot n)^2/[(g \cdot f)(g \cdot m)] \tag{48}$$

where

$g = (ik_x, ik_y, |k|)$ is the complex gradient vector in the 2D wave number domain

$g_H = (ik_x, ik_y)$ is the horizontal gradient vector in the 2D wave number domain

$f = (f_x, f_y, f_z)$ is the unit vector of direction cosines for the local geomagnetic field vector

$m = (m_x, m_y, m_z)$ is the unit vector of direction cosines for the resultant magnetization vector

$n = (n_x, n_y, 0)$ is the unit vector of direction cosines for the specified true north or magnetic north direction

As noted previously the direction cosines are measured using the International Geomagnetic Reference Field (IGRF) coordinate system. From equations (44) above it is possible to express the RTE transfer function in terms of the transformed 2D Hilbert operators $H_x(k), H_y(k)$ in the $k = (k_x, k_y)$ wave number domain, namely,

After puting $H = g/|k| = (H_x, H_y, 1)$, and $H_{2D} = g_H/|k| = (H_x, H_y)$, then:

$$G_{rte}(k) = (H_{2D} \cdot n)^2 / [(H \cdot f)(H \cdot m)] \tag{49}$$

Or, after puting $\Theta_f = H \cdot f$ and $\Theta_m = H \cdot m$ and $\Theta_n = H_{2D} \cdot n$, then:

$$G_{rte}(k) = \Theta_n^2 / [\Theta_f \ \Theta_m] \tag{50}$$

Whence for the induced magnetization case in which the direction cosines m of the resultant magnetization vector is identical to the direction cosines f of the ambient geomagnetic field, then equation (50) becomes

$$G_{rte}(k) = \Theta_n^2 / \Theta_f^2 \tag{51}$$

The FFT Pseudo-gravity Transform Filter

The pseudogravity filter is used to transform a grid of total magnetic field intensity measurements to a grid of vertical gravity component gz data. The expression for the pseudogravity transform operator $G_{psg}(k)$ at a point $k = (kx, ky)$ in the 2D wave number domain is given by Blakely (1995) as follows :

$$G_{psg}(k) = C_{psg} / (\Theta_f \ \Theta_m \ |k|) \tag{52}$$

where $\Theta_f = H \cdot f$ and $\Theta_m = H \cdot m$ are the same quantities as those defined above in and identical to those in Blakely (1995). The quantity C_{psg} is a constant given by

$$C_{psg} = (G/C_m)(r/M_{res}) \tag{53}$$

where

G is the Universal Gravitational Constant [= 6.6726 x 10-11 Nm^2/kg^2]
d is the density [kg/m^3]
M_{res} is the resultant magnetization in ampere/metre [A/m]
C_m is a constant equal to $\Theta_0 / 4\pi$ in henry/metre [H/m]
δ_0 is the magnetic permeability of free space = $4\pi \times 10^{-7}$ H/m.

Note for the induced magnetization case in which the direction cosines m of the resultant magnetization vector are identical to the direction cosines f of the ambient geomagnetic field, then equation (52) becomes

$$G_{psg}(k) = C_{psg} / (\Theta_f^2 \ |k|) \qquad (54)$$

The 2D FFT Pseudo-magnetic Transform Filter

The pseudomagnetic filter is used to transform a grid of vertical gravity component data $g_z(k, z_0)$ to a grid of total magnetic field intensity $B_t(k, z_0)$ data. The expression for the pseudomagnetic transform operator $G_{psm}(k)$ at a point $k = (k_x, k_y)$ in the 2D wave number domain is given by Blakely (1995) as follows:

$$G_{psm}(k) = C_{psm}|k| / (\Theta_f \ \Theta_m) \qquad (55)$$

where $\Theta_f = H \cdot f$ and $\Theta_m = H \cdot m$ are the same quantities as those defined above and in Blakely (1995). The quantity C_{psm} is a constant given by:

$$C_{psm} = (C_m/G)(M_{res}/r) = 1/C_{psg} \qquad (56)$$

where

G is the Universal Gravitational Constant [= 6.6726 x 10-11 Nm^2/kg^2]
r is the density [kg/m^3]
M_{res} is the resultant magnetization in ampere/metre [A/m]
C_m is a constant equal to $\mu_0 / 4\pi$ in henry/metre [H/m]
μ_0 is the magnetic permeability of free space = $4\pi \times 10^{-7}$ H/m.
C_{psg} is a constant given by equation (8.2).

Note for the induced magnetization case in which the direction cosines m of the resultant magnetization vector are identical to the direction cosines f of the ambient geomagnetic field, then equation (55) becomes

$$G_{psm}(k) = C_{psm}|k| / \Theta_f^2 \qquad (57)$$

FFT Magnetic Component or Phase Transformation Filters

The TMI General Phase Transformation Filter

The phase transformation filter is used to transform a grid of total magnetic field intensity (TMI) measurements BT(k;z0) to a grid of scalar magnetic field measurements $B_q(k;z_0)$ along a newly specified component direction q with direction cosines (q_x, q_y, q_z). The expression for the general phase transformation operator $G_{T2q}(k)$ at a point $k = (k_x, k_y)$ in the 2D wave number domain is given by the following relation (see for example, Blakely, 1995 or Schmidt and Clark, 1998):

$$G_{T2q}(k) = (g \cdot q)/(g \cdot f) \text{ for } B_T(k;z_0) \rightarrow B_q(k;z_0) \qquad (58)$$

or in terms of the transformed 2D Hilbert operator $H = g/|k| = (H_x, H_y, 1)$ as defined in equations 2.2a,b above:

$$G_{T2q}(k) = (H \cdot q)/(H \cdot f) \text{ for } B_T(k;z_0) \rightarrow B_q(k;z_0) \quad (59)$$

or in terms of Θf and Θq where $\Theta f = H \cdot f$ and $\Theta q = H \cdot q$, then:

$$G_{T2q}(k) = \Theta q/\Theta f \text{ for } B_T(k;z_0) \rightarrow B_q(k;z_0) \quad (60)$$

A pair of red-blue anaglyph glasses is available from Pitney Bowes Software upon request, which can be applied when viewing the output of the FFT TMI General Phase Transformation Filter. The use of these glasses enables the creation of stereo pairs from magnetic grids that allow direct depth perception.

The method is based on a technique proposed by Roger Clifton (Clifton, R. 2006, “Visualisating magnetic depths.” *AESC Conference*, 2-8 July, Melbourne). Our method allows for a direct east-west transformation of the magnetic grid which eliminates the need to rotate the images when using the reduction to pole method proposed by Clifton.

The BT(k) to Bx(k)Transformation Filter

From the general relations in (58 to 60) and after noting that unit vector of direction cosines q is $(1,0,0)$ for the North or X field component, then for the transformation to $B_x(k)$ then:

$$G_{T2x}(k) = ik_x/(g \cdot f) = H_x/(H \cdot f) = H_x/\Theta_f \quad (61)$$

The BT(k) to By(k)Transformation Filter

Similarly after noting that unit vector of direction cosines q is $(0,1,0)$ for the East or Y field component, then for the transformation to $B_y(k)$ then:

$$G_{T2y}(k) = ik_y/(g \cdot f) = H_y/(H \cdot f) = H_y/\Theta_f \quad (62)$$

The BT(k) to Bz(k)Transformation Filter

Similarly after noting that unit vector of direction cosines q is $(0,1,0)$ for the Depth or Z field component, then for the transformation to $B_z(k)$ then:

$$G_{T2z}(k) = |k|/(g \cdot f) = 1/(H \cdot f) = 1/\Theta_f \quad (63)$$

The FFT Vertical Continuation Filter

The vertical continuation filter is used to transform a 2D grid of scalar potential field (magnetic or gravity) measurements from one datum level at $z=z_0$ to another datum level at $z=z_1$ in either an upward or downward direction. The expression for the vertical continuation operator $G_{UDC}(k)$ at a point $k = (kx, ky)$ in the 2D wave number domain is given by the following relation (see for example, Blakely, 1995):

$$G_{UDC}(k) = \exp\{|k|\delta z\} \quad \delta z<0 \text{ for upward continuation to } z_1 \tag{64}$$

$$G_{UDC}(k) = \exp\{|k|\delta z\} \quad \delta z>0 \text{ for downward continuation to } z_1 \tag{65}$$

where $\delta z = (z_1 - z_0)$ is the change in vertical datum from z_0 to z_1 for the 2D grid and $|k|$ is the modulus of the 2D wavenumber vector k as defined in equation (38). Here it is noted that z is measured positive downwards which is in keeping with both the IGRF and the Encom grid coordinate systems.

The 2D FFT Low-Pass and High-Pass Butterworth Filters

The Butterworth filter is used to apply either low-pass or high-pass filtering to a grid of transformed potential field measurements. The high- or low-pass filters are employed to reject regional or residual scale features respectively in grids of scalar potential field data. The rate of amplitude roll-off for the Butterworth filter is controlled by its degree m whereas its cutoff wavelength is specified via the central wave number k_0 or wavelength λ_0 parameter, namely, $k_0 = 2\pi/\lambda_0$ (i.e. the half-amplitude point). The expression for the low-pass Butterworth filter of degree m and central wave number k_0 at a point $k = (k_x, k_y)$ in the 2D wave number domain is given by:

$$G_{lp}(k) = 1/[1+(|k|/|k_0|)^m] = 1/[1+(k/k_0)^m] \tag{66}$$

where

$$k = |k| = [k_x^2+k_y^2]^{1/2} \; ; \; k_0 = |k_0| = 2\pi/\lambda_0 \tag{67}$$

and λ_0 is the cutoff wavelength (metres).

The expression for the 2D high-pass Butterworth filter of degree m and central wave number k_0 at a point $k = (k_x, k_y)$ in 2D the wave domain is given by

$$G_{hp}(k) = 1-1/[1+(|k|/|k_0|)^m] = 1/[1+(k/k_0)^m] \tag{68}$$

The FFT Band-Pass and Band-Reject Directional Cosine Filters

The directional filter is used to pass or reject regional or residual scale features in grids of transformed scalar potential field data. These features have a preferred azimuth or strike direction which is specified by an angle α_0 while the angular bandwidth of the directional cosine filter is controlled by its degree m. The expression for the response function of a band-pass directional cosine filter of degree m and azimuthal strike direction α_0 (the maximum amplitude point) at a point $k = (k_x, k_y)$ in the 2D wave number domain is given by

$$G_{bpaz}(k) = (|\cos\{\alpha_0 - \theta(k)\}|)^m \tag{69}$$

where

$$\theta(k) = \arctan\{k_x/k_y\} \ ; \ -\pi/2 \le \theta(k) \le \pi/2 \tag{70}$$

The expression for the response function of a band-reject directional cosine filter of degree m and azimuthal strike direction α_0 (the zero amplitude point) at a point $k = (k_x, k_y)$ in the 2D wave number domain is given by

$$G_{braz}(k) = 1-(|\cos\{\alpha_0 - \theta(k)\}|)^m \tag{71}$$

The First Order Magnetic Moment Transformation

Consider a disturbing magnetic body or magnetic source centred at (x_c, y_c, z_c) whose anomalous magnetic field is $\Delta B(r) = [\Delta B_x(r), \Delta B_y(r), \Delta B_z(r)]$. Helbig (1963) has shown that the magnetic dipole moment $M = (M_x, M_y, M_z)$ of the disturbing magnetic source may be calculated from the components of the anomalous magnetic field at any point $r = (x, y, z_0)$ in the horizontal plane $z = z_0$. In this instance the three components M_x, M_y, M_z of the zero-th order magnetic moment M are the first order moments of $\Delta B_z(r)$, about the x, y axes and the first order moments of $\Delta B_x(r)$ and $\Delta B_y(r)$ about the x, y axes respectively. These relations between the magnetic moment of the body and the first order moments of the magnetic field may be expressed through the following double integrals, namely, for the M_x moment:

$$M_x = (1/2\pi)\iint_{-\infty}^{\infty} (x - x_0)\Delta B_z(x, y, z_0)\,dx\,dy \tag{72}$$

and for the My moment,

$$M_y = (1/2\pi)\iint_{-\infty}^{\infty} (y - y_0)\Delta B_z(x, y, z_0)\,dx\,dy \tag{73}$$

and for the Mz moment

$$M_z = (1/2\pi)\iint_{-\infty}^{\infty}(x - x_0)\Delta B_x(x, y, z_0)\,dx\,dy \tag{74}$$

or alternatively

$$M_z = (1/2\pi)\iint_{-\infty}^{\infty}(y - y_0)\Delta B_y(x, y, z_0)\,dx\,dy \tag{75}$$

Furthermore and importantly it is also noted that the following pair of integrals are identically zero.

