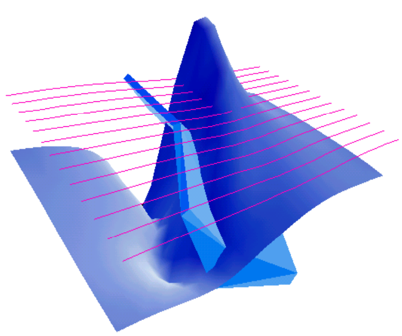
EMIGMA© /V8.1

3-D Electromagnetic, Magnetics, Gravity, Resistivity/IP, MT/CSAMT Interpretation Platform for Comprehensive Geophysical Imaging, Modelling & Inversion

for software release version 8.1

including:

….data importing, editing, correcting, filtering and compression
… survey analyses, gridding, mapping
…. Data management
.....robust and rapid 3D modelling
.....comprehensive survey simulation
.....3D Data and Structural Representation
....1D Inversion of FEM/MT/CSAMT, Resistivity and TEM
.....Contouring and Depth-section estimates
....3DMagnetics Inversions and Modelling and Processing
…..Gravity FFT Processing and 3D Modelling with Data Displays
…..Magnetic Processing including Gradients
….Simulation of magnetic and gravity derivatives
….Removal of Regional Trends
…Advanced Grid Interpolations using Natural Neighbor, Delauney Triangulation Shepard techniques, Minimum Curvature and Thin-Plate Splines
… and much,much more



This manual is the property of *PetRos EiKon Incorporated* and is a reference to licensed users of ***EMIGMA.*** It can be found in electronic form on your EMIGMA V8.1 CD and may be printed by the clients for their use. Hardcopies of this manual may be obtained from *PetRos EiKon* for $200.00 US including shipping. The manual has been checked for conformity with ***EMIGMA***, but absolute conformity cannot be guaranteed. Corrections and suggestions for its improvement can be sent to the address below and are actively solicited.

Trade names are used for illustrative purposes only and do not imply endorsement of any kind.

User license is contained as an Appendix.

2004

PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

FOREWORD

EMIGMA is not designed primarily as a data analyses tool but rather as an interpretation platform for a range of geophysical data. We have a wide range of 3D modelling algorithms within EMIGMA, but we also provide associated tools by which to aid in the understanding and interpretation of your data. For example, 3D visualization encompasses algorithms required for visualization and manipulation of data and model structures. Several types of inversion are also available as assistance to interpretation of your data.

While inversion is actively sought after in the geophysical community, EMIGMA provides tools by which to utilize the inversion while enabling the user to provide an analytic criticism of the inversion results. Comparison of model results or inversions to your data is, of course, critical and EMIGMA is designed to comprehensively and quickly look down the model dimension.

Tools for Electromagnetics

Modelling

Advances in electromagnetic (EM) modelling software over the past 20 years have enabled geophysicists to improve their intuitive understanding of electromagnetic scattering in the earth. Despite the development of many techniques and their application to a variety of models, these methods have been too slow, too difficult to use and too simplistic to be important for solving most of the modelling problems encountered by the practising geophysicist in his or her daily work.

The electromagnetic modelling software described in this manual represents a fundamental advance in the field of electromagnetic simulation. ***EMIGMA*** should not be considered a simulation program, but rather a simulation platform. It allows the importing of a multiplicity of scattering simulation routines as separate objects that can be energized with a large number of transmitter and receiver combinations. Combining these scattering objects with an eclectic collection of source fields and receiver geometries results in the capability to model virtually any geophysical system over a wide range of geological conditions.

The present version of ***EMIGMA*** consists of 4 fundamentally different simulation routines that in combination provide for a powerful capability for simulating geology and the near-surface realistically. One scattering simulation routine is based upon the Localized Non-Linear (*LN*) approximation (see Habashy, Groom and Spies, Journal of Geophysical Research, February 1993). This algorithm was extended in 1997 to include the effects of susceptibility variations as well as conductivity and permittivity variations (see Murray et al, SEG Expanded Abstracts 1999). The second simulation routine is based upon the *VH* formulation (see Walker and West, Geophysics, August 1991). The *LN* formulation brings unprecedented speed to electromagnetic scattering solutions, and so finally enables the geophysicist to simulate complex geological structure in time frames required for data interpretation. The *VH* formulation is highly robust and can be used reliably to simulate electromagnetic scattering phenomena over widely varying conditions, including highly conducting massive ore bodies. The third algorithm is the Inductive

Localized Non-Linear (ILN) approximator (Murray, EAEG Conference Proceedings, May 1997 and Murray et al, SEG Expanded Abstracts 1999).

Like all numerical solutions the authors have encountered, the solutions we employ have particular strengths and weaknesses. Although we attempt to describe those to the user, the user should bear in mind that no modelling software should be treated as a black box, in which results are taken at face value. With continued development and testing, we expect the range of solutions to be extended and better defined, and their accuracy and speed improved.

The fourth algorithm is a sphere algorithm (EMSPHERE) which is solved by a traditional separation of variable capability and is highly accurate but limited to a spherical geometry. At present, the solution can be obtained for either dipolar sources or plane wave sources but we are extending this to Loop sources particularly for UXO applications.

Inversions:

EMIGMA provides point-by-point one-dimensional inversion for both frequency and time-domain data. In each of these domains, inversion is available for airborne and ground data.

FEM data: Four (4) algorithms are provided plus a depth imaging technique based on the Sengpiel section technique.

TEM data: Two (2) different inversion algorithms are available with each having two (2) forward algorithms thus providing a total of 4 algorithms depending on your need.

Conductivity-Depth Imaging:

EMIGMA offers CDI's for both FEM (modified Sengpiel sections) and TEM (Eaton and Hohmann).

Tools for Magnetics

3D Modelling

In recent years, PetRos EiKon has changed its research focus to some extent towards magnetics problems. It has been an interesting period of time as our entrance to this was precipitated by the use of the LN algorithm to model magnetics. This was important as it allowed an accurate solution while limiting the magnetization to be parallel to the Earth's field. Over the last several years, we have experimented and extended our capabilities to include de-magnetization effects, magnetic channelling, magnetic interactions and then extended this to our inversion routines. Recently, we have included remanent effects and are now studying the problems of near-surface modelling and detection of magnetic anomalies.

There are in total three (3) basic algorithms for 3D modelling of magnetics in EMIGMA but these may be combined with different forms of magnetics interactions between multiple bodies.

Processing

EMIGMA contains a series of FFT tools for Magnetics that include derivative calculation, upward/downward calculation, wavelength filtering and reduction-to-the pole. Our product allows one to export the derivatives or any of the processing back to EMIGMA as a survey so as to work with the modelling or inversion. For example, one may upward continue and create a model for the processed grid and then model the data as if at another elevation and similarly for any other processing. It is very useful to generate the derivatives and then compare your modelled derivatives to the processed derivatives. This process greatly speeds up the modelling to data process.

Inversion

EMIGMA offers 3D susceptibility inversion but with some twists. We allow for both direct matrix inversion and for smooth optimized inversion. We also allow you to use your magnetic components as well as your magnetic derivatives and also to utilize data at multiple elevations. For example, you may invert your ground data with your airborne data.

EMIGMA also offers new Extended 3D Euler depth estimations and will soon provide you with an inversion for the magnetization source vectors.

Tools for IP/Resistivity

Modelling

EMIGMA offers two (2) algorithms for 3D resistivity modelling and the most extensive 3D modelling for IP available in the world. We also provide MIP and MMR modelling.

Inversion

At the present time, we only offer 1D Resistivity inversion as well as a range of pseudo-section tools for both IP and Resistivity.

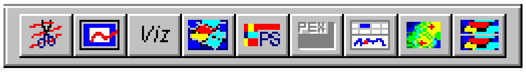
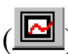
Tools for Gravity

Gravity in EMIGMA V8.1 has now been implemented. Although primary Gravity processing is yet to be included, advanced user functions are available. For example, EMIGMA contains a range of FFT functions for both gravity and magnetics, such as wavelength filtering, upward/downward continuation and derivative calculation. In addition, EMIGMA V8.1 offers some very good 3D modelling capabilities and 3D Euler deconvolutions with post-processing.

Table of Contents

FOREWORD **3**
SECTION I - INTRODUCTION **12**
WHAT EMIGMA V8.1 OFFERS 13
CHANGES FROM PREVIOUS VERSIONS 17
Features of V8- Release 1 17
V8 Tools 18
PETROS EIKON DIRECTORY 21
SECTION II - EMIGMA V8.1 OVERVIEW **22**
SECTION III - STEP-BY-STEP GUIDE **49**
INSTALLING EMIGMA v8.x 50
Upgrading from a Previous Version of EMIGMA 50
Performing a new installation 50
Installation Options 50
Requesting a TGS password 51
GETTING STARTED 52
First Use 52
To open an example database 52
To create a new database 52
Subsequent Use 53
To open an existing database 53
To create a new database 53
CREATING PROJECTS 55
Creating Projects From Imported Data 56
Example: Importing GEONICS TEM data 57
Step 1 58
Step 2 60
Step 3 62
Step 4 63
Creating Projects From Scratch 66
Specifying System Geometry 67
To specify a gravity system 67
To specify a magnetic system 67
To specify an MT system 68
To specify an EM/IP/Resistivity system 69
Specifying Transmitter type 69
Specifying Receiver Type 73
Specifying Separation 74
Using IP/Res Wizard 74
To define system components to be used in modeling 75
To specify the coordinate system 75
Specifying Profile Information 76
To add a new profile 76
To reorder profiles 76
To rename a profile 77
To replace a profile 77
To import a profile 77
ASCII Borehole Format 77

PEV files ... 80
Columnar files ... 81
UTEM4 format ... 81
To import topography ... 82
To split a profile ... 84
To merge profiles ... 84
To reduce the number of stations ... 84
To shift a profile along the axes ... 84
To add a single station ... 85
To replace a single station ... 85
To delete a single station ... 85
Specifying Waveform ... 86
To specify the frequency-domain mode ... 86
To specify the spectral mode ... 87
To transform spectral data to time-domain ... 88
Specifying Output ... 89
Specifying the Number and Properties of Layers ... 92
To insert a layer ... 92
To replace a layer ... 93
To delete/restore a layer ... 93
To import a layer ... 93
To build the Cole-Cole model: ... 94
Resistivity & Susceptibility Grid Data Files ... 94
Specifying the Number and Properties of Targets ... 95
To insert a target ... 95
To delete/restore a target ... 106
To import a target ... 107
To import topography ... 108
To specify the number of sampling points ... 109
To specify the model size and position ... 109
To replace a target ... 109
Saving your model ... 110
DATABASE PAGE: ORGANIZING PROJECT INFORMATION ... 111
Projects ... 111
To change the name of a project ... 111
To delete a project ... 111
Surveys ... 111
To add a new survey ... 111
To rename a survey ... 111
To provide a survey with comments ... 111
To view or change comments ... 112
To create a backup survey ... 112
To delete a survey ... 112
Data Sets ... 112
To change the name of a data set in the list ... 112
To delete a data set ... 112
To specify the name of a model ... 112
To add, view or edit a model ... 113

To view or edit the survey configuration 113
To view data set information 113
To view grid information 113
To export a grid into the Profile Data Set 115
To export a grid into the Geosoft format 116
To export a grid into the XYZ format 116
To remove “no data” points from your grid 117
• Select either a digital or spatial filter type. Select the filter from the drop down list and click Apply.DATA CORRECTION PAGE 119
DATA CORRECTION PAGE 120
EMIGMA’S MAIN TOOLBAR 122
Database Toolbar 122
Basic Tools Toolbar 123

.... 123
Processing Tools Toolbar 123
3D Visualizer () 124
Exercise 1: Editing Layers and Prisms/Plates 124
Viewing Data in VisRD (continuation of Exercise 1) 125
Exercise 2: Splitting and Manipulating Prisms 126
Plotting () 128
Static, Frequency- and Time-Domain Systems 128
To edit plot settings 129
To switch between the profile and spectrum/decay display 131
MT and CSAMT Systems 132
To edit plot settings 133
Loading Additional Data Set(s) 135
Viewing Plots 135
To switch between profiles and display multiple profiles 136
To switch between channels, separations and transmitters 137
To view multiple plots at a time 137
To view model properties 138
To adjust the scale 138
To switch grid on and off 139
To customize plot appearance 139
Converting to Various Displays 141
Saving Plot Settings 141
Printing Plots 142
Auto (Full-Screen) Mode 142
Scaled Graphic Mode 143
Contouring () 145
Working with Contour Display 146
2D and 3D display 148
Displaying Contour Lines and Intervals 149
Moving Your Plot. Zooming In and Out 149
Color and Black-and-White Representation 150
Viewing Profiles and Transmitters 151
Adjusting the Range of Data 151

Legend ... 151
Axes ... 151
Title and Captions ... 152
Print and Preview ... 152
Forward Simulation ( ) ... 155
Model Suite Generation ... 157
Location Load Mode ... 159
Batch Simulation ... 161
Green's function ... 161
Gridding ( ) ... 163
Grid Presentation ( ) ... 105
Adjusting Grid Display ... 106
To change the draw mode of your grid ... 106
To adjust the range of data to be displayed ... 106
To adjust the contour lines ... 106
To view your profiles ... 107
To view grid lines ... 108
To adjust axes ... 108
To toggle the coordinate grid on and off ... 109
To display and customize the legend ... 109
To provide the full-screen view of your grid ... 109
To toggle off any of the bars and panels ... 109
To zoom in on a grid fragment ... 110
Generating a map ... 110
Saving and Printing ... 112
To save your grid settings ... 112
To save your grid in a different format ... 112
To load previously saved grid settings ... 112
To print your grid as is ... 112
To specify the surroundings of your grid ... 112
Analytic Signal (AS) and Horizontal AS ... 113
Decay Rate ... 113
Euler Overlays ... 114
MultiGrid ( ) ... 116
To switch to another grid ... 116
To display multiple (up to 4) grids at a time ... 117
To compare various components of the same grid ... 117
To display and customize profiles ... 118
To switch between the Profile Data and Grid Data modes ... 118
To specify the draw mode ... 119
To display and customize contours ... 119
To display and customize the legend ... 120
To display and customize the coordinate grid ... 120
To display and customize grid locations ... 121
To load a map underlay ... 121
To add and customize titles ... 122
To switch to proportional axes ... 122

To zoom in and out 122
To move a grid 122
To hide a grid 123
To print/print preview a grid 123
PEX Show ( ) 124
To go through all locations of a line 124
To view cell information 124
To better view the CDI pattern: 125
To switch to another line 125
To switch between resistivity, conductivity and susceptibility 125
To switch between the linear and logarithmic scales 125
To specify the draw mode 125
To switch between the X and Y axes 125
To customize the range of data to be displayed 125
To carry out interpolation 125
To display and customize the legend 125
To customize the axes 126
To draw and customize contours 126
To save your grid settings 127
To load previously saved grid settings 127
To print your grid 127
SECTION IV - BEHIND THE USER'S INTERFACE 128
CHAPTER I - INSIDE THE SCATTERING ALGORITHMS 129
CHAPTER II - MODEL SPECIFICATION AND PROGRAM CONTROL 133
PRISM INPUTS 134
PLATE INPUTS 135
COLE-COLE INPUTS 139
MT STATION INPUTS 140
CONTROLLED SOURCE INPUTS 142
PROFILE INPUTS 148
NORMALIZATION INPUTS 148
CHAPTER III - USERS GUIDE TO EFFECTIVE MODELLING 160
CHOOSING PHYSICAL PARAMETERS 161
SETTING UP COORDINATE SYSTEMS 172
SAMPLING FIELDS IN PRISMS 173
MODELLING TIME DOMAIN SYSTEMS 175
CHAPTER III-A - EXTENDED TRANSMITTER SYSTEMS 176
DIPOLE-DIPOLE IP/RESISTIVITY 177
LOOP - BOREHOLE SYSTEMS 179
CENTRAL LOOP SOUNDING 179
FIXED LOOP - MOVING DIPOLE RECEIVER 180
SIROTEM DUAL LOOP CONFIGURATION 180
CSAMT 181
MELIS 182
MMR 183
CHAPTER III-B - DIPOLE TRANSMITTER SYSTEMS 184
HLEM (MAXMIN/SLINGRAM) 185
HEM 188

SAMPO 189
CROSSHOLE SURVEYS 190
CHAPTER III-C - NATURAL FIELD AND PLANE WAVE SYSTEMS 191
MAGNETOTELLURICS 192
SECTION V - EXAMPLE CALCULATIONS 193
INTRODUCTION 194
EXAMPLE 1: Max Min - Multiple PLATES 194
EXAMPLE 2: Induced Polarization - IP/Resistivity 196
EXAMPLE 3: Thin-Sheet Models - HLEMsystem 197
EXAMPLE 4: Comparison with a 3-D Code - Coplanar Helicopter System 199
EXAMPLE 5: Natural Field Modelling - Magnetotellurics System 201
SECTION VI – TROUBLE SHOOTING 202
SECTION VII - TUTORIALS 203
SECTION VI: GENERIC LICENSE 204
SECTION VIII - REFERENCES i
REFERENCES ii
Papers ii
Technical Talks In PowerPoint format iv

EMIGMA© /V8.1

3-D Electromagnetic, Magnetics, Gravity, IP, MT/CSAMT 2004 Comprehensive Geophysical Modelling and Inversion

SECTION I - INTRODUCTION

WHAT EMIGMA V8.1 OFFERS

3D VizRD:

- Interactive, flexible, and fully 3D visualization tool
- Completely mouse-driven
- Fully integrated with Windows 95/NT/98/2000
- Visually construct, display and manipulate geological models in 3D with respect to measured, simulated and inverted data, model structures (prisms, plates, layers, etc…) survey design (transmitters and profiles), boreholes and borehole logs
- Modify attributes of the prisms, plates, polyhedra, layers, topography and survey within the visualization module
- Import CAD geological specifications through a GEMCOM or Vulcan file importer and build and manipulate 3D models from them
- Reduce errors in model setups by visualizing the survey and model parameters
- Intuitive interface and its ability to give immediate physical meaning to data can increase the productivity of the geophysicist.
- Residual Plotting
- Plot Z data at one time
- Plot data as vectors, lines, surfaces and contoured surfaces

EiKPlot: x-y plotting software designed just for geophysical data.

- Integrated X-Y plotting for easy and quick viewing of your profile decay or spectral data.
- Allows comparison of actual data with simulated and inverted data
- Select the field components you would like to plot.
- Produces a professional graph to scale with labels and a comment box
- View both profile and decay representations of time-domain data or profile/spectral displays of frequency domain data
- Toggle through time or frequency channels, profiles, transmitters and data stations
- Display IP data as chargeability or as apparent resistivity
- CSAMT/MT data as apparent resistivity, phase, Tippers or field units.
- Residual Plotting
- Convert EM data to Apparent Resistivity
- Plot data and derivatives
- Plot Pseudo-Depths and Resistivities

EMContour: a 3D contour application including interpolated inversion volumes

- Accurate contouring of EM, Magnetic, Gravity MT or IP data over your survey grid

- Residual plotting
- Contour decay rates

GridPresentation: an enhanced grid tool

- displays your interpolated grids
- fills grids in 3 ways
- Equal Area or Equal Weight displays
- contouring and filling
- profile data overlays
- raster map overlays
- convert data to apparent resistivity

ProfileModifier: datapoint editing and viewing

- view data point locations with editing
- data overlays (stacked profiles)
- visual data subset selector
- analyse data over survey

ProfileCorrector: spreadsheet plus plotting for editing

Toolbox: filters, compression, processing

PseudoShow: generalized pseudo-section tool

- display pseudo-sections
- by separation, frequency or time
- grid and contour
- step profiles or transects

PEXShow: displays inversion sections

- show 1D Inversions
- contour and grid
- resistivity/conductivity/permeability/log/linear/equal weight/equal range

FFT Processing: perform 2D FFT functions

- integrated 2D FFT tools

- windowing, wavelength filtering,
- upward/down continuation
- reduction-to-the pole
- derivative generation

<u>Field Data Importers</u>: Import field data directly into *EMIGMA* V8.1 for

Ease of modelling:

- Automatic selection of data locations, transmitters, receivers and data channels eliminates the need for users to specify the system and survey configurations. The user is required only to define the 3D geological and electrical model before the survey can be simulated.
- Transforming imported simulated data to time domain is essentially automated as the base frequency, waveform configuration, ramp times, data channel times and normalization are automatically selected.
- *There is no need for you to know the system specification of your data, we have done the work for you!*

Data matching:

- Immediate viewing and investigation of field data. Once imported, you are ready to view and analyse your field data in our 2D Plotter, our 3D Visualizer or our Contouring tool. All of these are designed to help in the careful analyses of your data for insight into your geophysical anomalies and your geological targets.

EMIGMA is then relatively automatic for direct comparison of the modeled data to the measured data.

Based on a hypothesis of your geological environment, build a layered earth and add a target to your imported survey. Simulate the response in Forward Simulations and directly compare to your measured data.

<u>1D Inversion:</u>

- 1D Inversion, Depth Section Display and Data Fit Displays
- FEM – ground and airborne
- MT/CSAMT Impedance Inversions
- Resistivity Data Inversion
- TEM – ground and airborne

<u>3D Magnetics/Gravity Inversion:</u>

Generate 3D volume models of susceptibility from your magnetic data with linear and non linear inversion tools:

- Full 3D gridded inversions
- Magnetization Vector Inversions

- Optimization & Direct Matrix Inversion
- Physical sensitivity functions
- Iterative non-linear solutions
- Iterative born approximations
- Suite of minimization techniques
- Removal of non-susceptible cells
- 3D Extended Euler depth inversions

EIKMap Tools:

Apparent Resistivity Calculation for HEM Data: - Golden search plus Monte Carlo

Pseudo-Depth and Resistivity for HEM Data: - Combined Sengpiel and Fraser Pseudo-Depth

Polygenerator Tool:

Generate synthetic topography and anomalies

Data Exports:

To XYZ, PetRos EiKon's V6.X generation .pev format, to GeoSoft .grd files and export whole or partial databases to a new database file.

CHANGES FROM PREVIOUS VERSIONS

EMIGMA V7.x is a complete redesign from earlier EMIGMA V6.x generations. However, many conventions and principles are still maintained from the earlier versions and form fundamental concepts for all EMIGMA versions. Thus, this manual is designed as a fundamental reference book for the use of EMIGMA.

EMIGMA V8.1 can be run on either of the operating systems of Windows 95, Windows 98, Windows ME, Windows NT, Windows 2000 or Windows XP(Home and Professional) and now Windows Vista. It is entirely a Windows style program with integration between applications through a database.

In 8.1, applications share memory and interact directly through memory.

Features of V8- Release 1

Data Organization

In previous versions, the data file format was limiting. The user could easily generate many files in a modelling exercise which were difficult to organize and remember the details of the models. Recovery of interpretation status at a later time was awkward unless the user developed comprehensive file naming structures, or took copious notes on the model files. Additionally, integration of several data sets in the interpretation process was extremely difficult when a single file was generated for each model or inversion.

In V8, multiple datasets and models can be contained in a single database file. Multiple organizational levels are provided allowing for a variety of organization criteria depending on the user's preference. As examples, the user may organize by interpretation project, data type or simply organize all data and projects in one database file. The user may organize in a single project (one organizational level in the database file) several data sets for more ready analyses of the different data types and integration of models between data types. This framework will soon lead to integration of data types when viewing data within EMIGMA V8.1.

New data channels are provided such as GPS (x,y,z) , Altimeter, and FID .
The design of V7, will allow us to add data channels such as borehole geological or geophysical logs.

Data Processing and Reduction

Large data sets are now allowed (V6 was limited to 2000 data points per data set). In V8, the user can easily compress data or create subsets of surveys. You can import your entire data set and create subsets by compressing and/or selecting or removing part of the data. This allows you to more rapidly work with an interesting part of the data without modifying the entire original data set.

- An array of FFT tools for both gravity and magnetic data
- Two new reduction-to-the pole algorithms

- Wavelength filters plus upward/downward continuation.
- FFT tools may be performed on both measured and simulated data for evaluation of user processing
- Digital and spatial data filtering techniques allowing user-selectable filter sizes for both 1D and 2D filters

Data Editing and Correction

Correct the sign of entire components, fix individual values, remove bad data points. Features include, among others:

- Line sorting
- Line deletion
- Data compression with statistics calculation
- Select data through data display
- Remove a component Multiply or Shift data values
- Insert new data values
- Delete a subset of data
- Coordinate editing including GPS Z and FID

Tool Integration

All simulation, data display, inversion, gridding tools, etc are fully integrated in V8.1

3D Modelling

- First 3D gravity modeling algorithms
- Increased capabilities for magnetics modelling
- A new more accurate and faster 3D resistivity modeling algorithm
- Improvements to its electromagnetic modeling algorithms
- Extended batch mode simulation
- Greens' functions interpolation for faster multiple models
- Model suite generation and automatic "best model" analyzer

Inversion

- New 1D TDEM inversion and now both in-loop and out-of-loop configurations, including airborne sytems
- Improvements to 1D inversion capabilities for FDEM and Resistivity
- Extended and improved 3D magnetics inversion capabilities
- Gradient Magnetics Inversion
- 3D Euler Deconvolutions with Rodin Post-Processing

V8 Tools

- ***EM-Mapping*** – depth section determination and apparent resistivity tool for FEM and Resistivity
- ***Excitation field calculation and viewing tool*** – generate incident/freespace field and view in 3D
- ***Natural Neighbour, Delauney Triangulation, Minimum Curvature, Thin-Plate-Splines gridding and interpolation tool*** – interpolate your data onto a regular grid using the latest interpolation technologies

- ***3D Magnetics inversion extensions*** **-** utilize multiple altitude level data if available, automatic naming of inverse models including type, associated grid, etc
- ***1D inverse extensions for FEM, MT, CSAMT, Resistivity*** **–** included is a new Occam inverse with automatic layer reduction and joint susceptibility/resistivity inversions
- ***Direct comparison of model responses*** **–** compare model responses or compare field data to multiple models, view model changes in Visualizer as you step between models
- ***Data and configuration editing, data compression, data reduction*** **-** data compression with error estimate calculation, edit system configuration, edit and correct data, extract data
- ***Advanced Modelling Tools***
  a) Model suite generation – run a suite of layered earth or plate models (prism suites in development)
  b) Large data set simulation – new simulation engine allows user to model large data sets
  c) Extract previous model responses from data: e.g. extract regional magnetic response prior to inversion, remove overburden effects through modelling, build-up models for large data sets by modelling target by target
- ***Transform*****:**

a) Improved frequency interpolation allowing more accurate time domain calculations with fewer frequencies – up to 60% reductions in calculation time with improved accuracy, as much as 80% reductions with similar accuracy
b) Transforms of large airborne TEM profiles
c) Significantly reduced memory allocations to allow for transform of large datasets

- ***Data Analyses***
  a) EiKplot, Visualizer, Contour – plotting and display of the data from multiple models against each other or field data in the same plot
  b) Plot residuals between your model and field data or between models
  c) Stepping between models in Contour and Visualizer (with model views)
  d) Improved data display and new features in Visualizer
  e) Many new mapping capabilities
  f) New gridding algorithms and grid analyses tools
  g) A wide selection of datums for UTM projections as well as projections for polar latitudes
  h) Import and calibrate a raster map to UTM coordinates or latitude/longitude
  i) Overlay raster maps with data grids
  j) New graphical data editing tools
  k) Display by FID (fiducial)
  l) Display altitude, GPS Z, Apparent Resistivity, Estimated Error, etc
  m) FFT processing
- ***Visualization***
  a) A new tool for viewing data versus pseudo-depths
  b) A new tool for analyzing 1D inversions as geoelectric sections
  c) New tools for building 3D models in conjunction with data
  d) Improved image exports in both raster and vector formats
  e) GridPresentation and a new MULITIGRID tool
- ***New data imports***

  Airborne Time Domain
  Large data set FEM and Magnetics
  IP/Resistivity – Geosoft, BRGM, Res2D
  EM34/31/38
  AMIRA TEM EDI for Magnetotellurics

EM61-63, EM31-R,
Gradient Magnetics

- ***Compatibility with earlier versions*** EMIGMA V6.4 PetRos EiKon files (*.pev) can be
- directly imported into the database and organized by dataset. Datasets can also be exported from the V8.1 database back to a .pev file for use in EMIGMA V6.4.
- ***Sharing Models and Interpretations*** **–** the user may export an entire database or subsets of the database to another database either across the net to another user's database or zip the resulting directory and e-mail the database and attached flat files to your colleagues. You can include your field data, grids, models, inversions, plot settings, etc.

As well as:

- ***PEX Show*** - a new depth section viewing tool for 1D inversions
- ***Pseudo-Show*** - a new pseudo-depth section tool for data with multiple separations, frequencies or time windows
- ***Aeromagnetic Compensation***
- ***New Data Filters*** - Median, Mean, Polynomial (1D and 2D)
- ***Base Station Magnetic Corrections***

PETROS EIKON DIRECTORY

EMIGMA7.x expects to find a *PREiKon* directory on your disk, and will build one if one is not found. It utilizes this directory as a working directory for a variety of purposes.

THE SOFTWARE DONGLE

If you are running ***EMIGMA*** in a WINDOWS-based systems, a software dongle will be necessary. The software dongle contains different codes which control the availability of various aspects of EMIGMA. The dongle is read on startup, and ***EMIGMA*** is configured in accordance with the contents of the dongle. In this way, it is possible to customize ***EMIGMA*** to particular individual requirements.

The software dongle should be plugged into the parallel port of your computer before ***EMIGMA*** is started, and should remain in the port for the duration of the model run. The executable may be placed on a number of computers, but the dongle must be on the computer when executing.

Note: For new users using Windows NT, 2000 or XP a driver must be installed to see the dongle.
The driver installation software is found inside /EMIGMAV8.1/drivers.

Note: Parallel or USB dongles are available.

CONVENTIONS
The following conventions are used in ***EMIGMA*****:**

Right handed coordinates: *z* up, optionally *x* east, *y* north, but the selection of *x* and *y* depends on the model

Phase: Phase convention of exp(-iwt) unless otherwise specified. It can be switched when displaying.

Strike: Strike angles are measured clockwise in degrees from y (North) to x (East).

Dip: Dip convention is measured based on the right hand rule. If your right hand thumb is pointing in the direction of strike, your anomaly (your fingers) dip down to the right.

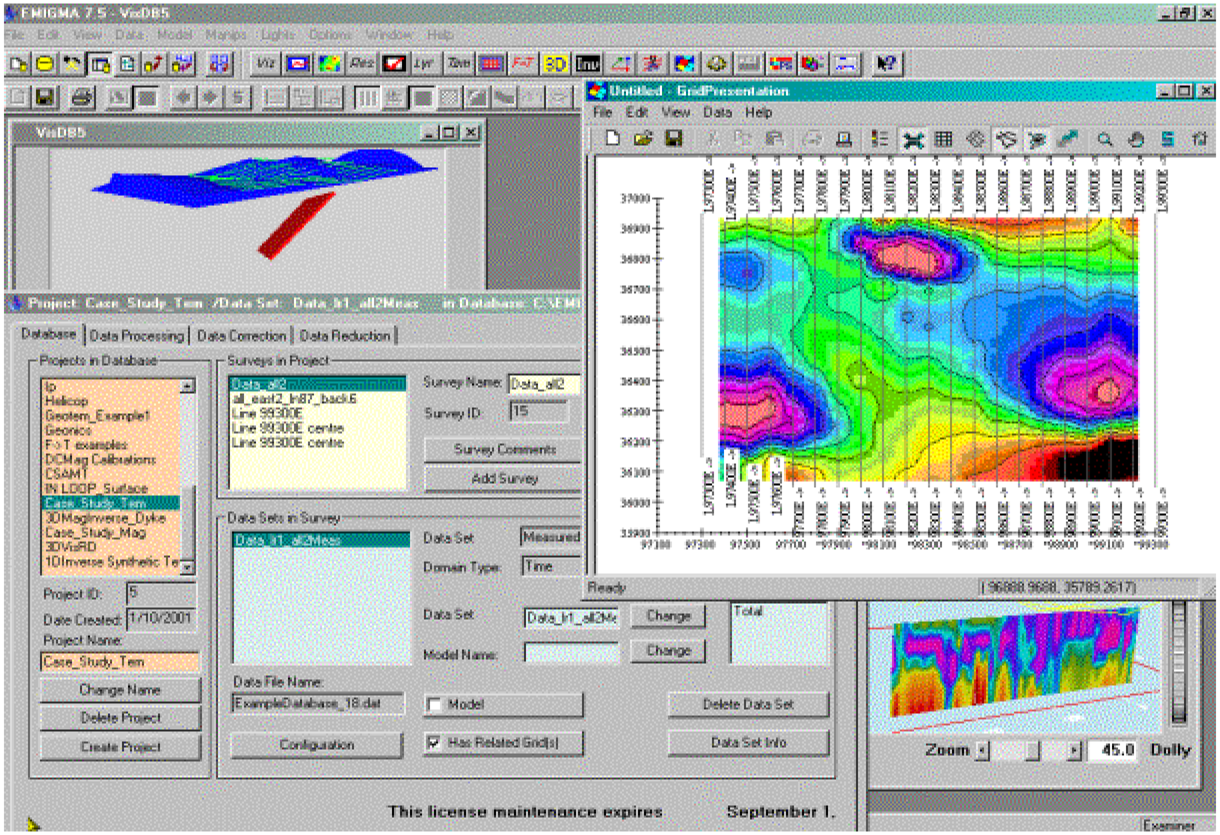
Declination: Declination angles are in degrees clockwise from y towards x.

Units: SI units. Distances are in meters, angles in degrees.

SECTION II - EMIGMA V8.1 OVERVIEW

EMIGMA

Premium and Professional editions



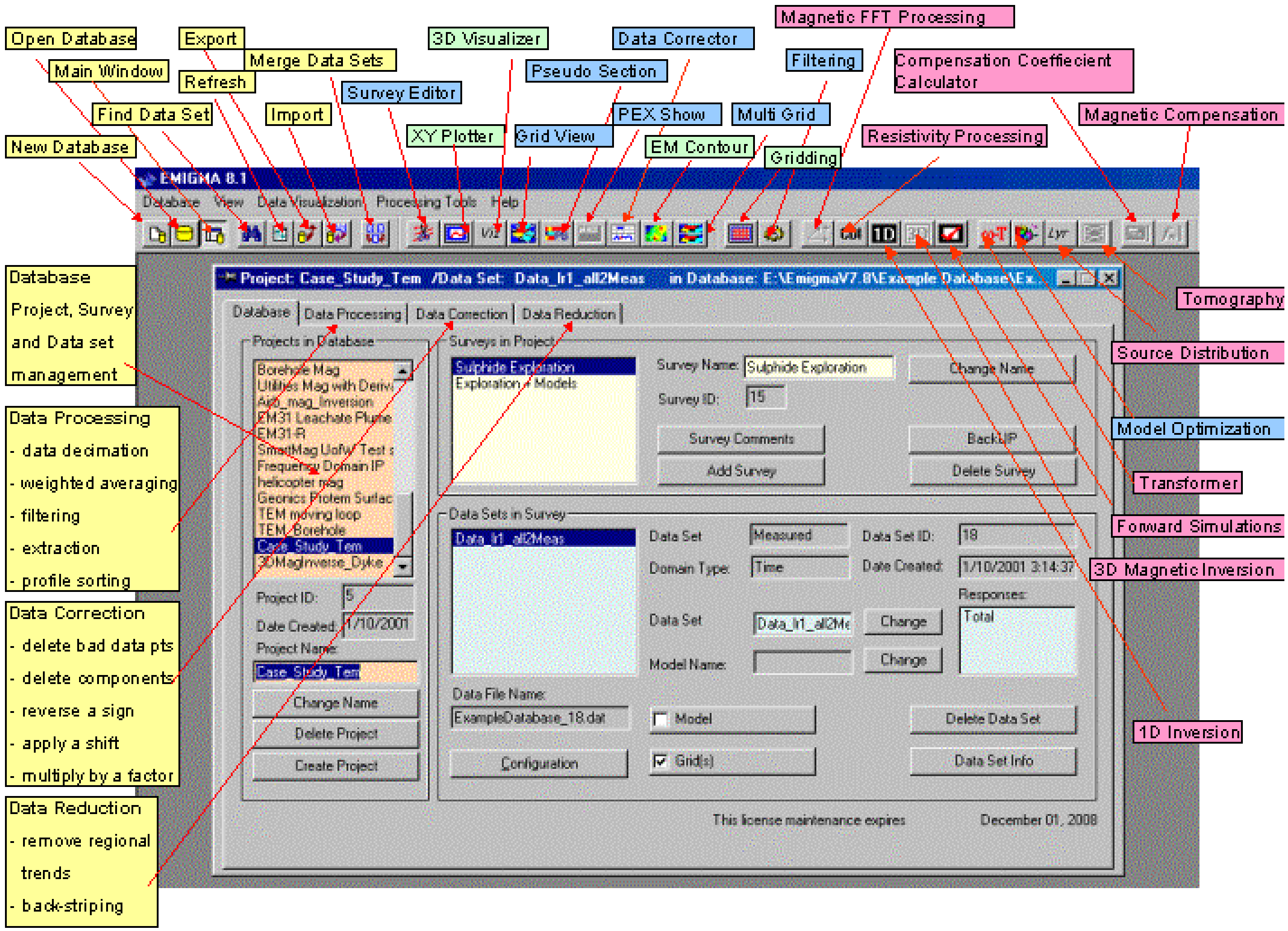
PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

EMIGMA Overview

EMIGMA – Database Version

Database Design Objectives

- Reduce user's time required for modelling and data analyses
- Enhance data analyses capabilities
- Improve ability to report on work and to recover interpretations at later times
- Integrate all tools into one platform
- Include friendly data editing, compression, filtering and processing
- Allow large datasets to accommodate airborne data analyses and large surface data sets
- Provide a framework for inversion tools and their integration with simulation tools



Open Database
Export
Merge Data Sets
Main Window
Refresh
Find Data Set
Import
New Database
3D Visualizer
Survey Editor
XY Plotter
Grid View
Data Corrector
Pseudo Section
PEX Show
EM Contour
Magnetic FFT Processing
Filtering
Multi Grid
Gridding
Compensation Coeffiecient Calculator
Resistivity Processing
Magnetic Compensation
Tomography
Source Distribution
Model Optimization
Transformer
Forward Simulations
3D Magnetic Inversion
1D Inversion
Database
Project, Survey and Data set management
Data Processing
- data decimation
- weighted averaging
- filtering
- extraction
- profile sorting
Data Correction
- delete bad data pts
- delete components
- reverse a sign
- apply a shift
- multiply by a factor
Data Reduction
- remove regional trends
- back-striping
EMIGMA 8.1
Database View Data Visualization Processing Tools Help
Project: Case_Study_Tem /Data Set: Data_lr1_all2Meas in Database: E:\EmigmaV7.8\Example Database\Ex
Database
Data Processing
Data Correction
Data Reduction
Projects in Database
Surveys in Project
Survey Name:
Sulphide Exploration
Change Name
Survey ID:
15
Survey Comments
BackUP
Add Survey
Delete Survey
Data Sets in Survey
Data Set
Measured
Data Set ID:
18
Domain Type:
Time
Date Created:
1/10/2001 3:14:37
Responses:
Total
Change
Model Name:
Project ID:
Date Created:
1/10/2001
Project Name:
Case_Study_Tem
Change Name
Delete Project
Create Project
Data File Name:
ExampleDatabase_18.dat
Model
Delete Data Set
Configuration
Grid(s)
Data Set Info
This license maintenance expires
December 01, 2008

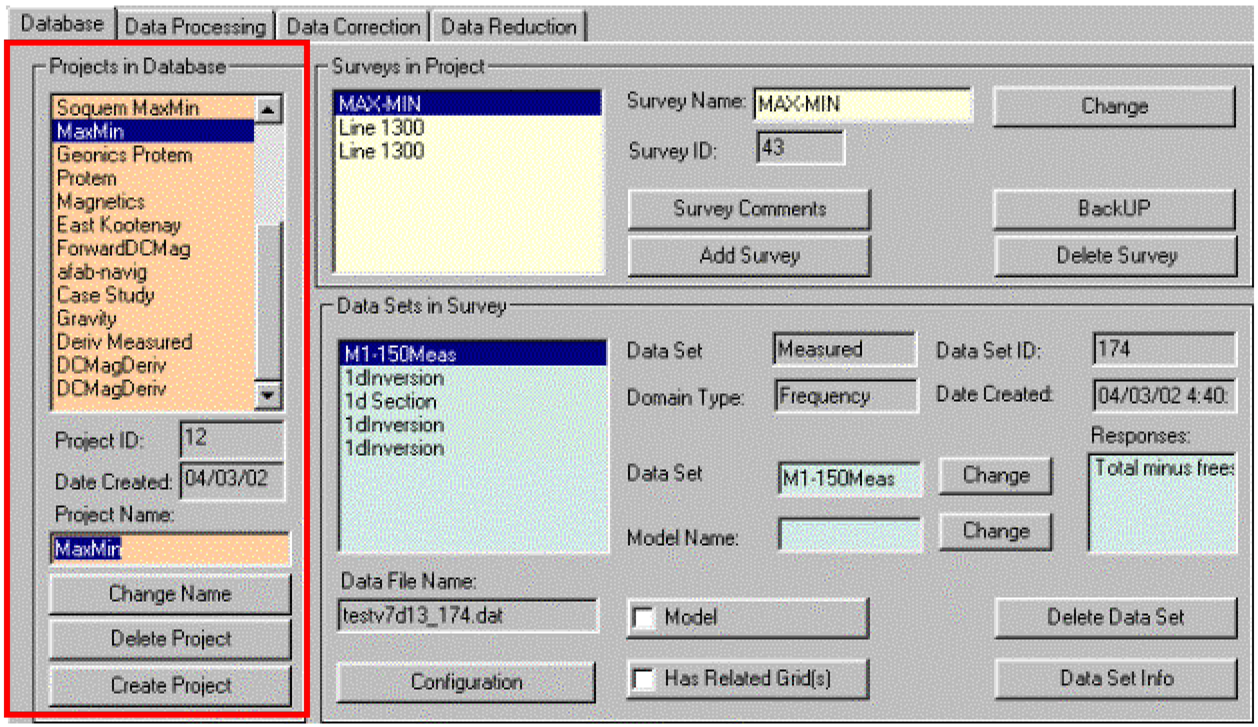
Data Organization

The user can easily generate many files in a modelling exercise, and the models are organized in the database structure making recovery of interpretation status at a later time easy.

The backend of EMIGMA is an object database. It allows you to organize data in three levels: projects, surveys and data sets. You can have different projects based on a variety of criteria, such as the type of your data or the interpretation method you are going to use. Or you may choose to create a single project for multiple datasets to provide more ready analyses of different data types and integration of your interpretation across all data sets within a project.

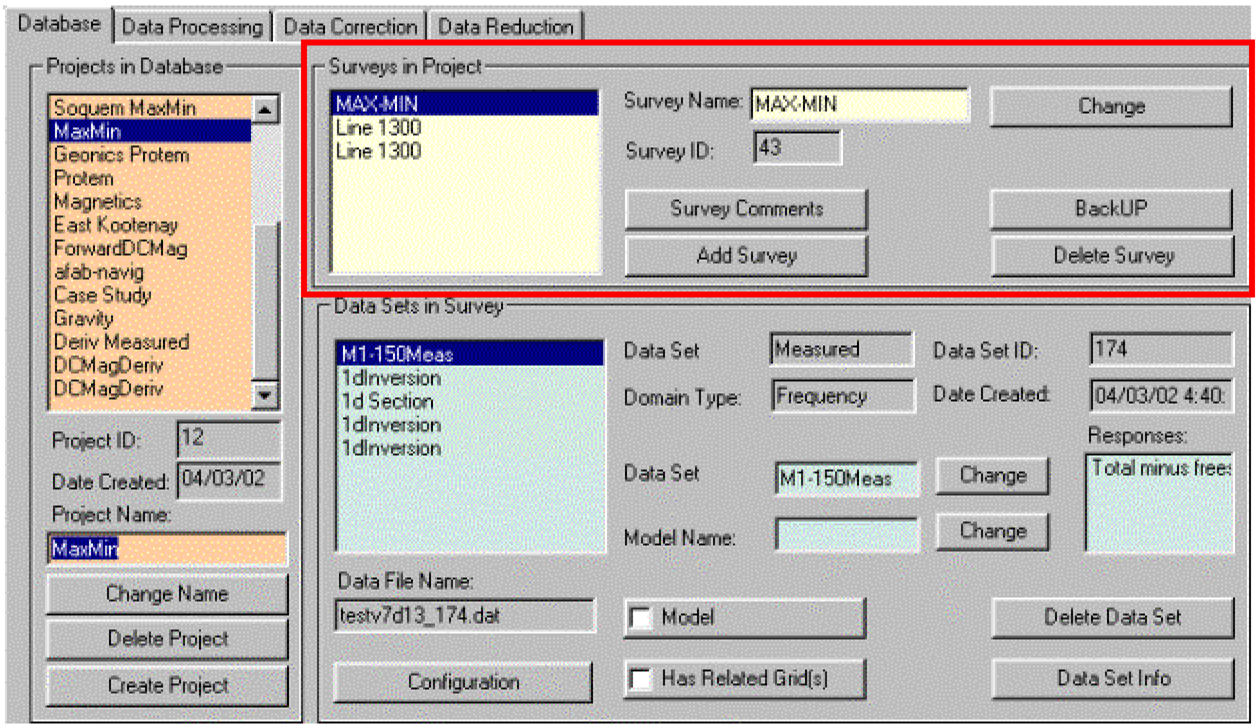
New data channels are now allowed such as: GPS (x,y,z) , Altimeter, and FID .

NEW – You can import and image your Gravity data as non-configured geophysical data. – V7 includes Gravity modelling



Projects in Database
First level of Project management

Change Name - the Project Name can be changed at any time by typing in the Project Name and selecting the Change Name button.
Delete Project – Projects can be deleted by selecting the Project, then selecting the Delete project button.
Create Project – Create a New Project by selecting the Create Project button.



<u>Surveys in Project</u> Second level of Project Management.

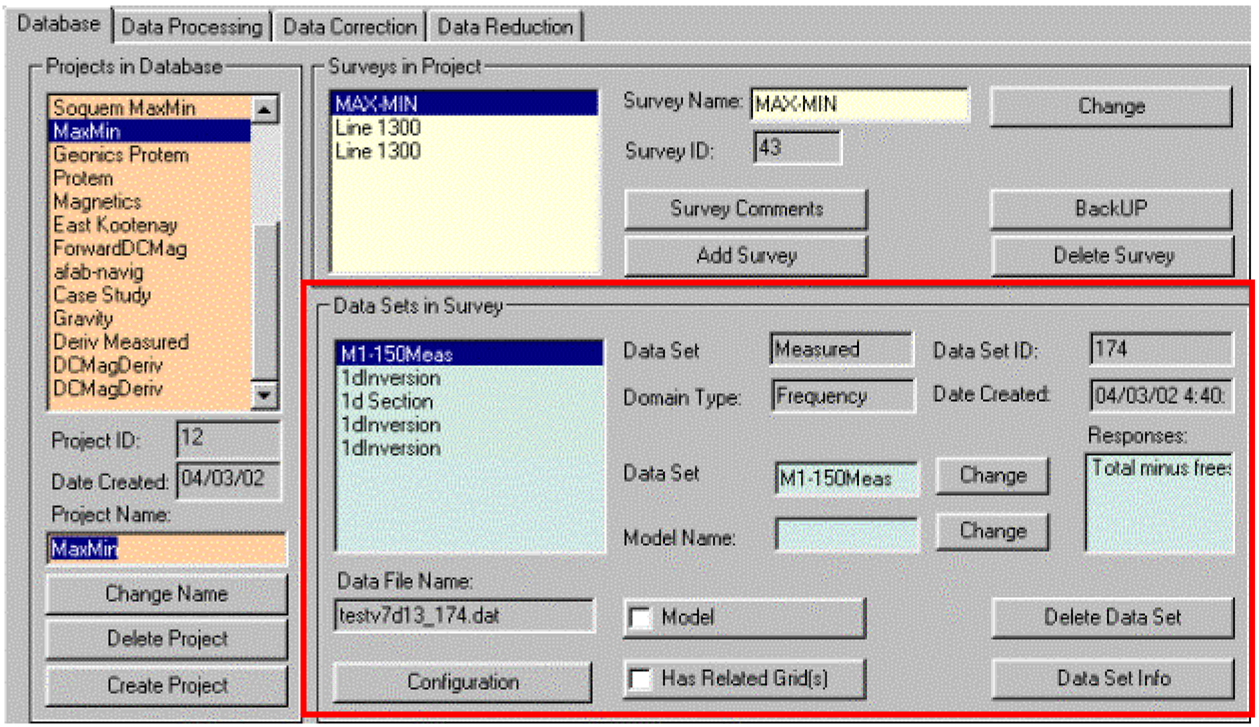
<u>Survey Name</u> – rename your surveys at any time by typing the new name and selecting Change

<u>Survey Comments</u> – enter and store notes about your survey.

<u>Add a Survey</u> – generate a new synthetic survey.

<u>Backup</u> – generate a duplicate of your survey including all Data Sets.

<u>Delete Survey</u> – delete a survey at any time by selecting the survey and clicking the Delete Survey button

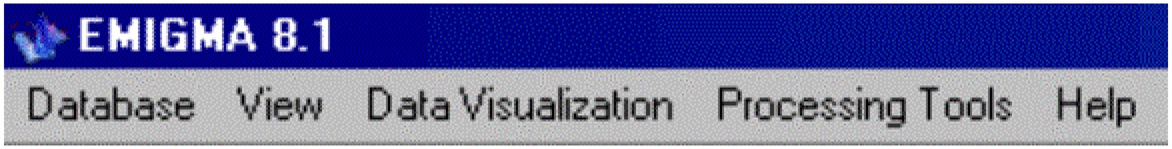


Data Sets in Survey

Third level of Project Management.

Data Set – displays whether the data is Measured (or field) data or Simulated Data
Domain Type – displays whether the data is Frequency domain, Spectral domain, Time domain or Static domain
Change Name - the Data Set name can be changed at any time by typing in a new name and selecting the Change Name button.
Model Name – a second identification string for your data set allows you to record additional details about your Data Set
Configuration – View and Modify information about your survey
Model – View and modify information about your layered earth and your prism/plate/polyhedra bodies.
Has Related Grid(s) – Grid management tool - contains all grids generated for this data set.

Database Tools



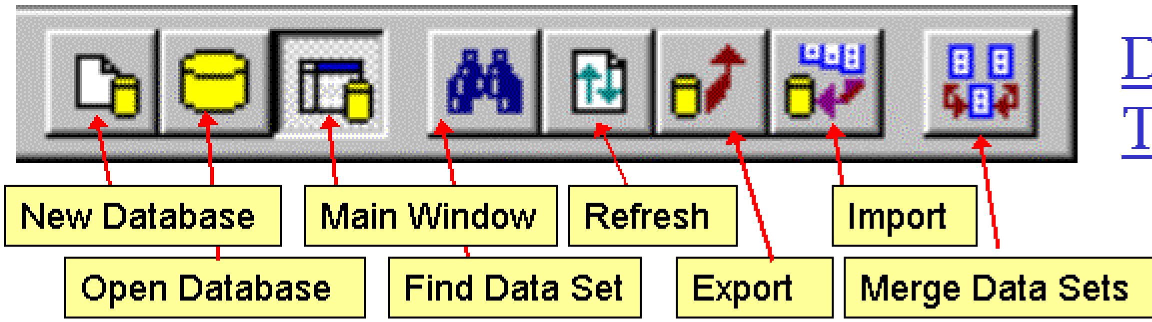
Database tools to manage your database *.mdb files. Allows you to create a new database file, open an existing database file, import data into the database, export data from the database, search for a data set, merge data sets and exit the EMIGMA program

View tools to manage your view options in EMIGMA including toggle display of the toolbars, status bar, and database colour.

Data Visualization allows you to view and display your data in various ways for analysis or reports

Processing Tools an extensive collection of tools to process the data in your surveys

Help displays online help topics, EMIGMA program version, and download the most current EMIGMA updates via proxy if necessary



Database Tools

New Database Open a new database file (*.mdb) into the EMIGMA program.

Open Database Open an existing database file (*.mdb) into the EMIGMA program.

Find Data Set a tool to locate a particular data set by date and EMIGMA data type

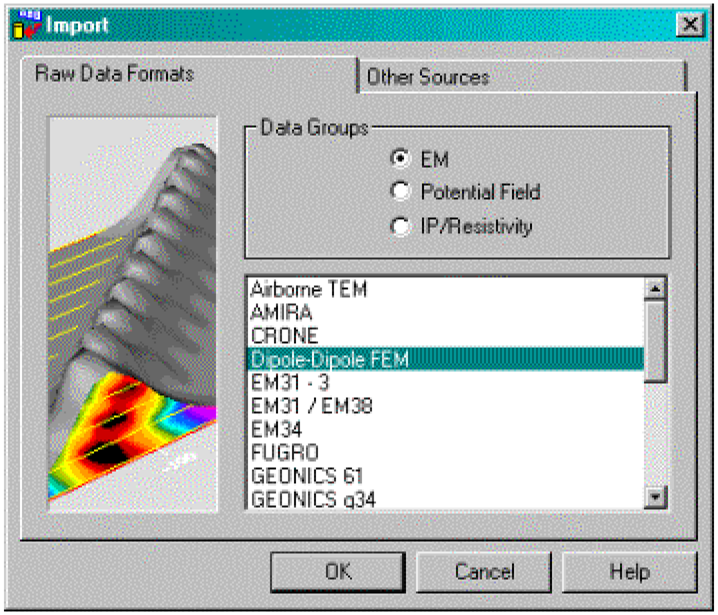
Main Window toggle display of the main Database window.

Refresh updates screen display

Import Suite of data imports to bring your data into EMIGMA as an ASCII file or manufacturers file. Import GeoTutor files into EMIGMA database.

Export Export your measured, simulated or inverted data to an .xyz file, to another database or to QCTool, GeoTutor and Geosoft applications.

Merge Data Sets a tool to merge two data sets from the same project but different surveys into one data set.



Data Imports

Import Utilities

A suite of imports are available which allow you to bring your measured data into the database in either the manufacturers file format or as an ASCII columnar data file.

Manufacturers Data Format

Enables you to bring measured data into the database in the manufacturers standard file format. Includes CRONE, GEONICS TEM, IP6, and AIRBORNE TEM (GEOTEM and QUESTEM) data.

ASCII Format

Enables you to bring measured data into the database in a ASCII columnar data format. Column labels are required for the program to recognize the data channels. If you don't have headers or want to modify the headers to our labels which can be read automatically by EMIGMA, an header editing tool is available in the import. Imports are available for DC MAGNETICS, GEONICS EM31/EM34/EM38, DIGHEM, MAX-MIN and IP/RESISTIVITY.

AMIRA Format

Enables you to bring measured TEM data into the database in the AMIRA file format. Includes GEONICS, CRONE, SIROTEM and ZONGE data.

PetRos EiKon

Enables you to import data (measured data with or without a simulation and pure simulated data) from our GeoTutor application. These files have an *.pev extension. You can bring in up to 10 files at a time.

Data Correction

Correct signs of entire components, fix individual values , remove bad data points.

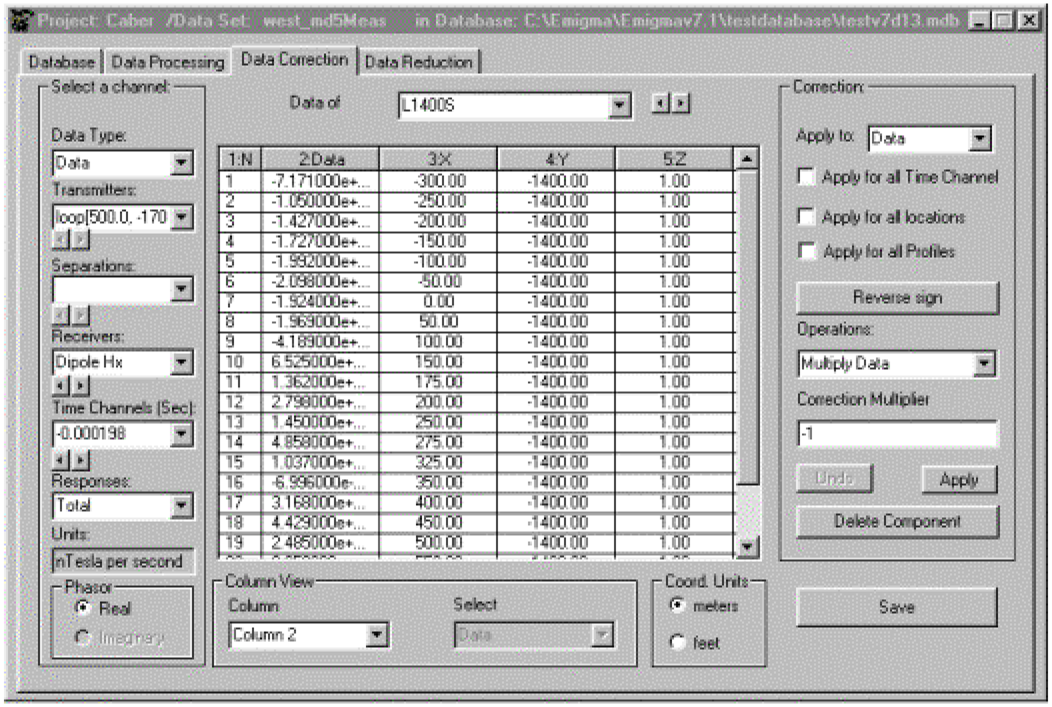
Some features:
Remove a component

Multiply or Shift data values

Insert new data values

Coordinate editing including GPS Z and FID

Display statistics information



Column View
defines the columns that are displayed.
Select a channel:
Select the Data Type, Transmitter, Receiver, Time Channel or Frequency, Response and Phasor of the data to correct

Correction Apply to
You can chose to apply to some or
-all time channels
-all locations
-all profiles

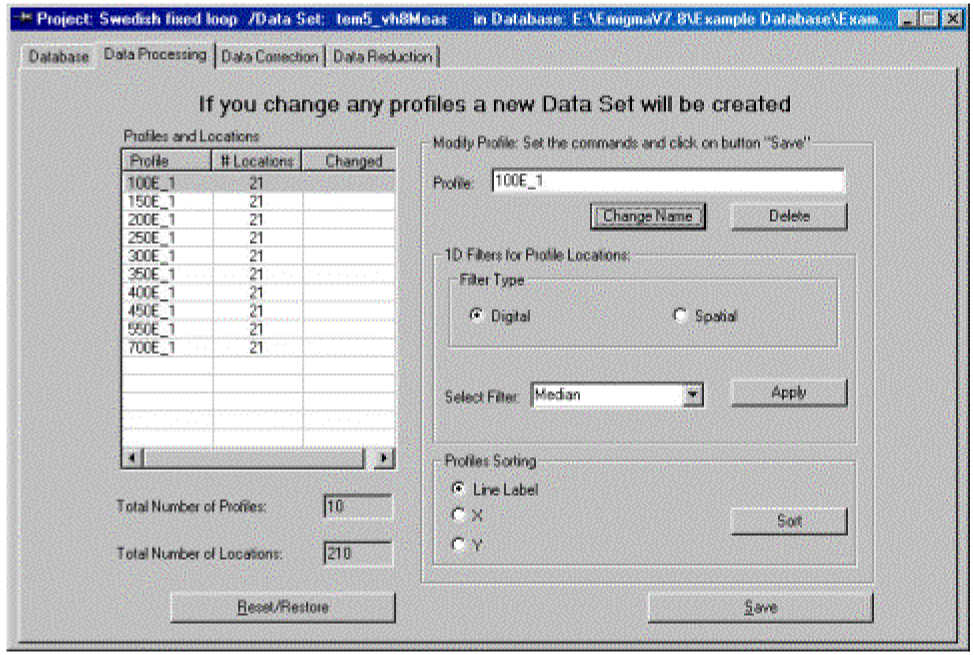
Operations
Select the Operation From
-Multiply Data
-Shift Data
-New Value
-Delete Points
-Delete Every

Reverse Sign
Can be applied at the touch of a button

Delete Component
A component can be deleted at the touch of a button

Once you are satisfied with your corrections, select **Save.**

Data Processing



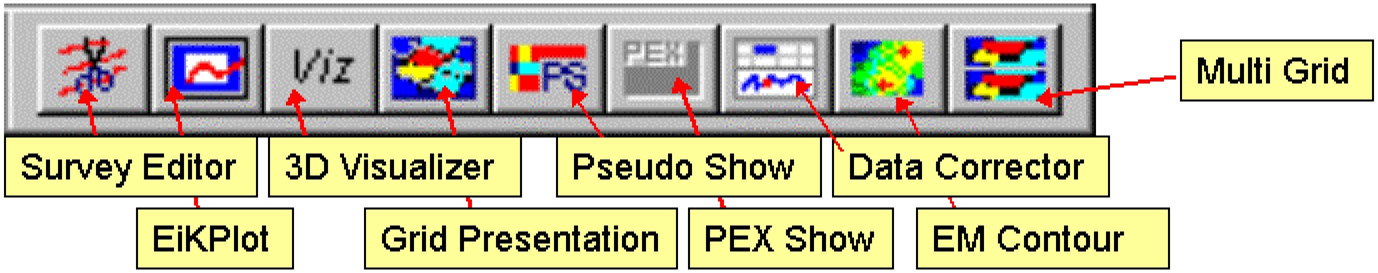
Profile sorting

by Line Label, X position or Y position.

Filters

Apply a Digital or Spatial filter to Data or Profile locations. Digital filters include Median, Gaussian, Mean and Savitzky-Golay (regular or irregular). Spatial filters includes Median, Gaussian, Mean and Savitzky-Golay (spatial radius).

Accessing Data Display Tools



Data Display Tools:

3D Visualizer State-of-the-art 3D Visualization and model building tool for viewing your data (measured, modelled and inverted) in 3D space, as profiles, vectors, true 3D surfaces or contoured surface with your 3D structure representation. Allows detailed analyses of anomaly position, shape and amplitude. View up to three data channels. Plot data from multiple models against each other or against field data. Step between models.

Data Corrector Edit data both graphically and in a spreadsheet.

EiKPlot Comprehensive XY Plotter for plotting your data, decays, positions and fiducials. Automated plotting. Plot to scale, multiple plots per page and save plotting defaults for rapid plotting of model suites.

EM Contour Gradient-enhanced gridding when measured magnetic gradients are available. Derive gradients from total magnetic field data. 3D volume contours of your 1D and 3D inversions with slicing/dicing tools.

Grid Presentation Display the data from grids created with the gridding tool.

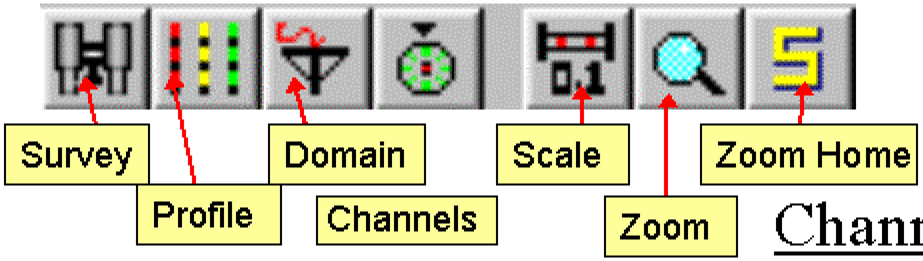
Multi Grid Display up to four grids at the same time.

Pseudo Show Display data sets created with CDI processor.

PEX Show Display data sets created with 1D Inversion and CDI processor.

Survey Editor Graphically edit the locations in the survey.

EiKPlot – Line Plotting



EiKPlot

Survey Allows you to chose a survey to plot when multiple surveys are available, e.g. MT surveys: surveys 1 and 2 = 1st and 2nd polarizations consecutively, survey 3 = impedances.

Profile Allow you to select a single profile to plot when you have multiple profiles. The Profile Selection window will open automatically. Either select a profile to plot or select multiple profiles to look at all of the profiles.

EiKPlot renumbers your profiles before plotting. Use the profile name or the Axis which determines the profile line to see which profiles correspond with which numbers. For a profile running North South, the X axis will tell you the start X position of the profile, which is usually the line name.

Domain Allows you to switch between the normal profile display and a decay or spectrum display. Mag data can only be displayed one way. Select OK.

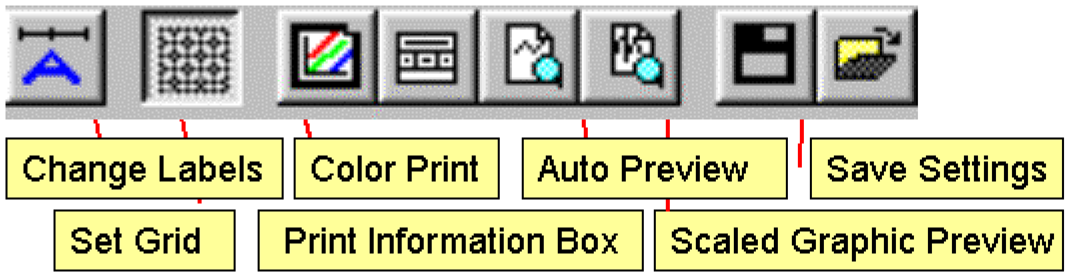
Channels EiKPlot will generally display your data plot automatically. To modify the data channels plotted, select the channels button, or simply double-click on the white space inside the plot. Click on the first white box under Plot #. Click on the yellow question mark under Field. Select the fields you would like to plot. Select measured or simulated data. Use fields available to view data contained in the file, and select from the X, Y, Z or total fields. Select OK, OK..

Scale Allows you to adjust the maximum and minimum settings for the X and/or Y axis. You can also adjust the units from fixed to exponential and change the axis to log scale or descending. Select the Scale button, or double-click on the axis you would like to rescale.

Zoom To zoom in on your data, select the zoom button and click and drag to select the area you would like to zoom in on.

Zoom Home Auto scales the plot using all positions on the profile

EiKPlot



Change Labels Title and axis labels can be adjusted. Their font size can also be adjusted.

Set Grid Turns grid on or off. The grid spacing is determined by the increment of each axis.

Color Print When selected, the plot will be printed in colour, otherwise, the plot will be printed in black and white.

Print Information Box

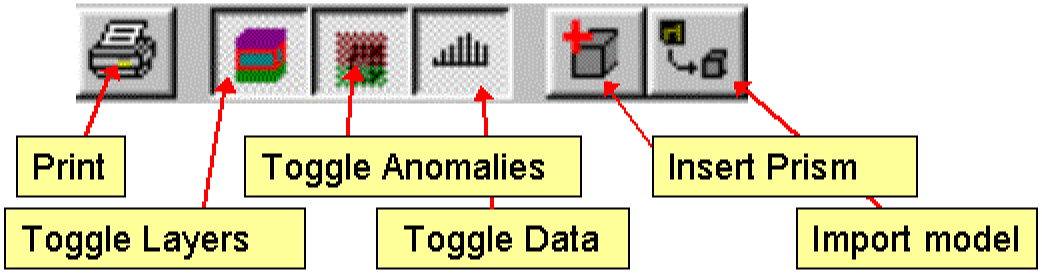
Auto Preview

Scaled Graphic Preview

Save Settings Allows you to save the defaults of your plot as a .plt file.

Get Settings You can then get the default setting files the next time you open this file, or any other model that has the same Tx-Rx configuration. This allows for rapid plotting of a number of models. For example, you could run a suite of models in batch mode, adjusting your layered earth, target positions, conductivity or size. Then plot the first model and save the default settings. Then open the next file and simply get the .plt files. The graph will be plotted for you automatically.

3D Visualization



3D Visualizer

Print Prints image in 3D visualizer using your windows print driver. The background will be printed white.

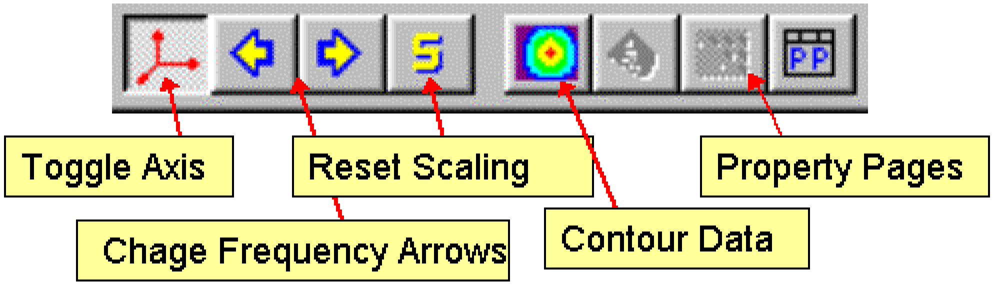
Toggle Layers Turns layer display on when pressed and off when released.

Toggle Anomalies Turns anomaly display on when pressed and off when released.

Toggle Data Turns data display on when pressed and off when released.

Insert Prism Model building tool to insert an LN Prism, ILN Prism or VHPlate. Once a prism is inserted, you can modify its size, position, orientation and conductance with the Properties interface.

Import model Allows you to import anomalies (prisms plates or polyhedra) from another dataset in your database, or from a *.pev file (GeoTutor III/EMIGMA V6.4 format).



3D Visualizer

Toggle Axis Turns on axis when pressed (default). To turn display of axis off, release button.

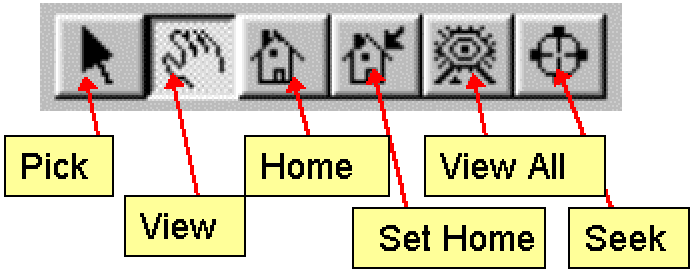
Change Frequency Arrows Allows you to step forwards and backwards through your frequencies or time channels.

Reset Scaling Auto scales your data to fit the screen

Contour Data When your data is displayed as a surface you can contour that surface by selecting the Contour Data button. Contours can be draped on the surface or flat.

Property Pages Interface to add and modify the properties of your anomalies and layers.

3D Visualizer



Pick Select Cursor. Use the arrow picking tool when selecting an object for modification in the Visualizer.

View Manipulate Cursor. Use the hand viewing tool when modifying the view of your model in the Visualizer, for example, when you are zooming or rotating.

Home Returns your view to the default home view.

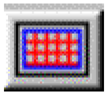
Whenever you modify your model you must Save to Database to apply the changes to your dataset.

Set Home Allows you to set a new home view,

View All Rescales your view so that the whole model can be seen on the screen

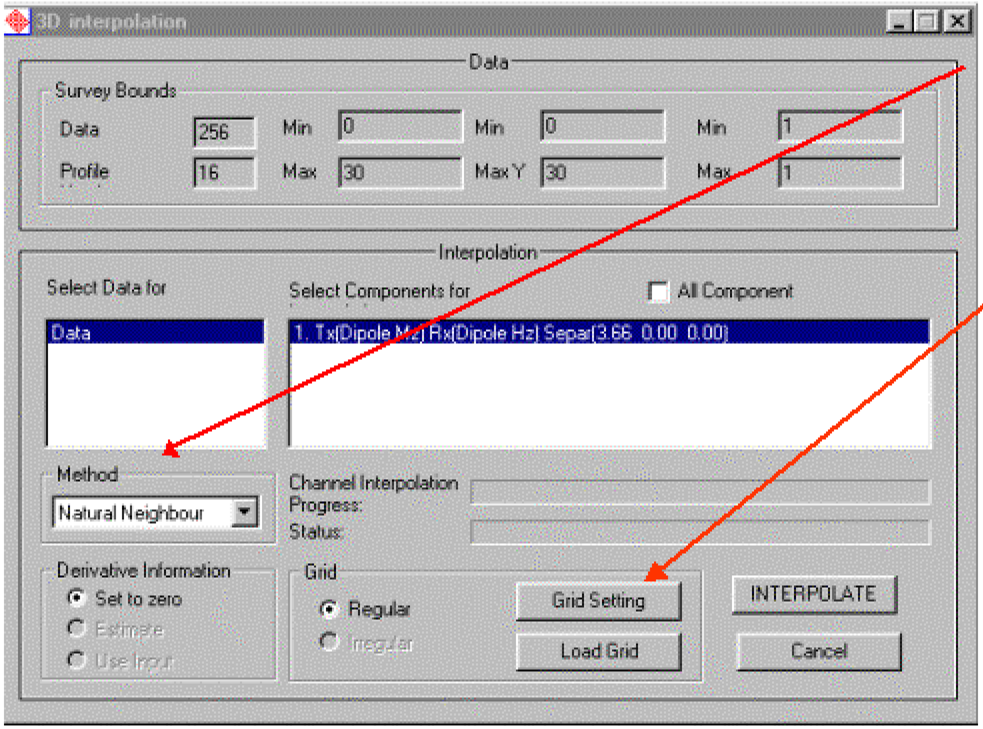
Seek Allows you to seek, or select a position, and then centres that position on the screen.

Gridding



Gridding Before contouring your data you need to grid your data first. Select your dataset and then select the Gridding button.

Gridding



Method Select your gridding algorithm from the list: Natural Neighbour or Delauney Triangulation, Thin Plate Spline and Minimum Curvature. Select Grid Settings to define your grid.