$$M_{xRy} = (1/2\pi)\iint_{-\infty}^{\infty}(x - x_0)\Delta B_y(x, y, z_0)\,dx\,dy = 0 \tag{76}$$

and also,

$$M_{yRx} = (1/2\pi)\iint_{-\infty}^{\infty}(y - y_0)\Delta B_x(x, y, z_0)\,dx\,dy = 0 \tag{77}$$

By inspection of equations (72 to 75), it is noted that the magnetic moment M is completely determined from measurements of a single pair of magnetic field components, i.e. either from $\Delta B_x(x,y,z_0)$, $\Delta B_z(x,y,z_0)$ or from $\Delta B_y(x,y,z_0)$, $\Delta B_z(x,y,z_0)$. In practice however a more stable estimator of the vertical component M_z of the magnetic moment is

$$M_z = [M_{z(x)}+M_{z(y)}]/2 \tag{78}$$

where $M_{z(x)}$ and $M_{z(y)}$ are given by equations (74) and (75) respectively. Also a further necessary condition for the existence of a reliable magnetic moment is that the integrals in equations (76) and (77) are both approximately zero, namely,

$$M_{x,By} \cong M_{y,Bx} \cong 0 \tag{79}$$

Other magnetic moment components which can be used are the horizontal moment,

$$M_H = [M_x{}^2+M_y{}^2]^{1/2} \tag{80}$$

and the total magnetic moment,

$$M_T = |M| = [M_x^2+M_y^2+M_z^2]^{1/2} = [M_H^2+M_z^2]^{1/2} \quad (81)$$

Furthermore the declination D_m and inclination Im of the magnetic moment vector M are given by

$$D_m = \arctan [M_x/M_y] \text{ for } 0 \leq D_m < 2\pi \quad (82)$$

$$I_m = \arctan [M_H/M_z] \text{ for } 0 \leq I_m < 2\pi \quad (83)$$

It is noted that the above formulae are for the zero order magnetic dipole moment of the disturbing body under the plane $z= z_0$ (see Helbig, 1963, p. 84). This is related to the reduced multipole moments of Grant (1952) and Grant and West (1965, p.222-225). Expressions exist for the zero, first and second order dipole moments of various simple bodies including spheres, ellipsoids of revolution, triaxial ellipsoids, finite length elliptic cylinders, rectangular prisms and wedges. Medeiros and Silva (1995) have estimated the total intensity, direction of magnetization and the 3D spatial orientation of a seamount source using inversion based on dipole moments up to second order.

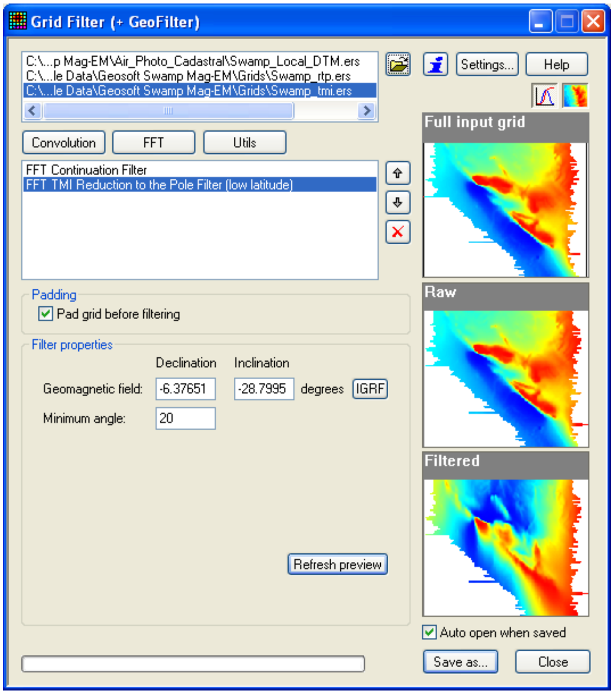
FFT TMI Reduction to the Pole (Low Latitude) Processing

In addition to the standard RTP filter provided for Reduction-To-the-Pole processing (see above), an additional filter specifically for use with magnetic data acquired at low magnetic latitudes is provided. This FFT filter uses analytic signal computation to calculate simplified responses from magnetic data acquired at magnetic latitudes less than approximately 30° of inclination. As the analytic signal computation produces response maxima over magnetic contacts irrespective of the direction of magnetisation, this can be used to assist the processing of observed magnetic data acquired near the magnetic equator.

The formulation used for the low latitude computation is described in *Low Latitude Processing Theory*.

The Low Latitude filter dialog appears as below. Operation of the filter computes both a phase and an amplitude component. A compensation applied to the phase component can be used to prevent the amplitude increasing and causing the equation to become unstable. This instability is primarily due to any anomalies oriented north-south within the observed data.

The Minimum angle factor defaults to a value of 20° but in the filter's implementation, if it specified less than the data's inclination (I), then the Minimum angle correction is reset to the value of 20° to maintain stability.



Application of the Low Latitude filter

Use the IGRF calculator to automatically assign the Declination and Inclination. A default Minimum angle of 20° is assigned.

The effect of changing the Minimum angle factor alters the relative sizes of the phase and amplitude. If the factor is increased (to a maximum of 90°), the phase component increases and amplitude adjustment is reduced (to be zero at 90°). If the correction factor is reduced, the phase and amplitude components become evenly applied.

Low Latitude Processing Theory

The Reduction-To-the-Pole (RTP) filter is used to transform a grid of total magnetic field intensity measurements to a grid of magnetic intensity measurements that would be observed at the north magnetic pole. The expression for the RTP transform operator `Grtp(k)` at a point `k = (kx,ky)` in the 2D wave number domain is given by Gibert and Guillamin (1985) and Blakely (1995) as follows:

$$G_{rtp}(k) = |k|^2/[(g\cdot f)(g\cdot m)] \quad (84)$$

where,

> $g = (ik_x, ik_y, |k|)$ is the complex gradient vector in the 2D wave number domain
>
> $f = (f_x, f_y, f_z)$ is the unit vector of direction cosines for the local geomagnetic field vector
>
> $m = (m_x, m_y, m_z)$ is the unit vector of direction cosines for the resultant magnetization vector.

Here it is noted that the direction cosines are measured using the International Geomagnetic Reference Field (IGRF) coordinate system, namely, X-North; Y-East, Z-vertically down. Furthermore, it is possible to express the RTP transfer function in terms of the transformed 2D Hilbert operators $H_x(k)$, $H_y(k)$ (Nabighan, 1983) in the $k = (k_x, k_y)$ wave number domain, namely,

$$H_x(k) = ik_x/|k| = i\cos\alpha(k)$$

$$H_y(k) = ik_y/|k| = i\sin\alpha(k) \qquad (85)$$

where $\alpha(k)$ is the horizontal azimuth of a point $k = (k_x, k_y)$ in the 2D wave number domain,

$$\alpha(k) = \arctan k_y/k_x \qquad (86)$$

Then after putting $H = g/|k| = (H_x, H_y, 1)$, we have omitting the k dependence of $H(k)$, $H_x(k)$ and $H_y(k)$ from now on

$$G_{rtp}(k) = [(H \cdot f)(H \cdot m)]^{-1} \qquad (87)$$

Or, after putting $\Theta_f = H \cdot f$ and $\Theta_m = H \cdot m$, we have in the notation of Blakely (1995, Ch 12) and omitting the k dependence of $\Theta_f(k)$ and $\Theta_m(k)$ from now on

$$G_{rtp}(k) = 1/[\Theta_f \ \Theta_m] \qquad (88)$$

Whence for the induced magnetization case in which the direction cosines m of the resultant magnetization vector are identical to the direction cosines f of the ambient geomagnetic field, then equation (88) becomes:

$$G_{rtp}(k) = 1/\Theta_f^2 \qquad (89)$$

Or, noting that $\Theta_f = H \cdot f$ is a complex quantity with complex conjugate $[\Theta_f^2]^* = [\Theta_f^*]^2 = (H^* \cdot f)^2$, we have multiplying the numerator and denominator by:

$$[\Theta_f^2]*G_{rtp}(k) = [\Theta_f*]^2/|\Theta_f|^4 \quad (90)$$

The direction cosines f of the geomagnetic field F are given by:

$$f = (f_x, f_y, f_z) = (\cos I_f \cos D_f, \cos I_f \cos D_f, \sin I_f) \quad (91)$$

where D_f is the declination of the geomagnetic field and If is its inclination. Now if we expand the expression for Θ_f we have,

$$\Theta_f = H \cdot f = [f_z + H_x f_x + H_y f_y] \quad (92)$$

or

$$\Theta_f = H \cdot f = [\sin I_f + \cos I_f [H_x \cos D_f + H_y \sin D_f] \quad (93)$$

And, after substitution for H_x and H_y from equations (1.2a,b) above, we have

$$\Theta_f = H \cdot f = [\sin I_f + i \cos I_f [\cos\alpha \cos D_f + \sin\alpha \sin D_f]$$

Hence, it is easily shown that

$$\Theta_f = H \cdot f = [\sin I_f + i \cos I_f \cos(D_f - \alpha)] \quad (94)$$

and

$$|\Theta_f|^4 = [\Theta_f \ \Theta_f*]^2 = [\sin^2 I_f + \cos^2 I_f \cos^2(D_f - a)]^2 \quad (95)$$

From equations (94 and 95) it is easily deduced that the transfer function for reduction to the pole $G_{rtp}(k)$ in a zero inclination If geomagnetic field is given by the relation,

$$G_{rtp}(k) = [\Theta_f*]^2/|\Theta_f|^4 = 1/\cos^2(D_f - \alpha) \text{ for } I_f = 0 \quad (96)$$

Therefore if the angle $|D_f - \alpha|$ is an odd multiple of? $\pi/2$?such that $\cos(D_f - \alpha)$ is zero, then $G_{rtp}(k)$ will become infinitely large. Even at low geomagnetic latitudes (say $0° < |I_f| < 20°$) there is very significant directionally selective amplification of any noise present within a grid of total magnetic intensity measurements (see for example, Blakely, 1995; Silva, 1989, Hansen and Pawlowski,1989). This amplification has the effect of producing short wavelength artifacts which trend parallel to the direction of the declination D_f (Blakely, 1995).

One means of overcoming this difficulty is to retain the phase information in the numerator term $[\Theta_f{*}]^2$ above but make a modification to the denominator $|\Theta_f|^4$ in equation (95 etc.) which prevents it from becoming too large. This may be achieved by introducing a second inclination angle $I_{f'}$ (referred to as the Minimum angle factor) as follows (see for example, McLeod et al, 1993):

$$G_{rtp}(k) = [\Theta_f{*}]^2/|\Theta_f|^2|\Theta_{f'}|^2 \ ; \text{ for } |I_f| < I'_f \qquad (97)$$

where

$$\Theta_{f'} = H{\cdot}f' = [\sin I'_f + i\cos I_{f'}\cos(D_f-\alpha)] \text{ for } |I_f|<I'_f \qquad (98)$$

Whence from equations (95, 97 and 98), the modified expression for the reduction to the pole operator at low latitudes, i.e. for $|I_f| < I'_f$, is given by,

$$G_{rtp}(k) = \frac{[\sin I_f - i\cos I_f\cos(D_f-\alpha)]^2}{[\sin I_f + \cos^2 I_f\cos^2(D_f-\alpha)][\sin^2 I_{f'} + \cos^2 I_{f'}\cos^2(D_f-\alpha)]}$$

Aspects of Numerical Computation

In most cases, magnetic survey data will consist of raw total magnetic intensity (TMI) measurements $B_m(x_i,y_j;z_0)$ which have been interpolated over a two-dimensional grid $1 \le i \le n_x$, $1 \le j \le n_y$. Therefore, in order to calculate the magnetic moments using Helbig's method, calculate the anomalous magnetic field components $\Delta B_x(x_i,y_j;z_0)$, $\Delta B_y(x_i,y_j;z_0)$, $\Delta B_z(x_i,y_j;z_0)$ from the original TMI data. This involves several steps namely:

1. Anomaly isolation.

2. Removal of a constant background field $B_p(x_i,y_j;z_0)$ from the original TMI data $B_m(x_i,y_j;z_0)$ to reveal the raw total magnetic field anomaly $\Delta B_{raw}(x_i,y_j;z_0)$, i.e.

$$\Delta B_{raw}(x_i,y_j;z_0) = B_m(x_i,y_j;z_0) - B_p(x_i,y_j;z_0).$$

3. Computation of a regional field $\Delta B_{reg}(x_i,y_j;z_0)$ for the raw anomalous total field $\Delta B_{raw}(x_i,y_j;z_0)$ over the area of interest.

4. Removal of the computed regional field $\Delta B_{reg}(x_i,y_j;z_0)$ from raw anomalous total field $\Delta B_{raw}(x_i,y_j;z_0)$ to reveal the final corrected total magnetic field anomaly $\Delta B_{cor}(x_i,y_j;z_0)$, i.e.

$$\Delta B_{cor}(x_i,y_j;z_0) = \Delta B_{raw}(x_i,y_j;z_0) - \Delta B_{reg}(x_i,y_j;z_0)$$

5. Pad the grid of corrected total magnetic field anomaly measurements $\Delta B_{cor}(x_i, y_j; z_0)$ ready for 2D Fourier transformation [new grid size is N_x*N_y].