Gridding

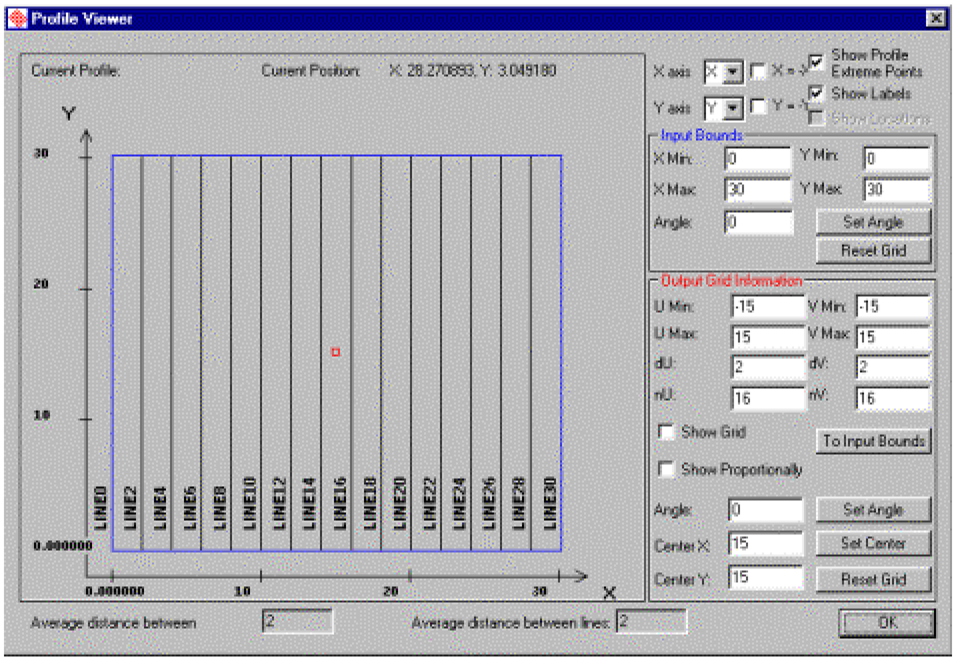
Profile Viewer

Displays your profiles and allows you to define the parameters of your grid.

You can control the X and Y orientation, whether the extreme points of your profile are show.

Input Bounds

You can use the whole survey as the input bounds or you can adjust the boundaries, either through the interface, or by clicking and dragging the blue input box. Controls also provided to set the grid angle. If you make an error, you can reset the grid.

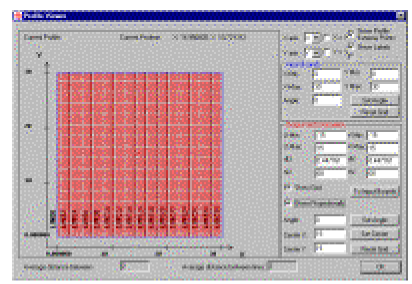


Output Grid

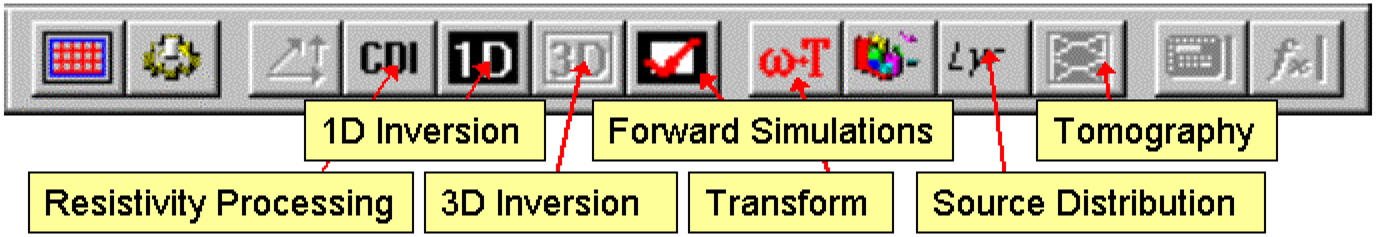
Likewise you can set the output grid by defining the U and V Min and Maximum (where U is the output X and V is the output Y), or by clicking and dragging the red output box (may be superimposed with the blue box). To define the grid resolution, set the width of the grid cells in the X and Y directions by setting dU and DV or set the number of grid cells by defining nU and nV.

To display the grid, turn on Show Grid. The grid will be auto scaled to fit the screen. To display the grid proportionally, use the Show Proportionally.

You can also set the angle, centre X and centre Y. If you make a mistake use the reset grid button.



Simulation and Inversion Tools



Resistivity Processing

1D Inversions and Sengpiel sections of your multi-line Resistivity data.

Forward Simulations

Perform simulations of a model.

Source Distribution

Generate a vector field describing the em field of the transmitter that can be viewed in the visualizer.

Tomography

Cross-hole EM tomography tool for licensed clients only.

3D Inversion

Full 3D inversions of your total magnetic and gravity data with iterative non-linear solutions and born approximations. Generate 3D volume models which can be viewed, "sliced" and "diced" in 3D VisRD. Euler solutions can be viewed in Grid Presentation.

1D Inversion

1D Inversions of your multi-line FEM, MT and CSAMT data. Generate 3D volume models which can be viewed, "sliced" and "diced" in EM Contour.

Transform

Time domain surveys are first simulated in Frequency domain and then automatically transformed to time domain in EMIGMA. For manual transformation users can generate the frequency domain response and then run the Transform.

SECTION III - STEP-BY-STEP GUIDE

This section of the manual deals with specific issues in regard to operating EMIGMA V8.x. For a more complete discussion of how to use these inputs to represent a particular geophysical system and geological structure, refer to Section III on Effective Modelling, Inversion and Data Analyses

INSTALLING EMIGMA v8.x

Before you begin, ensure that your dongle key is attached to your usb or parallel port.

Upgrading from a Previous Version of EMIGMA

Note for previous users of EMIGMA - EMIGMA V8.x will not operate properly with multiple versions installed on your computer so it is recommended that you uninstall any previous versions of EMIGMA before proceeding.

Performing a new installation

- Insert the CD into your CDRom

  If AutoRun is configured on your computer, the installation will begin automatically

  If not, run CD_Drive:\ setup.exe

- Insert Client License Diskette as prompted

Installation Options

Special Instructions for Windows 2000 and Windows XP and Windows Vista Users

1) Setting License Variable

You may need to set an environmental variable through Control Panel / System
SET environmental variable TGS_LICENSE_FILE to InstallDirectory\emigmav8.x\bin\ password.dat

Note: If you have an environmental variable set for a previous version, you will want to delete this earlier variable.

2) Setting your path

Ensure InstallDirectory \emigmav8.x\bin is on your path

Note: Ensure that password.dat and peuser.dll were copied successfully to the above installation directory. If they were not, manually copy the files from your Client License Disk to your installation directory.

3) Installing Sentinel Drivers

Sentinel Drivers must be installed on your hard drive in order to access the dongle. Drivers may already exist on your computer from another program. If, after a successful installation of EMIGMA you get the warning message error accessing key then you need to install the drivers.
InstallDirectory \emigmav8.x\drivers\Sentinel Protection Installer 7.4.0.exe:
Note that the Sentinel drivers can be lost by your system at a later date. If you get the error message error accessing key at any point, re-install the Sentinel drivers.

4) Reboot

For Windows2000 and XP and Vista you must re-boot to ensure the installation is completed.

Requesting a TGS password

In order to avoid the expense of licensing passwords, you will encounter an error message upon starting EMIGMA V8 TGS License File Error . This is just a warning message. Please click OK to this message and continue normal use. Passwords, licensed to a particular hard drive, are available upon request at cost.

To request a TGS password

- At the command prompt, go to your boot drive and type dir p
- E-mail the 8 character hex serial volume id to support@petroseikon.com

A password.dat file will be e-mailed back to you. Copy this password.dat file into your InstallDirectory\emigmav8.x\bin directory.

Note that you should you need to replace your hard drive or wish to install EMIGMA on another machine, a separate password.dat file is required to avoid the warning messages. Passwords are not necessary, but are available at cost.

GETTING STARTED

Start -> Programs -> Emigma V8.1 -> Emigma V8.1

Troubleshooting: If you get a TGS License Check Warning - *Product "Open Inventor" is not licensed for this host* - or - *Product "GraphMaster" is not licensed for this host* when you execute EMIGMA V8 (and/or open a file into Viz) - Click **OK**. This is a warning only and you will be able to continue using EMIGMA V8 with full functionality.

First Use

If you are using EMIGMA for the first time, the **Start** dialog will appear offering you four options – **Open an Example Database, Create your own Database**, **Convert your old database(s)** and **Open an Existing Database**.

To open an example database

- Select the **Open an Example Database** option. The **Database** dialog appears containing a number of demo projects

To create a new database

- Select **Create your own Database** . The **Save New Database** dialog opens.
- Enter the new database name. It is a standard recommendation to create a new folder to store your file in.
- Click **Create** . The blank **Database** dialog appears, bearing the name assigned by you.

  Note. *If you select an already existing database, it will be overwritten to produce a blank database*

To convert old databases:

This option is designed for the cases when you use the latest EMIGMA 8.1 version to work with the databases created in earlier versions.

- Select **Convert your old databases** . The **Start Dialog** appears, with your newly converted databases displayed in the lower **Open an Existing Database** field.

Note . *The conversion is carried out automatically provided the old databases are stored in the computer you are using.*

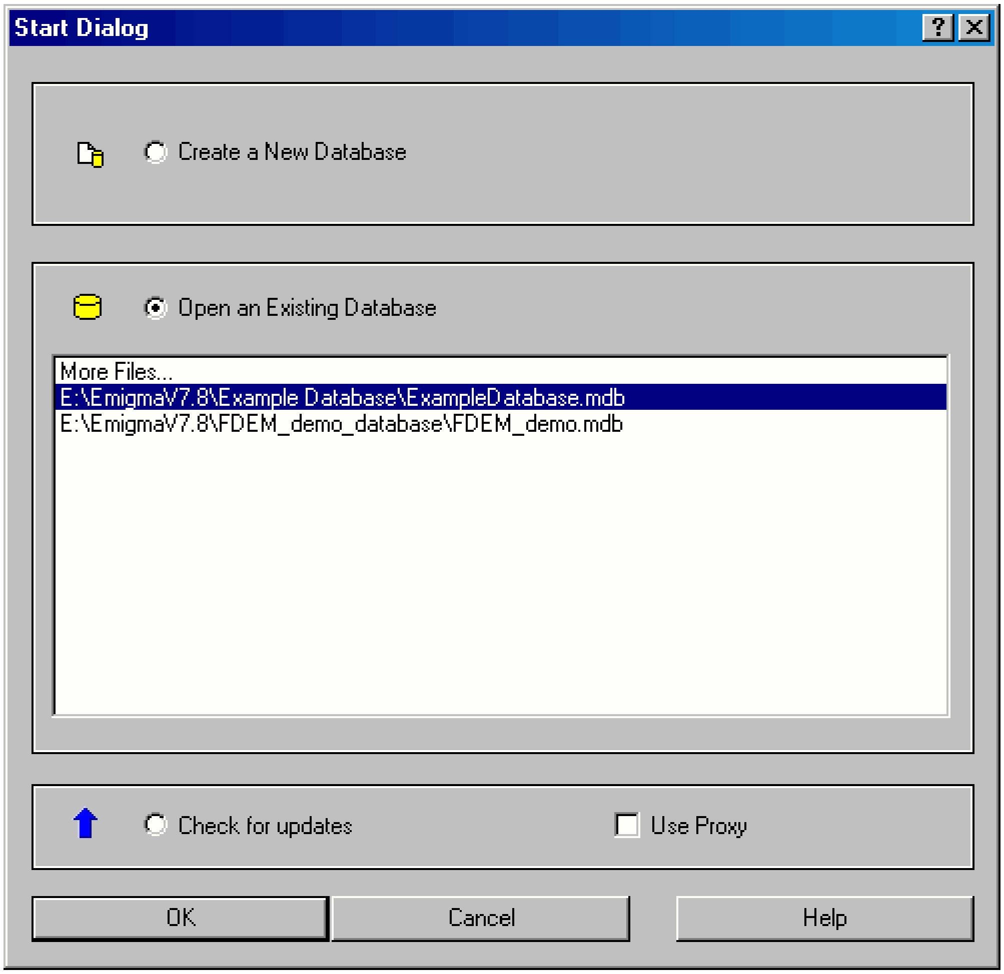
To open an existing database:

- Select **Open an existing database**. The **Open Database File** dialog appears
- Select desired database and click **Open**

.

Subsequent Use

In subsequent times, EMIGMA will offer another **Start** dialog with the following options for you to select:



<u>To open an existing database</u>

Since you are a regular user of EMIGMA now, the **Open an Existing Database** option will be selected by default. In the field below you will see the list of the databases you created in previous times, with the one on top being highlighted. You can also add other existing databases to your list:

- Double-click the **More Files** item that precedes the list to display the **Open Database File** dialog
- Select a required database and click **Open**

<u>To create a new database</u>

The **Start Dialog** allows you to create a new database in the same manner as when you started EMIGMA for the first time.

- Select the **Copy Template Database** option in the upper part of the **Start** dialog. The **Save New Database** dialog appears
- Create a new .mdb file and click **Open**. The blank **Projects in Database** dialog will appear, bearing the name you assigned

Note. *If you select an already existing .mdb file, it will be overwritten to produce a blank database*

To download EMIGMA updates:

• Select **Check for updates**
• If you have any problems connecting to the update server, activate the **Use Proxy** checkbox.
Otherwise, leave it unchecked since it will noticeably increase the length of the download process.
• Click **OK** , EMIGMA will close and the Update window will be launched

CREATING PROJECTS

EMIGMA suggests various ways of data organization depending on the user's preference. The backend of EMIGMA is an object database. It allows one to organize data into multiple levels based on different criteria. For example, you may divide your database into projects proceeding from the type of data or the simulation to be applied. You may also choose to create a single project for multiple datasets to provide more ready analyses of different data types and integration of models across all data types within a project. Use whichever way you think is best to meet your modeling purposes.

The **Projects in Database** dialog to appear on starting EMIGMA has three levels of data management. The first is **Project**, which is divided into one or more **Surveys**, which, in their turn, may contain one or more **Data Sets**. In one project, the surveys may represent different systems or data type, being associated with each other only, for instance, by a territorial criterion; as for the data sets in each survey, they may be numerous, measured or simulated, but must have the same strict structure defined by the system geometry, number of locations, etc.

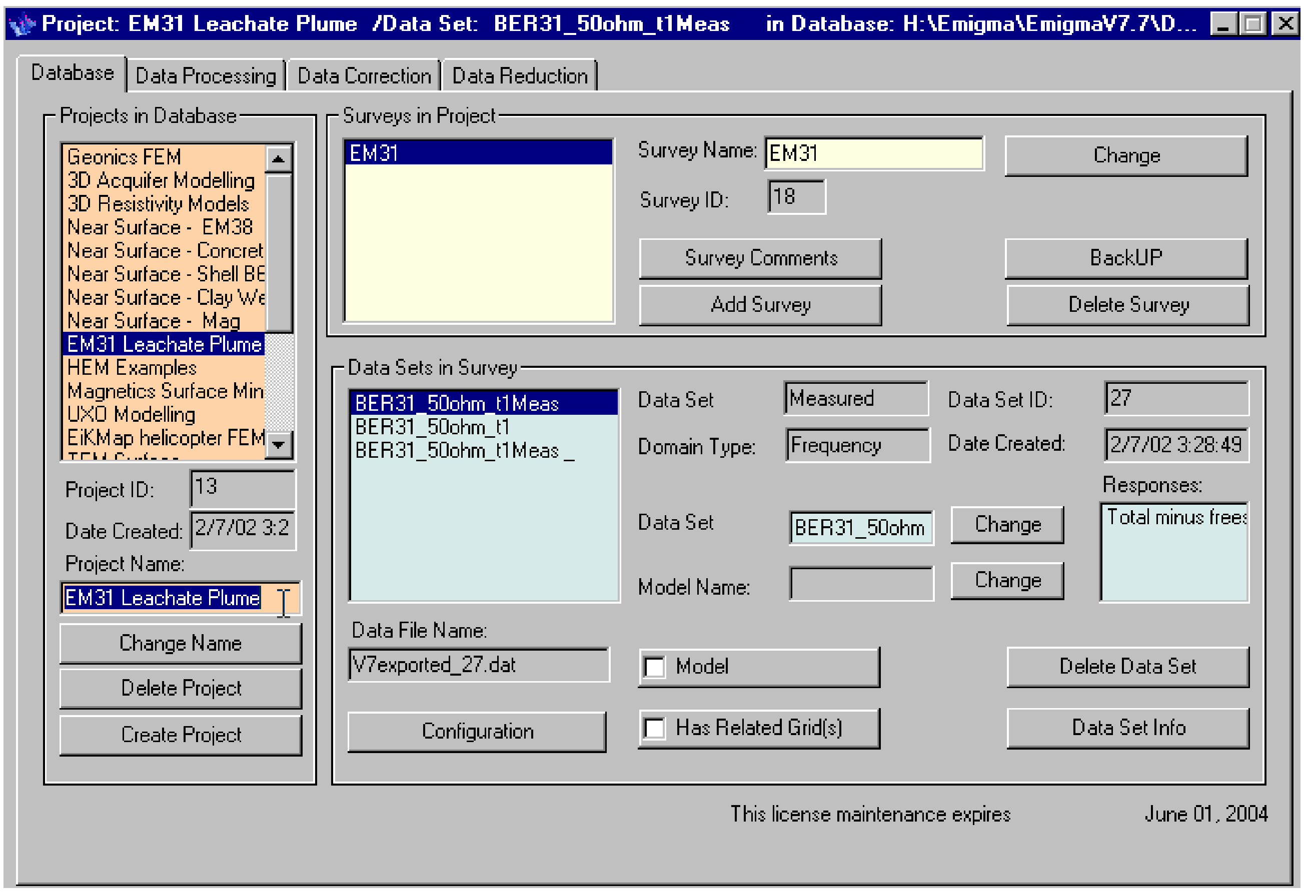


Fig. 1

Creating Projects From Imported Data

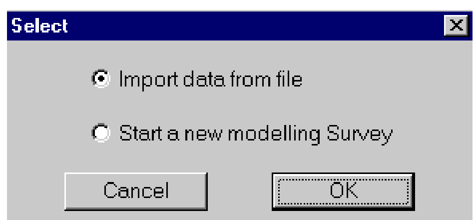
All import wizards in EMIGMA are customized, which means that each separate import procedure has been created to meet the specific requirements of our customers.

You can bring your measured data into the database either in the manufacturer's file format or as an ASCII columnar file. The first, for example, includes Crone, Geonics TEM, IP6, Airborne TEM (GEOTEM and QUESTEM) data, the second – DC Magnetics, Geonics EM31/EM34/EM38, DIGHEM, Max-Min and IP/Resistivity data.

You can also import TEM data in the AMIRA format (Geonics, Crone, SIROTEM and ZONGE) as well as the measured and simulated data in the PetRos EiKon format (from GeoTutor III). The PetRos EiKon files have a *.pev extension and can be brought in up to 10 at a time.

1. Click the **Import** button  on the main toolbar. The message will appear asking whether you want to create a new project

   - Select **Yes** if you are starting a new database or new project within a database.
   - Select **No** if you want to import a data file into a pre-existing project
   - If you do want to create a new project, you can also click the **Create Project** button in the bottom of the **Projects in Database** section.
   - The **Select** dialog appears



   - Select the **Import data from file** option in the **Select** dialog and click **OK**

   The **Name for New Project** dialog appears:

   - Insert a name to give to your project and click **OK**.

   The **Import file to database** dialog appears:

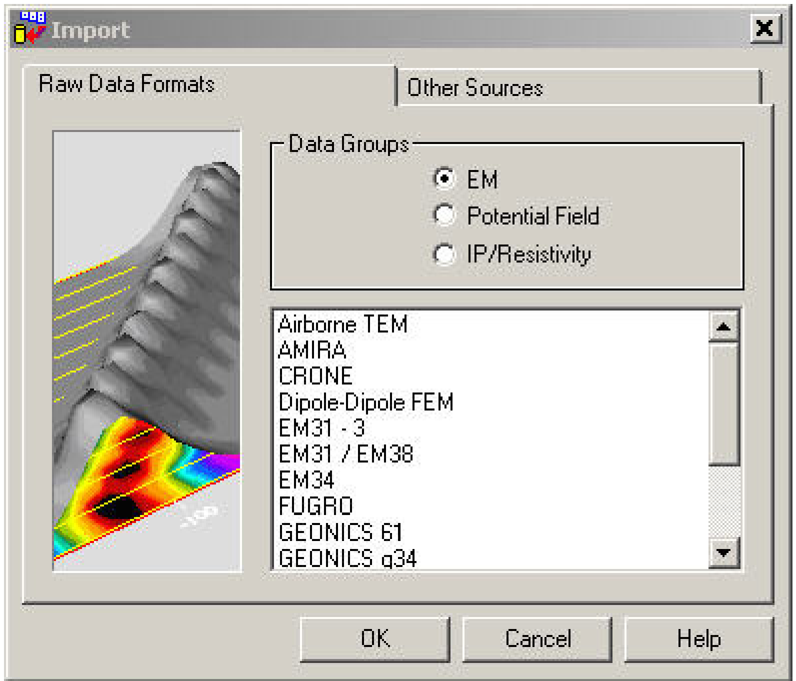


Fig. 2

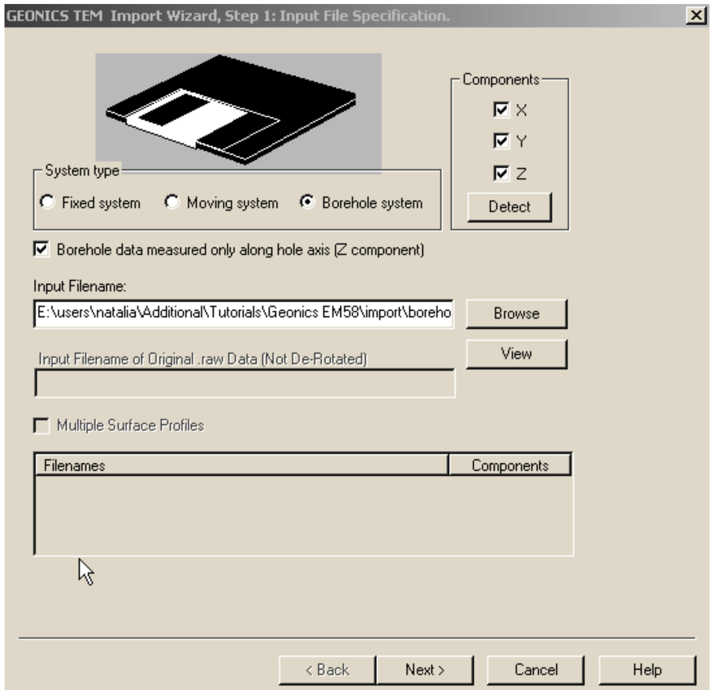
Note. *Meanwhile, your new project will be added to the **Projects in Database** list and EMIGMA will automatically assign its own internal number to the project (Project ID) and generate the date of its creation*

Example: Importing GEONICS TEM data

Select GEONICS TEM from the **Import Utilities** list of the **Import File to Database** dialog and click **Import File**

Step 1

The **GEONICS TEM Import Wizard, Step 1: Input File Specification** dialog offers you to specify your input file:

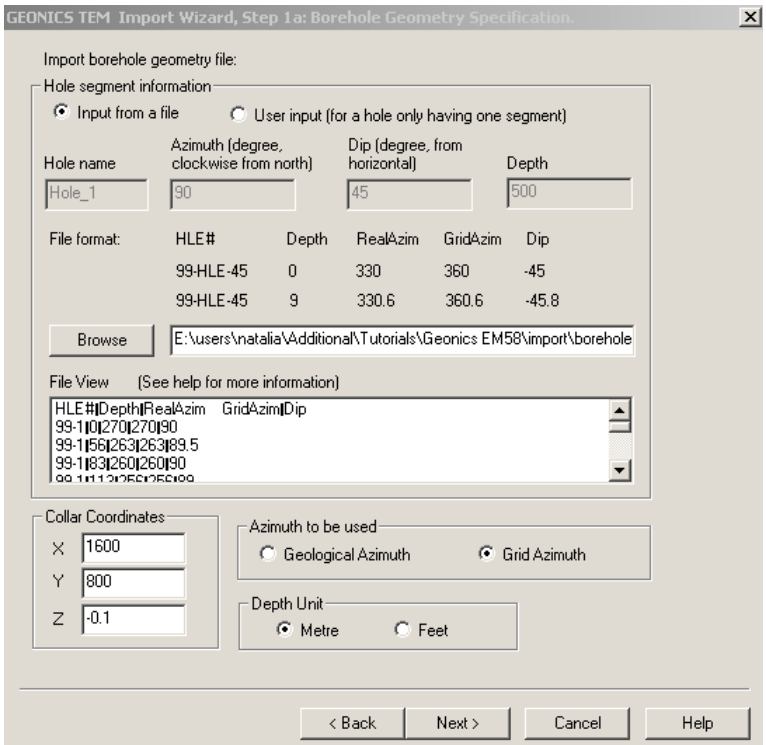


In this dialog:

- Select between the **Fixed System** and **Moving System** if you are importing surface TEM data or click **Borehole System** if you are importing borehole TEM data
- If the **Fixed** or **Moving System** options are selected:
    - Place you cursor in the **Input Filename** field and click **Browse** to bring up a standard Windows-style dialog for searching and opening files. Find the required de-rotated.raw file and click **Open**. The file name and path will appear in the **Input Filename** field
    - To import multiple surface profiles, check the respective box to activate the field below. Click **Browse** to search for as many files as necessary and add them to the list in this field. These files can be imported only if they have header lines. To delete a file from this list, just select it and press the Delete key
- If the **Borehole System** option has been selected:
    - The **Borehole data measured only along hole axis** box will be checked by default. In this case only the **Input Filename** field will be active. Browse for a processed de-rotated .raw file. Its name and path will appear in the **Input Filename** field.
    - To import additional (unprocessed) information which otherwise may be missing, de-select the **Borehole data measured only along hole axis** box. This will activate the **Input Filename of Original .raw Data (Not De-Rotated)**. Place your cursor in this field and click **Browse** to search for a not de-rotated.raw file and click Open. The file name and path will appear in the **Input Filename of Original .raw Data (Not De-Rotated)** field

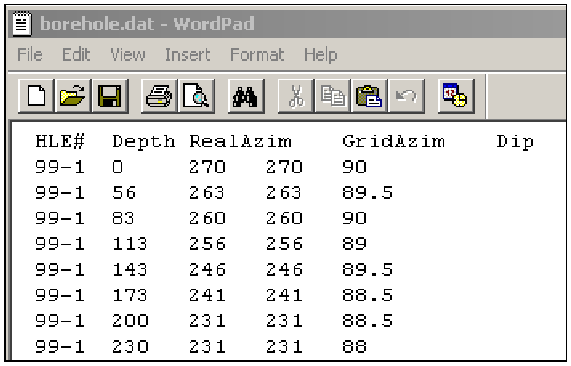
- To view the file to import, select it in the respective filename box and click **View**. The **File View** dialog will appear containing the file in the text format
- To view the components contained in the file to import, click **Detect** in the **Components** section of the dialog
- Click **Next**.

If you are importing borehole data, the intermediate **GEONICS TEM Import Wizard, Step 1a: Borehole Geometry Specification** dialog will appear:



Geonics borehole data files do not contain information on borehole geometry (i.e. dip and azimuth). You can recover this information from an ASCII borehole geometry file or input it manually. In the latter case, your entire hole will be considered as one segment.

- To load borehole geometry information from an existing ASCII file:
  - Select the **Input from a file** button in the upper part of the **Hole Segment Information** section. The **Browse** button will become active. Click this button to search for an ASCII file:



borehole.dat - WordPad

File Edit View Insert Format Help

HLE#	Depth	RealAzim	GridAzim	Dip
99-1	0	270	270	90
99-1	56	263	263	89.5
99-1	83	260	260	90
99-1	113	256	256	89
99-1	143	246	246	89.5
99-1	173	241	241	88.5
99-1	200	231	231	88.5
99-1	230	231	231	88

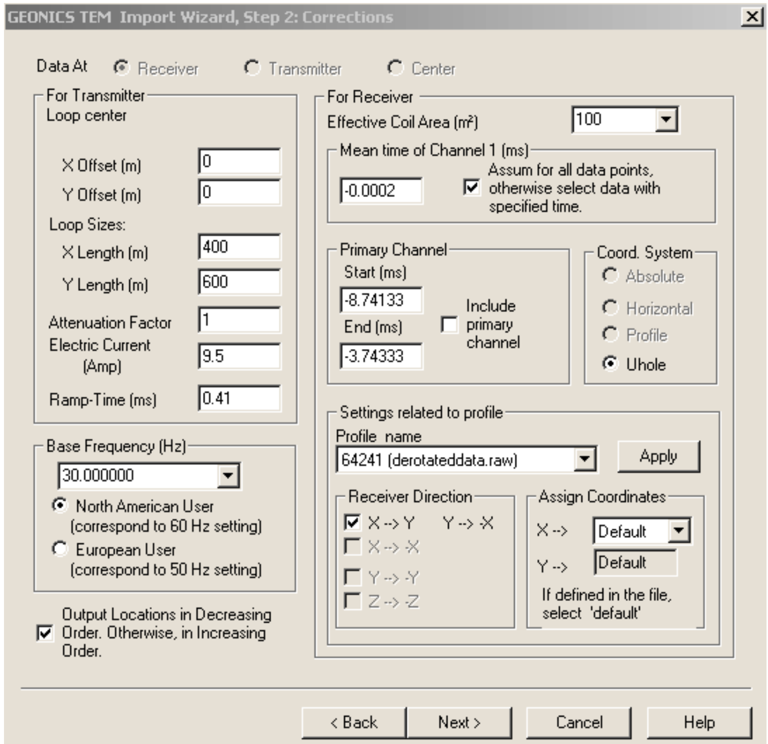
The file name and path will appear in the box across the **Browse** button, whereas the **File View** field will contain the loaded data in the text format

- To manually input the borehole geometry:
  - ➢ Select the **User Input** option. The four boxes below become enabled
  - ➢ Input your own values of azimuth, dip and depth in the respective boxes
- Specify the **X** and **Y** coordinates of the collar position, since neither Geonics borehole data files nor ASCII borehole geometry files contain collar coordinates, except for Z, which is always -0.1
- Select between **Geological Azimuth** and **Grid Azimuth** in the **Azimuth to be used** section. The second will be selected by default, since grid azimuths are commonly used for the local coordinate system
- Select between **Meters** and **Feet** in the **Depth Unit** section
- Click **Next** to proceed to Step 2.

Note: In borehole systems, a message may appear urging you to specify the receiver coil area; this is a warning that the current Geonics data file contains no coil area information and you will have to input it manually in the dialog to follow

Step 2

The **GEONICS TEM Import Wizard, Step 2: Corrections** dialog offers you to check or specify the settings related with the system geometry:



- In the **For Transmitters** section:
    - ➢ If you have a borehole or fixed system, the Geonics data file assumes the loop center to be 0, 0 unless specified otherwise. In the latter case, make sure the correct position is inputted
    - ➢ If you have a moving system, the **Loop Center** settings are replaced by the ones for X and Y coordinates of the Tx-Rx separation. The latter are either detected from the Geonics data file or inputted manually
    - ➢ The loop dimensions are usually detected from the Geonics data file. If they are not, input them manually
    - ➢ The attenuation factor, electric current and ramp time are detected from the Geonics data. Check for errors
- In the **Base Frequency (Hz)** section:
    - ➢ Select between the North American and European standard frequency settings to adjust the base frequency. This will also change accordingly the start and end times in the **Primary Channel** section of the dialog
    - ➢ In the **Primary Channel** section, check **Include Primary Channel** if you want to import it as well. In the boxes on the left, you can see the start and the end times of the on-time window detected from the Geonics data file
- In the **For Receiver** section:
    - ➢ Specify the effective coil area if it is not detected from the Geonics data file. If it is, check it for errors

- The mean time of **Channel # 1** is detected from the Geonics data file. Check it for errors
- Leave the box to the right checked if you want to apply the mean time detected from the Geonics data file to the entire Channel 1; de-select the box, if your objective is a certain time gate with the mean value specified by you in the **Mean Time for Channel 1** field

- In the **Coord.System** section:
  - Choose **Absolute**, **Horizontal** or **Profile** for surface systems and **Uhole** for borehole systems. In the latter case, **Uhole** must be selected by default
- In the **Receiver Direction** section:
  - To change the direction of any of the receiver coils, check the respective box. This will in effect change the sign of the response, which becomes necessary when profiles have been surveyed in opposite directions, while the direction of the coil has not been changed at the same time
  - In the Assign Coordinates section, leave default in the X dropdown list if the coordinates are set in the file to import as Column 1 for X and Column 2 for Y. The import recognizes the directions associated with the coordinate. For example, if it sees 500S in the first column, it will set it as the Y coordinate
- The **Output Locations** checkbox is set to the decreasing order. To organize your output information in the increasing order, de-select this box
- Click **Apply** in the **Settings related to profile** section to save all the changes before proceeding to the next step
- Click **Next**

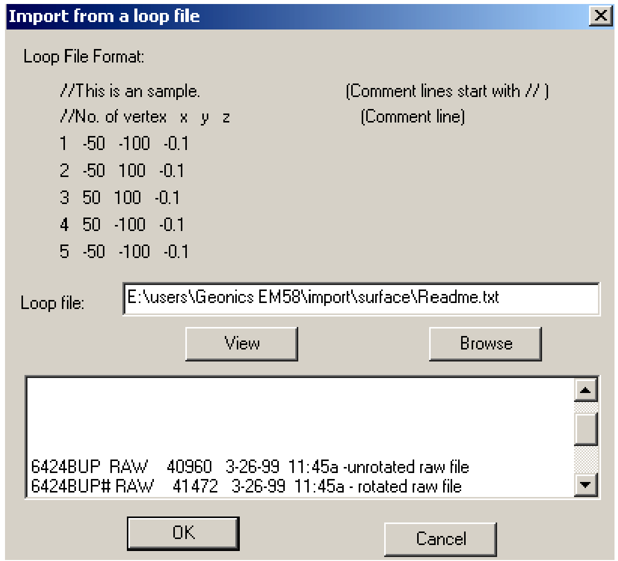
<u>Step 3</u>

The **GEONICS TEM Import Wizard, Step 3: Transmitter Loop Specification** dialog offers you to check, edit or import the loop configuration.

- Check the coordinates of the loop vertices in the spreadsheet-like table of the dialog. Their order coincides with the current flow direction, with the last corner repeating the first one to close the loop
- To add a vertex, specify its number and X, Y and Z coordinates in the boxes of the **Edit Loop Vertices** section and click **Insert**. You will see the vertex in the row you specified
- To edit a vertex, select it in the table, change its X, Y and Z coordinates in the respective boxes of the **Edit Loop Vertices** section and click **Modify**

Note. If after import and simulation the sign of your data is incorrect, re-import using the **Reverse Current Direction** button in this dialog

- To import a loop:
  - Click **Import** from a loop file. The respective dialog will appear:



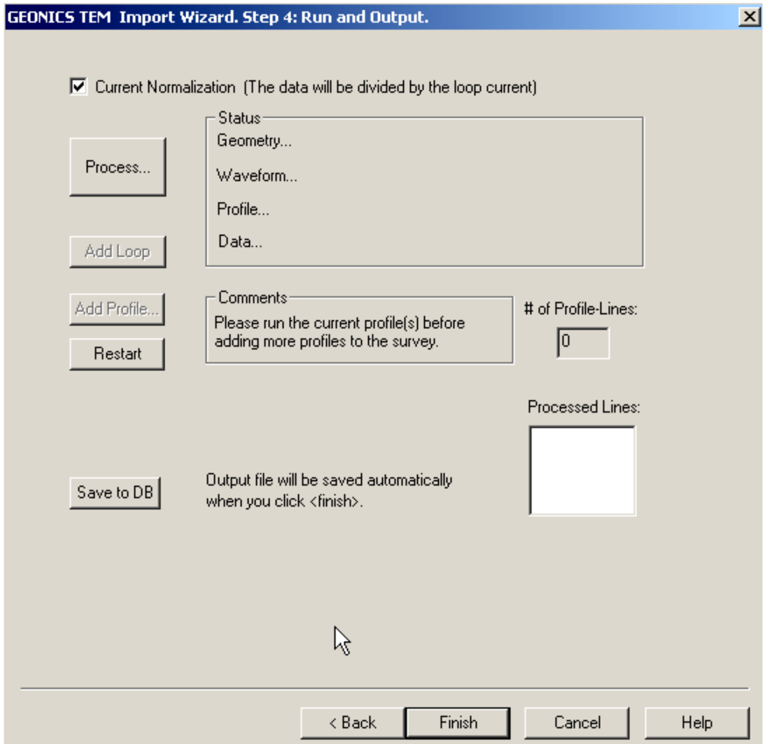
The loop file to import is to have a format as shown in the sample

- Click **Browse** to bring up a standard Windows-style dialog and open the required file. Its name and path will appear in the **Loop file** box and related comments will be displayed in the field below
- Click **View** to check the format of the file
- Click **OK** to complete loop import and return to the **GEONICS TEM Import Wizard, Step 3: Transmitter Loop Specification** dialog. The spreadsheet-like table will contain the vertices of the imported loop

Click **Next** to proceed to the Step 4.