6. Fourier transform the corrected anomalous TMI grid $\Delta B_{cor}(x_i, y_j; z_0)$ and keep a copy of it [see equation (32) for the 2D discrete Fourier Transform above].

7. Apply the $B_T(k)$ to $B_x(k)$ (x-North) transformation filter $G_{T2x}(k)$ to the Fourier transformed TMI grid, namely, compute

   $\Delta B_x(k) = G_{T2x}(k)\ \Delta B_T(k)$, where the transfer function

   $G_{T2x}(k) = ik_x/(g \cdot f) = H_x/(H \cdot f) = H_x/\Theta_f$

   Apply the $B_T(k)$ to $B_y(k)$ (y-East) transformation filter $G_{T2x}(k)$ to the Fourier transformed TMI grid, namely, compute

   $\Delta B_y(k) = G_{T2y}(k)\ \Delta B_T(k)$, where the transfer function

   $G_{T2y}(k) = ik_y/(g \cdot f) = H_y/(H \cdot f) = H_y/\Theta_f$.

   Apply the $B_T(k)$ to $B_z(k)$ (z-Down) transformation filter $G_{T2x}(k)$ to the Fourier transformed TMI grid, namely, compute

   $\Delta B_z(k) = G_{T2z}(k)\ \Delta B_T(k)$, where the transfer function

   $G_{T2z}(k) = |k|/(g \cdot f) = 1/(H \cdot f) = 1/\Theta_f$.

8. Apply the inverse 2D Fourier transform to each of the three transformed grids of magnetic component data ,namely, $\Delta B_x(k)$, $\Delta B_y(k)$, $\Delta B_z(k)$ in the x-north, y-east, z-down directions respectively.

9. Extract the real component parts of the inverse Fourier transformed grid scaled by the size of the padded grid, i.e. by the factor $1/(N_x*N_y)$. Importantly the extracted grids containing the anomalous magnetic field components $\Delta B_x(x_i, y_j; z_0)$, $\Delta B_y(x_i, y_j; z_0)$, $\Delta B_z(x_i, y_j; z_0)$ should exactly equivalent in size and location to the original and corrected TMI grids ,e.g. $\Delta B_{cor}(x_i, y_j; z_0)$.

10. Remove any DC trends from the $\Delta B_x(x_i, y_j; z_0)$, $\Delta B_y(x_i, y_j; z_0)$, $\Delta B_z(x_i, y_j; z_0)$ grids This is to ensure that the zero order moments of the $\Delta B_x(x_i, y_j; z_0)$, $\Delta B_y(x_i, y_j; z_0)$ and $\Delta B_z(x_i, y_j; z_0)$ field components are zero (see Helbig, 1963). The data are now ready for magnetic moment computation.

Computation of the Magnetic Moments

In order to calculate the magnetic moments using Helbig's method, replace the continuous double integrals in equations (72 to 75) with their discrete equivalents. This involves numerical integration of the anomalous magnetic field components $\Delta B_x(x_i, y_j; z_0)$, $\Delta B_y(x_i, y_j; z_0)$, $\Delta B_z(x_i, y_j; z_0)$ defined over the two-dimensional grid $1 \le i \le n_x$, $1 \le j \le n_y$. Here it is assumed that the anomaly of interest is quite compact and isolated. Furthermore, it assumes that the anomalous field components have been sufficiently closely sampled over a wide enough area of interest so that the first order moments can be accurately determined. A number of tests can be conducted to ensure this is approximately true. First the zero order moments of DBx(xi,yj;z0), DBy(xi,yj;z0) and DBz(xi,yj;z0) should be approximately zero as should the two cross moments Mx,By and My,Bx in equations (76) and (77) respectively. The double integrals in equation (72 to 77) may be each be evaluated numerically using a double application of the extended Simpson's Rule, closed 1D formula (Press et al, 1992, Ch 4, pp.127-129, equation 4.1.13) in the x and y grid directions. For the x grid direction then:

$$\int_{x_1}^{x_n} f(x)dx = h\left[\frac{1}{3}f_1 + \frac{4}{3}f_2 + \frac{2}{3}f_3 + \frac{4}{3}f_4 + \ldots \right.$$
$$\left. \ldots + \frac{2}{3}f_{n-2} + \frac{4}{3}f_{n-1} + \frac{1}{3}f_n\right] + O([x_n - x_1]^5 f''''/n^4)$$

$$\int_{x_1}^{x_n} f(x)dx = h\left[\frac{1}{3}f_1 + \frac{4}{3}f_2 + \frac{2}{3}f_3 + \frac{4}{3}f_4 + \ldots \right.$$
$$\left. \ldots + \frac{2}{3}f_{n-2} + \frac{4}{3}f_{n-1} + \frac{1}{3}f_n\right] + O([x_n - x_1]^5 f''''/n^4)$$

where f'''' indicates the fourth derivative of the function $f(x)$.

Once the numerical integrals corresponding to those in equations (72 to 77) have been computed the M_x, M_y, M_z computes of the magnetic moment may be calculated. The M_z magnetic moment is best computed using equation (78). The total magnetic moment M_T, its declination D_M and inclination I_M are calculated using equations (80 to 83). The declination and inclination are equivalent to the average values for the source body magnetization. The computed value of the magnetic moment may be increasingly underestimated as the grid size becomes progressively smaller. Schmidt and Clark (1998) have attempted to compensate for this in their paper (see the correction factors in Table 1).

Low Pass Filter

The FFT Low Pass Filter applies an operator to remove high frequency content with wavelengths above (that is, smaller than the defined wavelength cut-off). The cut-off rate specifies the severity of the filter at its wavelength margins. The higher the cut-off, the greater and the sharper the cut-off effect of removing a particular wavelength cut-off.

High Pass Filter

This FFT filter is the converse of the Low Pass filter. The High Pass filters pass frequencies that are higher than the specified cut-off.

Band Pass Filter

Band Pass filters remove wavelengths that lie between two specified wavelength limits.

A common cut-off is applied for both the high and low ranges of the wavelengths. Applying a simple cutoff filter to an energy spectrum (such as a Band Pass filter) almost invariably introduces a significant amount of ringing (referred to as the Gibbs' Phenomena).

Encom GeoFilters

In this section:

- *About Encom GeoFilters*
- *Edge Filters*
- *Block Filters*
- *Applying GeoFilters*

About Encom GeoFilters

The Encom GeoFilter filters are a specialist group of filters provided with Encom Discover PA. The Geofilters are only provided with the Utility Option of Encom Discover PA and require licensing of this module to be enabled before access is provided. A complete reference for these filters is provided in Zhi and Butt (2004)[1].

The GeoFilters comprise two types of filters that have been developed for the purpose of enhancing weak magnetic anomalies from near-surface sources while simultaneously enhancing low-amplitude, long-wavelength magnetic anomalies from deep-seated or regional sources. The Edge filter group highlights edges surrounding both shallow and deeper magnetic sources. The results are used to infer the location of the boundaries of magnetized lithologies. The Block filter group has the effect of transforming the data into zones which, similar to image classification systems, segregate anomalous zones into apparent lithological categories. Both filter groups change the textural character of a dataset and thereby facilitate interpretation of geological structures.

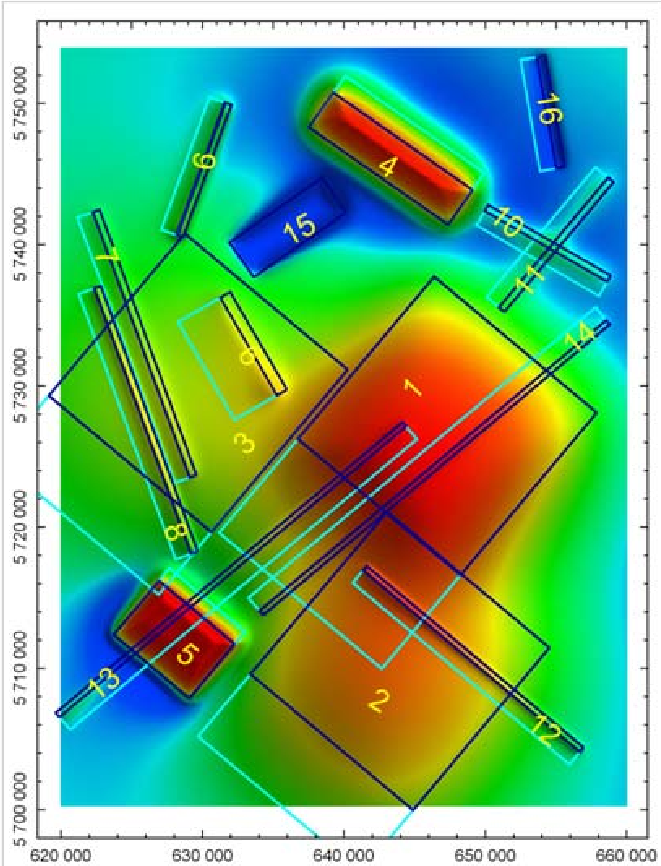
The effect of each filter is demonstrated using theoretical model studies in the paper by Zhi and Butt. The models include both shallow and deep sources with a range of magnetizations. Comparative studies are made with traditional filters using the same theoretical models.

Edge Filters

The Edge Filters increase the sharpness of anomalies used to map the edge of magnetic sources. The modulus of horizontal derivatives (MS) of grids yields anomaly peaks over the source edge locations, whereas these edges coincide with gradients in the First Vertical Derivative (1VD), Tilt and Analytic Signal (AS) filtered outputs. None of these filters produces easily interpreted edges in image form when the sources are weakly magnetized or are deep.

A new linear, derivative-based filter termed the ZS-Edgezone filter has been developed to improve edge detection in these situations. The advantages of the filter are greatly increased anomaly sharpness over source edges and compression of the amplitude range so that differences in the original TMI amplitudes do not persist to dominate the edge interpretation. This has the ancillary effect that the method can be modified to provide automated edge conversion to vectors for use in GIS systems.

1. Zhi, S. and Butt, G.R. 2004. “New Enhancement Filters for Geological Mapping”, *ASEG 17th Annual Conference Edition*, Preview, Sydney 2004



RTP image derived from multiple theoretical 3D magnetic sources, shown as wire frame outlines

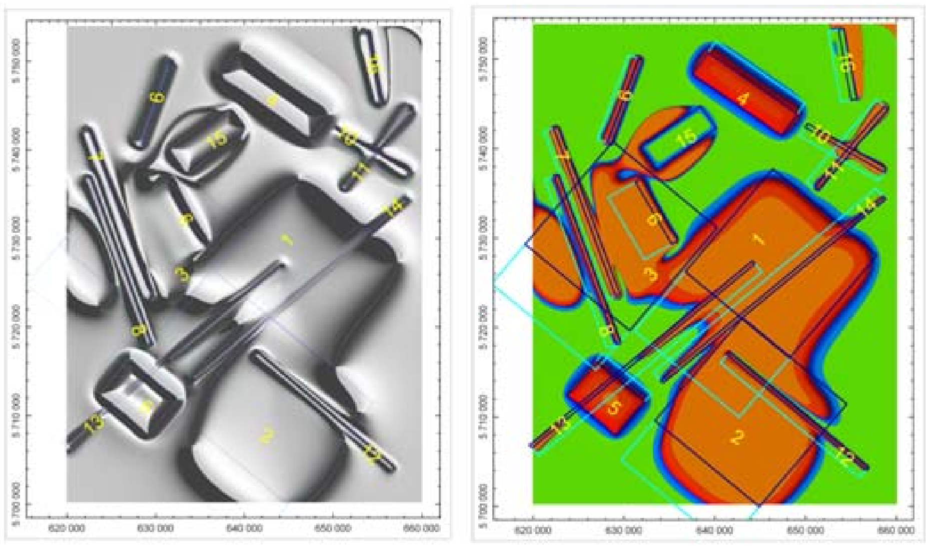
Although this filter significantly improves the precision of edge determination, it is subject to normal potential field limitations which determine that source edges cannot be resolved where the source is narrow relative to its depth. The filter also can produce a halo type artefact due to superposition of the response of a limited depth extent shallow source (see Model 6 in the diagram above) on that of deeper sources.

Block Filters

In attempting to improve edge detection filters, an obvious progression is to highlight the magnetic regions whose edges have been mapped. To do this, a set of filters called block filters has been developed.

The Block filter group has the effect of transforming the potential field data into zones which, similar to image classification systems, segregate anomalous zones into apparent lithological categories. These filters can be imported for use in image classification systems or displayed in RGB space with other grids for empirical classification purposes.