Step 4

The **GEONICS TEM Import Wizard Step 4: Run and Output** dialog is the concluding stage of the GEONICS TEM data import. It runs your files, saves them to the database, allows you to add loops or profiles to the already available data. In the upper part of the dialog, the **Current Normalization** box checked by default means that your data are divided by the loop current so that they could be directly compared to simulated data



- Click **Process** to run the data file to import

  If at any point more than one measurement has been detected, the **Duplicate Data** dialog will warn you that the average of all measurements will be taken. This dialog may appear up to 3 times (for the X, Y and Z components). To ignore it, click **OK**. Otherwise, click **Restart Import**. This will return you to Step 1 of the import procedure, while your previously selected settings will be lost

  Note. *Before re-import, go to the original data file and delete the data stations you do not want*

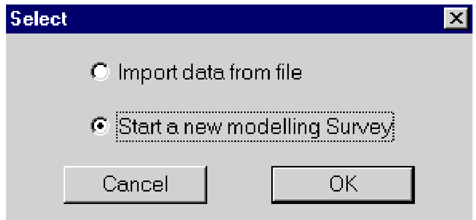
- To add another loop, save your processed data by using the **Save to DB** button and click **Add Loop** to go back to the Step 1 dialog. Follow steps 1, 2, and 3
- To add another profile:
  - Click the **Add Profile** button to bring up the Step 1 dialog
  - Browse for another file to recover the necessary profile information from and click **Next**
  - Check the settings in the Step 2 dialog as described above and click **Next**. This will bring you to the Step 4 dialog
  - Click **Process** to read in the profile and see it appear in the Processed Lines field
- All done, click **Finish** in the bottom of the dialog.

  If you added a loop, the **Profile Order** dialog will appear asking you whether you want to keep the resulting data in separate profiles or merge them into one profile. If you select the first option, you will have 2 surveys and data sets in your newly created project, one containing the one-loop geometry and the other – the two-loop geometry. If you select to merge the data, you will have only one survey and data set with the two-loop geometry.

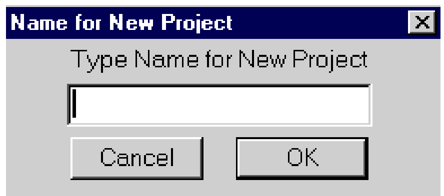
If you added a profile, the **Join Profile** dialog will appear asking you whether you want to merge the two (or more, dependently of the number of profiles you added) sets of data into 1 profile. Select the profiles you want to merge and click **Join**. To keep the data in separate profiles, click **Close**.

Creating Projects From Scratch

1. Click the **Create Project** button in the bottom left-hand section of the **Projects in Database** dialog. In the **Select** dialog to appear, select the **Start a New Modeling Survey** button:



The **Name for New Project** dialog appears



2. Type a name to give to your project and click **OK**.

The blank **Tx-Rx** tab of the **Property Pages** dialog opens offering you to specify the system:

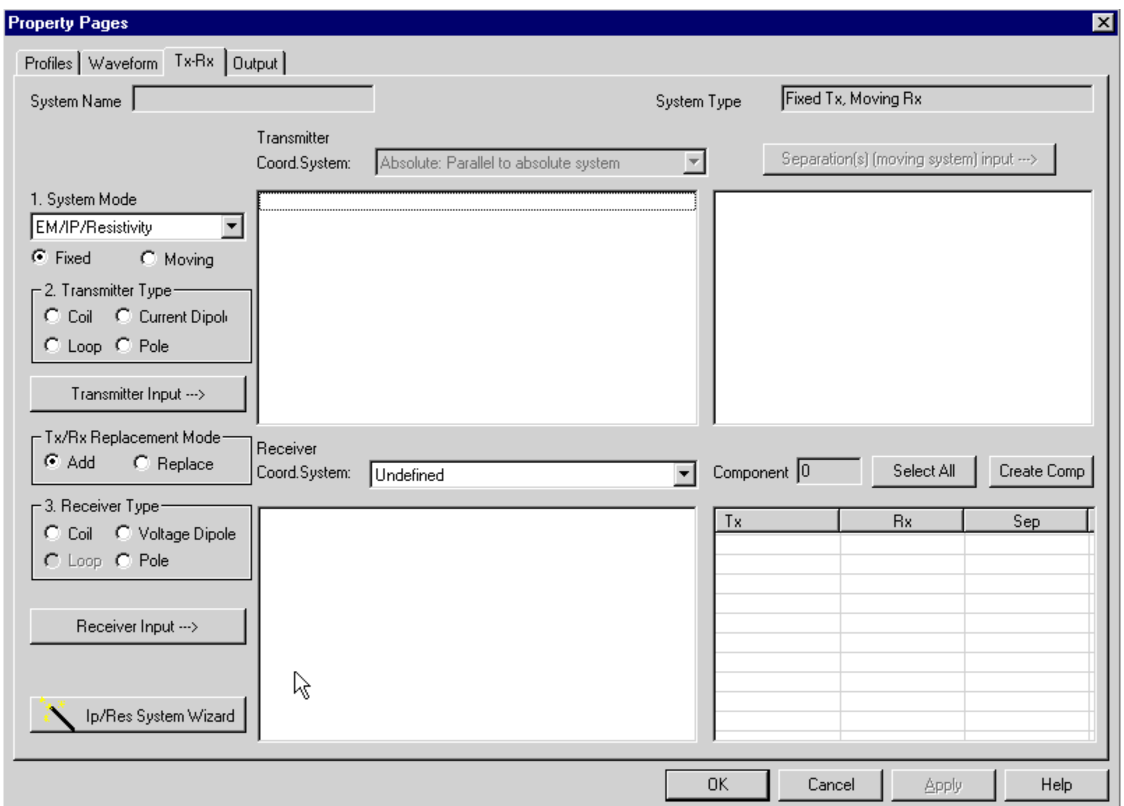


Fig. 3

Note*. Meanwhile, your new project will be added to the Projects list of the* ***Database*** *page and*

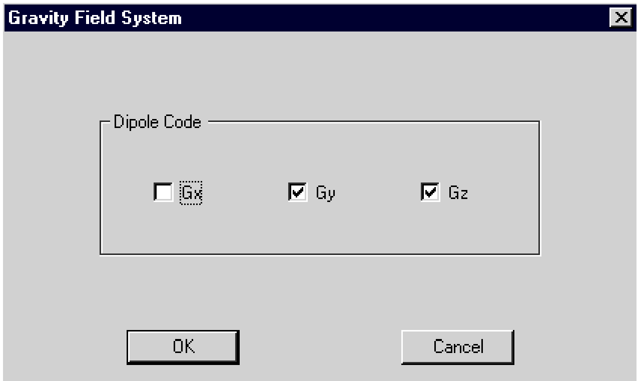
EMIGMA will automatically assign its own internal number to the project (Project ID) and generate the date of its creation

Specifying System Geometry

In the **Tx-Rx** tab, select the system mode from the respective dropdown list. You have four options: **Gravity**, **Magnetic**, **MT** and **EM/IP/Resistivity**.

To specify a gravity system

- Select **Gravity** from the **System Mode** list (Fig. 3). The **Gravity Field System** dialog appears

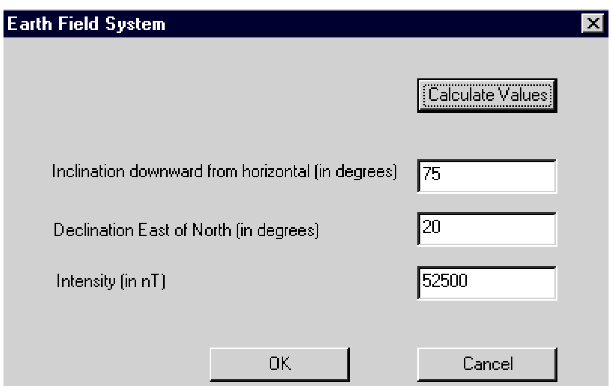


- Specify the number and orientation of the dipoles and click **OK**. The rest of the **Tx-Rx** tab will be filled out automatically
- To add a dipole, click the **Receiver Input** button to reopen the **Gravity Field System** dialog and check dipoles as needed

To specify a magnetic system

- Select **Magnetic** from the **System Mode** list in the **Tx-Rx** tab, Fig. 3.

The **Earth Field System** dialog appears

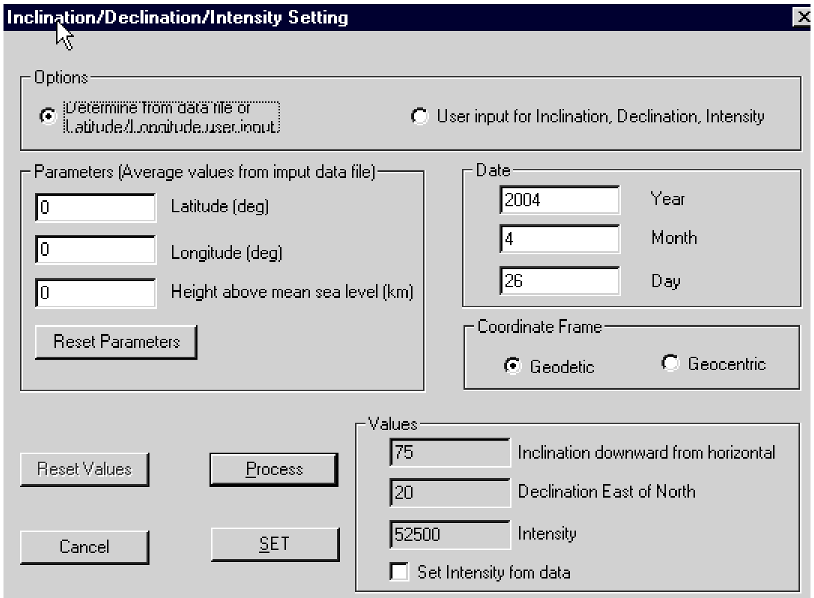


- Specify the inclination, declination and intensity in the respective fields and click **OK**

OR

- Click the **Calculate Values** button to compute these parameters.

The **Inclination/Declination/Intensity Setting** dialog appears



- Select the **Determine from Data File or Latitude/Longitude User Input** option to activate the **Parameters**, **Date** and **Coordinate Frame** fields below

Note. If your data are imported, these fields will already be filled

- Replace with or type in new values and click **Process**

  The **Values** section will be updated accordingly

- To reset your latitude, longitude and height, click the **Reset Parameters** button
- To return to the initial inclination, declination and intensity in the **Values** section, click **Reset Values**
- To change the values manually, select the **User Input for Inclination, Declination, Intensity** button and insert your values in the respective fields

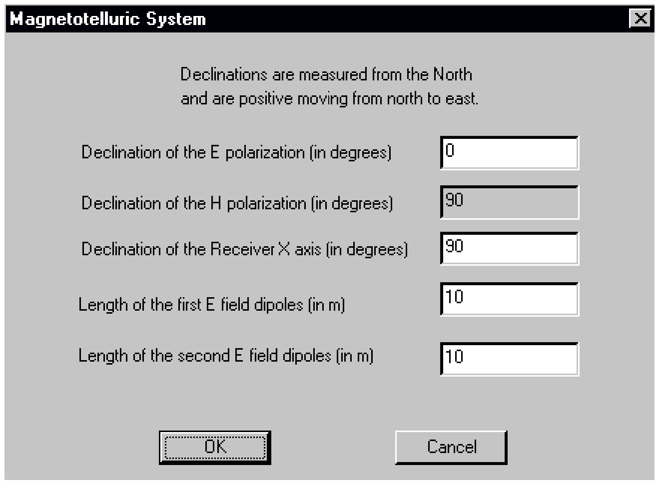
Note. If your data are imported, check the Set Intensity from Data box to have the intensity value to be taken from available data

- Click **Set** to return to the **Earth Field System** dialog
- Click **OK** in the **Earth Field System** dialog. You will proceed to another dialog with the same name
- Set dipole codes and click **OK** to close the dialogs and view the system created
- To reopen the dialogs, use the **Transmitter Input** and **Receiver Input** buttons in the **Tx-Rx** tab, Fig. 3.

<u>To specify an MT system</u>

- Select **MT** from the **System Mode** list of the **Tx-Rx** tab, Fig. 3.

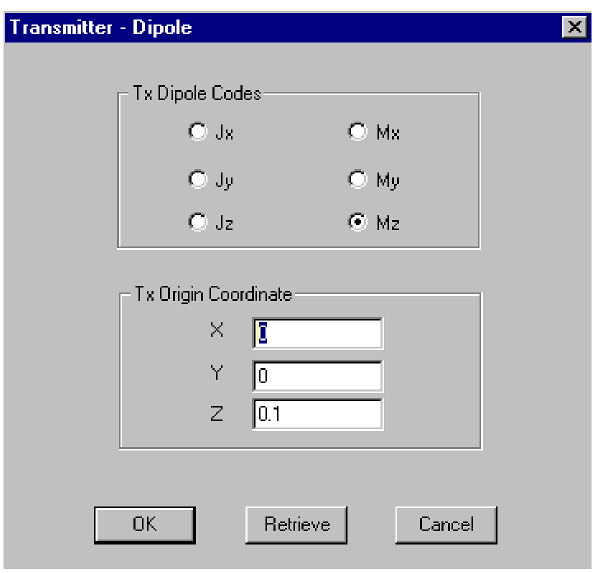
The **Magnetotelluric System** dialog appears



- Specify the declination of the E polarization in the respective field. This is the polarization of the electric field in the source plane wave. The declination of the H polarization will change accordingly by 90 degrees
- Set the values for the X-axis declination of the receiver. The measurement setup may be oriented at any angle
- Set the lengths of the first and second E field dipoles
- Click **OK**
- To change your settings, click **Transmitter Input** or **Receiver Input** buttons, Fig. 3, to reopen the **Magnetotelluric System** dialog and to insert new values.

To specify an EM/IP/Resistivity system

- Select **EM/IP/Resistivity** from the **System Mode** list of the **Tx-Rx** tab, Fig. 3
- Select between the **Fixed** and **Moving** systems



- Specify your transmitter type in the respective section and click **Transmitter Input**. The dialog to appear depends on the transmitter type you selected

Specifying Transmitter type

Coil Transmitter

If you select **Coil**, the **Transmitter – Dipole** dialog will appear:

- Specify the Tx dipole codes

Should you have an electric dipole, its z coordinate will automatically set to its maximum, which is –0.1. Should you have a magnetic dipole, its z coordinate will automatically set to its minimum, which is 0.1.

- For fixed systems, specify the origin coordinates in the respective section
- Click **OK** to close the dialog and view your transmitter information on the **Tx-Rx** page
- To add a new transmitter, make sure that the **Add** option is on in the **Tx/Rx Replacement Mode** section of the **Tx-Rx** page and click the **Transmitter Input** button to reopen the **Transmitter – Dipole** dialog and specify the codes of a new dipole
- To replace a transmitter, switch to the **Replace** option in the **Tx/Rx Replacement Mode** section and click the **Transmitter Input** button to bring up the **Transmitter – Dipole** dialog and change your settings

Note. *If at this stage you change your mind, click* ***Retrieve*** *to return to initial values (this button works only in the* ***Replace*** *mode)*

Current Dipole

If you select **Current Dipole**, the **Transmitter Input** button will bring up the **Bipole Length** dialog:

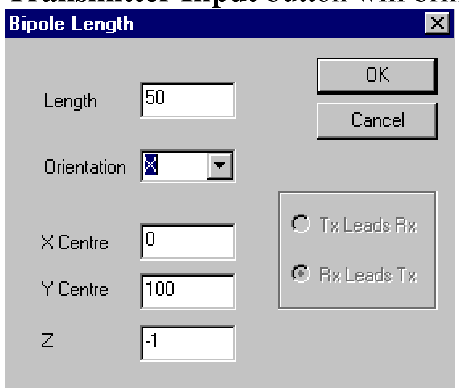


Fig. 3a

- Specify the bipole length and select its orientation from the list.
  The list contains 4 items: X, Y, Z following a blank item provided for those cases when bipole orientation is to be specified each separate time
- In a fixed system, specify the position of the bipole center in the respective fields
  Should it be an X orientation, the **X Center** value will define the position of its center relative to the X-axis. The z coordinate will be set automatically to –1
- In a moving system, specify the position of the transmitter and receiver relatively to each other in the right-hand section of the dialog
- Click **OK**

The **Extended Source** dialog appears containing the settings you defined in the **Bipole Length** dialog:

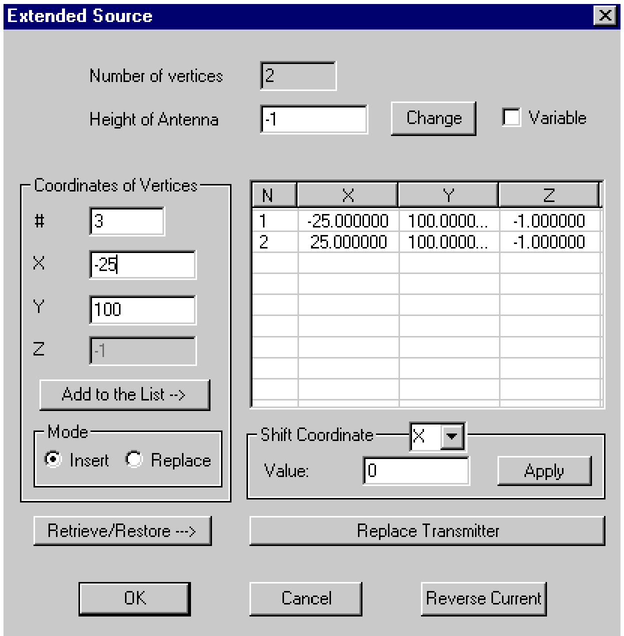


Fig. 3b

- To change the settings:
  - ➢ Select the vertex row and type in new coordinates
  - ➢ Switch to **Replace** in the **Mode** section and click **Add to the List**

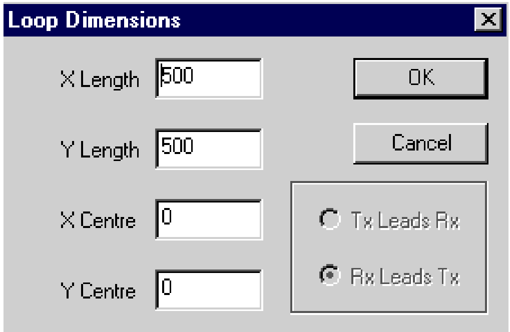
Note*. If you change your mind, click the* ***Retrieve/Restore*** *button to return to your previous values*

  - ➢ If necessary, alter the value of the antenna height in the respective box and click **Change** or, if it is to vary, check the **Variable** box
- To replace the coordinate:
  - ➢ Select it and turn the **Replace** button on in the **Mode** section of the dialog
  - ➢ Change the coordinate as needed and click **Add to the List**
- To change the coordinate for all vertices by the same value
  - ➢ Select the axis from the **Shift Coordinate** list
  - ➢ Type in the value you want to change the coordinate by in the **Value** box and click **Apply**.
- To input a bipole from another survey within the same project
  - ➢ Click **Replace Transmitter** to open the **Browse For** dialog
  - ➢ Select the required survey and bipole in this dialog and click **Replace**
- To change the current to reverse, click the respective button in the bottom right-hand corner of the **Extended Source** dialog
- Click **OK** to close the dialog and view your transmitter information on the **Tx-Rx** page

- To replace a bipole:
  - ➢ Select the **Replace** option in the **Tx/Rx Replacement Mode** section of the **Tx-Rx** tab and click **Transmitter Input**
    This will bring up the **Extended Source** dialog (Fig. 3b)
  - ➢ Make your changes in the dialog as described above, click **Add to the List** and **OK**

Loop

If you select **Loop**, the **Transmitter Input** button will bring up the **Loop Dimensions** dialog:



- Specify the length of the loop sides in the **X Length** and **Y Length** fields and the position of the loop center relative to the X- or Y-axis
- In a moving system, specify the position of the receiver and transmitter relative to each other
- Click **OK**.

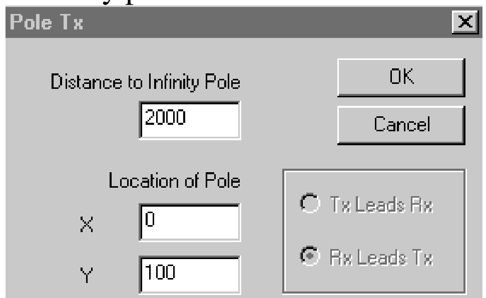
The **Extended Source** dialog appears (Fig. 3b):

- To add more loop vertices, specify the X and Y coordinates in the respective section and click **Add to the List**
- To replace a loop with one from another survey within the same project
  - ➢ Click **Replace Transmitter** to open the **Browse For Tx Loop** dialog
  - ➢ Select the required survey and loop in this dialog
  - ➢ Click **Replace** to return to the **Extended Source** dialog
- For the rest of operations in the **Extended Source** dialog, follow the respective procedures in the *Current Dipole* section and click **OK** to return to the **Tx-Rx** page, Fig. 3
- To add a new loop, make sure that the **Add** option is on in the **Tx/Rx Replacement Mode** section and click the **Transmitter Input** button to reopen the **Loop Dimensions** and then the **Extended Source** dialog to specify the dimensions and vertices of the new loop
- To replace a loop, select the **Replace** option in the **Tx/Rx Replacement Mode** section and click **Transmitter Input**. This will bring up the **Extended Source** dialog right away. Make your changes, click **Add to the List** and **OK**.

Pole

If you select **Pole**, the **Transmitter Input** button will display the **Pole Tx** dialog:

- Specify the distance to the infinity pole and its coordinates



- Click OK to proceed to the **Extended Source** dialog (Fig. 3b). Continue as described in the *Current Dipole* section.

Specifying Receiver Type

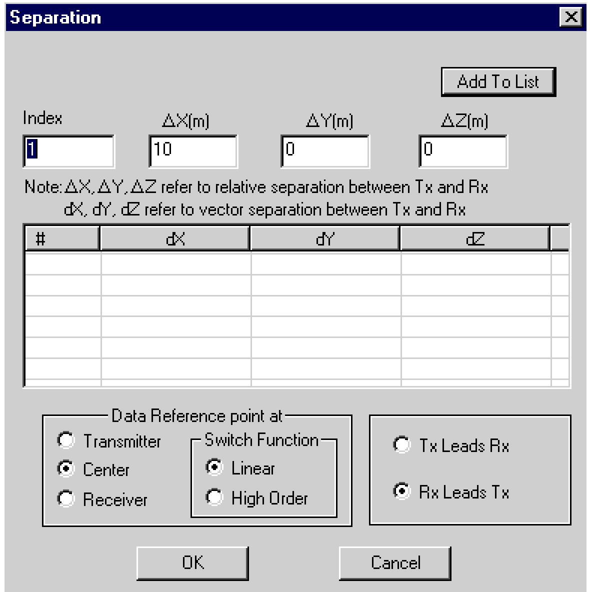
Select the receiver type in the respective section of the **Tx-Rx** page, Fig. 3, and click **Receiver Input**. As with transmitters, the dialog to appear depends on the receiver type you selected.

- If you select **Coil**, the **Receiver – Dipole** dialog will appear. Specify dipole codes in the respective section. To change back to initial settings, click the **Retrieve/Restore Data** button
- If you select **Voltage Dipole**, the **Bipole Length** dialog (Fig. 3a) will open to be followed by the **Extended Source** dialog (Fig. 3b). Go through the steps described in the *Current Dipole* section
- If you select **Pole**, the **Rx Pole** dialog will open to be followed by the **Extended Source** dialog (Fig. 3b). Go through the steps described in the *Pole* section of Specifying Transmitter Type

In case your system is fixed, you will return to the **Tx-Rx** page right from above dialogs. If it is moving, you will have to specify the separation parameter.

Specifying Separation

In moving systems, after you have specified the receiver type as described above, the **Separation** dialog will appear:



- In this dialog, **Index** is the record number of separation and is 1 by default.
- Specify the relative separation in the **ΔX**, **ΔY** and **ΔZ** boxes and click **Add to List**. Repeat as many times as the number of separations you have
- Select between the **Transmitter**, **Center** or **Receiver** options, which are conventions used as a reference point
  The center point convention means that the transmitter is located +1/2 the separation parameter from the plot point. In the transmitter point convention, the transmitter and the plot point occupy the same position, whereas in the receiver point convention, it is the receiver that coincides with the plot point
- To switch between the three options, select a required switch function in the respective section. **Linear** is set by default
- Select between the **Tx leads Rx** and **Rx leads Tx** options to define the sign (negative or positive) of separation
- To delete a separation, select it in the table and press the Delete key
- Click **OK** to close the dialog and return to the **Tx-Rx** page, Fig. 3.

  You can see the specified separation data in the upper right-hand field of the page. To change these data, click the **Separation(s) (Moving System) Input** button above this field. This will immediately display the **Separation** dialog.

Using IP/Res Wizard

You can also use the **IP/Res Wizard** to automatically add the geometry of your system. As of now, only four standard configurations are available: Wenner, Schlumberger, Dipole-Dipole, Pole-Dipole:

- Click the **IP/Res Wizard** button in the bottom left-hand corner of the **Tx-Rx** page, Fig. 3

  The **Select System** dialog appears

- Select one of the four options and click **Next**
  - If you select **Wenner**, the **Wenner Sounding System** dialog will appear containing the respective diagram. Change the suggested length of the transmitter dipole as needed; this will automatically change the length of the receiver dipole. Specify the separation and click **Finish**
  - If you select **Schlumberger**, the **Schlumberger Sounding System** dialog will appear containing the respective diagram. Change the suggested length of the transmitter dipole as needed; this will automatically change the length of the receiver dipole. Specify the separation and click **Finish**
  - If you select **Dipole-Dipole**, the **Dipole-Dipole System** dialog will appear containing the respective diagram. Change the suggested length of the transmitter/receiver dipole and specify the number of receivers. Select which of the two – transmitter or receiver – is leading and click **Finish**
  - If you select **Pole-Dipole**, the **Pole-Dipole System** dialog will appear containing the respective diagram. Change the suggested distance to the infinity pole, length of the transmitter/receiver dipole and specify the number of receivers. Select which of the two – transmitter or receiver – is leading and click **Finish**

<u>To define system components to be used in modeling</u>

If you have a number of transmitters, receivers or separations, you may choose to use either all of them or only their concrete combinations in further modeling:

- Click the **Select All** button to add all transmitters, receivers and separations into the **Component** table in the bottom right-hand corner of the **Tx-Px** page, Fig. 3. This will result in all possible combinations

- Select a required transmitter, receiver or separation in the respective field and click the **Create Component** button. This will add only those combinations that are based on your selections

Note. *In the case of gravity, magnetic and magnetotelluric systems, the list of components will be generated automatically.*

<u>To specify the coordinate system</u>

In the **Tx-Rx** tab, Fig. 3, there are two dropdown lists: the **Transmitter Coordinate System** and the **Receiver Coordinate System**. The first is not active, so to change it, you have to specify the coordinate system in the second. However it is true only for moving transmitter geometries. In the case of fixed geometries, the transmitter will always be located in the absolute coordinate system and the option in the respective list will read **Absolute: Parallel to Absolute system** no matter which coordinate system is chosen for the receiver.

In gravity and magnetic systems, the **Transmitter** and **Receiver Coordinate System** lists will both be **Absolute: Parallel to Absolute system**, whereas in MT, the magnetotelluric option will be selected.

For more information on the coordinate systems in EMIGMA, go to the *Specifying Output* section of the present manual.

Specifying Profile Information

To specify profile information, go to the **Profiles** tab of the **Property Pages** dialog:

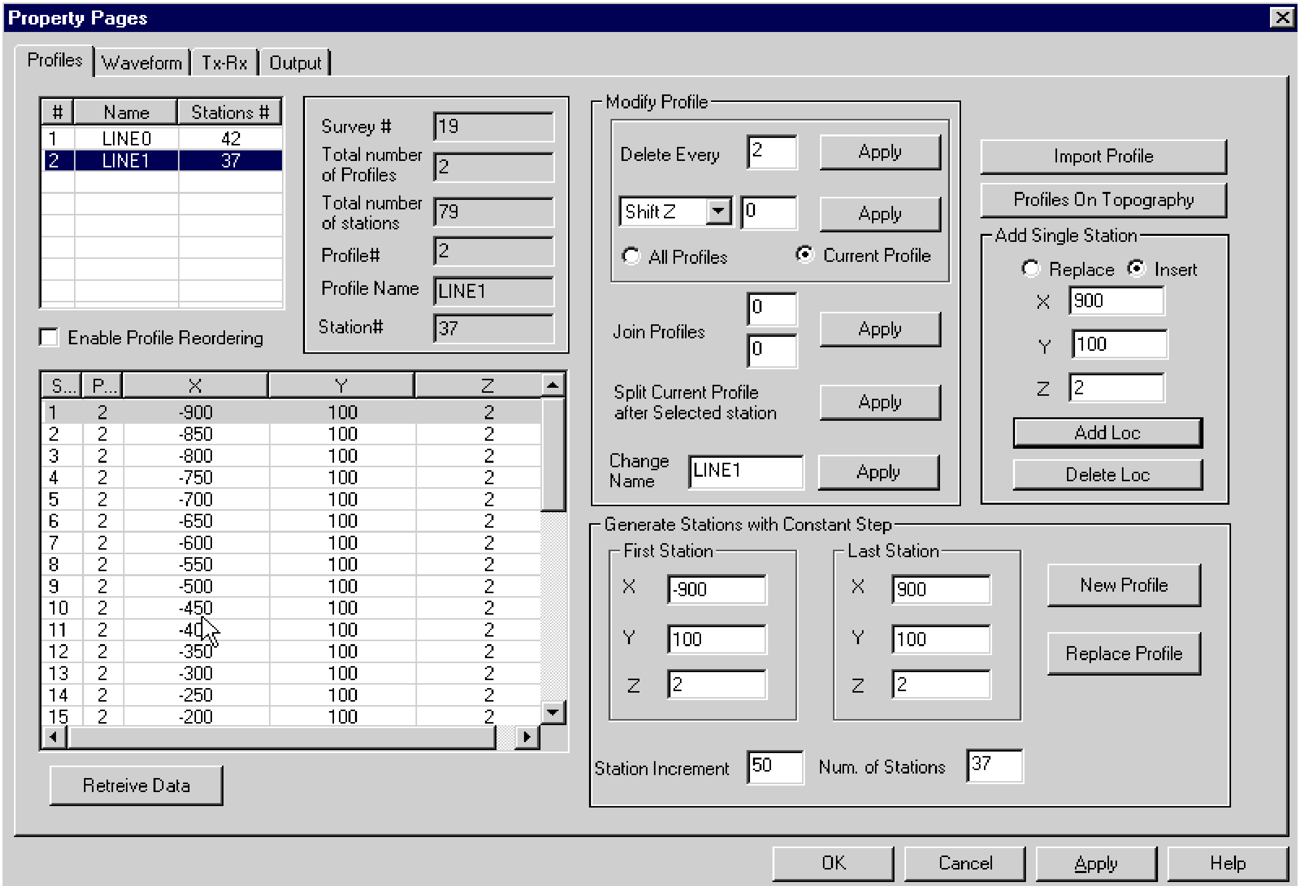


Fig. 4

<u>To add a new profile</u>

- In the **Generate Stations with Constant Step** section, specify the **X**, **Y** and **Z** coordinates of the first station and click in the respective boxes for the last station to change its coordinates accordingly
- Specify the step in the **Station Increment** box and click in the **Num. of Stations** box to change the number of stations accordingly
- Click **New Profile**.

The new profile will appear in the table in the top left-hand corner of the tab. By convention, this table will be referred to as the *List of Profiles*. The spreadsheet-like table below will show all the stations and their coordinates and will be referred to as the *List of Stations*.

Information on each profile can be viewed in the section to the right of the *List of Profiles*. Select a required profile in the *List of Profiles* and this information will be displayed automatically.

<u>To reorder profiles</u>

- Check the **Enable Profile Reordering** box under the *List of Profiles*
- Click and drag profile to whichever position you need

To rename a profile

The profile name and number will be generated automatically, with the very first profile to be named Line 0

- Select the profile you want to change the name of in the *List of Profiles* (Fig. 4)

  Its name will be displayed in the **Change Name** box of the **Modify Profile** section
- Replace the former name with a new one and click the **Apply** button to the right. The name of the profile you selected will change respectively

To replace a profile

- Select the profile you want to replace in the *List of Profiles* (Fig. 4)
- Make your changes as required in the **Generate Stations with Constant Step** section and click **Replace**

  The profile you selected will be replaced with the one you specified. If necessary, change its name as described above.

To import a profile

- Click **Import Profile** in the top right-hand corner of the **Profiles** tab, Fig. 4

  The **Import Profile** dialog will open offering you to select one of the four formats available for import:

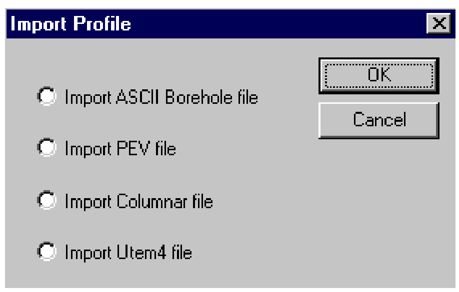
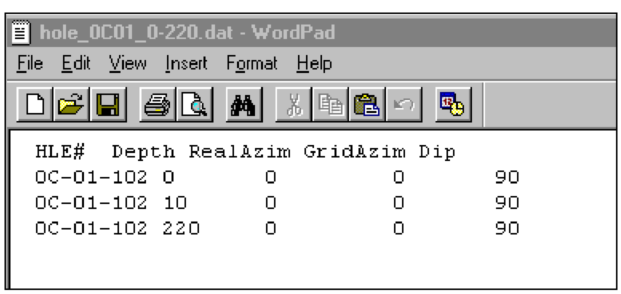


Fig. 4a

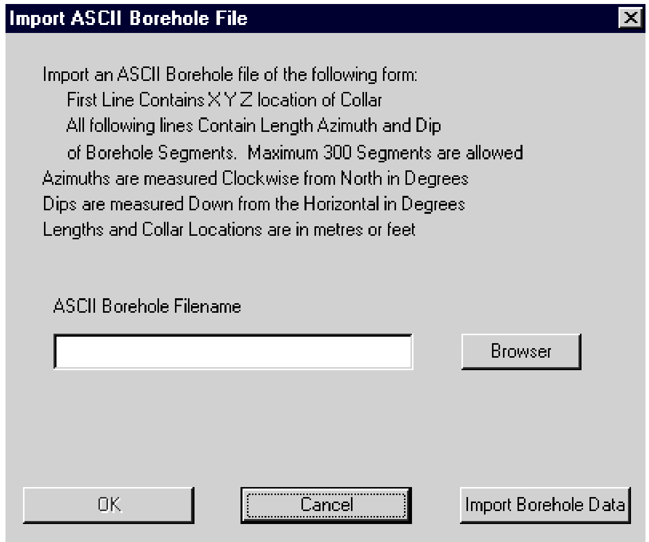
ASCII Borehole Format

To be efficiently imported, your file should be a text file of the following appearance:



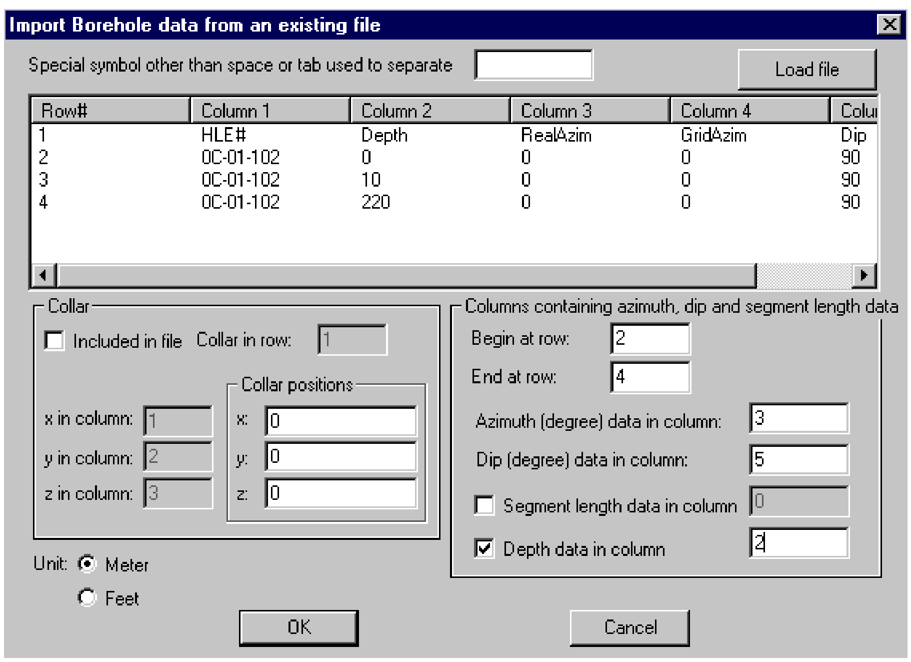
- Select **Import ASCII Borehole File** in the **Import Profile** dialog (Fig. 4a) and click **OK**

The **Import ASCII Borehole File** dialog appears describing the format of the file to be imported:



- Click **Browse** to open the **Borehole Data File** dialog, a standard Windows-style dialog for searching and opening files
- Select a required file and click **Open**. Your file path will appear in the **ASCII Borehole Filename** field in the **Import ASCII Borehole File** dialog
- Click **Import Borehole Data**

The **Import Borehole Data from an Existing File** dialog appears:



- If in the file you are importing, a symbol other than a space or tab is used to separate data, specify this symbol in the upper box and click **Load File.** In the table below, you will see the data you are about to import

In the **Collar** section of the dialog:

- If your data contain collar information, leave checked the **Included in File** box (it is checked by default) and specify the row and columns containing this information
- If your data do not contain collar information, de-select the **Included in File** box and specify the collar position in the respective section

In the **Columns containing azimuth, dip and segment length data** section:

- Specify the columns containing azimuth and dip information
- Select between depth and segment - two ways of determining location coordinates in a borehole
- Specify the units (meters or feet) in the bottom left-hand corner of the dialog, and
- Click **OK**. The **Import ASCII Borehole File** dialog reappears. Click **OK** in this dialog to confirm import

This will bring up the **Locations per Borehole Segment** dialog:

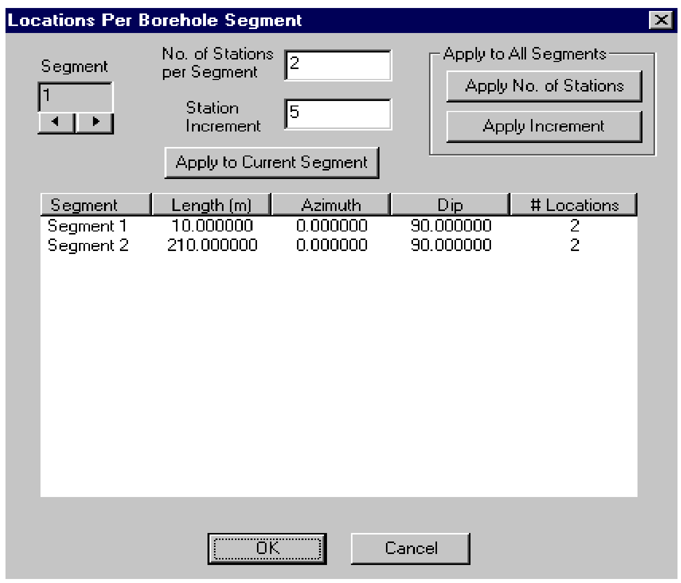


Fig. 4b

To change data of a segment:

- Select a segment in the table or from the **Segment** list using the scroll arrows
- Specify the number of stations per segment in the respective box and click **Apply to Current Segment**. The segment data in the table will change accordingly as well as the step in the **Station Increment** box

  You can also choose to edit the station increment, which will lead to the respective change in the number of stations per segment and the table data

To change data of all segments:

- Specify the number of stations per segment in the respective box and click **Apply No. of Stations** in the **Apply to All Segments** section

  OR

- Specify the station increment in the respective box and click **Apply Increment** in the **Apply to All Segments** section

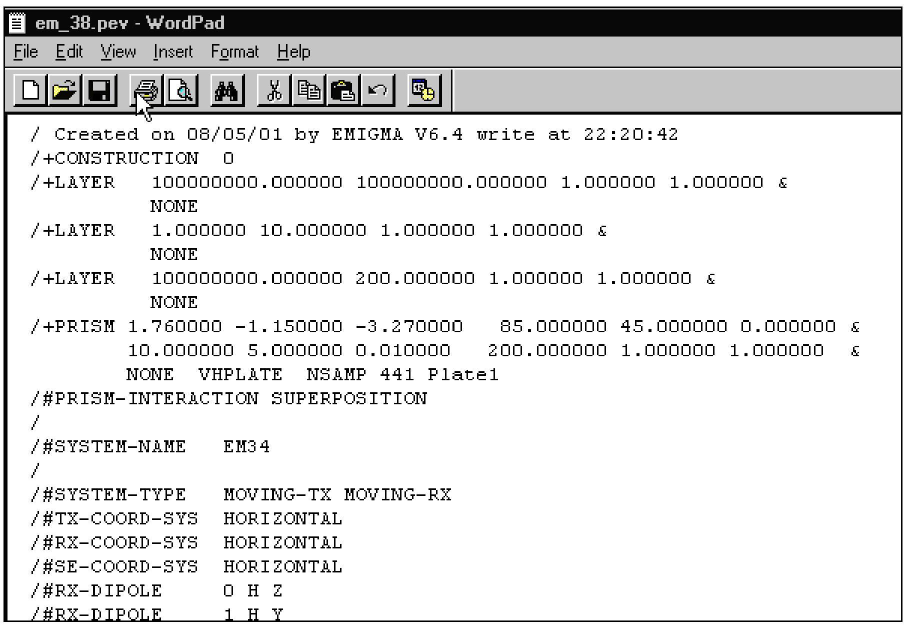
  As a result all segments will have the same number of locations and step, each changed accordingly relative to the other

- Click **OK** to complete the import procedure and return to the **Profiles** tab of the **Property Pages** dialog

Your ASCII profile will be added to the *List of Profiles* (Fig. 4). If necessary, change its name as described above.