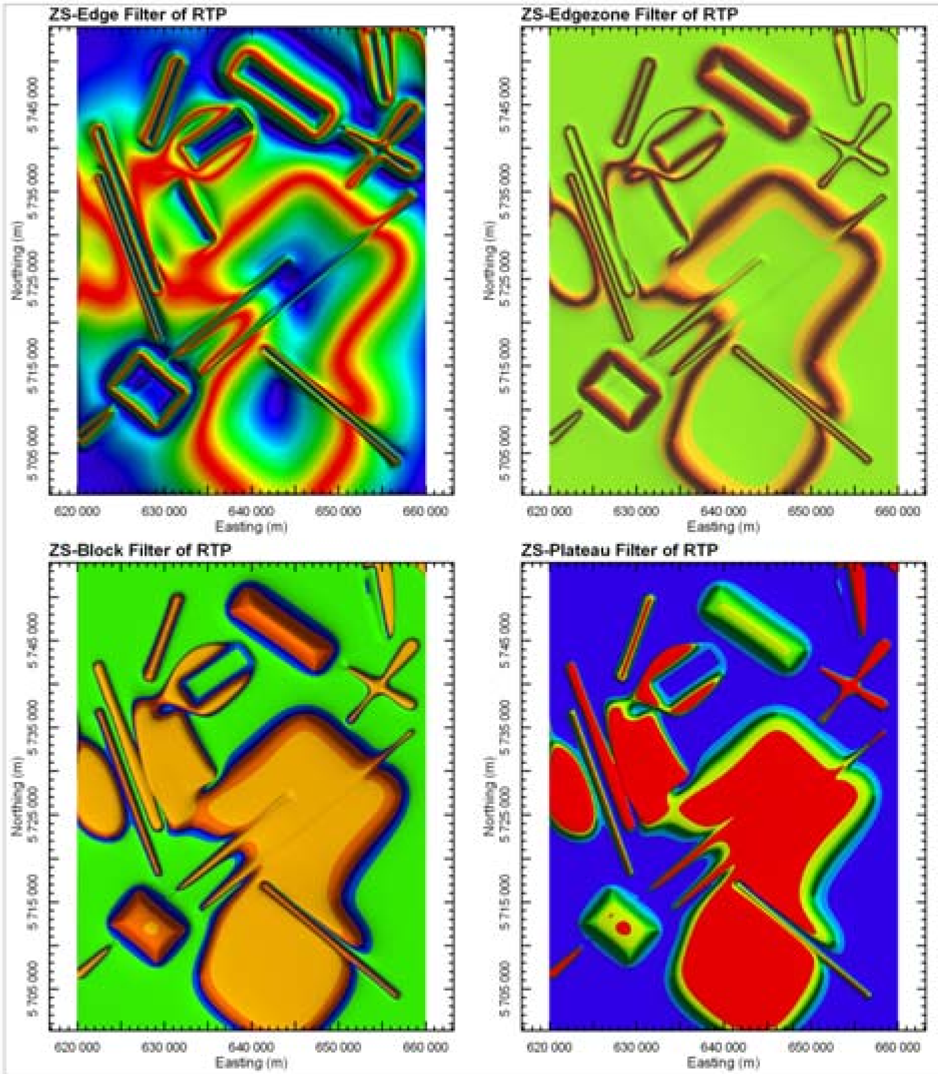
The block filters, like the edge filters, are linear, derivative-based filters which use a combination of derivative and amplitude compression techniques to render the magnetic data into regions whose edges are sharply defined and whose amplitudes have a reduced range in comparison to the original TMI.



Anomaly edge and block enhancements using the ZS-Edgezone (left) and ZS-Block filters (right). Model positions are shown using wire frames

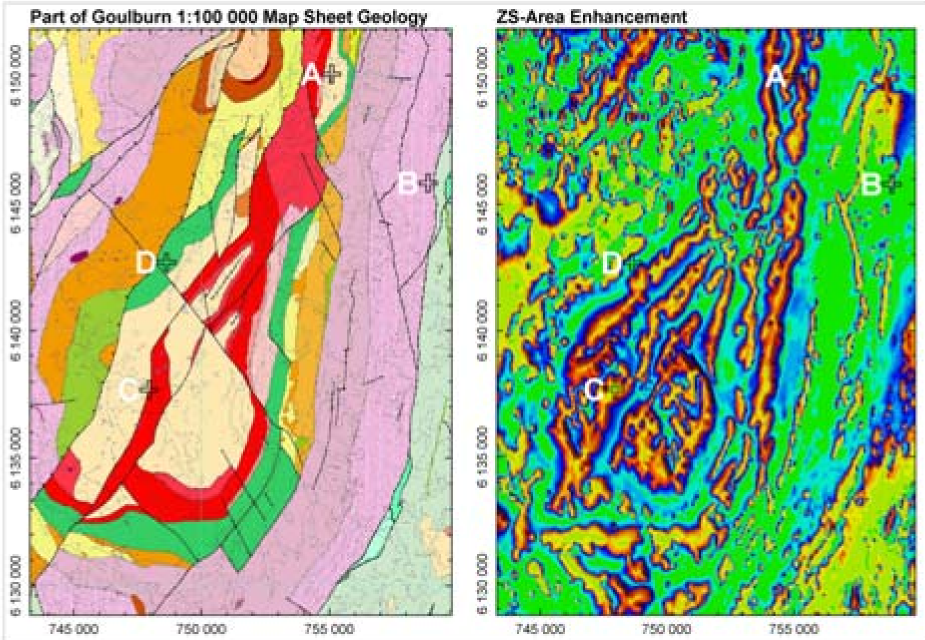
The ZS-Block filter and the ZS-Plateau filter depict the magnetic data as a 2D plan of apparent magnetic source distribution. Artefacts may occur as discussed for the edge filters.

The choice of ZS-Block, ZS-Plateau or ZS-Area filters will depend on the data characteristics of each magnetic survey and on the end-use requirement. The ZS-Plateau filter, for example, yields less variation in amplitude “texture” over a magnetic unit that either the ZS-Block or ZS-Area filters.



Comparison of ZS-Edge, ZS-Edgezone, ZS-Block and ZS-Plateau filtered outputs of RTP data

An example of some edge filter enhancements are shown below.



Comparison of geology and ZS-Area enhancement

Traditional filters used to enhance magnetic data, including the more recently developed potential field tilt filter, are currently used to assist in determination of the location and extent of magnetic units.

Newly developed derivative-based filters may be used to improve the precision of source edge detection and, by extension, the determination of the spatial extent of magnetic units. These filters are demonstrated to perform successfully on both strongly magnetized features as well as on weakly magnetized or deep magnetic features. Artefacts may result particularly where anomaly superposition occurs.

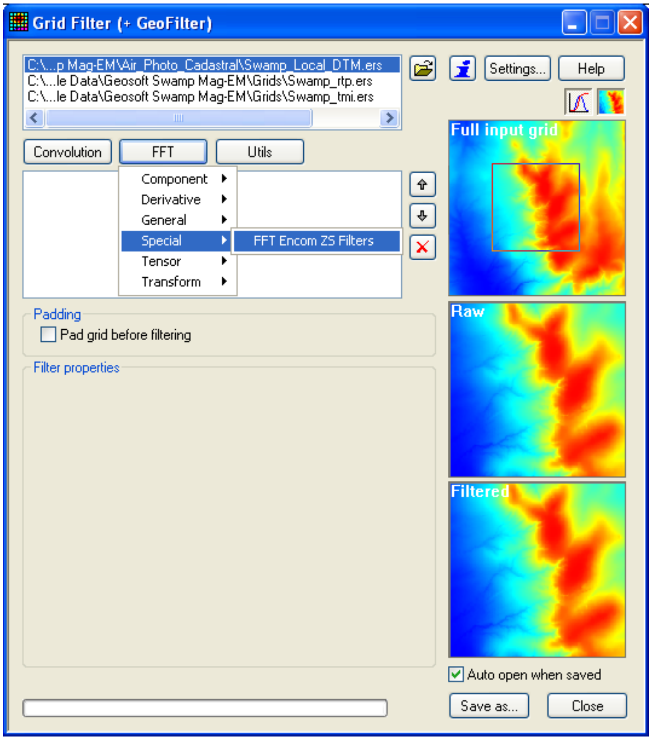
The impact of noise in real data may be accommodated by these new methods provided noise-reduction techniques are employed.

The new filter outputs may be used as part of regional or detailed geological mapping projects, including in classification systems or in RGB space, to improve lithological discrimination and mapping.

The speed of magnetic unit mapping can be considerably increased through reliance on edge detection filters. Further improvements in mapping speed can be envisaged through automated conversion of edge anomalies to vector files.

Applying GeoFilters

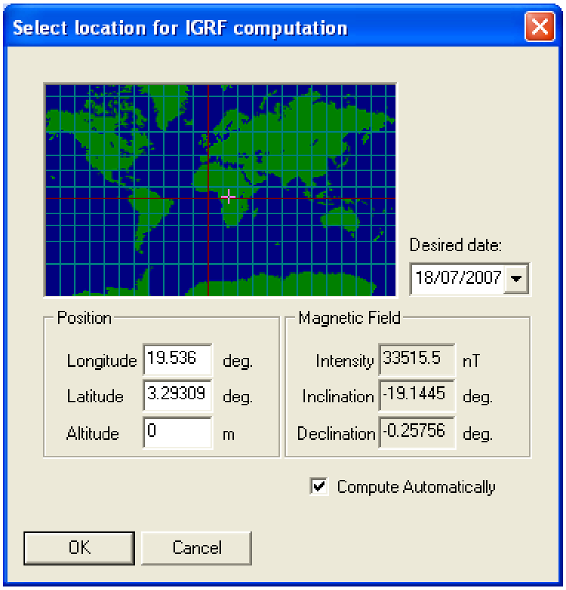
The GeoFilters are only available with a GeoFilter module licence of Encom Discover PA. Access to the various GeoFilters is through the Grid Filter tool (see *Grid Filter*). After accessing the required grid, select the **FFT** button and the **Encom ZS Filters** option from the pull-down filter list.



The Grid Filter dialog and the selection of the specialist Encom GeoFilter options

Once the Encom ZS filters are chosen, the Filter properties area alters with:

- Input specification for the **Geomagnetic Field Declination** and **Inclination** in degrees. If you are unsure of these for the area of your grid, select the **IGRF** button. You can enter the location of the grid or use the cursor to position it. When this is done, for the date specified, the total magnetic field (TMI), Declination and Inclination are computed automatically.



For computation of IGRF for the grid area for use by the GeoFilters

- Single or multiple GeoFilters can be chosen to be computed concurrently. The available filters include:
  - Analytic Signal (AS)
  - Modulus of Horizontal Derivative (Mod)
  - Tilt
  - Edge
  - Edge Zone
  - Area
  - Block
  - Plateau

  Refer to Zhi and Butt (2004) for additional details of these filters. All filters can be selected from the **Select All** button, or you can choose which are to be computed.

- The **Refresh Preview** button can be clicked when a change of filters is selected. This button forces the processing to update.

- The grid output shown in the bottom preview window shows the GeoFilter selected in the **Filter to preview** list.

When the specification of filters and processing is satisfactory, use the **Save As** button to output the resulting grid. Note that some filter combinations can take significant time to process across a large grid. In this case, a progress bar is displayed to indicate output status.

Bibliography

Blakely, R.J., 1995. *Potential Theory in Gravity and Magnetic Applications*, Cambridge University Press, Cambridge, 441p.

Clifton, R., 2006. “Visualisating Magnetic Depths”, *AESC Conference*, 2-8 July, Melbourne.

Hansen, R.O. and R.S. Pawlowski 1989. “Reduction to the pole at low latitudes by Wiener filtering”, *Geophysics 54*, 1607-1613.

Macleod, I.N., Jones, K. and Ting Fan Dai, 1993. “3-D analytic signal in the interpretation of total magnetic field data at low magnetic latitudes”, *Exploration Geophysics 24*, 679-688.

Silva, J.B.C., 1986. “Reduction to the pole as an inverse problem and its application to low-latitude anomalies”, *Geophysics 51*, 369-382.

D Fonts

Encom Discover PA can use True Type fonts for symbol, labelling and other text or symbol drawing requirements.

In addition, two font datasets are provided with Encom Discover PA

- *Encom Discover PA Basic Font*
- *Geological Font*

Encom Discover PA Basic Font

File name is ETBASIC__.TTF.

ASCII	ET Basic	ASCII	ET Basic
33	!	47	/
34	"	48	0
35	#	49	1
36	$	50	2
37	%	51	3
38	&	52	4
39	'	53	5
40	(	54	6
41	)	55	7
42	*	56	8
43	+	57	9
44	,	58	:
45	-	59	;
46	.		

Geological Font

File name is ETGEE__.TTF

Encom Discover PA includes a Geological True Type Font (ET GeoExplore) that contains non-rotatable geological symbols. These symbols may be used with the Encom Discover PA.