PEV files

PEV is originally the PetRos EiKon format. PEV files can be imported, for example, from GeoTutor. In WordPad, they have the following appearance:




```
/ Created on 08/05/01 by EMIGMA V6.4 write at 22:20:42
/+CONSTRUCTION  0
/+LAYER   100000000.000000 100000000.000000 1.000000 1.000000 &
          NONE
/+LAYER   1.000000 10.000000 1.000000 1.000000 &
          NONE
/+LAYER   100000000.000000 200.000000 1.000000 1.000000 &
          NONE
/+PRISM 1.760000 -1.150000 -3.270000   85.000000 45.000000 0.000000 &
        10.000000 5.000000 0.010000   200.000000 1.000000 1.000000  &
        NONE  VHPLATE  NSAMP 441 Plate1
/#PRISM-INTERACTION SUPERPOSITION
/
/#SYSTEM-NAME   EM34
/
/#SYSTEM-TYPE   MOVING-TX MOVING-RX
/#TX-COORD-SYS  HORIZONTAL
/#RX-COORD-SYS  HORIZONTAL
/#SE-COORD-SYS  HORIZONTAL
/#RX-DIPOLE     0 H Z
/#RX-DIPOLE     1 H Y
```

- Select **Import PEV file** in the **Import Profile** dialog (Fig. 4a) and click **OK**

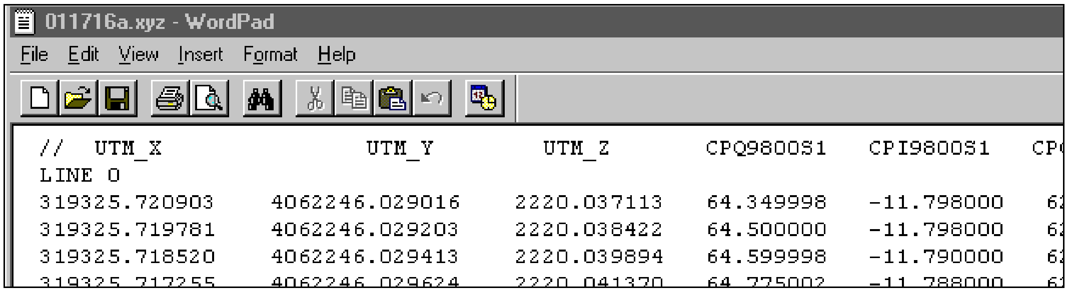
  This will bring up the **Import ASCII File with Profile Data** dialog, a standard Windows-style dialog for searching and opening files

- Select a required .pev file and click **Open**

The profile data from the file will be imported into your **Profiles** tab, Fig. 4, automatically.

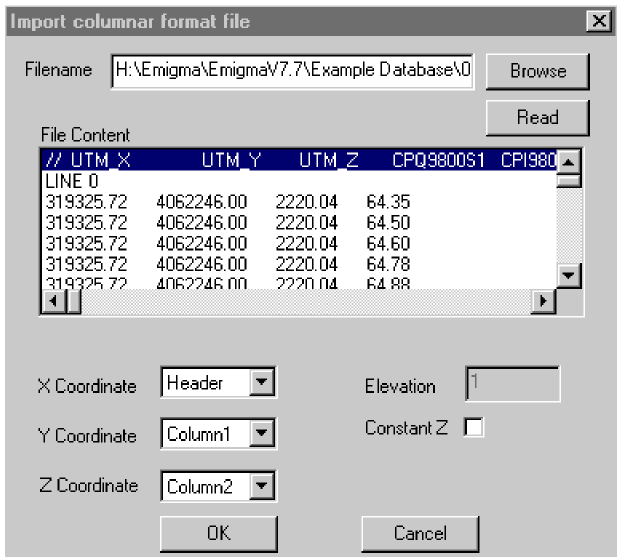
Columnar files

Columnar files have an xyz format and in WordPad they look as follows:



- Select **Import Columnar File** in the **Import Profile** dialog (Fig. 4a) and click **OK**

  This will display the **Import columnar format file** dialog:



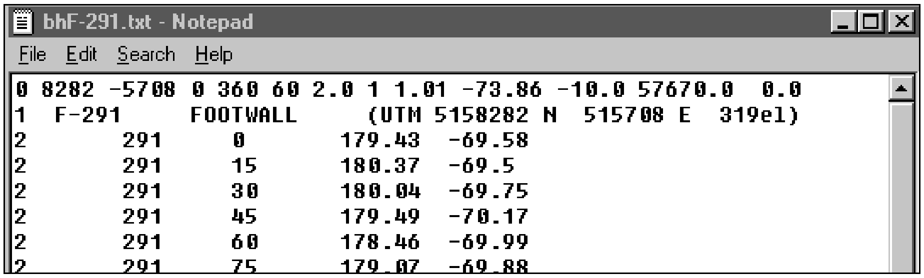
- Click **Browse** to search for a file to import. In a standard Windows-style dialog to open, select a required file and click **Open**

  The selected filename will appear in the respective field of the **Import columnar format file** dialog

- Click **Read** to display the profile information in the **File Content** field
- Select the columns for X and Y coordinates from the respective dropdown lists; in the present version, **Header** stands for the first column
- If you want the Z coordinate to be constant, leave the **Constant Z** box checked (it will be checked by default). If you want it to be taken as is, de-select the **Constant Z** box and choose a required column from the **Z Coordinate** dropdown list
- Click **OK** to complete import and see the results on the **Profiles** tab.

UTEM4 format

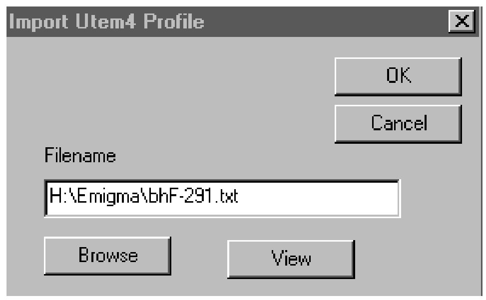
UTEM4 files contain borehole data of the following format:




```
0 8282 -5708 0 360 60 2.0 1 1.01 -73.86 -10.0 57670.0  0.0
1  F-291      FOOTWALL      (UTM 5158282 N  515708 E  319el)
2        291       0        179.43  -69.58
2        291       15       180.37  -69.5
2        291       30       180.04  -69.75
2        291       45       179.49  -70.17
2        291       60       178.46  -69.99
2        291       75       179.07  -69.88
```

- Select **Import UTEM4 file** in the **Import Profile** dialog (Fig. 4a) and click **OK**

  This will bring up the **Import UTEM4 Profile** dialog:



- Click **Browse** to display the **UTEM Data File dialog**, a standard Windows-style dialog for searching and opening files. Select a required file and click **Open**

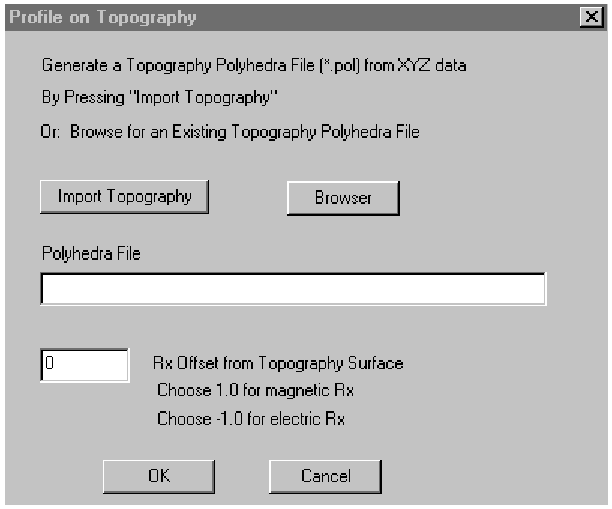
  The selected filename will appear in the respective field of the **Import UTEM4 Profile** dialog
- Click **View** to open the file in its original (text) format
- Click **OK** to proceed to the **Locations per Borehole Segment** dialog, Fig. 4b. Continue as described above (the ASCII Format section)
- Click **OK** to complete import

The profile data from the file will be imported into your **Profiles** tab, Fig. 4. You can see them displayed in the *List of Profiles* and *List of Stations*, with the coordinates of each location recalculated using azimuth and dip.

To import topography

- Click the **Profiles on Topography** button in the upper right-hand corner of the **Profiles** tab, Fig. 4

The **Profile on Topography** dialog will open:

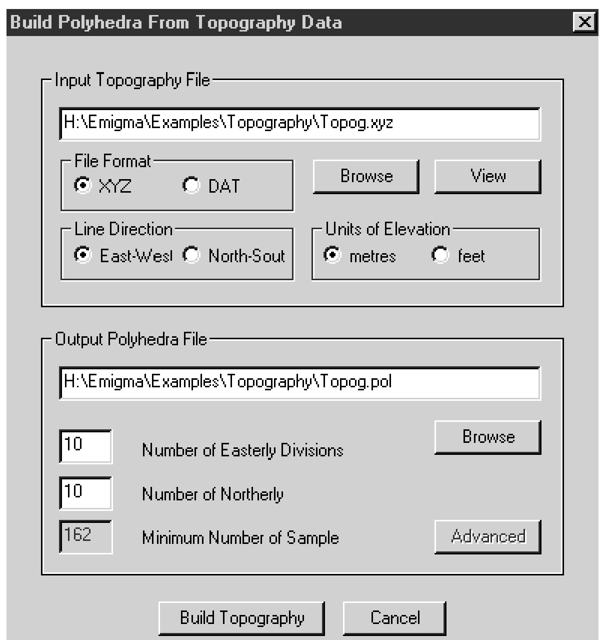


- Click **Browse** to find an existing polyhedra file (.pol)

OR

- Click **Import Topography** to create a polyhedra file from an available topography (.xyz) file

In the latter case, the **Build Polyhedra from Topography Data** dialog appears:



- Click **Browse** in the upper **Input Topography File** section to display the **Topography Data File** dialog, a standard Windows-style dialog for searching and opening files. Select a required .xyz file and click **Open**

  The filename will appear in the **Input Topography File** field. Simultaneously, a polyhedra file (.pol) will be generated and saved in the same directory, as seen from the example. Its path will be written in the **Output Polyhedra File** field of the dialog

- In the **Input Topography File** section:

- To see the input topography file in the WordPad format, click **View**
- Select the line direction and units of elevation in the respective sections

- In the **Output Polyhedra File** section:
  - Specify the number of Easterly or Northerly divisions (conventional units into which a polyhedron is divided for the simulation purposes) in the respective boxes. The minimum number of samples below will change accordingly. To properly carry out simulation, this number cannot exceed 250 (for ILN) and 1000 (for LN)
- Click **Build Topography**

  This will bring back the **Profile on Topography** dialog, with the newly created .pol file displayed in the **Polyhedra File** field
- Specify the receiver offset from the topography surface and click **OK**

Back in the **Profiles** tab (Fig. 4), see the changes in the Z column of the *List of Stations*.

To split a profile

- Select a profile to split from the list of profiles in the upper left-hand corner of the **Profiles** tab (Fig. 4) and click on the station you want to split at in the Table of Stations below
- In the **Modify Profile** section, click **Apply** to the right of the **Split Current Profile at Selected Station** command

  In the List of Profiles, you now have two profiles instead of the former one
- If necessary, rename these two profiles as described above

To merge profiles

- In the **Modify Profile** section, type the number of the first profile you want to merge in the upper and the number of the second profile in the lower box to the right of the **Join Profiles** command
- Click **Apply**

  In the List of Profiles, you now have one profile instead of the former two
- If necessary, rename this profile as described above

To reduce the number of stations

- Select the profile from the List of Profiles in the upper left-hand corner of the **Profiles** tab (Fig. 4)
- In the **Modify Profile** section, set the **Delete Every** box to 2 to delete every 2nd, 3 to delete every 3rd position and so on
- Click **Apply**

  In the List of Profiles, you now have one profile with reduced sampling

To shift a profile along the axes

- Select the axis you want to shift a profile along from the **Shift** dropdown list in the **Modify Profile** section of the **Profiles** tab (Fig. 4)

- Type the value you want to shift by in the box to the right
- Choose the **All Profiles** option to shift data of all profiles and **Current Profile** to shift data only of the profile selected
- Click **Apply**

See the coordinates in the List of Stations change accordingly.

<u>To add a single station</u>

- Select the station in the List of Stations *before* which you want to insert a new location
- Leave the **Insert** option on in the **Add Single Station** section
- Specify the X, Y and Z coordinates of the new location
- Click **Add Loc**

<u>To replace a single station</u>

- Select the station you want to replace in the List of Stations
- Turn the **Replace** option on in the **Add Single Station** section
- Specify the X, Y, and Z coordinates of the substitute
- Click **Add Loc**

<u>To delete a single station</u>

- Select the station you want to delete in the List of Stations
- Click **Delete Loc** in the **Add Single Station** section.

Specifying Waveform

To specify the waveform mode, go to the **Waveform** tab of the **Property Pages** dialog:

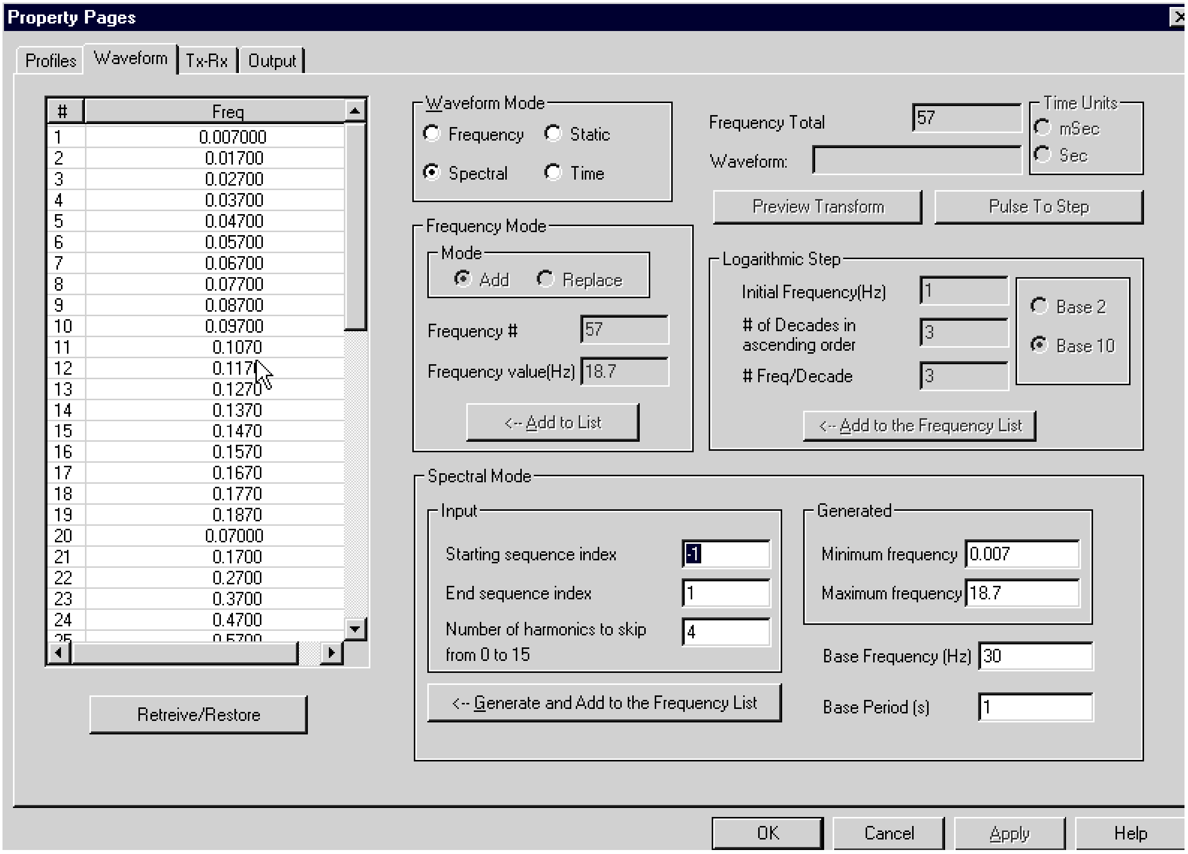


Fig. 5

In this tab, you can specify only the frequency-domain and spectral mode settings. If your data are magnetic or gravity, you will automatically have the **Static** option selected for you, with all the rest sections of the dialog being inaccessible. If you want to model a time-domain system, you first have to specify the spectral mode, run the forward simulation and only then you can transform your simulated spectral data to time-domain.

<u>To specify the frequency-domain mode</u>

This mode is useful for direct frequency-domain calculations as it allows a list of desired frequencies to be inputted directly:

- Select **Frequency** in the **Waveform Mode** section. This will enable the **Frequency Mode** and the **Logarithmic Step** sections
- In the **Frequency Mode** section:
  - ➢ Leave the **Add** option on to create a new frequency in the list of frequencies to the left

    If you are creating your first frequency, the **Frequency #** box will be set automatically to 1
  - ➢ Specify the value of your first frequency in the respective box below and click **Add to List**.

The consequent number and value of frequency will be added to the table, whereas the **Frequency #** and **Frequency Value** boxes will switch to the next number and value. Create as many frequencies as needed

- To replace a frequency, select the frequency to change in the list of frequencies, turn the **Replace** option on, specify the value of the frequency in the respective box and click **Add to List**
- To delete a frequency, select it in the list of frequencies and press the **Delete** key.

- In the **Logarithmic Step** section:
  - Choose the initial frequency in the respective box
  - Specify the number of frequency decades in the ascending order in the respective box and the number of steps within one decade in the **#Freq/decade** box.
  - Click **Add to the Frequency List**. In the example below, the initial frequency is 10.00, the number of frequency decades is 3 (10-100, 100-1000, 1000-10000) and the number of steps is 3, as well:

#	Freq
1	10.000
2	21.5443
3	46.4159
4	100.000
5	215.4430
6	464.1590
7	1000.000
8	2154.4399
9	4641.5898
10	10000.000

  - To change the step, type in your new values in the above boxes and click **Add to the Frequency List**. The message will warn you that the previous data will be lost. Click **OK** to display a new list of frequencies.

<u>To specify the spectral mode</u>

The spectral mode was designed to provide faster time-domain simulations. Spectra are generated at a fixed number of frequencies, which are sampled linearly in every decade. After having passed forward simulation (see the **Forward Simulation** section), they are subject to the standalone fast frequency to time domain transform (see Appendices: FSEMTRS).

- Select **Spectral** in the **Waveform Mode** section. The **Generate Spectral Suite** dialog opens
- Specify the base frequency in this dialog and click **OK**.

  The frequencies will be added automatically to the list on the left. The **Input** section of the **Waveform** tab (Fig. 5) will display the starting and ending spectral sequence indices dependently of the base frequency you indicated. For example, if your base frequency is less than 20, you will automatically have 1 and 3 set for the starting and ending sequences, respectively; if the base frequency is over 20, these indices will be 2 and 4. The number of harmonics will be set to 8 by default. It means that each sequence (decade) will contain 12 frequencies.

- If necessary, change the spectral settings and click **Generate and Add to the Frequency List**

In the **Generated** section, you will see the minimum and maximum frequencies dependent on the index of the first and last sequences as well as the base frequency and base period.

<u>To transform spectral data to time-domain</u>

- Specify the spectral mode as described above
- Click **Apply** in the **Property Pages** dialog to save the configuration and **OK** to close the dialog

  Note. *At this point you may be asked to specify the output information, if you have not done it earlier. Go to the* ***Output*** *tab and select required settings as described in the Specifying Output section.*

- In the **Project in Database** dialog, Fig. 1, select the survey your new spectral data set belongs to and the data set itself in the **Surveys in Project** and **Data Sets in Survey** lists, respectively, and click the **Forward Simulation** button  on the main toolbar to run simulation (for details see the **Forward Simulation** section)
- After the forward simulation has been completed, select the newly simulated data set in the **Data Sets in Survey** list of the **Projects in Database** dialog, Fig. 1, and click the **Transformer** 

button on the main toolbar to convert your data to time-domain (for details, see Appendices: FSEMTRS)

If you already have a time-domain model, but want to modify it, you will have to run a new forward simulation and transform operation. However you do not need to run the standalone transform to convert your spectral data into time-domain. It can be done automatically from the **Forward Simulation** dialog:

- Click the **Forward Simulation** button on the main toolbar to open the dialog
- In the bottom part of the dialog, click the **Simulation Settings** button
- In the **Spectral Waveform Configuration** dialog to appear, check the **Transform Spectral Domain Data to Time Domain Data** box

  The transform operation will be performed automatically without your having to run it separately.

Your data having been converted to time-domain, select the newly created data set in the **Data Sets in Survey** list of the **Projects in Database** dialog and click **Configuration** to open the **Property Pages** dialog. On the **Waveform** tab, you can view your time-domain data. Select between the **Sec** and **mSec** representation in the **Time Units** section of the tab, which is now active.

Specifying Output

The parameters related with data representation, normalization and fields are set in the **Output** tab of the **Property Pages** dialog:

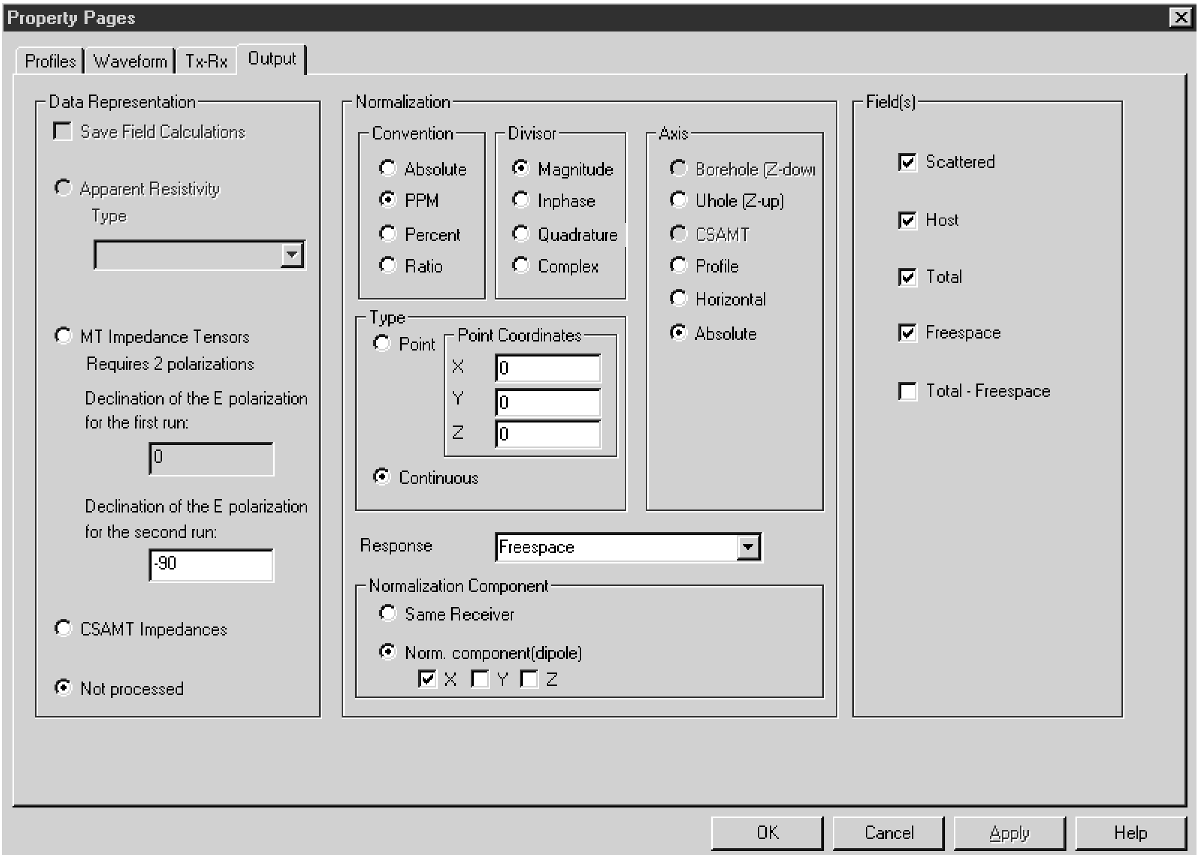


Fig. 6

If you have already specified system geometry, profile information and waveform mode, this tab will be filled out automatically. However you will be able to edit settings, if necessary. In frequency-domain systems, you can change all settings in all sections. In magnetic or gravity systems, where no normalization is used, the **Normalization** section will not be accessible, but you will be able to specify different fields. In the spectral mode, all the sections of the dialog will be disabled. In this case, normalization is performed in FSEMTRS, and since the choice of the fields is very specific, it is done for you and cannot be changed.

In the **Data Representation** section, Fig. 6, three options are accessible for the time being:

- If your data are MT or CSAMT, the **MT Impedance Tensors** or **CSAMT Impedances** options will be selected for you
- In all other cases, the **Not processed** button will be turned on

In the **Normalization** section, Fig. 6:

- To apply no normalization, click **Absolute** in the **Convention** subsection. This will disable all other buttons and boxes in the **Normalization** section
- To apply normalization:

- ➢ Select **PPM**, **Percent** or **Ratio** to specify the units of your normalized data representation. All other subsections will become active
- ➢ Select the component to normalize your data to in the **Divisor** subsection. You can normalize to the complex field, its magnitude or in-phase and quadrature components by selecting the respective option in this subsection

Note. *When you normalize to the magnitude of the field, the sign may be lost and outputs may be reversed from what is expected*

- Choose a required coordinate system in the **Axis** subsection

  If you have done it in the **Tx-Rx** tab while specifying the system geometry, the axes will be chosen for you automatically. Otherwise, there are four choices here: absolute, profile, borehole and horizontal:

  - ➢ Select **Absolute** to have the orientation of your coordinate system parallel to absolute model coordinates. This coordinate system is the simplest to visualize and is useful for model studies. However it is not very practical for modeling actual field data
  - ➢ Select **Profile** to have the orientation of your coordinate system parallel to the direction of the profile; at that, the y-axis will be horizontal and on the left-hand side when viewing down the profile and the z-axis will be pointing up and perpendicular to the x- and y-axes. This coordinate system is useful when the orientation of system components is relative to the profile, even if the profile has a variable elevation
  - ➢ Select **Uhole (Z-Up)** to model a borehole system. You will have the same orientation as in the profile system, but with the z-axis being in the line of the profile. The x-axis will be horizontal, whereas the y-axis will be orthogonal to the two and its projection on the horizontal will be the azimuth direction of the borehole
  - ➢ Select **Horizontal** to have a coordinate system in which the x-axis will be the horizontal projection of the profile, the y-axis is also horizontal and on the left-hand side when viewing down the profile and the z-axis is parallel to the absolute z-axis. Thus, it is just a rotation of the absolute system about the absolute z-axis by an angle equal to the strike angle of the profile. It is useful for modeling systems oriented along the profile, but leveled with respect to true vertical and horizontal

- In the **Type** subsection:

  There are two types of normalization supported by EMIGMA

  - ➢ Select **Point** to perform the point normalization

    This type of normalization is, as a rule, calculated using a free-space model with the same transmitter-receiver geometry (moving transmitter-receiver modes) or the same transmitter loop (fixed transmitter modes) as was used in the simulation. In the latter case, specify the coordinates of a point from which to normalize the data

  - ➢ Select **Continuous** to perform the continuous normalization

    This type of normalization is only meaningful in fixed transmitter modes. The normalizing fields are recalculated for each orientation and position occupied by a receiver. Thus the fields are normalized to the fields that would have been present if there had been no anomalous structures. It is especially useful for amplifying anomalies that may be a significant fraction of the normalizing field, but have low amplitude because of a very small host (incident) field.

- In the **Normalization Component** subsection:
    - Select **Same Receiver** if you want to normalize the field of a receiver to the equivalent free-space field modeled for the same receiver
    - Select **Norm.Component (dipole)** to specify one-, two- or three-component normalization. In three-component normalization, the sum of the total free-space field from all components is used as the normalizing field

  Since you normalize fields to equivalent free-space fields, **Freespace** will be automatically selected in the **Response** list of the **Normalization** section.

In the **Field(s)** section, Fig. 6, specify the field you want to model:

- **Scattered** fields are generated from currents flowing inside prisms, plates or other anomalous structures. In the absence of these structures, no scattered field will be present. Scattered fields can be thought of as “anomalies” sought after in exploration geophysics
- **Host** (incident) fields are reflected from the electrical property contrasts in the layered earth.

  EMIGMA offers the choice of outputting the sum of scattered and host fields. It is especially useful when you want to simulate the signal from surface or airborne prospecting systems that remove the primary field.

- **Total** fields include scattered and host fields plus the field transmitted directly from the transmitter to the receiver through the layer interfaces (if the transmitter and the receiver are in different layers) or the field that could be measured were the transmitter and the receiver in a uniform whole space (if the transmitter and the receiver are in the same layer). In other words, the total field represents the entire signal sensed by the receiver and is often useful for simulating surface-to-borehole and cross-hole surveys.
- **Freespace** fields are generated as the response of the system to an absolutely resistive environment or vacuum. In EMIGMA, both half-spaces are given the same resistivity (1e008 Ohm·m), so in effect the free-space field is the response to the system in a whole-space air model.
- **Total – Freespace** field is useful for simulating some borehole surveys

  In surface and airborne surveys, one would expect that the total minus freespace output should be equivalent to the sum of the scattered and host fields. In principle, it is true, but numerical round off errors can make difference, and the calculation of the sum of the scattered and host fields from the difference of the total and free-space fields can yield poor results.

After having specified your system geometry, profile information, waveform mode and output, click **Apply** in the bottom of the **Property Pages** dialog to save your configuration. The **Model Configuration** dialog will open offering you to build a model.

Specifying the Number and Properties of Layers

In the **Layers** tab of the **Model Configuration** dialog, Fig. 7, you can define the settings of a layered-earth model to simulate the general background conductivity structure of the ground. This model is conceptually simple and accounts for such geological effects as weathering, lateralization, overburden and conductive ground water.

To build this model, one has to specify the number of layers and assign certain physical properties (conductivity, magnetic permeability and electrical permittivity) to each layer. There is an option to include Cole-Cole polarization parameters in each layer, as well. The model consists of two half spaces with a number of layers sandwiched in between.
In the **Layers** tab:

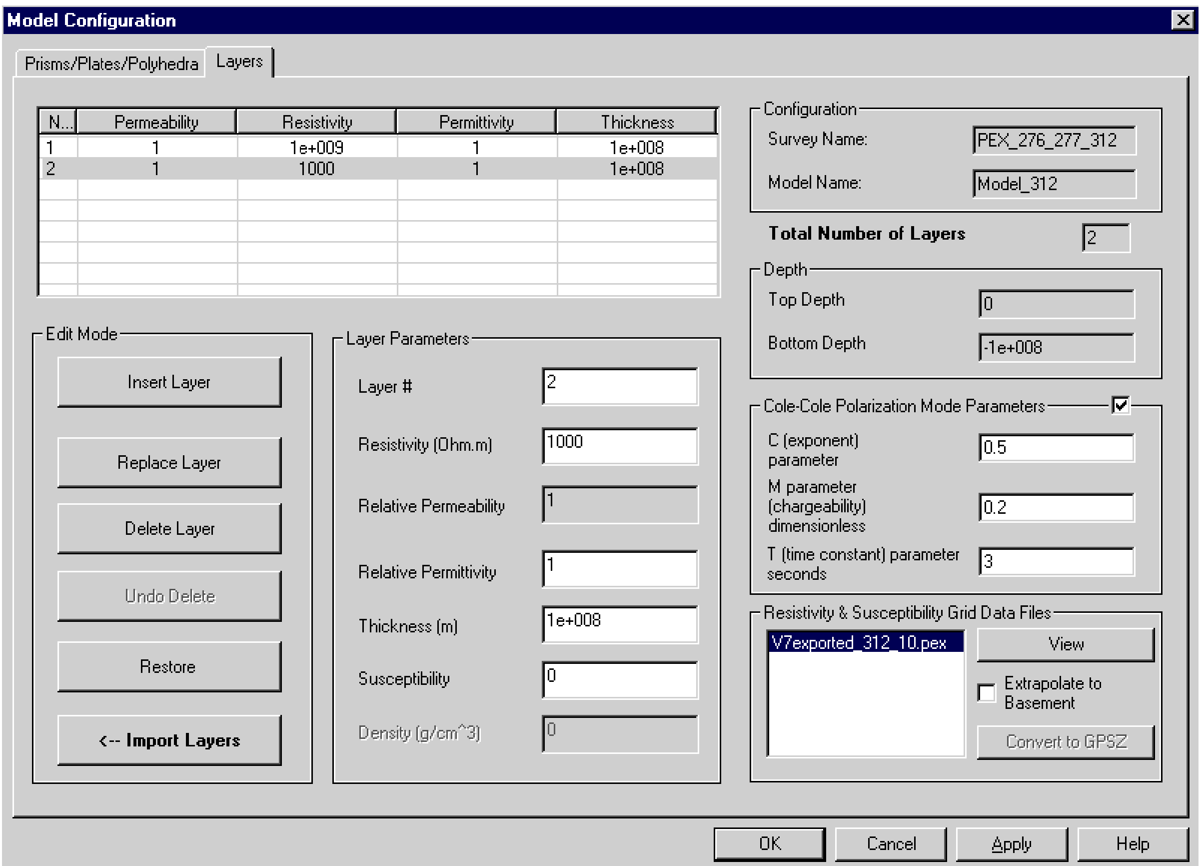


Fig. 7

The first item in the *List of Layers*, a table in the top left-hand corner of the tab, will represent the upper half space (usually air) and the last - the lower half-space (ground). The index (**N**) of the upper half-space will always be 1, whereas the index of the lower half-space will depend on the number of layers requested in the model. Since the half-spaces are to be compulsorily included in the model, the total number of layers in the table cannot be less than 2.