Contained below is a list of the ASCII and symbol equivalent list of all geological symbols available in this font.

Name	ET GeoExplore	ASCII
Petroleum exploration well, proposed site		33
Petroleum exploration well, dry, abandoned		34
Petroleum exploration well with show of oil		35
Petroleum exploration well with show of oil, abandoned		36
Petroleum exploration well with show of gas		37
Petroleum exploration well with show of oil and gas, abandoned		38
Petroleum exploration well with show of oil		39
Petroleum exploration well with show of oil and gas		40
Stratigraphic hole for petroleum exploration		41
Oil well, shut in or suspended		42
Oil well, abandoned		43
Gas well		44
Gas well, shut in or suspended		45
Gas well, abandoned		46
Oil and gas well		47
Oil and gas well, shut in or suspended		48
Oil and gas well, abandoned		49
Gas and condensate well		50
Gas and condensate well, shut in or suspended		51
Gas and condensate well, abandoned		52
Fossil locality		53
Macrofossil locality		54
Microfossil locality		55
Trace fossil locality		56
Fossil wood locality		57

Name (Continued)	ET GeoExplore	ASCII
Oncolite locality		58
Palynomorph locality		59
Plant fossil locality		60
Stromatolite locality		61
Vertebrate fossil locality		62
Sample location for isotopic age determination		63
Type locality		64
Drill hole		65
Unworked deposit		66
Prospect or mine with little production		67
Abandoned prospect or mine with little production		68
Mine; may be abandoned		69
Major mine		70
Mine abandoned or not being worked		71
Minor open cut or quarry		72
Major open cut or quarry		73
Abandoned open cut or quarry, or not being worked		74
Minor alluvial workings		75
Major alluvial workings		76
Abandoned alluvial workings, or not being worked		77
Treatment plant		78
Treatment plant not operating, or abandoned, or former site		79
Main shaft showing number of compartments		80
Shaft extending above and below plan level		81
Accessible shaft extending below plan level		82
Accessible shaft extending above plan level		83
Head of rise or winze T		84
Foot of rise or winze		85
Rise or winze extending through level		86
Inclined accessible shaft extending below plan level (small scale)		87
Inclined accessible shaft extending below plan level (large scale)		88
Cross section of cross-cut or drive; same side of plane of section as observer		89

Name (Continued)	ET GeoExplore	ASCII
Cross section of cross-cut or drive; opposite side of plane of section		90
Cross section of cross-cut or drive extending across plane of section		91
Ore chute		92
Filled workings		93
Portal and approach of tunnel or adit		94
Natural surface		95
Grab-sample locality		96
Costean or trench		97
Oil seep		98
Gas seep		99
Oil and gas seep or show		100
Oil seep reported (by geoscientist) but not relocated		101
Gas seep reported (by geoscientist) but not relocated		102
Oil and gas seep reported (by geoscientist) but not relocated		103
Mud volcano or mud volcano without with hydrocarbons		104
Mud volcano with hydrocarbons		105
Relative gravity high		107
Relative gravity low		108
Proterozoic symbol		109
Cambrian symbol		110
Photo point		111
Drillhole		112
Registration cross		113
Scarp		114
Inclined drill hole		115
Inclined drill hole		116
Inclined drill hole		117
Inclined drill hole		118
Major eruptive centre with recorded eruption		119
Major eruptive centre with no recorded eruption		120
Minor eruptive centre with recorded eruption		121
Minor eruptive centre with no recorded eruption		122

Name (Continued)	ET GeoExplore	ASCII
Astrobleme or impact structure or cryptoexplosive structure		123
Trigometrical	△	124
Astronomical station	⊕	125
Major volcanic centre		126
Volcanic plug residual		127
		128
Basalt capped residual hill		129
Residual hill		130
Crater wall		131
Pediment		132
Landslips		133

E Supported File Formats

In this section:

Supported Grid Formats

A wide range of industry-standard grid formats are supported and can be imported for use in Encom Discover PA. Compatible grids include:

- Arc ASCII grid files (.ASC)
- Arc Binary grid files (.ACF)
- ASEG GXF (.GXF)
- BIL files (.BIL)
- Encom grids (.GRD)
- ER Mapper (.ERS)
- ESRI/Arc triangulation grid format (.TIN)
- Intrepid (.ERS)
- Geopak grid (.GRD compressed and uncompressed)

- Geosoft compressed and uncompressed Oasis montaj (.GRD)
- GeoTIFF georeferenced TIFF files with file header (.TIF, .TIFF)
- Landmark Seismic grid format (.GRD)
- MapInfo grid files (.MIG)
- Minex grid files (.XYZ)
- Surfer ASCII grid files (.GRD)
- Surfer binary grid (.GRD)
- USGS grids (.USG)
- USGS DEM grid file (.DEM)
- USGS SDTS grids (.TAR)
- Vertical Mapper grid files (.GRD, .GRC)

Supported Located Image Formats

Read

The Located Image option allows display of a variety of positioned raster and vector data formats:

- 3D Studio files (.3DS)
- Arc ASCII grid files (.ASC)
- Arc Binary grid files (.ACF)
- ASEG GXF (.GXF)
- AutoCAD 3D DXF files (.DXF)
- BIL files (.BIL)
- Compuserve GIF (.GIF)
- Datamine vector files (.DM)
- Encom grid files (.GRD)

- Encom located image files (.EGB)
- ER Mapper algorithm files (.ALG)
- ER Mapper compressed wavelet format files (.ECW)
- ER Mapper grid files (.ERS)
- ESRI triangulation files (.TIN)
- Geopak grid files (.GRD)
- Geosoft grid files (.GRD)
- GeoTIFF format files (.TIF, .TIFF)
- GoCAD (.TS, .PL, .VS)
- Gemcom (.BT2)
- JPEG (.JPG, .JPEG)
- Landmark grid files (.GRD)
- MapInfo grid files (.MIG)
- MapInfo raster files (.TAB)
- Minex grid files (.XYZ)
- ModelVision (.TKM)
- Portable Network Graphics (.PNG)
- Surfer ASCII grid files (.GRD)
- Surfer binary grid files (.GRD)
- Surpac vector files (.DTM)
- USGS grid files (.USG)
- USGS DEM grid files (.DEM)
- USGS SDTS grid files (.TAR)
- Vulcan vector files (.00T)

- Vertical Mapper grid files (.GRD, .GRC)
- Windows Bitmap (.BMP)

EGB Image Sources

Image formats .BMP, .JPEG, .PNG, .TIFF, and .GIF files are supported as image sources within .EGB files.

Write

Located images can be saved by Encom Discover PA in the following formats:

- PNG – Portable Network Graphics
- TIFF – Tagged Integrated Format
- BMP – Windows standard bitmap format
- JPG – Compressed bitmap format
- ECW – ER Mapper Compressed Wavelet format

Located Bitmap Surfaces

These located bitmap surface formats can be associated with a grid (see Located Bitmap Surface option in *Transparency and Colour-Intensity Balance*):

- ERMapper Algorithm files (.ALG)
- ERMapper ECW files (.ECW)
- GeoTiff files (.TIFF)
- JPEG 2000 files (.JP2)
- MapInfo files (.TAB)

Supported 2D Vector Formats

- MapInfo Professional (.TAB, .MIF)
- AutoCAD (.DXF)
- ESRI (.SHP)

- ER Mapper (.ERV)
- Encom GPinfo (.GSF)

Supported 3D Vector Formats

- AutoCAD (.DXF)
- GoCAD (.TS, .PL, .VS)
- Gemcom (.BT2)
- ESRI TIN (.ADF)
- ESRI 3D shape (.SHP)
- Datamine wireframe (.DM)
- Surpac (.DTM)
- Vulcan Triangulation (.00T)
- 3D Studio (.3DS)

Supported Databases

- Geosoft Oasis montaj
- DFA Intrepid
- MapInfo .TAB file

Supported Image Formats

- .PNG – Portable Network Graphics
- TIF – Tagged Integrated Format
- .BMP – Windows standard bitmap format
- .JPG – Compressed bitmap format
- .EMF – Enhanced Windows Metafile

Supported Lookup Table Formats

- ER Mapper (.LUT)
- MapInfo Professional (.CLR)
- Geosoft Oasis (.TBL)
- Engage3D Legend (.LEG)

Supported Drillhole Formats

- ASCII collars, survey and downhole data files
- ASCII XYZ desurveyed data files
- Geosoft .GDB
- Encom Discover .TAB
- acQuire databases - Microsoft Excel, FoxPro dBase, Microsoft Access, SQL Server, VisualFoxPro databases, and text files.

Supported Voxel Model Formats

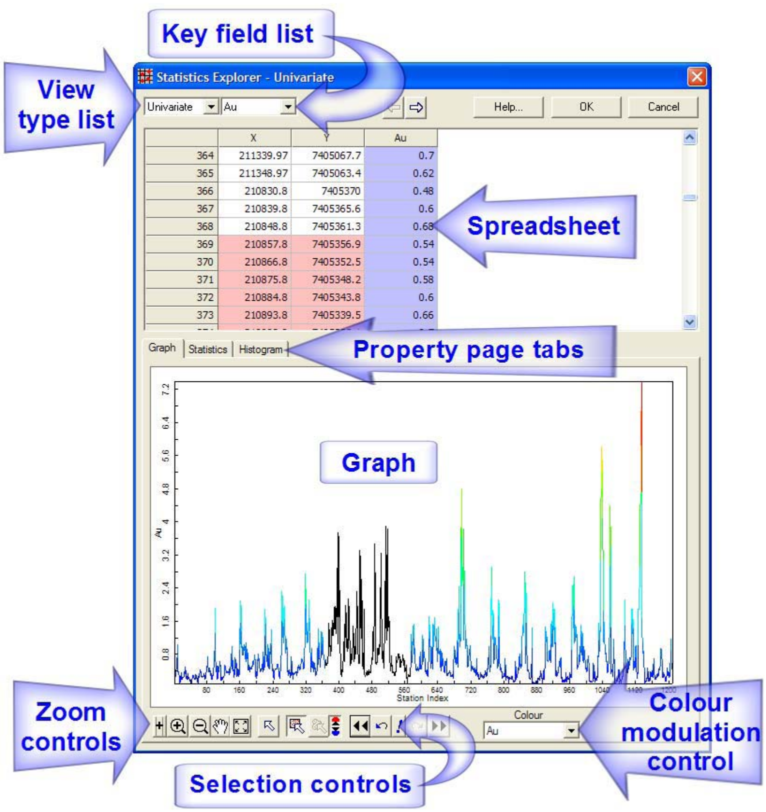
- UBC 3D (University of British Columbia) (.MSH, .DAT)
- CEMI (University of Utah) (.CEM)
- Datamine (single precision) (.DM)
- Surpac (versions 1.0 to 3.0) (.MDI)
- Vulcan (Warning: support may be incomplete) (.BMF)
- Gocad voxet (Warning: support may be incomplete) (.VO)
- Gemcom (Warning: support may be incomplete) (.BPR)
- Noddy (.G00 and .GXX)
- Encom 3D grid (.E3D)
- ASCII XYZ (.XYZ)

F Statistics Explorer

The **Statistics Explorer** is a series of dialogs that enables you to examine any dataset both statistically and spatially.

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.

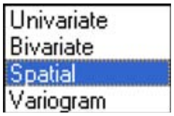
The Statistics Explorer is generally accessed via the statistics button in the various Field Data Conditioning dialogs (see *Appendix G: Field Data Conditioning*) accessed from many Properties dialogs, as well as the 3D Gridding tool.



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:

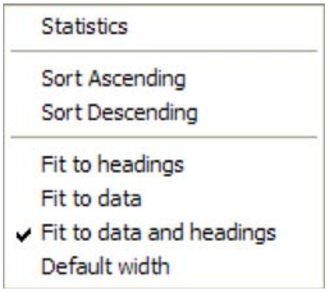
- The currently selected data points are highlighted red.

- 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.

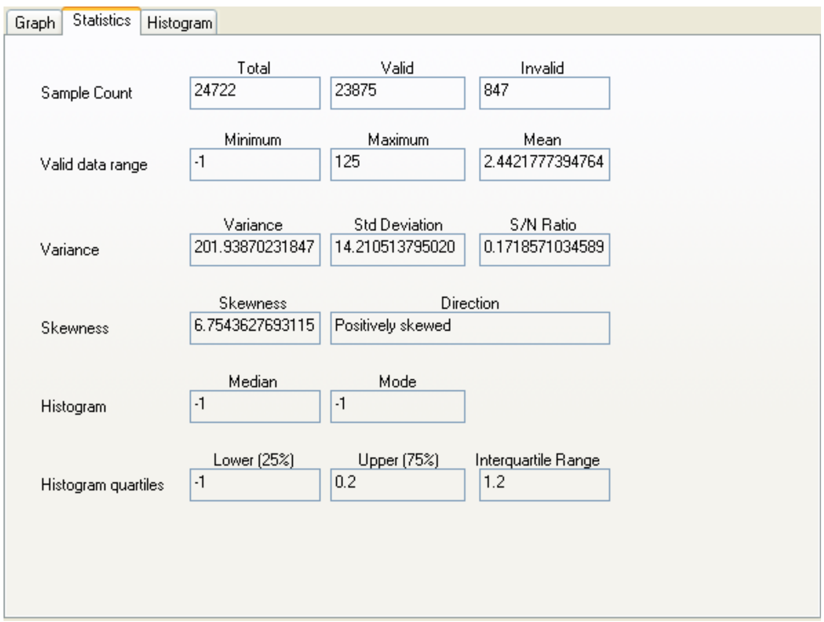
The spreadsheet shortcut menu can be accessed by right clicking on any column header. This provides a number of column resizing options (including the various ‘Fit to’ options). Alternatively, columns and rows can be resized by clicking on their boundaries and dragging.



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 a gold 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

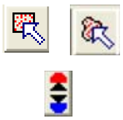
The Zoom controls toolbar

- 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.

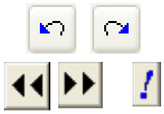
Selection Tools

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.

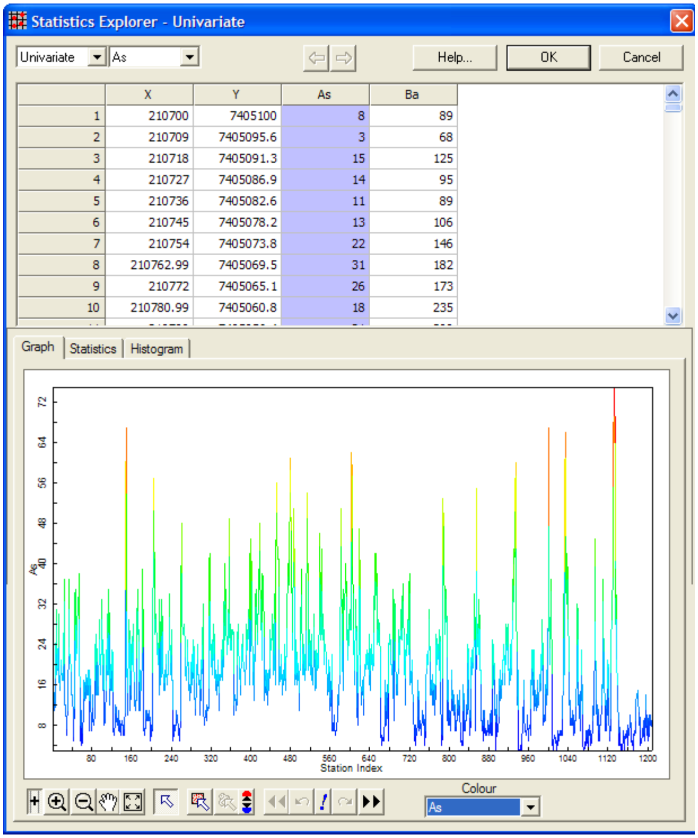
This button extracts the selected data points from the dataset to create a new subset. The spreadsheet and graph views will be redrawn to focus on this subset. Only the subset will be considered in any statistical computations or graphs. Further selections can be made to examine the data in greater detail. 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.

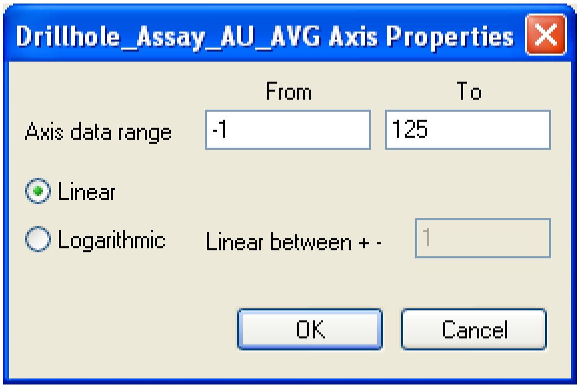


Univariate graph display

The **Graph** page plots the key field versus station index. Graph controls are detailed in *Zoom, Selection and Display Controls*.

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.

Note These axes property pages are not available for univariate or spatial graphs.

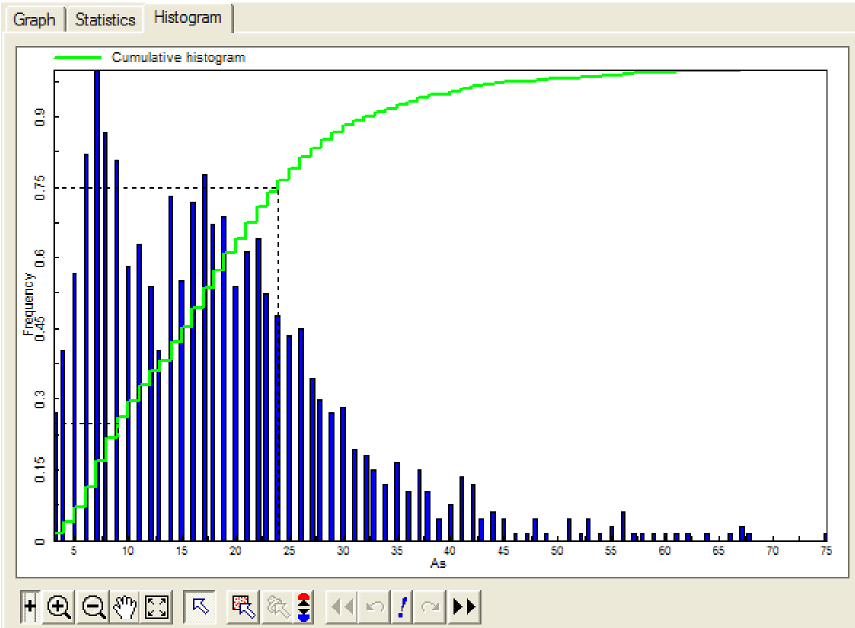


Specifying an axis range

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.

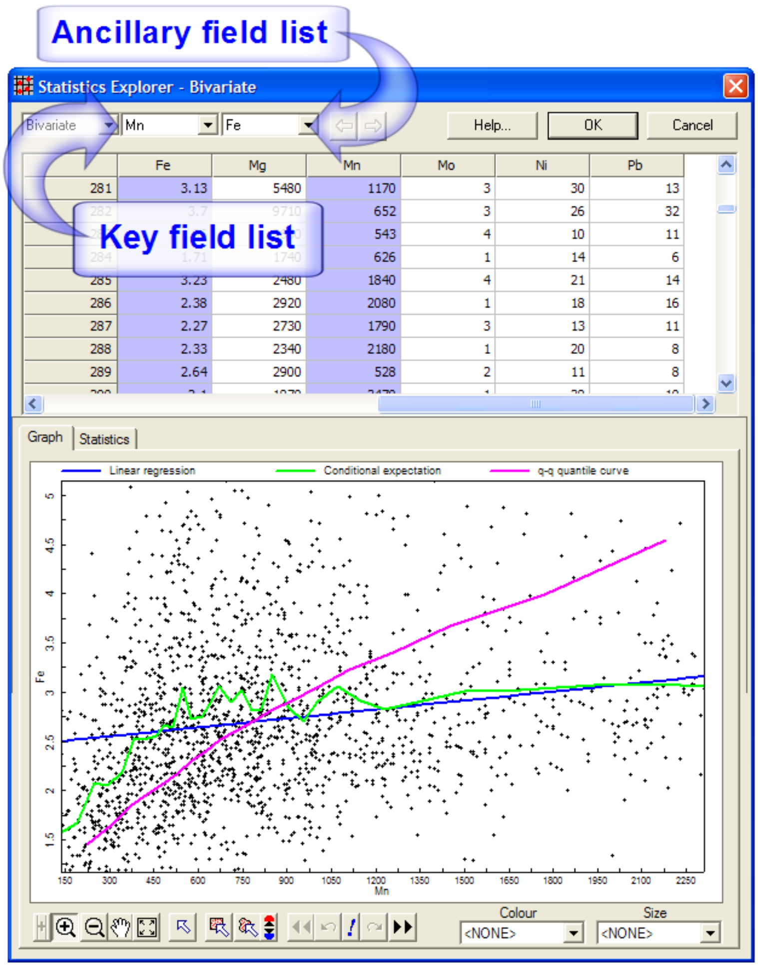
To be effective a large number of bins are used and these are grouped together depending on the scale at which the histogram is viewed. When multiple bins are being grouped together each consolidated bin is drawn with a back border. When zoomed in sufficiently to see individual bins, no black border 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:

below:

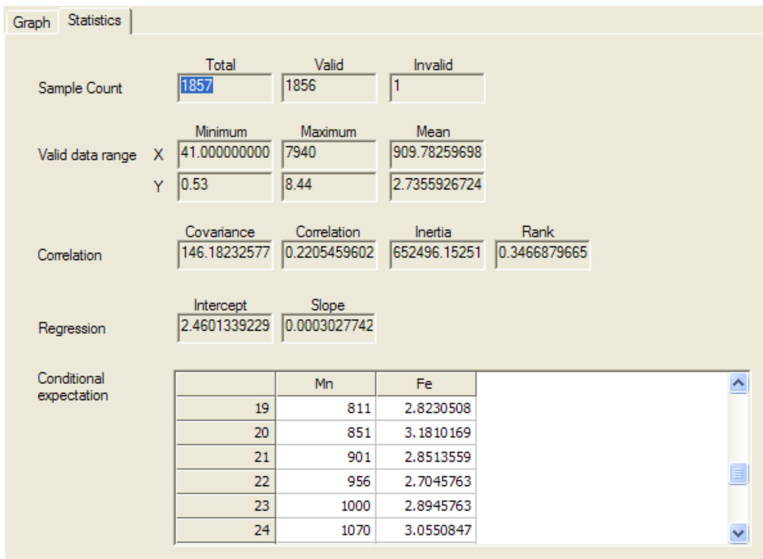
Sample Count	The total number of samples in both fields.
Valid Sample Count	The total number of valid (non-null/ selected)samples in one or both fields.
Invalid Sample Count	The total number of invalid (null/unselected) samples in one or both fields.
Minimum	The minimum value of all valid samples.
Maximum	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.
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.

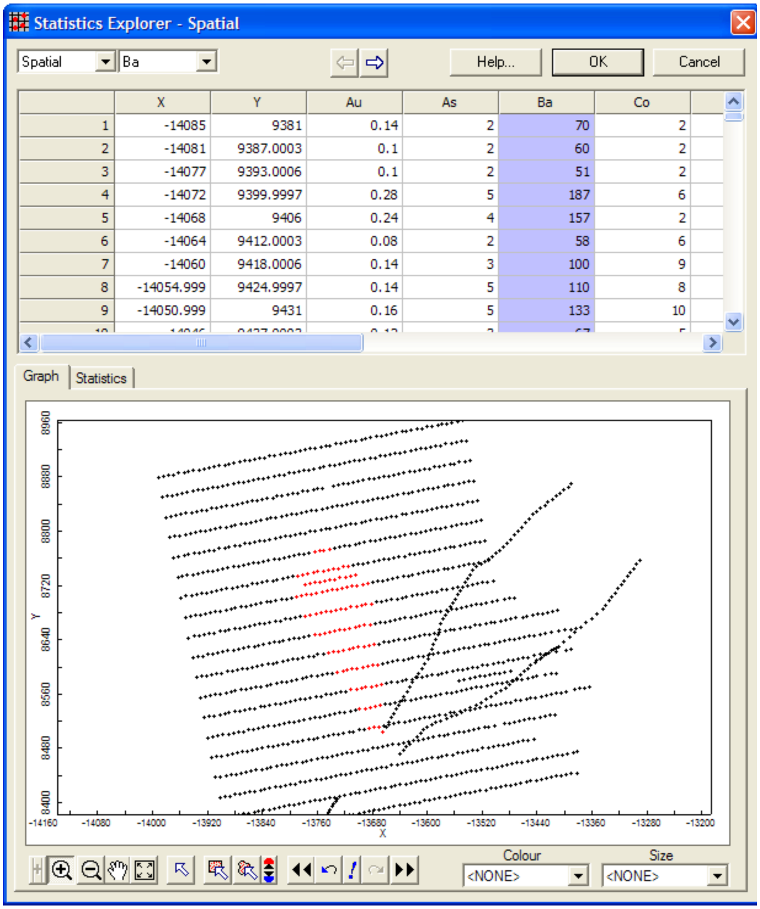


Bivariate statistical data ranges and distribution

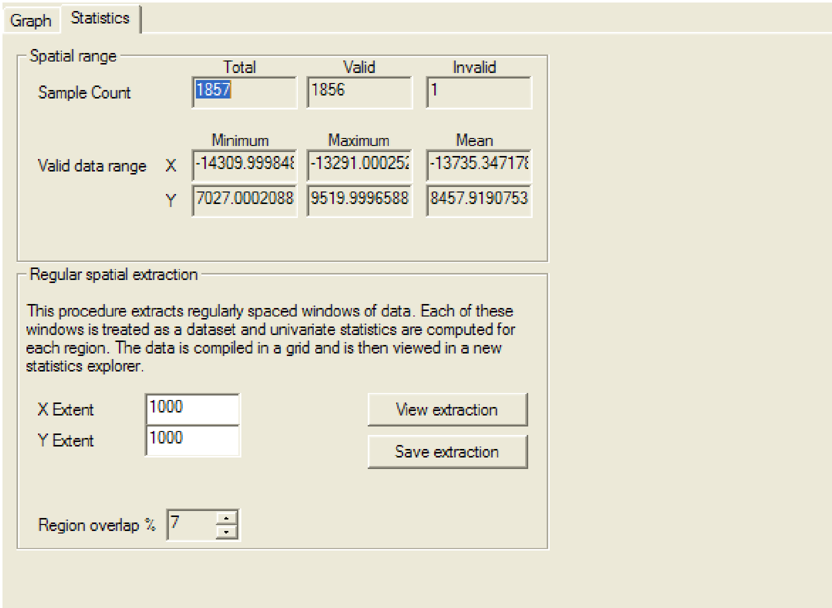
Spatial

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.