To insert a layer

On opening the **Layers** tab, you see the *List of Layers* containing the first half-space by default. To add a layer:

- Specify the index (N) and properties of the new layer in the **Layer Parameters** section and click **Insert Layer** in the **Edit Mode** section

  The new layer will appear in the row specified by the index

<u>To replace a layer</u>

- In the *List of Layers*, select the layer you want to replace and change the properties in the **Layers Parameters** section
- In the **Edit Mode** section, click **Replace Layer**

  The replaced layer will have the newly set properties.

<u>To delete/restore a layer</u>

- In the *List of Layers*, select the layer you want to delete and click **Delete Layer** in the **Edit Mode** section

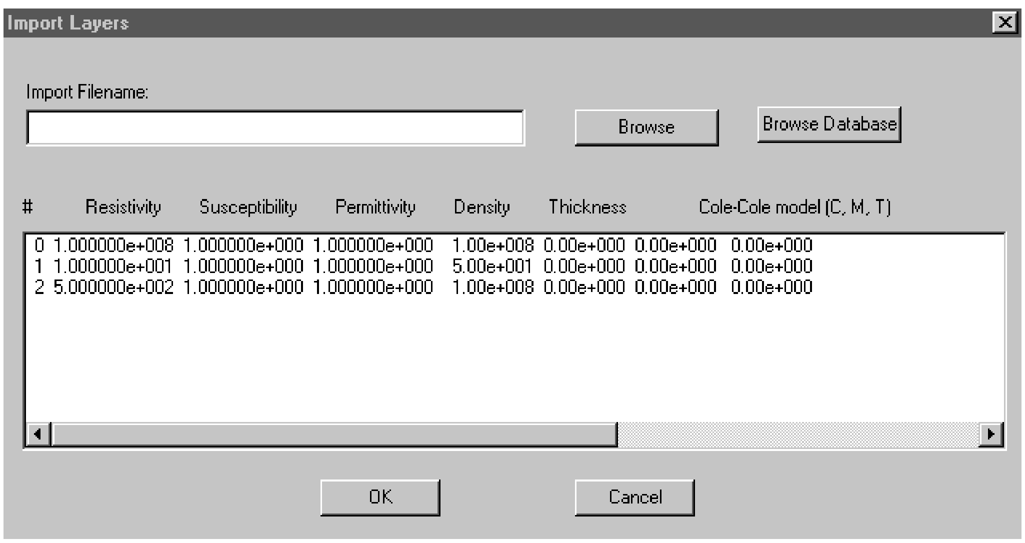
  The layer will be removed from the *List of Layers*

- To restore the layer, click **Undo Delete**.

  The layer will reappear in the *List of Layers*

<u>To import a layer</u>

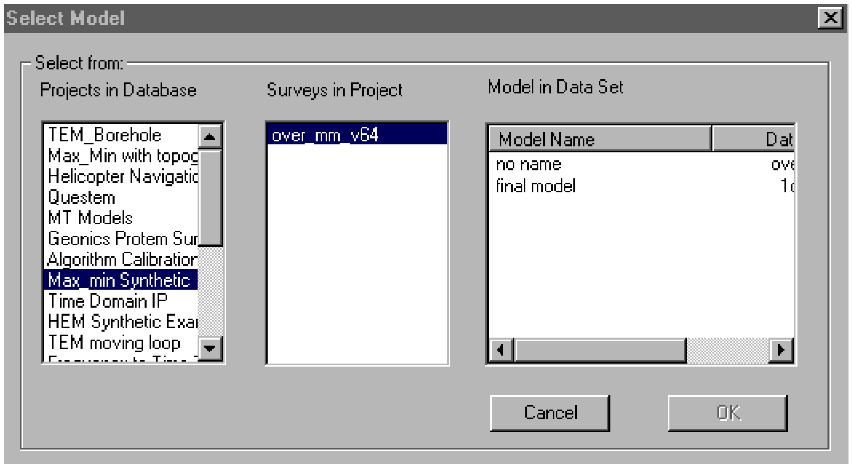
- Click **Import Layers** in the **Edit Mode** section of the **Layers** tab, Fig. 7. The respective dialog will appear:



- In this dialog, click **Browse** to open the standard Windows-style **Open** dialog and select a PetRos EiKon file (.pev, .dat, .spt) to import layers from. Click **Open**. This returns you to the **Import Layers** dialog, in which the **Import Filename** box now contains the filename you selected and the field below shows the layers to import

OR

- Click **Browse Database** to bring up the **Select Model** dialog and search for the model within the current database:



- Select the project, survey and model in the respective lists and click **OK** to return to the **Import Layers** dialog

  The list of layers will be displayed in the respective field of the dialog

- Click **OK** in the dialog to complete import and return to the **Layers** tab, Fig. 7. You now see the *List of Layers* containing the imported layers and their properties

The **Configuration** section, Fig. 7, displays the name of your current survey and the **Depth** section - information on the depth of your new layered-earth model.

<u>To build the Cole-Cole model:</u>

The Cole-Cole model is useful for representing the electrical conductivity or resistivity of a polarizable material and, therefore, allows one to account for frequency dependent conductivity effects associated with low frequency IP phenomena.

- Check the **Cole-Cole Polarization Mode Parameters** box to activate the respective section of the **Layers** tab, Fig. 7
- Specify chargeability, m, a time constant, τ, and a frequency dependence, c, - the three parameters that describe conductivity/resistivity, magnetic permeability and electric permittivity in EMIGMA

<u>Resistivity & Susceptibility Grid Data Files</u>

In the **Resistivity & Susceptibility Grid Data Files** section, you can view .pex files created as a result of the 1D inversion within EMIGMA. The names of such files, if available, will be displayed in the field of the section

- Click **View** to look at the data in the text format. A text file contains resistivity data as well as coordinate and depth information.

  Note. *To view these data in a grid form, close the **Property Pages** dialog, select the PEX file from the **Data Sets in Survey** list, Fig. 1, and click the **PEX file***  *button on the main toolbar*

- Click **Convert to GPSZ** to include topography in the resistivity pattern available. This command will be active if your PEX file contains GPS data. When launched, it recalculates Z coordinates by subtracting depth values from the GPS data

  OR

- Click at first in the **Extrapolate to Basement** checkbox to prevent the resulting pattern from being ragged and disrupted at the bottom. Then click **Convert to GPSZ**

  A new GPSZ data set will be added to the **Data Sets in Survey** list of the **Projects in Database** dialog, Fig. 1. Click the **PEX file** button on the main toolbar to see the results of the conversion.

Specifying the Number and Properties of Targets

EMIGMA allows simulation of different complex conductivity structures that can be represented by right rectangular prisms (overburden thickness variations, fractures, lithology changes, etc.) and plates (dipping tabular bodies). The prisms and plates can further be converted into polyhedra in the Visualizer to build more complicated geometrical shapes. Polyhedra can also be created from scratch.

In the **Prisms/Plates/Polyhedra** tab of the **Model Configuration** dialog:

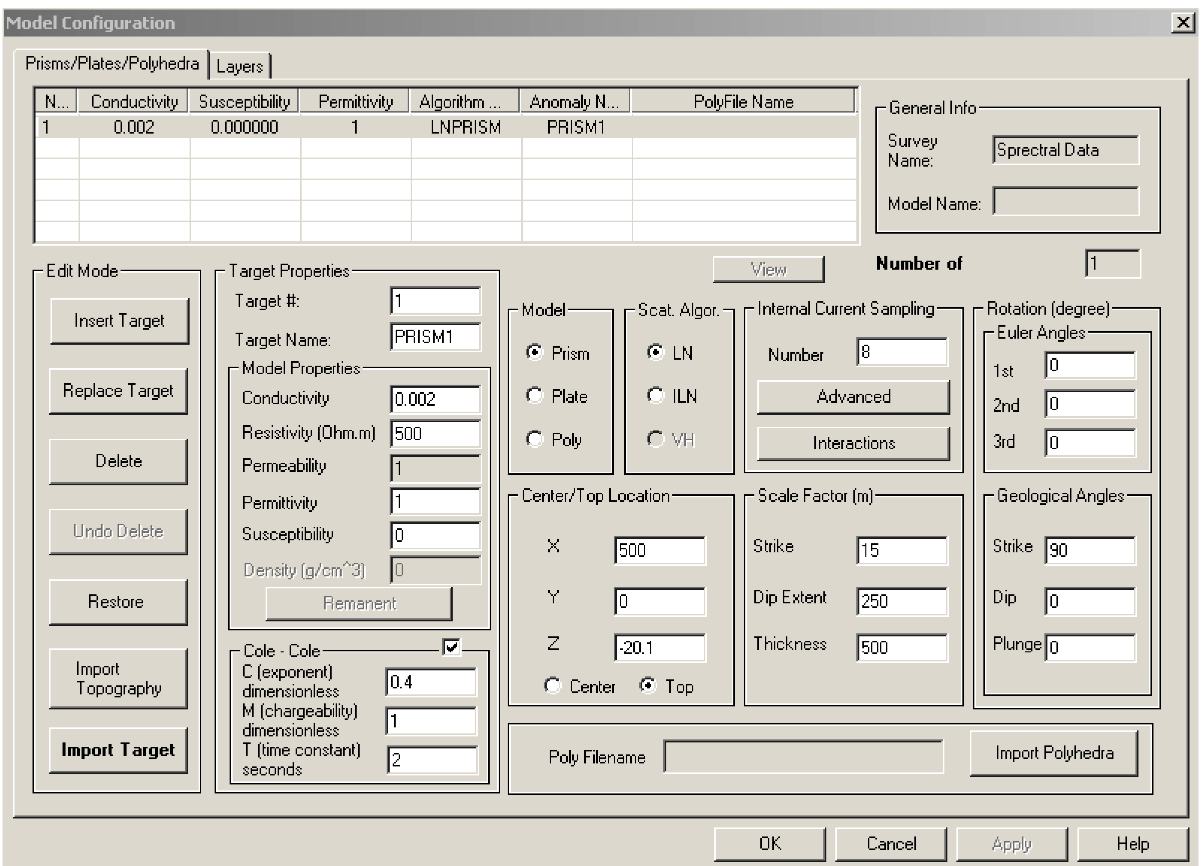


Fig. 8

<u>To insert a target</u>

- Select between **Prism**, **Plate** and **Poly** in the **Model** section of the tab

The **Target Properties** section will automatically display the number and name of the model. Change the name, if necessary

- Specify the scattering algorithm to be used in the **Scat. Algor.** section. If your model is a prism or polyhedron, two options are available, **LN** and **ILN**, whereas in the case of a plate, the **VH** option is selected by default

  Note*. You can find more information on these algorithms in the manual "Forward 3-D Electromagnetic and Magnetic Simulation Platform for Comprehensive Geophysical Modeling", Feb. 2000*

- In the case of a plate, select between the **Center** and **Top** options in the respective section. Both options represent the reference points of a plate in the absolute frame, the first coinciding with its center and the second – with its top center
- In the case of a polyhedron, click the **Import Polyhedra** button in the bottom right-hand corner of the tab. This will open the standard Windows-style **Open** dialog offering you to select an existing polyhedra file (.pol). Select a required file and click **Open**. The name and path of this file will appear in the **Poly Filename** box

  Note*. To create a polyhedra file, use the standalone polyhedra generator available in EMIGMA (PolyGen.exe)*

- In the **Model Properties** section:
  - Specify the physical properties of the target (conductivity, magnetic permeability, electrical permittivity)
  - Set density in the **Density** box, which becomes active in gravity systems
  - To account for remanent magnetization in magnetic systems, click the **Remanent** button. In the **Remanent Magnetization** dialog to open, specify the inclination, declination and relative magnetization parameters
- To account for frequency dependent conductivity effects which are often associated with low frequency IP phenomena, check the **Cole-Cole** box to activate the respective section and specify chargeability, m, a time constant, τ, and a frequency dependence, c, the three parameters describing above effects:

  Chargeability is the proportional drop in voltage immediately after the current is turned off. **1** is a maximum, **.6** is a large chargeability, **.004** is common for many background materials

  Frequency dependence (exponent) generally ranges from 0.5 to 1.0, with 1 indicating one IP decay for the anomaly

  Time constant (decay in seconds for a scattering object) generally ranges from 0.5 to 3.0 seconds

- Click **Insert Target** in the **Edit Mode** section. Your target will be added to the *List of Targets*, the table in the upper left-hand corner of the tab, Fig. 8.

<u>To delete/restore a target</u>

To delete a target:

- Select it in the *List of Targets* and click **Delete Target**

To restore a target:

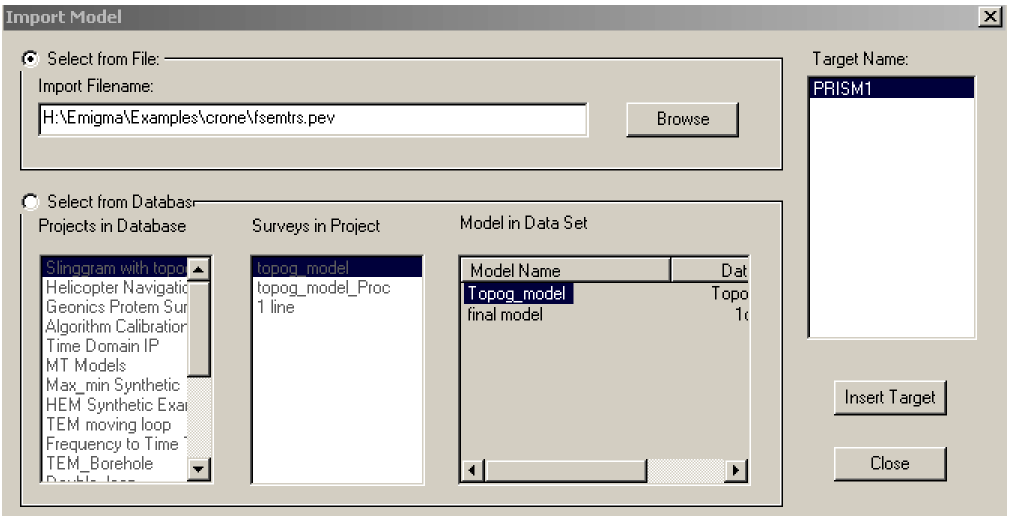
- Click **Undo Delete**, if you have deleted your target and now decide to bring it back

OR

- Click **Restore**, if you have deleted, replaced or changed your target and now want to cancel all the changes made ever since and return to the initial version

To import a target

- Click the **Import Target** button in the bottom of the **Edit Mode** section, Fig. 8. The **Import Model** dialog appears



To import a model from an existing PEV file:

- Click **Browse** in the **Select from File** section to open the standard Windows-style **Open** dialog and select a PetRos EiKon file (.pev, .dat, .spt) containing a required model
- Click **Open**. This will return you to the **Import Model** dialog. The **Import Filename** box will now contain the filename you selected, and you will see the name of the target to import in the respective field on the right
- Click **Insert Target** to complete the import

To import a model from the current database:

- Click **Select from Database** and select the project, survey and model from the respective lists

  The name of the model will be displayed in the **Target Name** field on the right

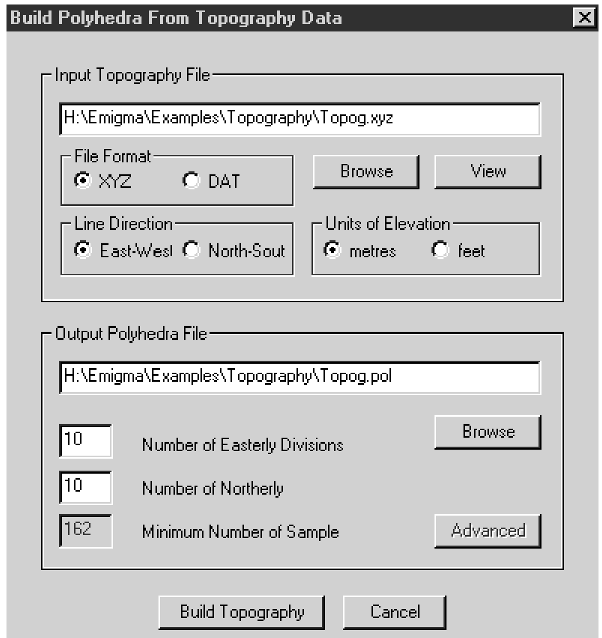
- Click **Insert Target** to complete the import

As you return to the **Prisms/Plates/Polyhedra** tab, Fig. 8, you find the imported model added to the *List of Targets*.

To import topography

For more realistic representation of your model, you can synthesize topography and polyhedra

- Select **Poly** in the **Model** section and specify the physical parameters of your target in the **Target Properties** section
- Click the **Import Topography** button in the **Edit Mode** section. The **Build Polyhedra from Topography Data** dialog appears:



- Click **Browse** in the upper **Input Topography File** section to display the **Topography Data File** dialog, a standard Windows-style dialog for searching and opening files. Select a required topography XYZ file and click **Open**

  The filename will appear in the **Input Topography File** field. Simultaneously, a polyhedra file (.pol) will be generated and saved in the same directory, as seen from the example. Its filename will be written in the **Output Polyhedra File** field of the dialog

- In the **Input Topography File** section:
  - To see the input topography file in the WordPad format, click **View**
  - Select the line direction and units of elevation in the respective sections
- In the **Output Polyhedra File** section:
  - Specify the number of **Easterly** or **Northerly** divisions (conventional units into which a polyhedron is divided for the simulation purposes) in the respective boxes. The minimum number of samples below will change accordingly. To properly carry out simulation, this number cannot exceed 250 (for ILN) and 1000 (for LN)
  - Click **Advanced** to specify the boundaries of the topography you are about to import in the **Advanced Settings** dialog to appear

- Back in the **Build Polyhedra from Topography Data,** click **Build Topography**

  This will return you to the **Prisms/Plates/Polyhedra** tab, Fig. 8. In the **Poly Filename** field, you will see the newly created .pol file

- Click **Insert Target** to add your polyhedron, topography inclusive, to the *List of Targets*

To specify the number of sampling points

If your model is a prism or polyhedron, the **Internal Current Sampling** section, Fig. 8, will be enabled suggesting 100 points as a default value for LNPRISM. You can change it as needed, with a maximum of 600, which is often more than enough. The **Advanced** button allows for a more accurate setting:

- Click **Advanced** to open the **Advanced Prism Sampling** dialog
- Type your values in the **X**, **Y** and **Z** boxes. The multiplication product of these values can be viewed in the **Total Number** field of the dialog
- Click **OK** to close the dialog and view the result in the **Number** box of the section

If your model is a plate, the **Internal Current Sampling** section will be disabled and the respective number will be set to 441. This value represents a mesh of 21 by 21 used to calculate the scattered current distribution in the VH scattering algorithm and cannot be changed.

In this section, you can also specify the type of interaction between various prisms or plates in your model:

- Click the **Interaction** button. This brings up the **Prism Interactions** dialog
- Select between the four options available in this dialog:

  The **Superposition** option simply adds the target responses that do not interact

  The **Near-Field** option accounts for the current flow between your targets and regional conductors in contact with them

  The **Far-Field** option includes the primary effects of multiple conductors not in contact with your target

  The **Automatic Mode** option sets interaction for you automatically

- Click **OK**

To specify the model size and position

- In the **Scale Factor (m)** section, specify the geological dimensions of your target, including its strike, dip and thickness
- In the **Rotation (degree)** section, specify the Euler and geological angles in the respective fields to provide the required orientation of the model

To replace a target

- Select the target you want to replace in the *List of Targets*, Fig. 8
- Change settings in the rest of the sections as needed and click **Replace Target**

Saving your model

- Click **Apply** in the bottom right-hand corner of the **Prisms/Plates/Polyhedra** tab
- Click **OK** to close the **Model Configuration** dialog and return to the initial **Projects in Database** dialog, Fig. 1. The box on the **Model** button in the bottom of this dialog will be ticked off to signal that your data set now contains a model

***Note**. To reopen the **Model Configuration** dialog, click the **Model** button*

DATABASE PAGE: ORGANIZING PROJECT INFORMATION

The **Database** dialog, Fig. 1, is divided into 3 levels of data management.

Projects

In the **Projects in Database** section of the dialog, you can create or delete projects and change their names. To create a new project, see Creating Projects from Imported Data and Creating Projects from Scratch.

<u>To change the name of a project</u>

- Select it in the **Projects in Database** list in the upper left-hand corner of the section and type a new name in the **Project Name** box below
- Click the **Change Name** button

<u>To delete a project</u>

- Select it in the **Projects in Database** list and click the **Delete Project** button.

  The message will ask you to confirm deletion
- Click **OK**

Surveys

In the **Surveys in Project** section, Fig. 1, you can see a list of surveys available in each project. Here, you can add or delete surveys, provide them with comments, change their names or create back up files. The **Survey ID** number is generated automatically.

<u>To add a new survey</u>

- Click **Add Survey** in the **Surveys in Project** section. The **Property Pages** dialog will open suggesting you to specify your survey parameters
- Perform all the steps described in the Creating Projects from Imported Data and Creating Projects from Scratch sections. Back in the **Projects in Database** dialog, you can see your new survey added to the **Surveys in Project** list

<u>To rename a survey</u>

- Select it in the **Surveys in Project** list. Its name will appear in the **Survey Name** box to the right
- Replace the old name with a new one and click **Change**

<u>To provide a survey with comments</u>

- Select the survey in the **Surveys in Project** list and click the **Survey Comments** button.

  The respective dialog will open. For your convenience, you can pin it to prevent it from closing during other operations, if any, in the **Projects in Database** dialog
- Write necessary comments in the empty field of this dialog, click **Save** and **Exit**

To view or change comments

- Double-click the survey you want to display your comments for or select it in the list and click the **Survey Comments** button

  The **Survey Comments** dialog will open containing the earlier written comments

- Change them as needed, click **Save** and **Exit**

To create a backup survey

- Select the survey in the list and click the **BackUP** button. This will automatically create a backup file and add it to the **Surveys in Project** list

To delete a survey

- Select the survey to delete and click the **Delete Survey** button.

  The message will ask you to confirm deletion

- Click **OK**. The survey will be removed from the **Surveys in Project** list

Data Sets

In the **Data Sets in Survey** section, Fig. 1, there is a list of data sets available in each survey. They all are characterized by the same configuration (system geometry, waveform, profile and output settings). Whenever you do any operations with your data set – subject it to simulation, transform or inversion - the program will ask your permission to overwrite it; otherwise it will create a new data set, with a name differing from the initial one by an automatically generated number attached to its end.

The **Data File Name** box below the list contains the name, which EMIGMA assigns to each data set to store it with in the respective database directory

The boxes to the right of the list display information on the type of data (measured or simulated; frequency-domain, time-domain, spectral, static, gravity), data set ID (EMIGMA internal number), the date of creation, and output fields.

To change the name of a data set in the list

- Select the required data set. Its name will be displayed in the **Data Set** box
- Type a new name in the box and click **Change**

To delete a data set

- Select it in the list and click the **Delete Data Set** button

  A message will ask your confirmation. Click **OK**

To specify the name of a model

- Select the data set containing a model in the list. The model name will be displayed in the **Model Name** box to the right
- Type a new name in the box and click **Change**

This option may be very useful, since it allows you to provide additional details about your data set.

To add, view or edit a model

- Select the data set in the list and click **Model**. The **Model Configuration** dialog will open
- Edit or view our layered earth model on the **Layers** tab and your target on the **Prisms/Plates/Polyhedra** tab
- To add a new model, follow the steps described in the Specifying the Number and Properties of Layers and Specifying the Number and Properties of Targets sections

*Note. Once you added a model to your data set, the box on the **Model** button in the **Projects in Database** dialog will be checked.*

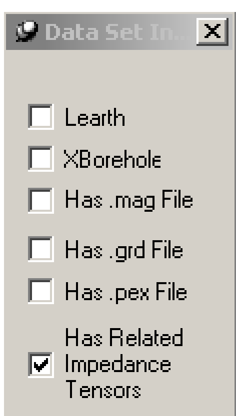
To view or edit the survey configuration

- Select the survey or data set in the respective lists and click **Configuration**. This will display the **Property Pages** dialog, Fig. 3-6, with the system geometry, profile, waveform and output information

To view data set information

To check what kind of information a data set may contain, click the **Data Set Info** button in the bottom of the **Project in Database** dialog, Fig. 1. The respective dialog will appear:

Pin it not to let it close if you want to search through a number of data sets



- The checked **Learth** box means that the data set contains a layered earth model.
- The checked **Xborehole** box means that the data set represents a cross-hole survey.
- The checked **Has .mag File** box means that the data set has an attached MAG file resulting from 3D magnetic inversion within EMIGMA
- The checked **Has .grd File** box means that the data set has an attached *.grd file resulting from inversion in EMIGMA
- The checked **Has .pex File** box means that the data set has an attached PEX file resulting from 1D inversion within EMIGMA
- The checked **Has Related Impedance Tensors** box means that the data set contains files for different configurations of an MT/CSAMT survey.

To view grid information

If your data have earlier been subject to interpolation, the resulting grid is attached to the initial data set. Whether or not a data set has a grid will be seen from the **Has Related Grid(s)** button in the **Project in Database** dialog, Fig. 1. The box on the button will be checked to signal the availability of a grid/grids. To view grid information, select the data set in the **Data Sets in**

Project list, Fig. 1, and click the **Has Related Grid(s)** button. This will bring up the **Grid Information** dialog:

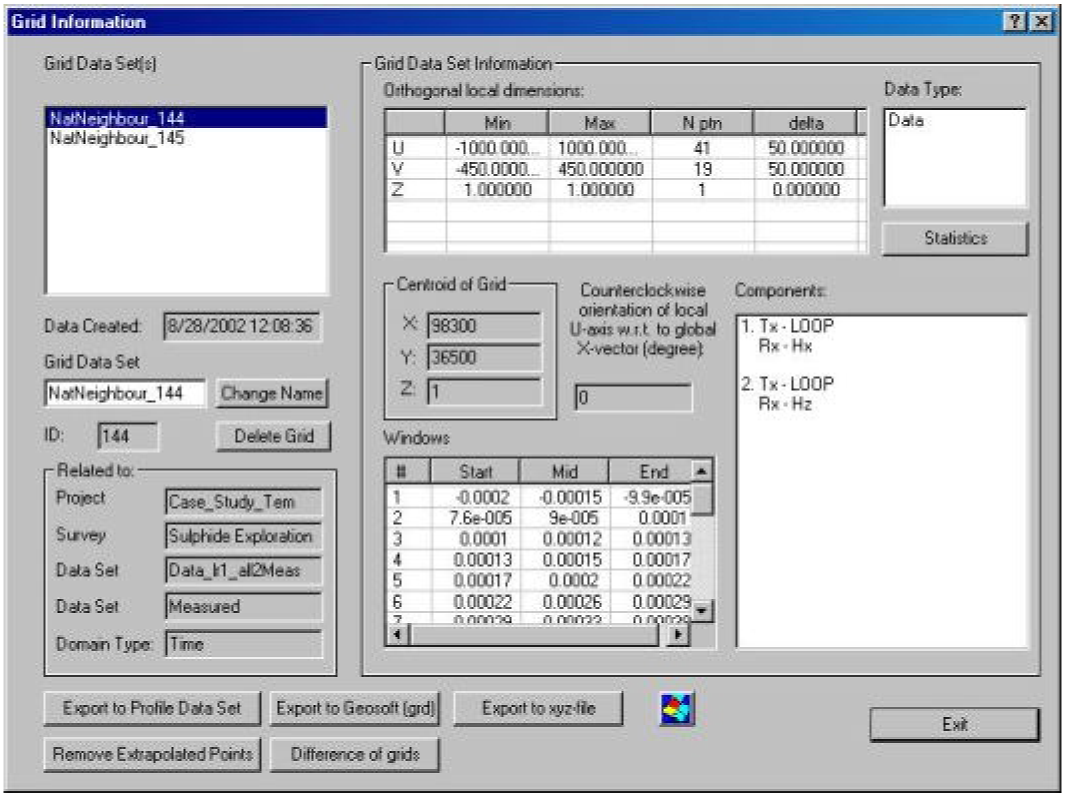


Fig. 9

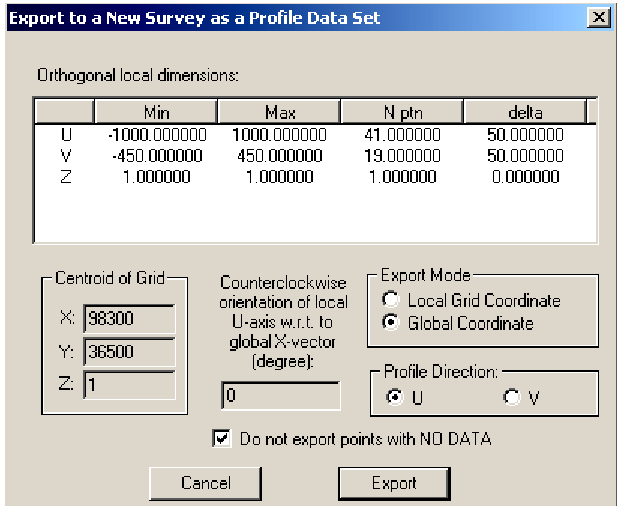
In this dialog, the **Grid Data Set(s)** list contains all the grids available with your data set

- To view information on a given grid, select it in the list. The rest of the fields and sections of the dialog will display thorough information on this grid and the data set it belongs to. This information cannot be edited
- To see more details, i.e. the grid statistics per channel or configuration component (transmitter, receiver, and separation) in a desired format (fixed or exponential), click the **Statistics** button

  The **Grid Statistics** dialog will open. Toggle through all the available channels and configuration components to view respective grid parameters
- To change the name of the grid, type the new name in the **Grid Data Set** box in the left-hand part of the **Grid Information** dialog and click **Change Name**
- To delete a grid, select it in the **Grid Data Set(s)** list and click the **Delete Grid** button

To export a grid into the Profile Data Set

To export your grid into a separate survey/data set, click the **Export to Profile Data Set** button in the bottom of the **Grid Information** dialog, Fig. 9. The **Export to a New Survey as a Profile Data Set** dialog appears with the **Orthogonal Local Dimensions** field displaying the coordinates of your grid



- To export as a profile data set using the local grid coordinates:
  - ➢ Select the **Local Grid Coordinate** button in the **Export Mode** section
  - ➢ Select between the two options for profile direction in the respective section. If you turn the **U** option on, the profile direction will coincide with the X-axis; with the **V** option on, the profile direction will coincide with the Y-axis

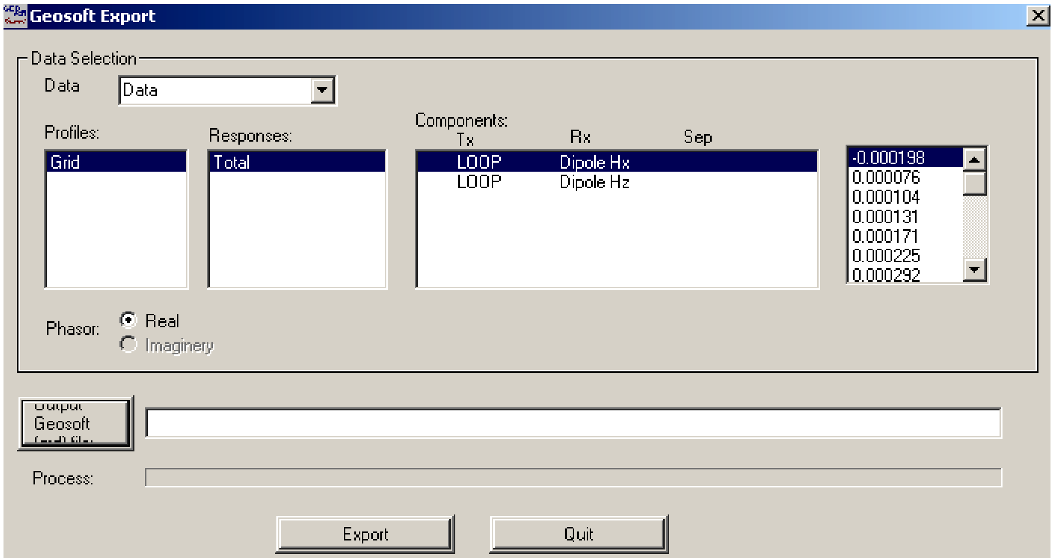
    In the example above, if you select the **Local Grid Coordinate** option, with the **U** option turned on in the **Profile Direction** section, your profile in the resulting data set will start at –1000 and end at 1000, with 41 stations spaced at 50 m.

- To export as a profile data set using the coordinates of the original data set:
  - ➢ Select the **Global Coordinate** button
  - ➢ Select between the two options for profile direction in the respective section.

    In the example above: the profile in the original data set, as seen from the number of stations (41), the step (50 m) and the X coordinate of the profile center (98300) in the **Centroid of Grid** section, starts at 97300 and ends at 99300. The exported profile will have the same coordinates and, since the **U** option in the **Profile Direction** section is turned on, will coincide with the X-axis
- To prevent export of points containing no data, check the respective box in the bottom of the dialog
- Click **Export**. The new survey appears in the **Surveys in Project** list, Fig. 1, containing a new data set. Change the name of the survey and data set, if necessary

To export a grid into the Geosoft format

To use your data in Geosoft's Oasis Montaj, export them into the Geosoft format. Click the **Export to Geosoft (.grd)** button in the bottom of the **Grid Information** dialog, Fig. 9. This will bring up the **Geosoft Export** dialog:



- Select the data type you want to export from the **Data** dropdown list

  All data obtained by means of import or simulation in EMIGMA, subjected to normalization, etc., are considered as core data and referred to as ***Data***; all the rest - calculated through various algorithms - are considered as optional and referred to in accordance with their type, e.g. ***Apparent Resistivity***, ***Apparent Depth***, ***IP***, etc.