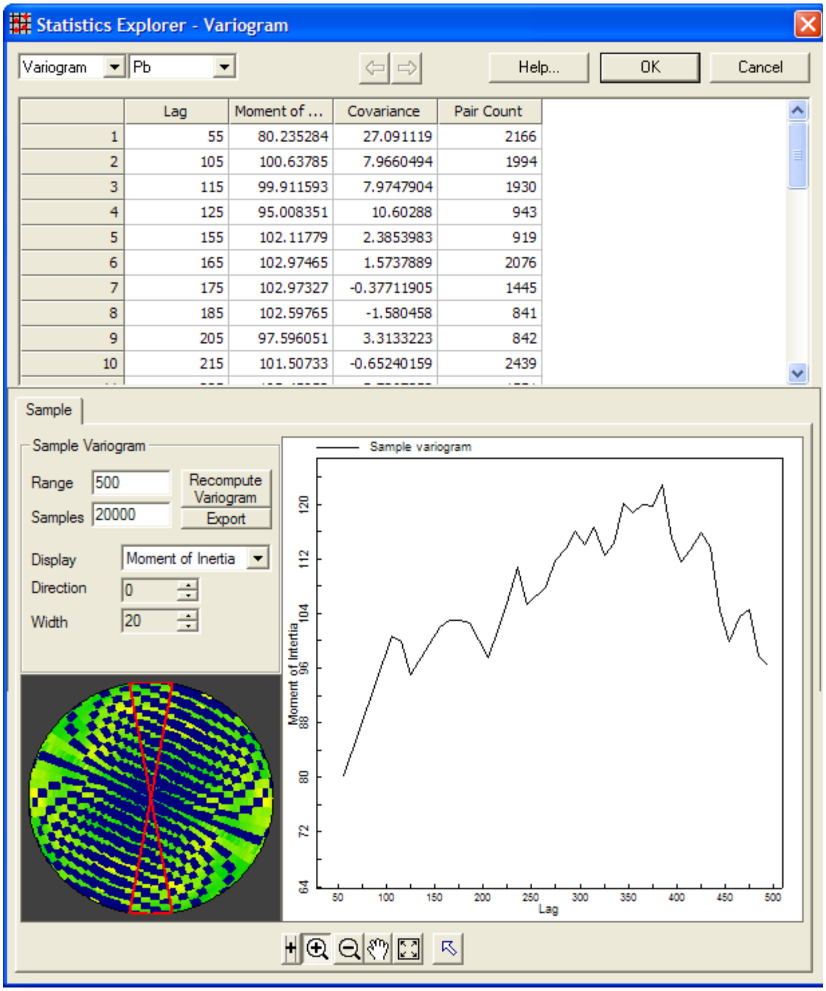
Variogram

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 (semivariogram), 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

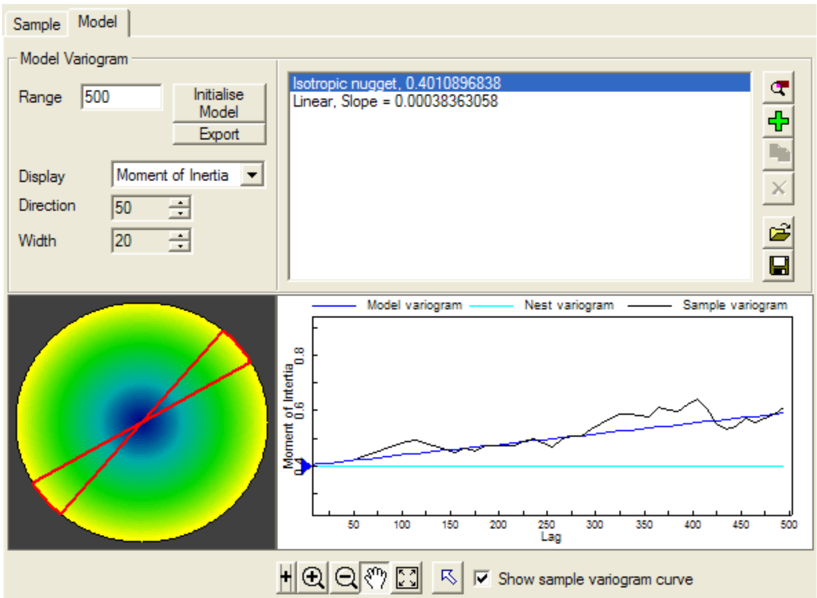
If the Statistics Explorer has been opened via the **Variogram** button within the Kriging wizard of the Gridding Tool, a Model Variogram also can be created, edited and displayed. In this case an option is added to the **Sample** page to show the model variogram curve in addition to the sample variogram 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.

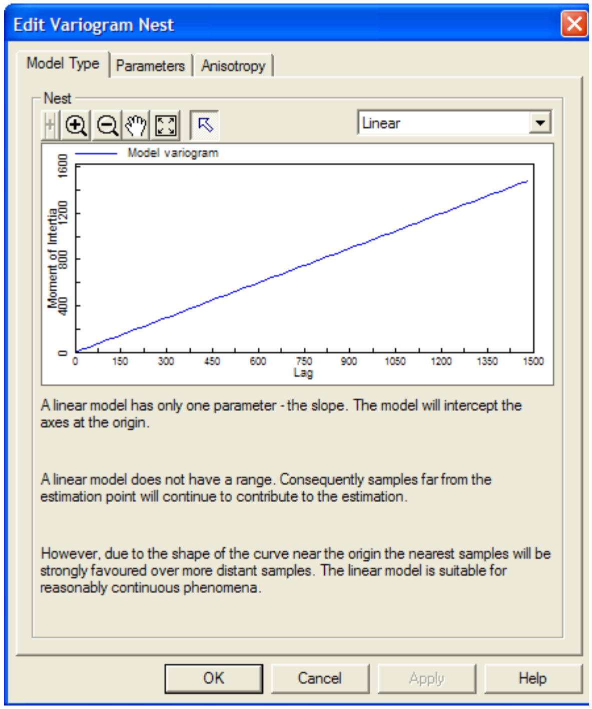
The model variogram is displayed as a grid and also as a directional extraction – just like the sample variogram. The sample variogram curve can be plotted for comparison. As before, the extraction direction can be controlled via the edit buttons or by dragging the cursor across the variogram grid display.

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.



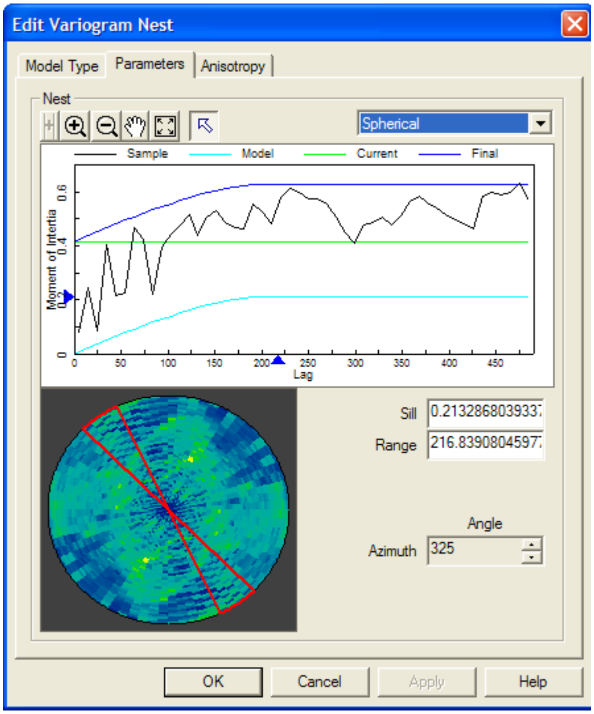
Displaying the sample and model variograms

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.

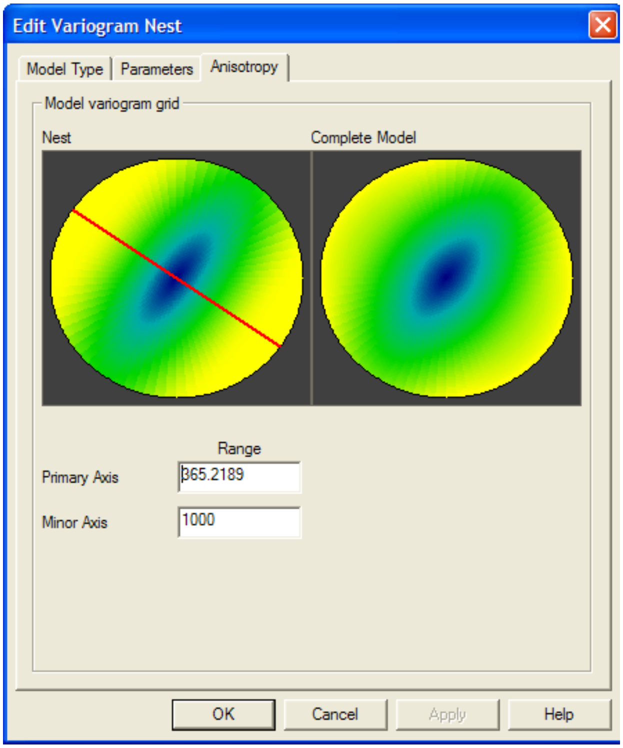


Controlling the parameters of the variogram

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

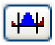
The **Anisotropy** tab controls the range of the model along two axes – the major and minor axes. The direction of the major axis is shown on the nest grid preview on the left. The final model preview is shown on the right.

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.

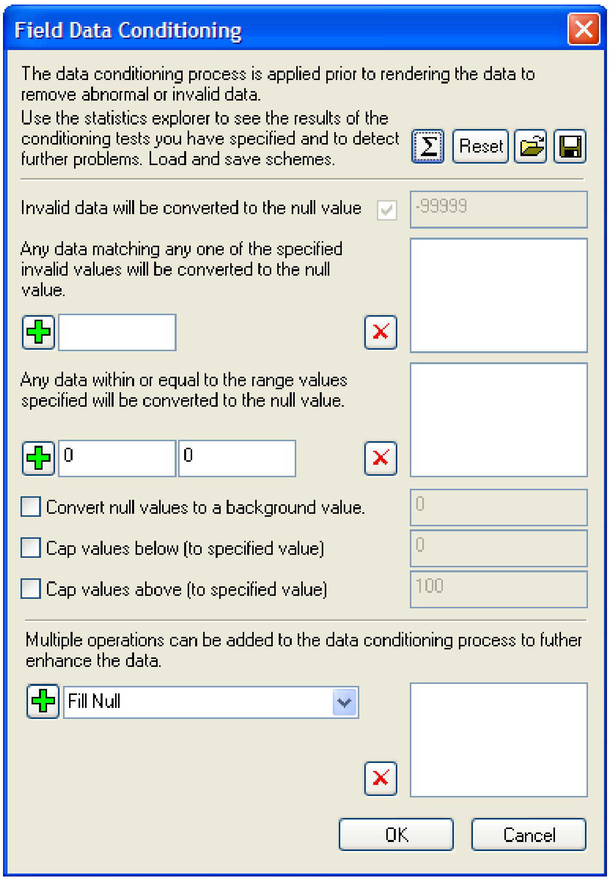
If it is larger than the major axis range then the model will have a smaller contribution in the minor axis direction.

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.