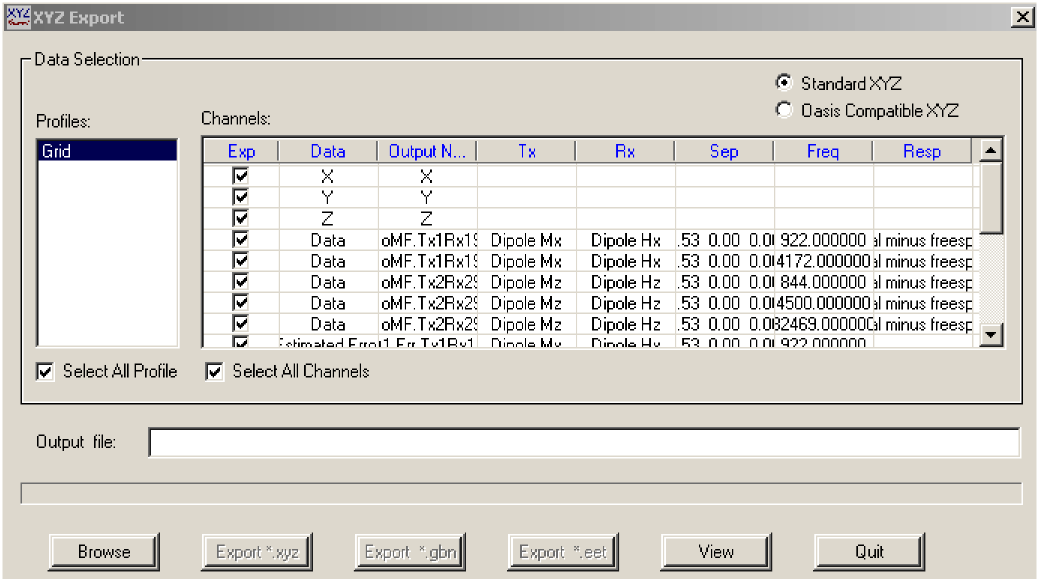
- Make necessary selections in the **Profiles**, **Responses** and **Components** boxes. The Geosoft format can take only one component or channel at a time

- Click the **Output Geosoft (.grd) File** button to bring up a standard Windows-style **Open** dialog, browse for the directory and folder to save the output file into and click **Open**. The output filename will appear in the respective box

- Click **Export** to complete export to the Geosoft format

To export a grid into the XYZ format

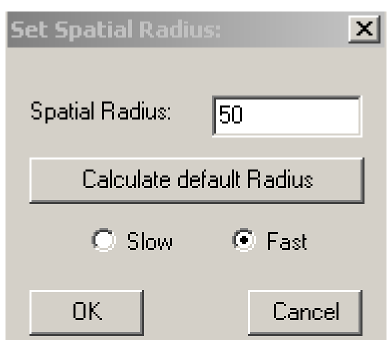
- Click the **Export to xyz-file** button in the bottom of the **Grid Information** dialog, Fig. 9. The **XYZ Export** dialog will open offering two different options of export: one transfers data to the standard XYZ format and the other to the Geosoft's Oasis Compatible XYZ format:



- Select the profile and channel to export in the respective lists. You can choose to select all available profiles and channels by checking the respective boxes below the lists
- Click **Browse** to bring up a standard Windows-style **Open** dialog, specify the directory and folder to save the output file into and click **Open**. The output filename will appear in the **Output File** box
- To view the output file in the text format, click **View** in the bottom of the dialog
- Click **Export *.xyz** if you export your data into the standard XYZ format, **Export *.gbn** if you export them into the Geosoft's Oasis compatible XYZ format and **Export *.eet (.qct)** if you export your data into the XYZ format to further subject them to quality control in PetRos EiKon's QCTool

<u>To remove "no data" points from your grid</u>

- Click the **Remove Extrapolated Points** button in the bottom of the **Grid Information** dialog, Fig. 9. This will bring up the **Set Spatial Radius** dialog:



- Set a required spatial radius to restrict the area of interpolation. You can also click the **Calculate Default Radius** button to use the radius set by default

In the present example: a spatial radius of 50 means that if there are no data in the radius of 50 m around a given point – a grid cell center, – this cell will be removed from interpolation

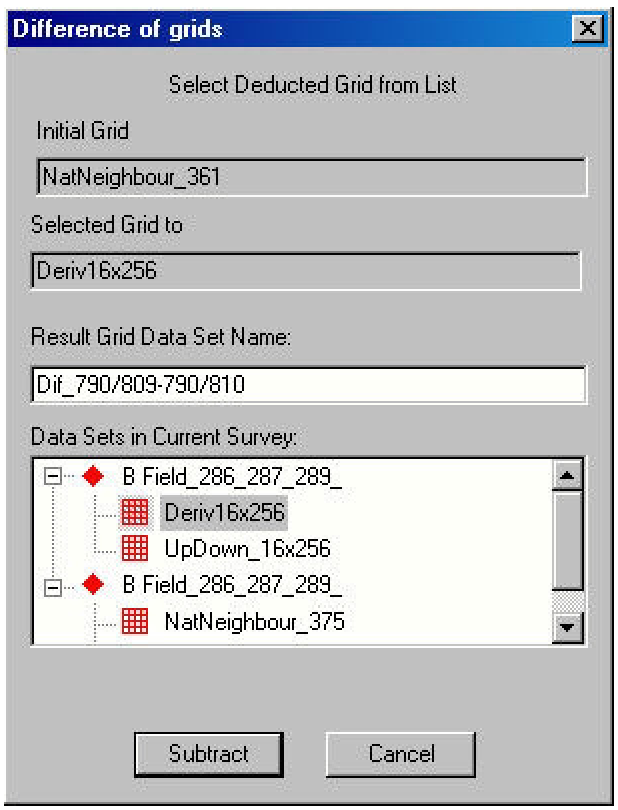
- Select between the slow and fast interpolation algorithms (slow is more accurate, but fast is almost always sufficient) and click **OK**

  Your new grid will appear in the **Grid Data Set(s)** list of the **Grid Information** dialog, Fig. 9.

Note. *To view your grid, select the data set and click the* ***GridPresentation*** *button on the main toolbar.*

Calculate the Difference of Two Grids

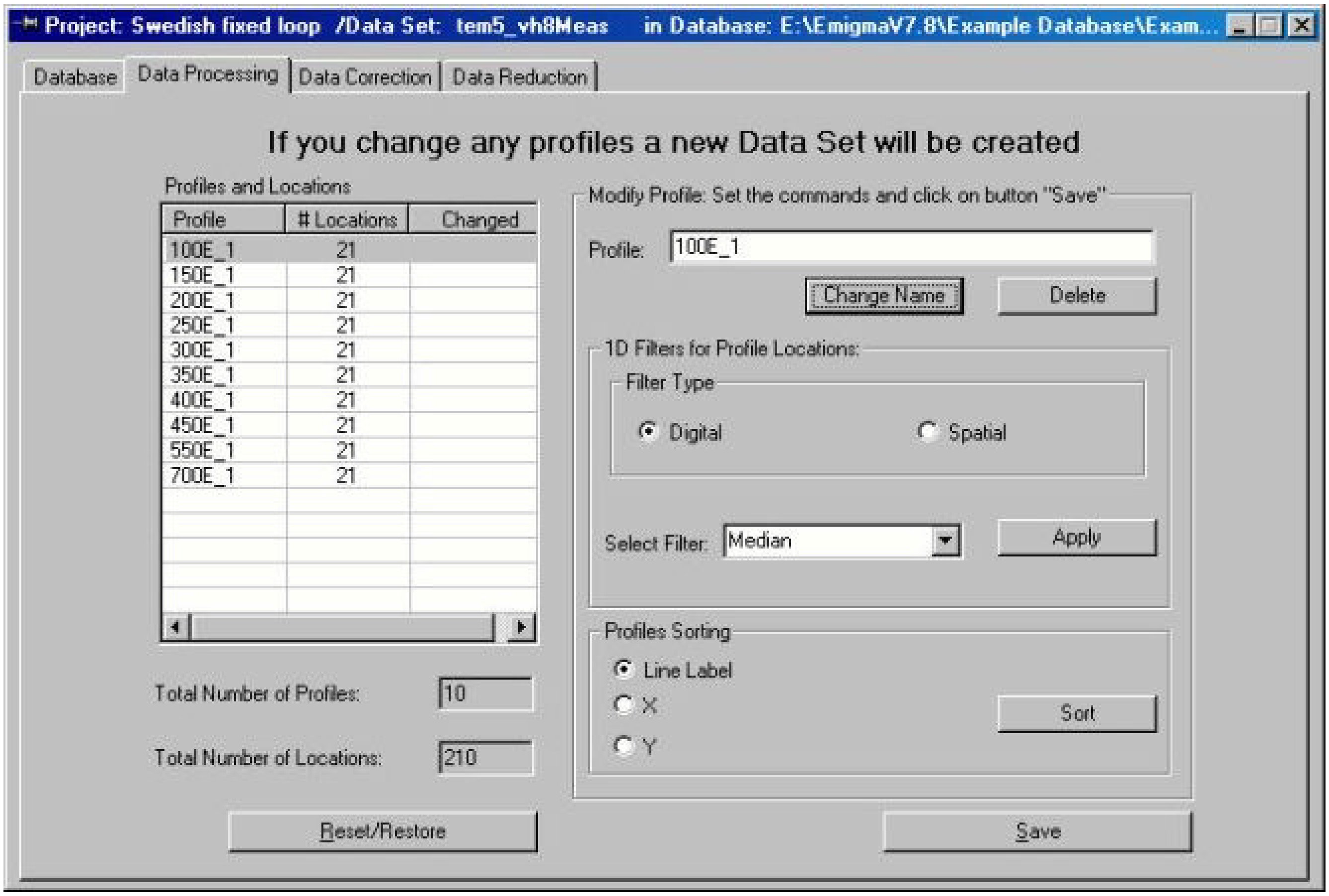
To create a new data set which contains the difference of two grids, first select the grid data set that you would like to subtract from, then click the **Difference of grids** button at the bottom of the **Grid Information** dialog. The **Difference of grids** dialog appears:



In this dialog, the grid data set that you selected is displayed under **Initial Grid**. The name of a default grid which is to be subtracted from the initial grid is displayed under **Selected Grid To**. This grid can be changed by selecting a different one from the **Data Sets in Current Survey** list. The data sets which are not selectable are displayed with a red diamond next to the name. The selectable grids are listed under the data set name. The name of the resulting data set containing the difference of data is displayed under **Result Grid Data Set Name**. This default name may modified. Click **Subtract** to create the new grid data set.

Data Processing

Click on the **Data Processing** tab on the Database dialog to reach this tool:



Use this tool to make changes to the profiles in a survey.

- Click on the profile you would like to modify in the Profiles and Locations list.

To change names:

- Edit the name that appears beside the Profile label and click Change Name. Click Save to keep your changes
- Click Delete to delete the profile. Click Save to keep your changes.

To sort profiles:

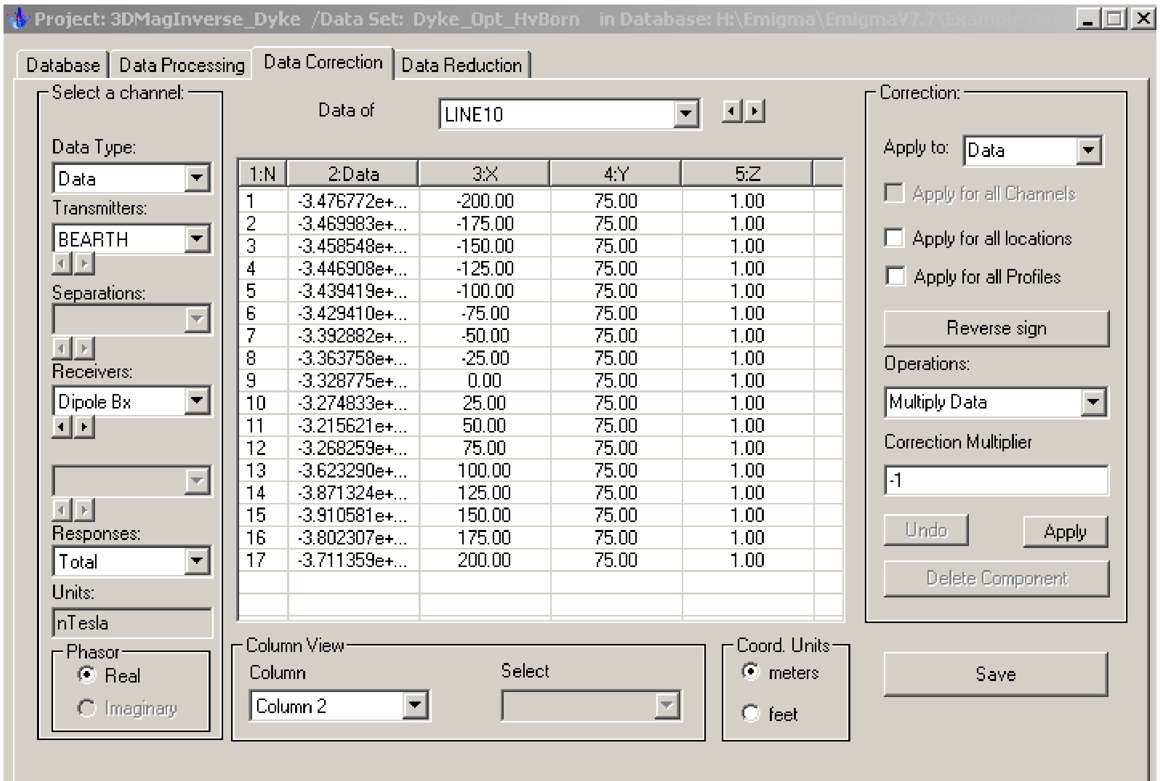
- Select the sorting index and click the Sort button.

To apply 1D Filters to the locations:

- Select either a digital or spatial filter type. Select the filter from the drop down list and click Apply.

DATA CORRECTION PAGE

The Data Correction Page allows you to edit the data values as well as the X, Y and Z values of the data set.



Column View defines the columns that are displayed

Select a channel:
Select the Data Type, Transmitter, Receiver, Time Channel or Frequency, Response and Phasor of the data to correct

Correction Apply to: You can chose to apply to some or
-all time channels
-all locations
-all profiles

Operations:
Select the operation from
-Multiply Data
-Shift Data
-New Value
-Set NODATA

Define the Correction Multiplier and click Apply

Reverse Sign: Can be applied at the touch of a button

Delete Component: A component can be deleted at the touch of a button

Once you are satisfied with your corrections, select **Save**

EMIGMA'S MAIN TOOLBAR

The database Design of EMIGMA V8 allows all tools to be fully integrated and to share/access the data in the database. Thus there is no need to open and save flat .pev files. There is also no need to save a Database .mdb file, once it has been created. EMIGMA continuously updates the .mdb file as changes are made.

The main toolbar can be divided into three parts: the database toolbar, the basic tools toolbar and the processing tools toolbar. Depending on your preference, you can move these toolbars to different places of the screen; for example, you can leave the first on top and move the second to the bottom, right or left, changing its position from horizontal to vertical and vice versa.

Database Toolbar

 - The **New Database** button allows you to save a new database

 - Click the **Open Database** button to open another database. Emigma will display the Open dialog box, in which you can locate and open the desired database

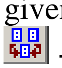
 - The **Tools** button provides a number of operations with your files and data (file cutting, column interpolation, Lat/Lon to UTM conversions, etc.)

 - The **Main Window** button is engaged when you open a database and the **Database** page of the main dialog appears. You can switch this dialog off only by disengaging this button. Otherwise it will always be either opened or minimized in the bottom left-hand corner of your screen

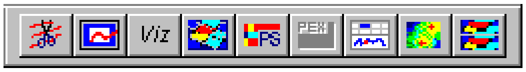
 - The **Refresh** button updates the tables of the database so the effects of deleting or adding data sets will be seen in the database project page. Usually the database is automatically updated after a change and this function is not needed

 - This button exports data sets to various formats, such as *.pev files for use with Emigma v6.4, other databases as well as to *.xyz columnar files. You only need to select the data set to be exported before this button is pressed in the case of the *.xyz and *.pev files

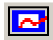
 - The **Import** button opens the **DBImport** dialog with a list of available imports. You are asked if you would like to create a new project. If you select no, a new data set will be added to the currently selected project. Otherwise, a new project will be created. A default name will be given to the project if one is not given

 - The **Merge Data Sets** button combines data from two different data sets together into one data set

Basic Tools Toolbar



 - Visually edits the profiles of the selected survey

 - Launches the plotter with the selected data set

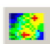
 - Launches the visualizer with the selected data set. You can see the data set's data and add a model using visual tools

 - Provides presentation of your grids

 - Pseudo-Section

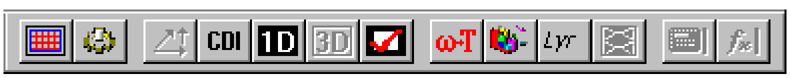
 - PEXShow tool provides 2D representation of geoelectric sections

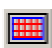
 - Data Corrector combines dynamic spreadsheets and line plotter for data cleaning, missing data interpolation and simultaneous plotting of different data channels for fast cross-analyses

 - Launches the contour application with the selected data set.

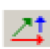
 - MultiGrid tool allows viewing and comparing up to 4 grids at a time

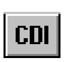
Processing Tools Toolbar



 Generates an interpolation of the selected data set on a regular grid to facilitate the viewing of the data in the contour application

 - Data Decimation and Filtering

 - Calculates derivatives along the three principal axes for the selected data set. Data set must be gridded where the number of data points along an edge is a power of two.

 - Calculates resistivity halfspace model for airborne data or calculate a sengpiel section (EikMap)

 - Provides 1D inversion for EM, Resistivity, MT/CSAMT data

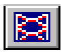
 - Generates a 3d susceptibility model from dc mag data in the selected data set that can be viewed in the visualizer and contour application

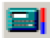
 - Calculates the simulated data for the selected model

 - Converts the frequency domain data in the selected data set to time domain data

 - Model Optimization

 - Calculates a background field and shows the resulting vectors in the visualizer. Only available for data sets with fixed transmitters

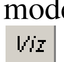
 - Calculates a tomogram of the selected data set. Available only for crosshole data sets

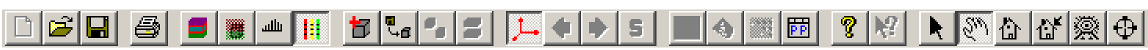
 - Calculates Magnetic Compensation Coefficient

 - Applies magnetic compensation

Please find below guides to some of the Interpretation tools. Those not covered by this manual are underway; do not hesitate to call us and ask about the updates.

3D Visualizer ()

You can build models in the Visualizer (**3D VisRD**) as in EMIGMA V6.4. Select the Data Set and choose the  icon. Your model will appear in the 3D graphical viewer. Simultaneously the VisRD toolbar appears under the main toolbar of EMIGMA:



You can access the **Property Pages** by clicking the  button on the Visualizer toolbar, or select the prism, or layer, right click and bring up the properties window. You can import layers, models, polyhedra and topography from datasets in your current database, or you can browse for a .pev file.

You can also build or modify models through **Configuration** on the **Database** Page of the main dialog. **Configuration** has the same functionality as the **Properties Pages** had in EMIGMA V6.4 VisRD.

All properties of the Survey, including the profile, frequency/waveform, transmitter/receiver, Prisms/Plates/Polyhedra, Layers and Output sections can be viewed and modified.

I**MPORTANT** - Once you have made your changes in Visualizer, be sure to **Save to Database** to update the database before running a simulation. If you do not save to database, your changes will not be updated.

We offer you a number of exercises below to get familiar with the ways of modifying, editing and manipulating objects in EMIGMA's Visualizer.

Exercise 1: Editing Layers and Prisms/Plates

TEM_Borehole Project in ExampleDatabase

To view the model, rotate it 90 degrees by doing one of the following:

- Rotate the dial bars in the bottom left hand corner of the window

or

- Select the **View Mode** button, click the model and, holding the button down, rotate the field of view.

To edit the layers:

- Click the **Property Pages** button on the top toolbar and go to the **Layers** page
- Select the 2nd layer from the **Layer List**
- In the **Layer Parameters** section, change the resistivity to 25 and thickness to 30
- Move to the **Mode** section, click **Insert Layer**

  The **Layer List** now displays 3 layers instead of the initial 2.
- Select the 3rd layer and in the **Layer Parameters** section, change the resistivity to 5000
- Move to the **Mode** section and click **Replace Layer**
- Click **Apply**, then **OK** in the lower right hand corner
- On the top toolbar, click the **Toggle Layers** button to turn the layers on and view the model
- Use the **Pick** button to switch from the manipulator hand to the selection arrow
- Click on a layer (the layer will be outlined in white), right-click and select Model/Layer/Properties from the popup menu to display a small dialog with the layer properties. Selecting other layers updates the properties in the dialog. You can change the properties of the layers as needed
- Use the **Toggle Layers** button to turn the layers off

To edit the plate:

- Click on the plate, right-click and select **Properties** from the popup menu to display the **Anomaly Properties** dialog
- In the **Geological Angles** section, change the dip to 45 degrees
- In the **Material Properties** section, change the conductivity to 200
- Click **Apply**, then **Close**.

Save as a new filename.

Viewing Data in VisRD (continuation of Exercise 1)

To plot the data:

- Open the **Data Selection** dialog by selecting **Show** on the **Data** menu or clicking the **Toggle Data** button on the top toolbar
- Select **Plot Settings** from the **Data** menu to open the **Data Settings** dialog. Change **Scaling** to **Constant** and **Appearance** to display the amplitude in the Y direction, click OK

- Toggle through the time channels using the yellow arrow buttons and on the top toolbar or scroll arrows under the **Time Channel** list in the **Data Selection** dialog
- As the data display scales to the first channel plotted, use the **Scaling** button to rescale when amplitudes are too small to display

With the **Data Selection** dialog displayed, you may need to shift the model into your field of view:

- Click the **View Mode** button on the top toolbar to switch to the hand manipulator
- Shift-click on the model and, holding the button down, move your model in a desired direction

  In order to see both the **Data Selection** dialog and the data display over the model, you can choose to move the model to the left and the **Data Selection** dialog to the right side of the window

Exercise 2: Splitting and Manipulating Prisms

In the last data set of the Italian_TEM Project in ExampleDatabase**.**

Change to the simple axis:

- Click the **Toggle Axis** button on the top toolbar to turn the grid axis off
- On the **Options** menu, select **Axis/Axis Simple/On**

To split the prism:

- Click the **Pick** button to switch back to the arrow manipulator
- Click on the prism, right-click and select **Split Prism** from the popup menu. The prism will be cut equally in its local X_Y plane
- Select a split plane of X_Z and a split ratio of 1:1. The cutting tool position will change respectively. Click **Split.**
- Click on the northern prism, right-click, and select **Split Prism** from the popup menu
- Select a split plane of Y_Z and a split ratio of 1:1, then click **OK**.

You now have 3 prisms: northeastern, northwestern and southern.

To change the position of the prisms:

- Click on the northeastern prism, right-click and select **Properties** from the popup menu.

  The **Anomaly Properties** dialog appears
- In the **Geological Angles** section, change **Strike** to -20 degrees
- In the **Center/Top** section, turn the **Center** option on, change the East position to -101 and the North position to 80, click **Apply**
- Click on the northwestern prism, right-click, select **Properties** from the popup menu to display the **Anomaly Properties** dialog

- In the **Geological Angles** section, change **Strike** to -10 degrees
- In the **Center/Top** section, with the **Center** button turned on, change the East position to –112. Click Apply, then Close

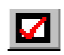
Both the northeastern and northwestern prisms have changed their position.

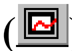
- Click the **Property Pages** button  on the top toolbar and go to the **Prisms/Plates/Polyhedra** page
- Click **Interactions** in the **Internal Current Sampling** section and select the **Near Field** button in the dialog to appear. This will change the type of interaction between all of the prisms to near field, i.e. electrical contact
- Select the first prism in the list of models, move to the **Internal Current Sampling** section and click **Advanced**.

  The **Advanced Prism Sampling** dialog appears

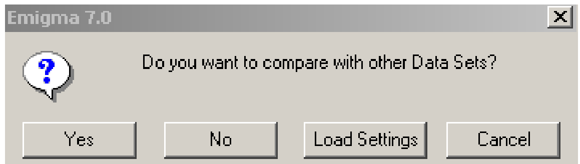
- In this dialog, change X, Y and Z to 2, 5 and 2, respectively, click **Apply**, then **OK**

Note: Only the number of points is saved in the file; special gridding, like the above one, is applied only when simulation is run before saving

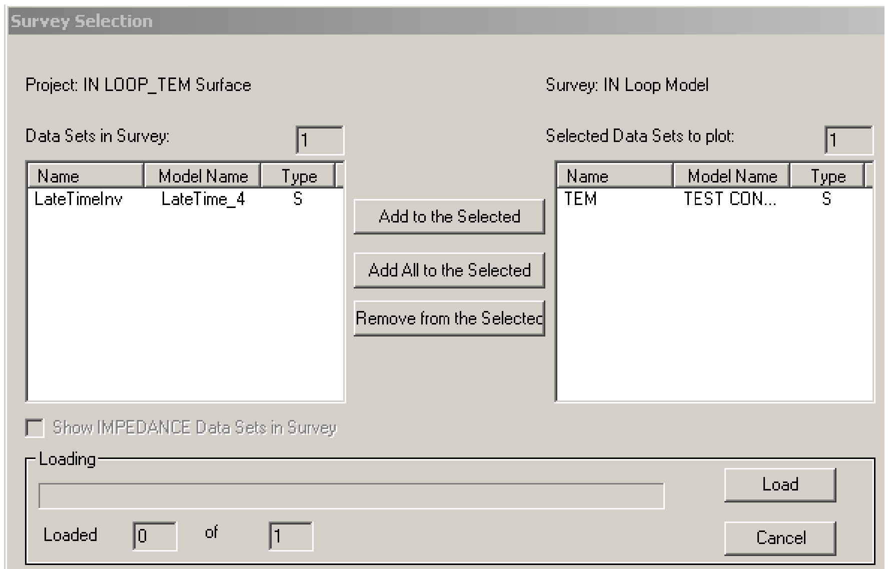
- Click **Apply**, then **OK** on the main page
- Now you can start simulation by clicking the  button on the top toolbar.

Plotting ()

To plot your data, click the EiKPlot button  on the main toolbar. If your survey contains several data sets, a message will appear asking you whether you want to compare the current data set with the other data sets in the survey:



1. If you click **Yes**, the **Survey Selection** dialog will open offering you to choose data sets to be compared with your current data set:



- Select the data set from the list on the left and click **Add to the Selected**. In case you want all the available data sets to be used for comparison, click **Add All to the Selected**
- To remove a data set from the list on the right, select it and click **Remove from the Selected**.
- Click **Load**

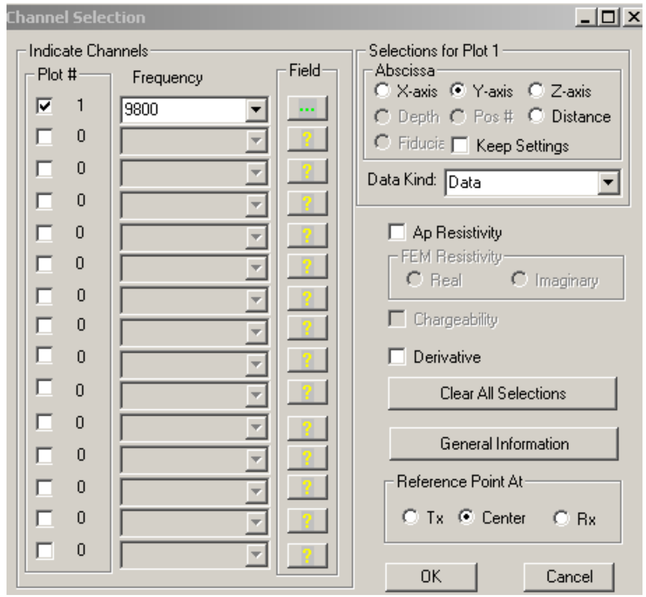
2. If you click **No**, the initially selected data set will be plotted automatically.
3. To load the settings of a previously created plot, click the **Load Settings** button to open the respective dialog and load the settings (see below)

Static, Frequency- and Time-Domain Systems

In the case of static, frequency- or time-domain systems, the first available channel of your data will be plotted automatically.

To edit plot settings

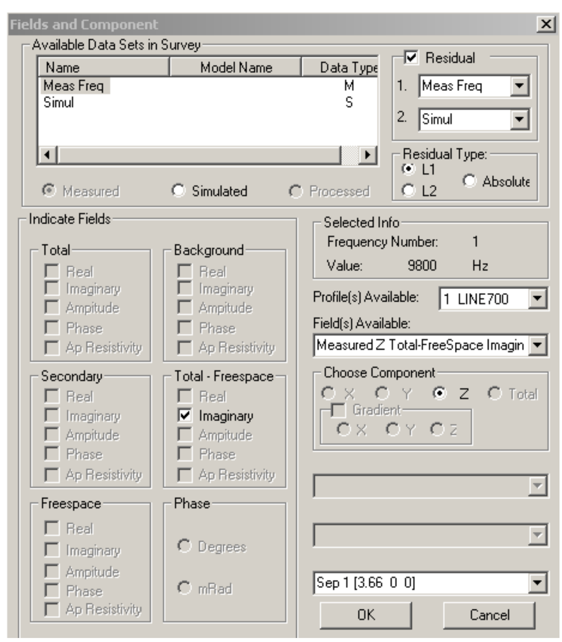
- Click the  button on the EiKPlot toolbar or double-click anywhere in your plot. The **Channel Selection** dialog will open:



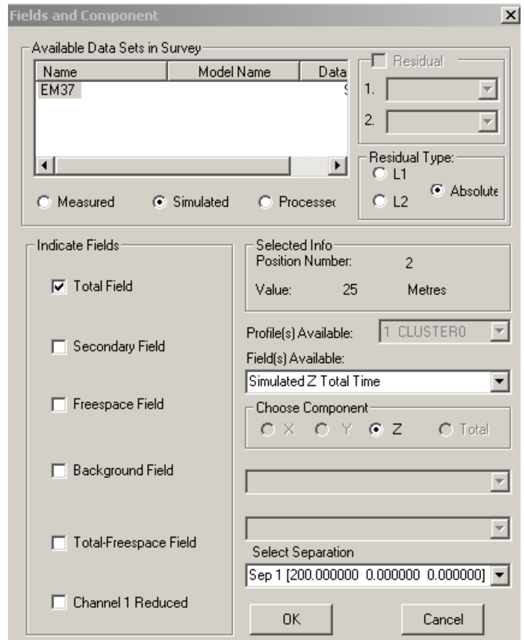
- Select the channel you want to display from the **Frequency** (**Time (msec)** in the case of time-domain data or **Static** in the case of DC Magnetic or Gravity) dropdown list and click the **Field** button  to specify the fields and components

If you want several channels to be displayed on the same plot, select as many **Plot #** boxes as you need. For example, if two channels are to be plotted at a time, check the second **Plot #** box and select the second channel from the respective dropdown list.

In the case of static or frequency-domain systems, the **Fields and Component** dialog to appear will be as follows:



If your data is time-domain, the **Fields and Components** dialog will differ only in the **Indicate Fields** section:



In both dialogs, **t**he **Available Data Sets in Survey** section shows the name of the data sets and models you have loaded and the type of data (Measured, Simulated, Processed) in these data sets. You can see the number of channels you selected and their value in the **Selected Info** section.

- Select the data set(s) to plot if you have loaded several data sets
- Check the fields and phases (for frequency-domain and static) to plot in the **Indicate Fields** section
- Select the X, Y, Z or **Total** component in the **Choose Component** section if your receivers are dipoles
- Select the bipole from the respective dropdown list to appear in the bottom right-hand corner of the dialog if your receivers are bipoles. In this case, the **Choose Component** section will be disabled

You can also select the field and component from the **Fields Available** dropdown list on the right. This automatically checks the required field and selects the respective component/bipole

Note. *Checking more than one field will display the respective number of responses on the same plot*

- Select a transmitter from the **Select Transmitter** dropdown list, which becomes active when multiple transmitters are used
- Select a separation to be used by the plotter from the **Select Separation** dropdown list, which becomes active in the case of a moving transmitter survey

- Provided you have both measured and simulated data, check the **Residual** box in the respective section in the upper right-hand corner of the dialog. The two dropdown lists below will be enabled. Select the data sets from these lists. In the **Residual Type** section, select the algorithm to be used for your data recalculation
- In case total derivatives have been measured or modeled (in magnetics and gravity surveys), select **Total** to enable the **Gradient** X, Y and Z buttons. Choose the gradient you want to plot
- Click **OK** to return to the **Channel Selection** dialog

In the **Channel Selection** dialog, you can specify to plot the Apparent Resistivity response:

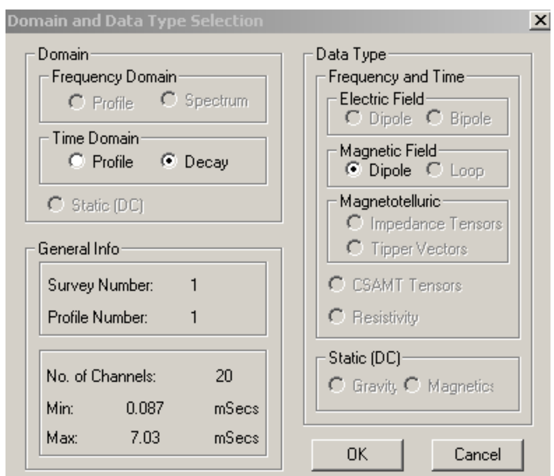
- Check the **Ap Resistivity** box. This will activate the **FEM/TEM Resistivity** section. Select between **Real** and **Imaginary** for FEM and **Late-T** and **All-Time** for TEM
- Check the **Derivative** box to plot the derivative of the data selected. The derivative is calculated by a simple inline finite difference
- Click **OK** in the **Channel Selection** dialog to view the plot.

Note*. The warning "Curves containing non-data only are not plotted" means that your selection cannot be plotted, since it requests data that are not available.*

To switch between the profile and spectrum/decay display

- Select **Configure/Domain** or click the **Domain** button  on the EiKPlot toolbar.

 The **Domain and Data Type Selection** dialog opens:



- In the **Domain** section of the dialog, select the **Spectrum** button if your data are frequency-domain or the **Decay** button if they are time-domain
- Click **OK**. The **Channel Selection** dialog appears, with the first profile position selected by default
- Click the **Field** button  to the right of the selected position to display the **Fields and Component** dialog

- Check or edit the previously selected settings and click **OK** to return to the **Channel Selection** dialog
- Click **OK** to close the dialog and view the **Spectrum/Decay** response

Note. Use the **Next** and **Previous Channel or Position** buttons (and) on the EiKPlot toolbar to toggle forward and back through the available profile locations

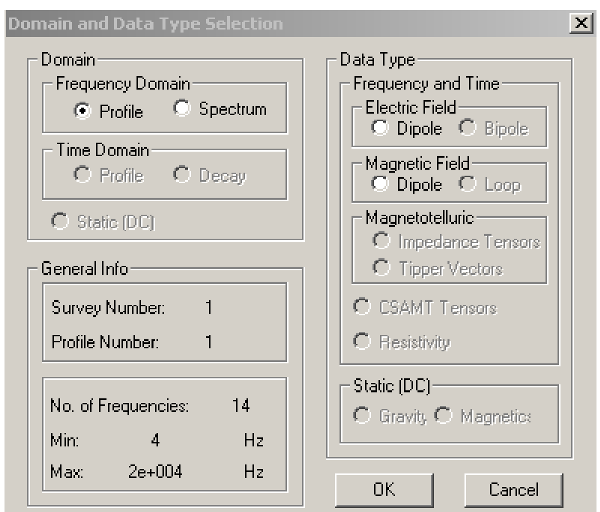
Use the **Next** and **Previous Profile** buttons (and) to toggle forward and back through the available profiles

- To switch back to the profile display, click the **Domain** button on the EiKPlot toolbar again, select the **Profile** button in the **Domain** section of the **Domain and Data Type Selection** dialog and repeat all the further steps as described in this section.

MT and CSAMT Systems

If your MT or CSAMT data contain impedance information, the plot will be generated automatically. However you can always view your initial data:

- Click the **Survey** button on the EiKPlot toolbar. The **Survey Selection** (see above) dialog will appear
- De-select the **Show IMPEDANCE Data Sets in Survey** box below the list of data sets on the left. All initial EM data sets will appear in the list of data sets
- Select a data set (or both data sets) to plot and click **Add to the Selected**. The **Domain and Data Type Selection** dialog will appear offering you to select between the electric and magnetic fields:

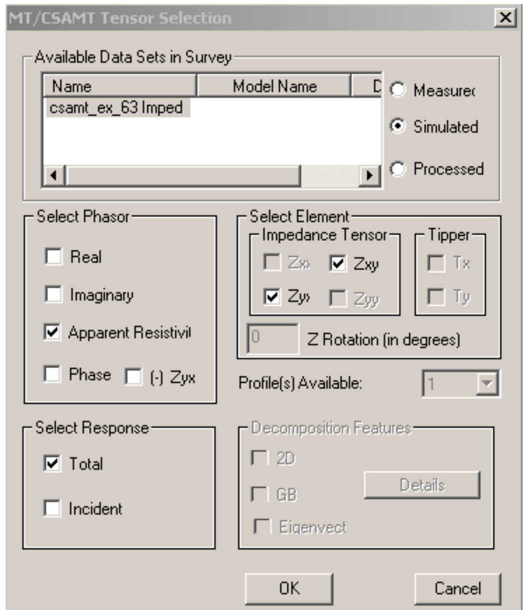


If your data set contains no impedance information, the **Domain and Data Type Selection** dialog will open prior to plot generation.

To edit plot settings

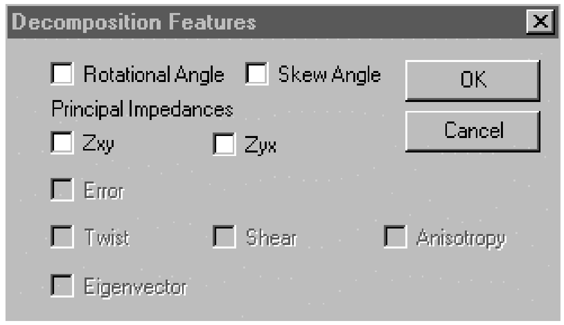
- Click the  button on the EiKPlot toolbar or double-click anywhere in your plot. The **Channel Selection** dialog will appear. Select the channel you want to plot from the **Frequency** dropdown list and click the **Field** button 

  This will bring up the **MT/CSAMT Tensor Selection** dialog:



In this dialog, the **Available Data Sets in Survey** section shows the name of the data sets and models you are currently in and the type of data in these data sets. You can see the number of channels you selected and their value in the **Selected Info** section.