G Field Data Conditioning



The **Field Data Conditioning** dialog can be accessed from a number of Encom Discover PA 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

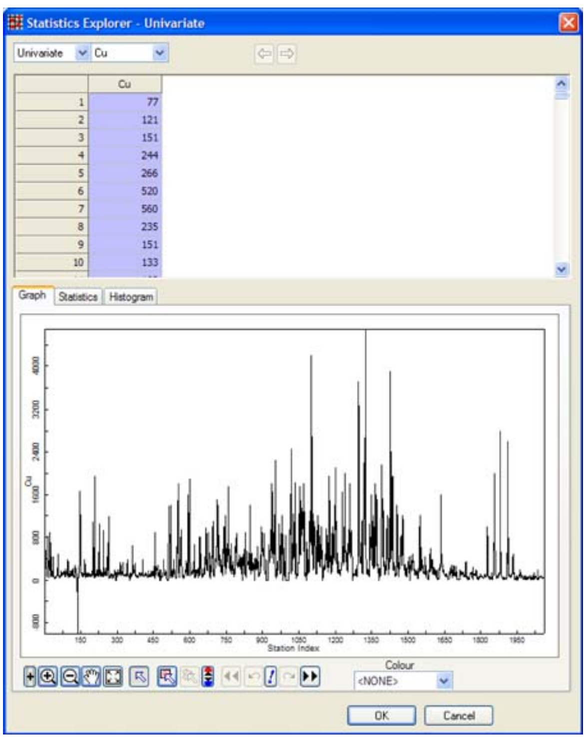
The **Statistics** button at the top of the dialog opens the Statistics Explorer. This tool provides a powerful means of analysing the dataset for geochemical outliers, invalid data ranges, data distribution, etc via Graphical, Statistical and Histogram views. This dialog is dynamic: it will reflect 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.

Within the Graph and Histogram views of the Statistics Explorer, 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.

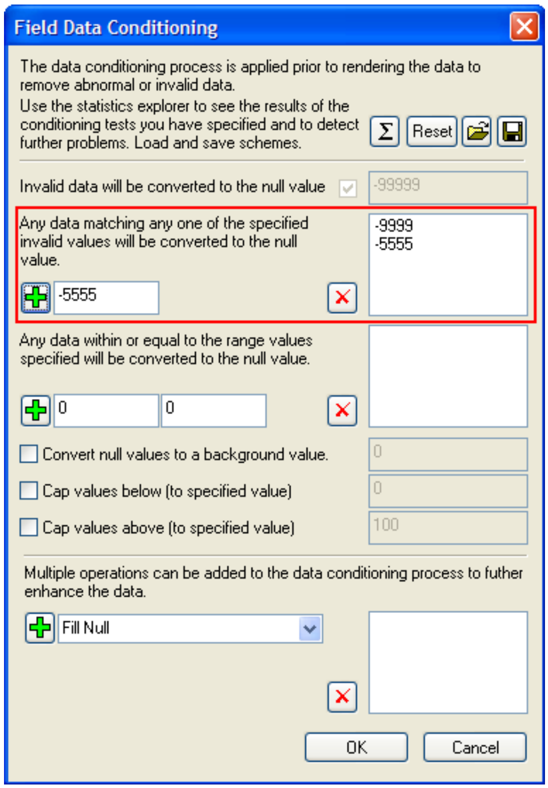
The central part of the Field Data Conditioning dialog allows the specification of invalid data and/or data ranges. The first option auto-populates with the dataset's default null value: generally -1.0×1032 or -99999. Invalid data (either user-defined or pre-existing null input data) will be handled as a null value internally and not displayed.



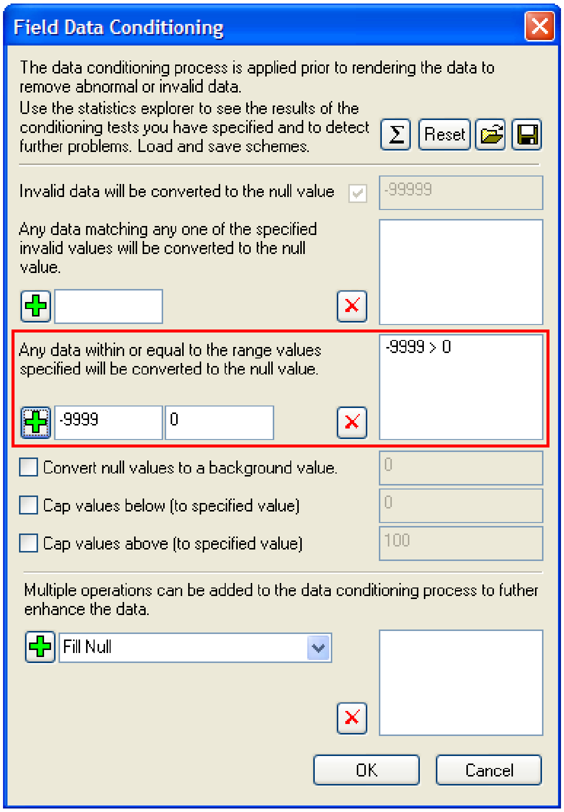
Specifying individual invalid data values. This is appropriate for removing negative values representing BDL (Below Detection Limit), SNR (Sample Not Received), etc.

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.

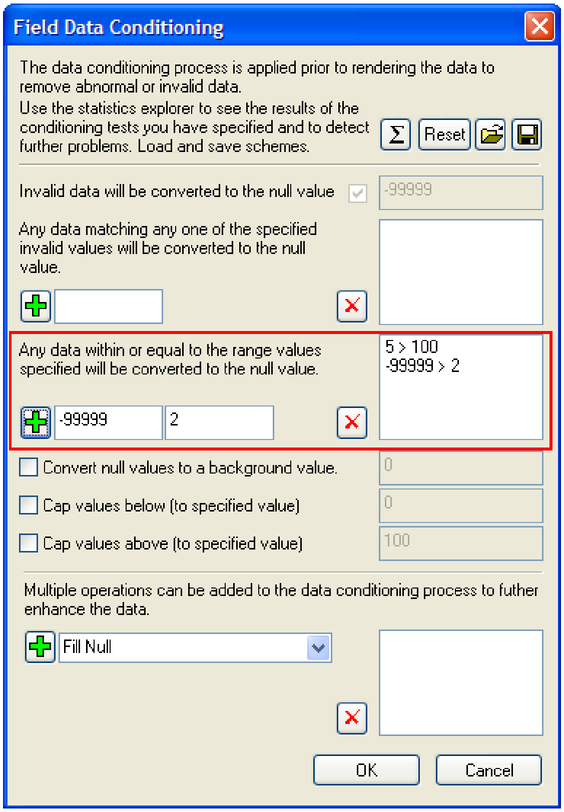
List entries can be removed using the **Delete** button. Some examples of use are presented below:



Specifying individual invalid data values. This is appropriate for removing negative values representing BDL (Below Detection Limit), SNR (Sample Not Received), etc.

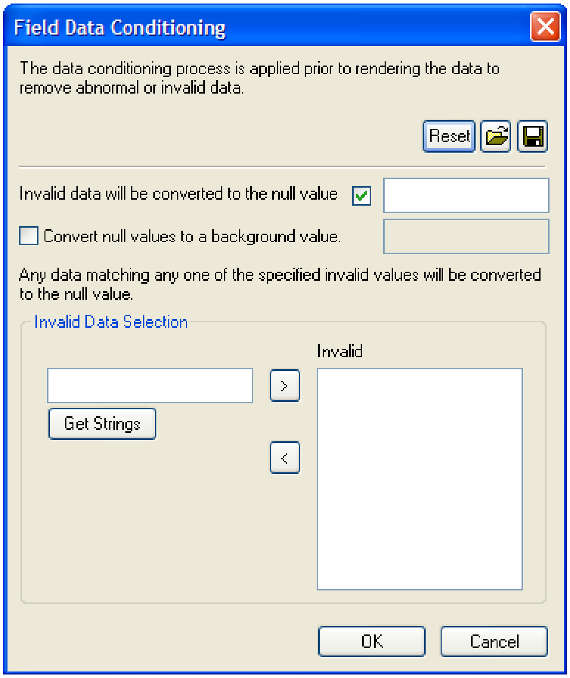


Specifying an invalid data range to remove a range of negative values in the dataset from the 3D gridding process. This is a useful way to remove multiple negative values, rather than entering each one individually.



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.

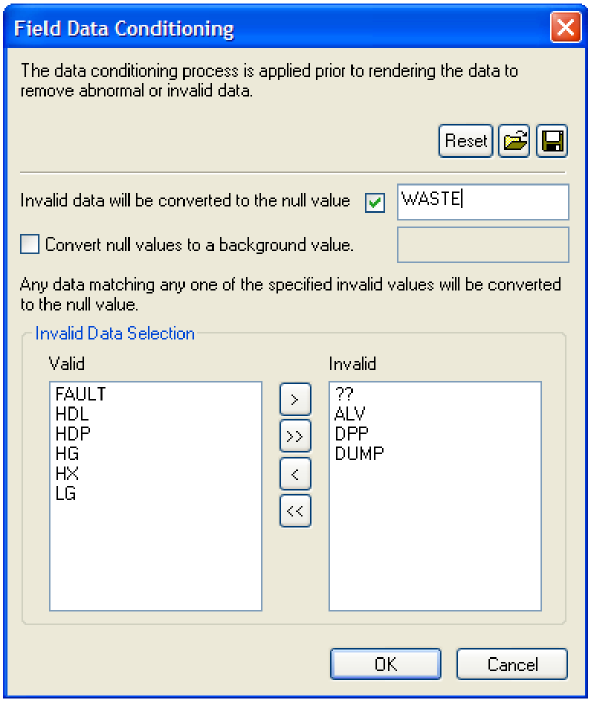
Setting a **Cap values below** or **values above** will cap source data outside the set limit to the limiting value. For example, with **Cap values above** set to 500, a gold assay on 725ppm will be handled as a 500ppm value. It is also possible to **Convert null values to a background value** specified 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.



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 Discretised 3D gridding), the Field Data Conditioning dialog display will be altered accordingly.

This allows individual text strings to be specified. The **Get Strings** button will populate the left list with all Valid unique attributes (default all), allowing a list of invalid text strings to be easily created in the right window. The null value for invalid data can be set, and a background value to set nulls to.

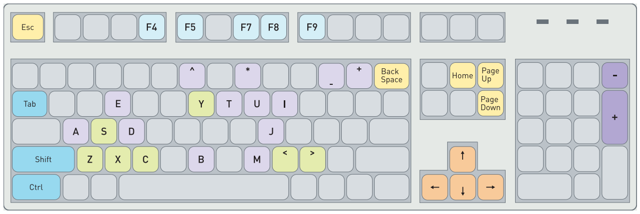


Creating a list of invalid lithological codes (so that only the valid codes on the left e.g. rated ore grades are displayed when colour modulating drillholes)

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.

H 3D Cursor Keyboard Shortcuts

3D Cursor Keyboard Shortcuts



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

	Rotates the view
	Zooms in and out
	Moves the viewing centroid horizontally in the XY plane
Shift +	Moves the viewing centroid vertically and parallel to the screen plane
Ctrl +	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

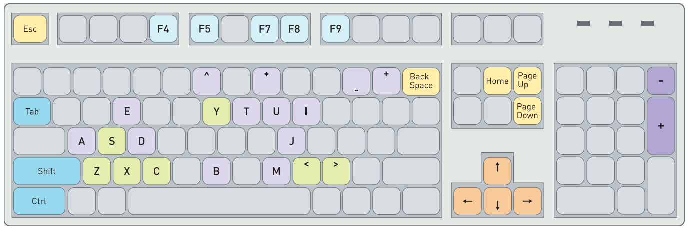
Tab	Switches the cursor plane orientation between three standard view planes: XY, YZ and XZ
Shift + X	Orientates the cursor plane parallel to YZ plane
Shift + Y	Orientates the cursor plane parallel to XZ plane
Shift + Z	Orientates the cursor plane parallel to XY plane
Page Up Page Down	Moves the cursor 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
+ -	(numeric keypad) Iterates through all selected images
Shift + + or -	Switches bonding between the first and last images in a series

3D Cursor Keyboard Shortcuts



Creating Features

	Creates a point feature or new points of polygon/ polyline (depending on the current default)
	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

	Selects a feature
Ctrl +	Adds a feature to the current selection

Editing Features

Applies to a reshapable feature

	Depends on the current edit mode (Add, Delete, Break, None) and the type of element (node, edge, face)
	Node: moves node on the cursor plane **Edge:** inserts a node
Ctrl +	**Node:** moves node perpendicular to the cursor plane **Edge:** inserts a new node and moves it perpendicular to the cursor plane
Shift +	Moves feature on the cursor plane
Shift + Ctrl +	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

Index

Symbols

Numerics

A

B

C

D

E

F

G

H

I

J

K

L

N

O

P

Q

R

S

U

V

W

X

Y

Z