- Select a required phasor in the respective section. Checking the **Real** and **Imaginary** phasors will display them both on the same plot. Selecting **Apparent Resistivity** or **Phase** will cancel all other selections
- Check the **(-) Zyx** box to de-select the respective element in the **Impedance Tensor** section
- Check a required response(s) in the **Select Response** section
- Select either one or more elements in the **Impedance Tensor** section, which is active by default
- Specify the rotation angle in the **Z Rotation (in degrees**) field. This option is active when all the **Impedance Tensor** elements are available
- Click in the **2D** box in the **Decomposition Features** section to bring up the respective dialog:

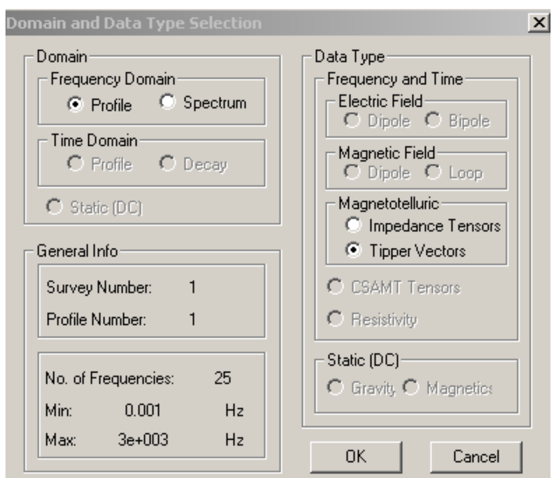


- Select to calculate and plot **Rotational Angle** and **Skew Angle** or the principal impedances

Note. *The* ***Decomposition Features*** *section is yet underway. As of now, only a standard Swift 2D decomposition is provided.*

- In MT, to switch from **Impedance Tensor** to **Tipper** elements:
    - o Click **OK** in both the **MT/CSAMT Tensor Selection** and **Channel Selection** dialogs to close them
    - o Select **Configure/Domain** or click the **Domain** button  on the EiKPlot toolbar

      The **Domain and Data Type Selection** dialog opens:



    - o In the **Magnetotelluric** section of the dialog, select the **Tipper Vectors** button and click **OK**. The **Channel Selection** dialog reappears
    - o Click the **Field** button  to the right of the selected channel to reopen the **MT/CSAMT Tensor Selection** dialog

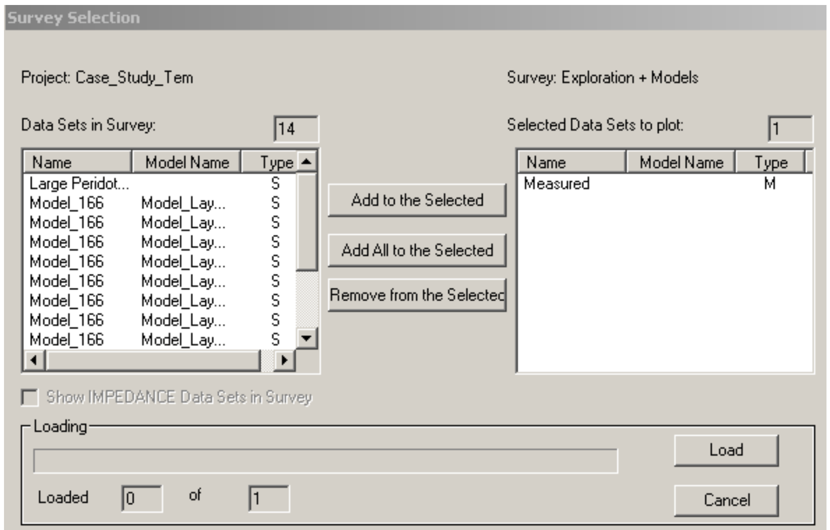
      You now have the **Tipper** section activated, whereas the **Impedance Tensor** boxes are disabled. The **Z Rotation** box will be disabled as well, however it may contain an angle setting if one was specified earlier
    - o Select the **Tipper** vector you want to plot

- Click **OK** to return to the **Channel Selection** dialog
- Click **OK** in the **Channel Selection** dialog to view the plot.

Loading Additional Data Set(s)

If you have more than one data set in your survey, but you did not load them (e.g., for comparison purposes) at the very beginning, you can always do it from your plot display:

- Select **Configure/Survey** or click the **Survey** button on the EiKPlot toolbar. The **Survey Selection** dialog appears:



- Select the data set(s) from the list on the left and click **Add to the Selected**. The data sets selected will move to the list on the right
- Click **Load**. The **Channel Selection** dialog appears
- Select the channel(s) and click **Field** ( )
- In the **Fields and Component** dialog to appear, select the data set from the respective list (now containing the data set(s) you added) in the upper left-hand corner of the dialog and specify fields and components as described above
- Click **OK** in both **Fields and Components** and **Channel Selection** dialogs to view the plot

Note. To plot several data sets at a time, check as many **Plot #** boxes in the **Channel Selection** dialog as the number of data sets you want to display. Select the channels from the activated **Frequency/Time/Static** dropdown lists. Click **Field** across each activated list and specify in each case the data set, field and components in the **Fields and Components** dialog to appear.

Viewing Plots

Your data having been plotted, you get an easy access to different view options, such as switching between profiles, channels, separations transmitters and receivers, customizing plot appearance, viewing model properties. All this is available from the EiKPlot menu and its toolbar offering a wide range of buttons. Or, you can also use the hot keys popping up when you

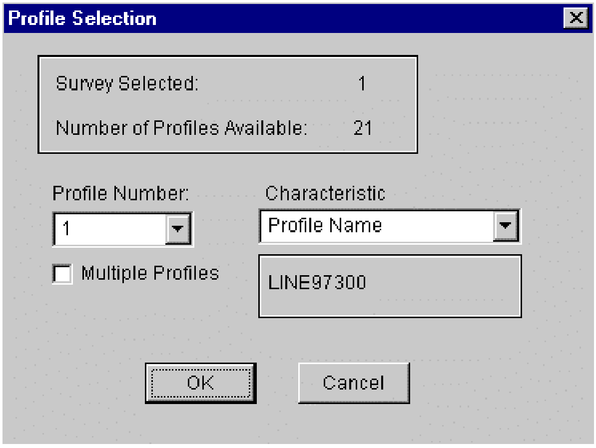
hold your mouse cursor over the toolbar buttons.
To view the coordinates of a plot point, click it and hold the button down. You will see the X- and Y-axis values displayed over the cursor.

To switch between profiles and display multiple profiles

If you have more than one profile, you may need to switch from one profile to another or view multiple profiles at a time. Your first plotted profile is 1 by default.

To switch to another profile:

- Select **Configure/Profile** or click the **Profile** button on the EiKPlot toolbar. The **Profile Selection** dialog opens:



- Select another profile from the **Profile Number** dropdown list. In the **Characteristic** dropdown list on the right, you can select the profile name or the profile (starting) X, Y and Z coordinates to view the respective information in the frame below

  Note. You can also switch to another profile right from the EiKPlot toolbar. Click the **Next Profile** and **Previous Profile** buttons to toggle forward and back through all available profiles.

To view multiple profiles on the same plot:

- Select **Configure/Profile** or click the **Profile** button on the EiKPlot toolbar. The **Profile Selection** dialog opens (see above)
- Select the **Multiple Profiles** box and click **OK** to close the dialog
- Double-click anywhere in the plot or click the **Channels** button on the EiKPlot toolbar to open the **Channel Selection** dialog
- In this dialog, check the next **Plot #** box to activate the **Frequency/Time/Static** dropdown list and select the channel
- Click the **Field** button to display the **Fields and Components** or **MT/CSAMT Tensor Selection** dialog. You now see the **Profile(s) Available** dropdown list enabled
- Select the number of the profile you want to add to your plot from this list
- Click **OK** to return to the **Channel Selection** dialog

- Click **OK** to close the dialog and view the plots.

Repeat all the steps above but the first two to display as many profiles as needed on the same plot.

Note. To switch to another set of channels or profiles, use respectively the **Previous** and **Next Channel or Position** buttons (and) or the **Previous** and **Next Profile** buttons (and) on the EiKPlot toolbar.

To switch between channels, separations and transmitters

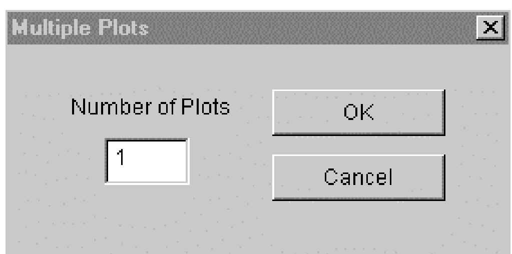
- To switch between available channels, click the **Previous Channel or Position** or **Next Channel or Position** buttons on the EiKPlot toolbar.
- To switch between separations use the **Next** and **Previous Separation** buttons (and) on the EiKPlot toolbar.
- To switch between transmitters, use the **Next** and **Previous Transmitter** buttons (and) on the EiKPlot toolbar.

To view multiple plots at a time

You may need to view multiple plots at a time. It is especially convenient when you deal with responses that cannot be displayed on the same plot.

- Select **Settings/Custom/Number of Plots** from the menu

  The **Multiple Plots** dialog appears:



- Type the number of plots you want to display and click **OK**

  The window will be divided into the respective number of plots. The first plot will be the one you are currently in.
- If you want to check or edit the settings of your current plot, double-click anywhere in it to display the **Channel Selection** dialog. Make necessary changes as described in
- Repeat this operation for all the other plots

To switch back to the single-plot display:

- Select **Settings/Custom/Number of Plots** from the menu to bring up the **Multiple Plots** dialog
- Change the number of plots to 1 and click **OK**

PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

This will switch you back to the full-screen single-plot view again.

To view model properties

To view model properties without leaving the plotter, click the **Model** button on the EiKPlot toolbar. In the **Model Description** dialog to appear, you will find:

- Location details

  If you have a number of bodies, select the one of interest from the **Anomalies** to see its parameters

- Information on the conductivity, susceptibility, Euler angles, scale sizes, sample points and interactions of your model

- Resistivity and thickness of available layers.

  To view the properties of a layer, select it from the **Layers** list.

If you have loaded several data sets, you can switch between your models using the **Previous** and **Next** Model buttons on the EiKPlot toolbar.

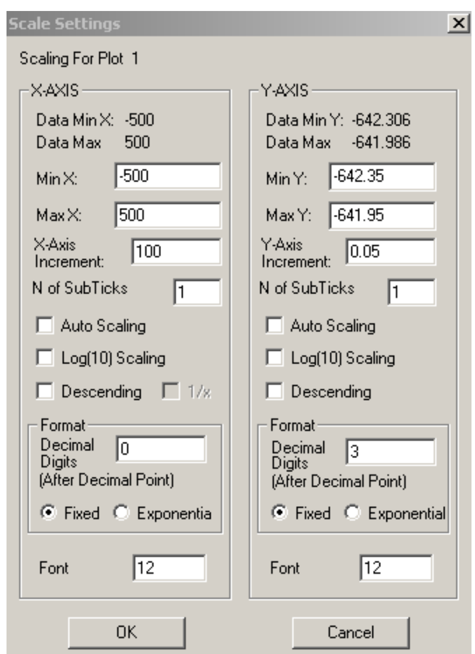
To adjust the scale

EiKPlot allows you to customize your plot scale and to zoom in on a specific fragment of your plot.

To adjust the scale settings:

- Select **Settings/Scaling** or click the **Scaling** button on the EiKPlot toolbar

  The **Scale Settings** dialog appears:



The minimum and maximum X- and Y-axis settings and the X- and Y-axis increments are generated automatically from your input data

- To change these settings, type your values in the respective fields

  To return to initial scaling, select the **Auto Scaling** checkbox. You can also do it later, right from the EiKPlot menu or toolbar. Select **Settings/To Initial Scale** or simply click the **Rescale** button .

  Change to the descending scale by selecting the respective checkbox. De-select to change back

- To customize the appearance of the axis labels, change the number of digits to be displayed after the decimal point, adjust units (fixed or exponential) and set a required font size in the respective fields

- To change to the logarithm scale, select the **Log(10) Scaling** checkbox.

  Note. If your data contain negative or zero values, you will see a warning message indicating negatives plotted as positives and zeroes not plotted. Click **OK**

- Click **OK** in the **Scale Setting** dialog to close the dialog and return to your plot.

To zoom in on a fragment of your plot:

- Select **Settings/Zoom** or click the **Zoom** button on the EiKPlot toolbar

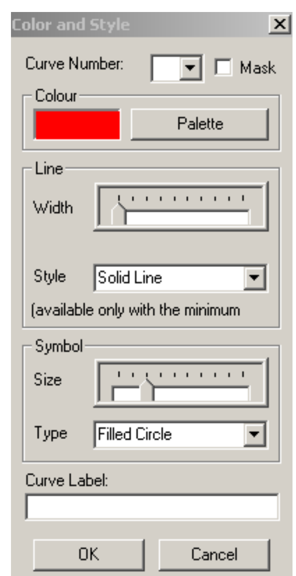
- Click in the area of your plot where you want the fragment to start and, without releasing the button, drag right/left and up/down until the required fragment is selected (outlined in green). Release the button

- To zoom in further on, repeat the operation.

  The two buttons on the EiKPlot toolbar, **Zoom Back** and **Zoom Forth** become active

- Use these buttons to toggle through all the fragments you have zoomed in

- Click the **Rescale** button to zoom out and return to the initial scaling.



<u>To switch grid on and off</u>

To toggle the grid on and off, select or de-select **Settings/Custom/Grid** or click the **Set Grid** button on the EiKPlot toolbar.

To cancel the grid, you can also use the **Default** command from the **Settings** menu.

<u>To customize plot appearance</u>

EiKPlot offers you a whole set of tools for customizing your plot appearance. You can change the color, style and label of the curve, adjust the appearance of the symbols, edit and move the axis labels, mask any of the plots.
To change the color and style of the curve:

- Select **Settings/Colour and Style**

OR

- Double-click the curve label (top left corner). The **Colour and Style** dialog appears

In the **Colour** section:

- Click the **Palette** button to open the standard palette of basic and custom colors. Change or add new colors and hues

In the **Line** section:

- Define the width of the line using the slider

  This option is applicable only to the **Solid Line** style. Other styles do not show on the plot unless the minimum line width has been selected

- Select the style of your curve from the respective dropdown list

In the **Symbol** section:

- Use the slider to increase or decrease the size of used symbols
- Select the shape you want to apply from the **Type** dropdown list.

Note. To reset all your color and style changes, select **Settings/Default**.

To change the curve label:

- Double-click on the curve label to change

  The **Curve Label** field of the **Color and Style** dialog to open will contain this label
- Change it as desired and click **OK**
- To change back to the former label, open the **Color and Style** dialog again and delete the new curve label from the respective field. Click **OK**. The former label will reappear

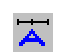
To mask a curve:

- Double-click on the label of the curve to mask
- Check the **Mask** box in the **Color and Style** dialog to open and click **OK**

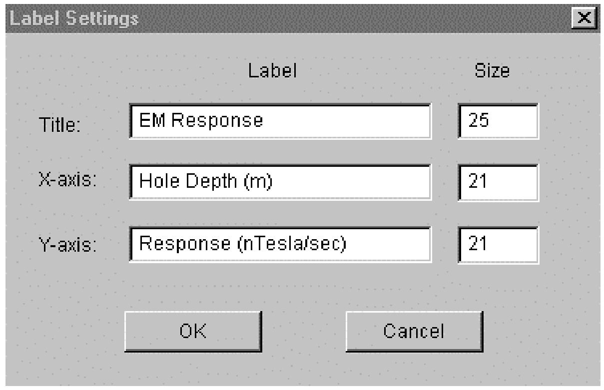
  The curve and its label will become gray

- To bring the masked curve back, de-select the **Mask** box

To adjust the axis labels and their font size:

- Double-click on the axis label to change (or select **Settings/Labels** or click the  button on the EiKPlot toolbar)

The **Label Settings** dialog appears:



- Make your adjustments and click **OK** to close the window and view the changes.

Note. To change the position of an axis label on the plot, simply click and drag the label wherever you want.

Converting to Various Displays

EiKPlot allows you to convert plots to various displays:

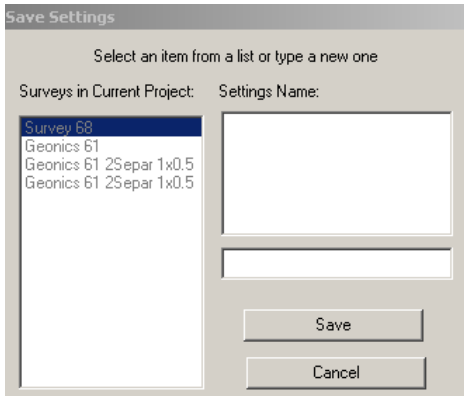
- Select **Settings/Custom/Flip Sim Data** to multiply all simulated data by –1 and thus to flip your plot. De-select this command to switch it back to the initial view
- Select **Settings/Custom/Sim Quad Conv** to flip the simulated quadrature data only
- Select **Settings/Custom/Crone (X,Y)⇐PetRos EiKon (-Y,X**), if you are the user of Crone systems and need to convert coordinates between Crone and PetRos EiKon formats
- Select **Settings/Custom/Smooth Meas Data** to process your measured data and make the plot less jagged
- Select **Settings/Custom/App Conductivity** to switch between the apparent resistivity and apparent conductivity displays.

Saving Plot Settings

This option allows for the rapid plotting of a number of models. For example, you could run a suite of models in batch mode, adjusting your layered earth, target positions, conductivity or size. Plot the first model and save the default settings as a .plt file. Then proceed to the next model and simply open the .plt file. The graph will be plotted for you automatically.

To save the settings of your current plot(s) as default:

- Select **Default/Save As** or click the **Save Settings** button  on the EiKPlot toolbar. The **Save Settings** dialog appears



- Select the survey you want to save the settings file in
- In the field below the **Settings Name** box, type the name of your new .plt filename and click **Save**.

To open a default setting file:

- Select **Defaults/Get** or click the **Get Settings** button on the EiKPlot toolbar. The **Get Settings** dialog appears
- In the **Surveys in Current Project** list, choose the survey your settings file is in
- Select the **Settings Name** and click **Load**.

To save changes to an existing setting file

- Select **Defaults/Save** or click the **Save Settings** button on the EikPlot toolbar
- In the **Save Settings** dialog, click **Save**, if you want to save the file under the same name, or type a new name if you want to save it as a separate file.

Printing Plots

EiKPlot offers two output modes: Auto (Full-Screen) and Scaled Graphic. The former automatically makes your printed plot look as you see it on the screen. The latter enables you to change the scale of the plot to be printed and to add an information box.

Note. To print your plot in color, select **File/With Colour** or turn the **Colour Print** button on before setting print properties.

<u>Auto (Full-Screen) Mode</u>

To print the plot as you see it on the screen:

- Select **File/Print** to display the **Print** dialog
- In this dialog, specify the printer, the print range and the number of copies in the respective fields
- Click **Properties** to specify the document properties in the dialog to appear and click **OK**
- Click **OK** in the **Print** dialog to start printing

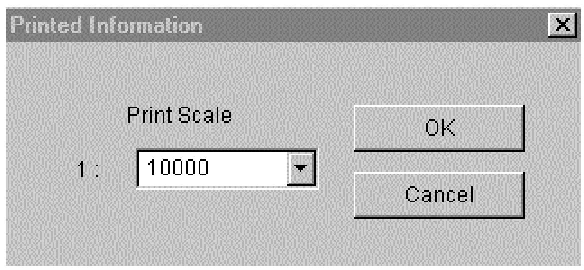
To preview your plot in the Auto (Full-Screen) mode:

- Select **File/Print Preview** (or **File/Output Settings and Preview/Auto**) or click the **Auto Preview** button  on the EiKPlot toolbar
- Use the **Next Page** and **Previous Page** buttons on the **Print Preview** toolbar to toggle through available pages
- Click the **Two Pages** button to preview two pages at a time, click it again to switch back to the **One Page** mode
- Click the **Zoom In** button to take a closer look at your plot and the **Zoom Out** button to move it away
- To close the **Preview** mode, click **Close**
- To print the plot, click **Print**. In the **Print** dialog to open, specify print properties and click **OK**.

Scaled Graphic Mode

To print your plot in the **Scaled Graphic** mode:

- Select **File/Output Settings and Preview/Scaled Graphic** or click the **Scaled Graphic** button  on the EikPlot toolbar. The **Printed Information** dialog appears:

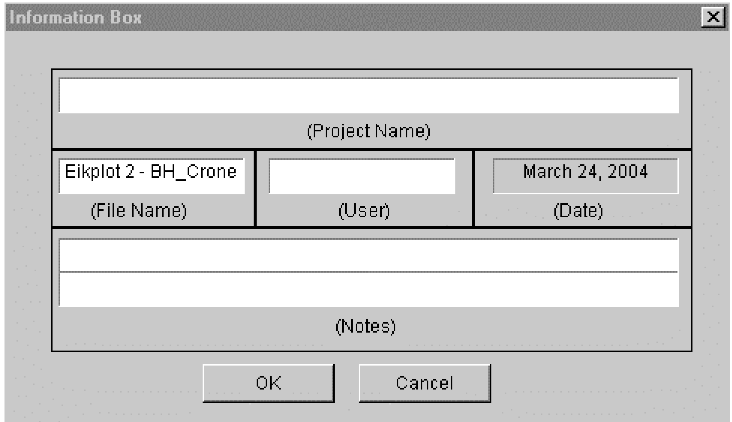


- Select the print scale from the respective list and click **OK**. This will display the **Print Preview** mode, with your plot changed to the scale you selected

  Note. The **Print Preview** toolbar in the **Scaled Graphic** mode offers the same options as in the **Auto (Full-Screen)** mode.

The **Scaled Graphic** mode also allows you to add an information box to your plot:

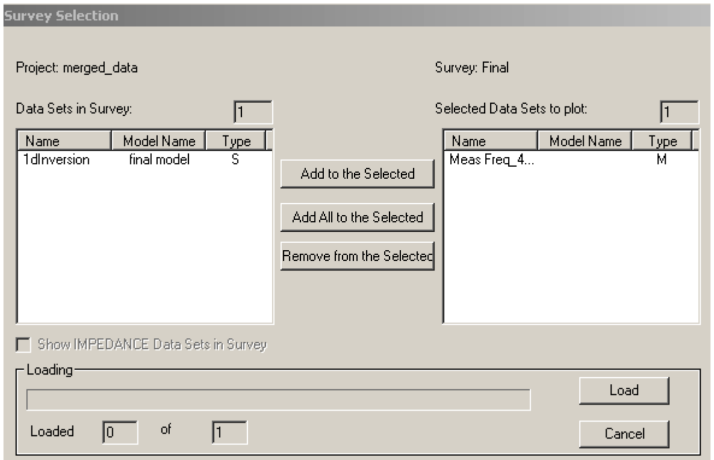
- Click the **Print Information Box** button  on the EiKPlot toolbar. The respective dialog appears:



- Fill in the project name and user in the respective fields. The filename and the date are generated automatically
- Write your comments in the **Notes** field
- Click **OK**. The information box will be printed in the upper right-hand corner of the page.

Contouring ()

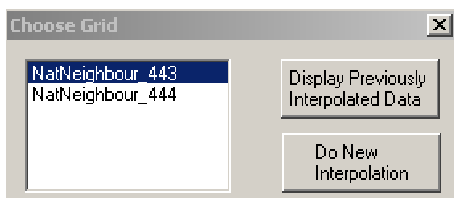
Click the **EM Contour** button on the main toolbar of EMIGMA. The **Survey Selection** dialog will open:



If your survey contains only one data set, the latter will be loaded automatically; if it has two or more data sets:

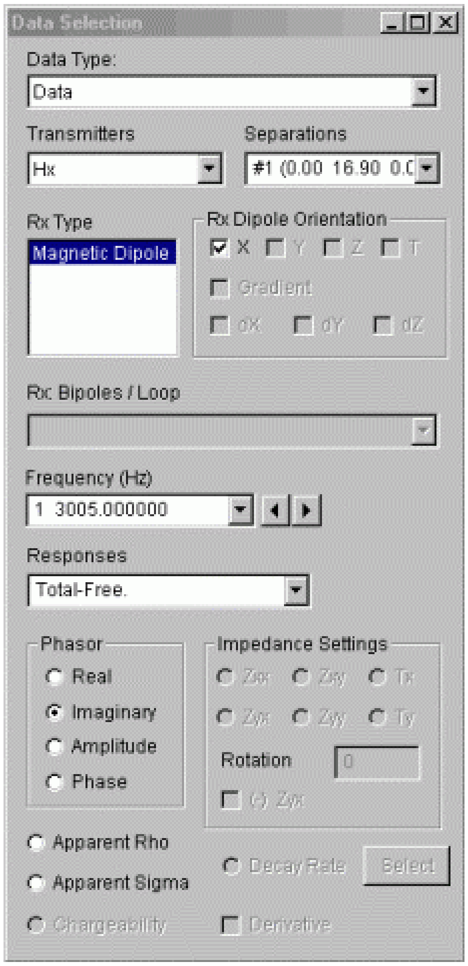
- To view only the current data set, click **Load**
- To compare your current data set with some other data set in the same survey, select this other data set in the left table, click **Add to the Selected** and **Load**
- To compare your current data set with all the data sets available in your survey, click **Add All to the Selected** and **Load**

Your data needs to be already interpolated, the **Choose Grid** dialog will open offering you to select the grid to display:



Working with Contour Display

Your contour displayed, the EM Contour toolbar will appear below the main EMIGMA toolbar:



To the right of your contour display, you will see the **Data Selection** dialog that opens automatically at the same time with contour generation.

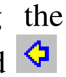
Note**. To close the **Data Selection** dialog, disengage the **Select Data  *button on the EM Contour toolbar.*

In the **Data Selection** dialog:

Active fields will show default selections. E.g., if there is a choice of channels, the channel selected by default will be the first one

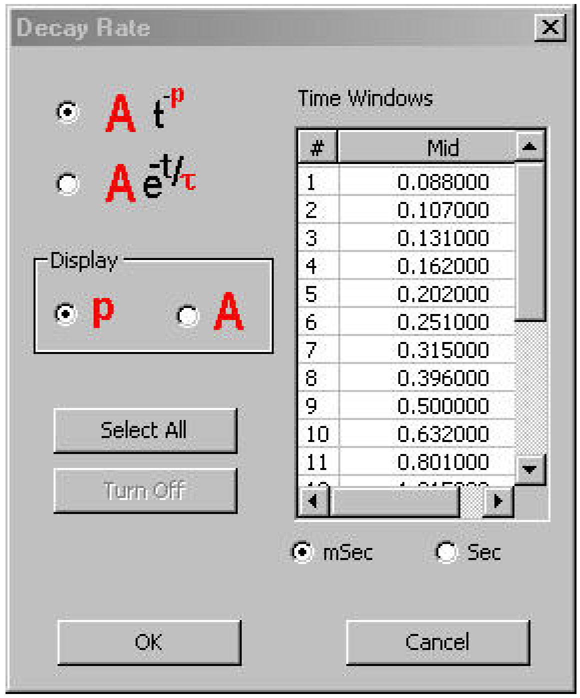
Adjust the selections as needed, viewing simultaneously the changes in your contour display:

- To switch to a different transmitter, separation, receiver or response, select it from the respective dropdown list
- To switch to a different channel (frequency, time), select it from the respective dropdown list or use the scroll buttons below

Note. You can also toggle through available channels right from the EM Contour toolbar by using the **Change Channel Forward** and **Back** buttons  and 

- Dependently of your system, make required selections in the **Dipole Orientation**, **Gradient**, **Phasor** or **MT Settings** sections
- Select **Apparent Rho**, **Apparent Sigma** or **Chargeability** to flip to respective response
- The **Static (DC)** section will be active if your data are Resistivity, Magnetic or Gravity

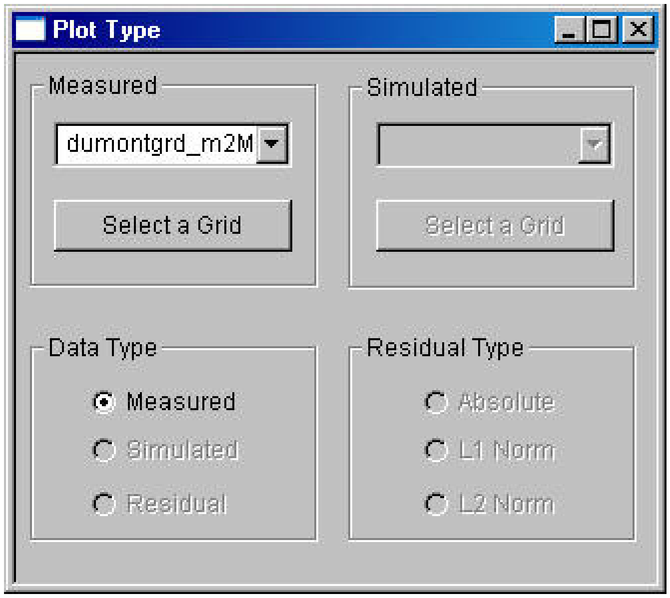
- Click the **Decay Rate** button to display a contour of decay rates (for time-domain systems only). In the dialog to open:



  - Select between the two algorithms and display options on the left
  - Select the range of decay windows on the right to be used in the decay rate calculation. To select all windows, click the **Select All** button. To cancel selections, click **Select None**

    Note. *Only multiple selections are applicable.*
  - Click **OK** to close the dialog and view the **Decay Rate** contour display.

By default, EM Contour generates a plane (2D) display of your first channel or separation. Other grids that you have loaded are available by maximizing the **Plot Type** dialog (in the bottom left-hand corner of the screen):



In this dialog:

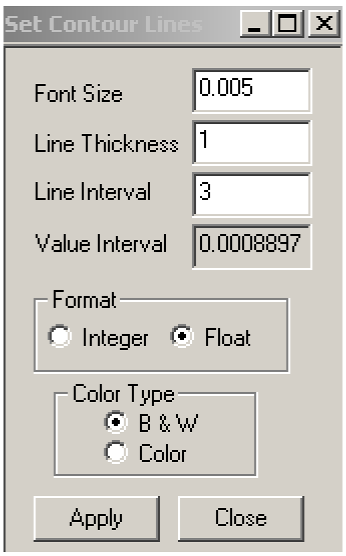
1. In case there are a number of grids in the same data set, choose the one you want from the appropriate drop down box or click Select a Grid and the **Choose Grid** dialog will appear (see above) offering you to select a required grid
2. In the **Data Type** section, you can view the type of loaded data sets:
   - If there is either measured or simulated data, then respectively the **Measured** or **Simulated** button will be turned on, whereas all the rest options will be disabled
   - If there is both measured and simulated data, you will have the third option - **Residual** – enabled. In this case, the **Residual Type** section becomes accessible. Select the algorithm to be used for your data recalculation

2D and 3D display

There are two modes of contour display: **Plane** (2D) and **Examiner** (3D). To switch to one or the other, select **View/Plane** or **View/Examiner**.

Displaying Contour Lines and Intervals

To display contour lines, click the **Toggle Contour Lines** button on the EM Contour toolbar. The contour lines will be generated automatically.



You can edit their appearance in the **Set Contour Lines** dialog to open simultaneously in the bottom left corner of the window:

- Type an integer interval between the lines and an integer value of their thickness in the respective fields
- Select between the integer and float formats of contour values in the **Format** section and indicate their font size in the **Font Size** box
- Select between the black-and-white and color representation of contour lines in the **Color Type** section
- Click **Apply** to view the changes and **Close** to close the dialog.

To toggle between the black-and-white and color display of your contour lines, use the Grayed/Color View button .

Moving Your Plot. Zooming In and Out

To move you plot in the **Plane** view:

- Switch to the hand manipulator
- Ctrl- or Shift-click the contour display and, without releasing the button, drag it right/left and up/down.

To zoom in or out on your plot:

- Click anywhere in your contour display and, without releasing the button, move your mouse down to zoom in and up to zoom out

Note. You can also use the Zoom dial bar in the bottom right-hand corner of the window

To move your plot in the **Examiner** view:

- Switch to the hand manipulator. This will allow you to examine your contour display from various sides.
- Click the **Set Home** button  on the EM Contour toolbar to define the "home" position of your display, which is the position you can always switch back to from any other positions
- Click on the plot and, without releasing the button, rotate it in any direction. You can also rotate it relative to the X- or Y-axis by using the dial bars in the bottom left-hand corner of the window
- Press the Shift or Ctrl keys to move the plot up/down and right/left

To make the plot rotate by itself:

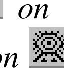
- Click on the plot and, holding the button down, move your mouse in the desired direction of rotation
- Release the button while moving the mouse. The plot will continue rotating
- To stop it, click anywhere in the screen.

To zoom in on your plot, use the **Zoom** dial bar.

To zoom in on a certain area of your contour display:

- Click the Seek button on the EM Contour toolbar. The message will prompt you to press 's' on the keyboard to seek
- Click OK and press 's' on the keyboard. The hand will change into the crosshairs cursor
- Click the crosshairs cursor over the area you want to examine more closely

  The display will move nearer, with the area you clicked shifted to the center of your view.

***Notes**. To return your display to the home position, click the Home button*  *on the EM Contour toolbar. To bring everything into your field of view, click the **View All** button* .

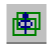
Color and Black-and-White Representation

To switch between the color and black-and-white contour display, click the Grayed/Color View button  on the EM Contour toolbar.

To change the color of the background, select **View/Edit Background Color.** This will bring up the standard color palette.

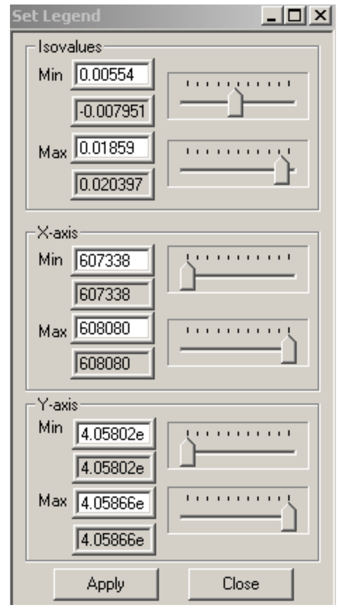
Viewing Profiles and Transmitters

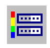
Use the **Toggle Profile** button  on the EM Contour toolbar to switch the profile display on and off.

Use the **Toggle Transmitter** button  to switch the transmitter display on and off.

Adjusting the Range of Data

To adjust the range of data to be displayed:

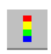


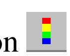
- Click the **Adjust Legend** button  on the EM Contour toolbar.

The **Set Legend** dialog appears. In the **Isovalues** section of this dialog, you can see the maximum and minimum values of your data to be used as cutoffs. By default, this range covers 97.5% of data

- If required, change the maximum and minimum isovalues. You can do it by typing your values in the active **Min** and **Max** boxes or by using the slider on the right. The disabled boxes below show respectively absolute minima and maxima of your data
- Click **Apply**. The plot will change accordingly
- To change back to initial settings, click the **Reset Scaling** button  on the EM Contour toolbar.

Legend

- Click the **Toggle Legend** button  on the EM Contour toolbar to display the legend in the frame on the left of your plot. The range of contour values and colors is generated automatically.

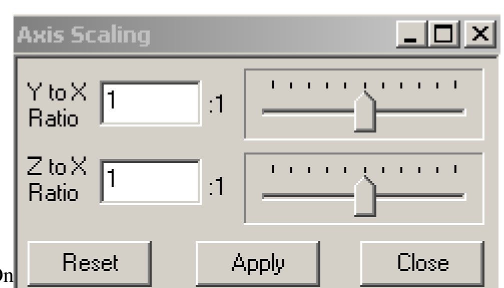
  If you have made changes in the **Isovalues** section, this will show in your legend
- To move the legend to a different place on the screen, click and drag it with your hand manipulator
- To switch the legend off, click the **Toggle Legend** button  again.

Axes

The axes are on by default. To turn them off, click the **Toggle Axis** button  on the EM Contour toolbar. Click it again to bring the axes back.

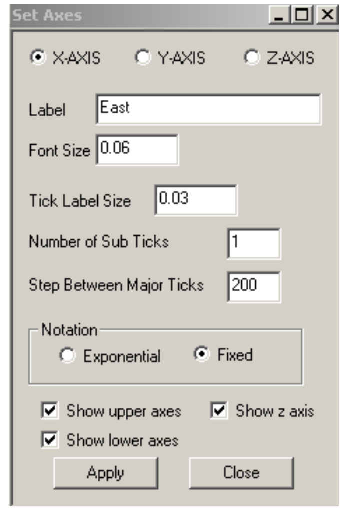
To change the length of your axes:

- Maximize the **Axis Scaling** dialog that appears in the bottom left-hand corner of the screen simultaneously with contour generation



- Type in the required values of Y to X and Z to X ratios (or use the slider to the right) and click **Apply**
- To change back to the initial value, click **Reset**

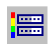
Note. *The **Z-Scaling** button*  *on the EM Contour toolbar switches the **Axis Scaling** (minimized) dialog on and off*



To change the axis labels:

- Select **Plot/Plot Settings/Axes** to bring up the **Set Axes** dialog
- Type new labels for your plot and edit their font size, determine the step between major ticks and the size of their labels, and specify the number of subticks in the respective boxes
- Click **Apply** to view the changes
- Click **Close** to close the dialog.

To change your coordinate settings:

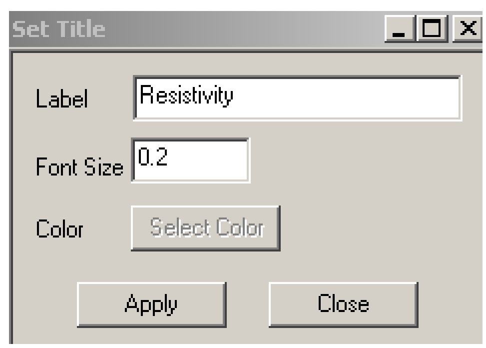
- Click the **Adjust Legend** button  on the EM Contour toolbar.

The respective dialog appears (see **Adjusting the Range of Data**)

- Edit your settings in the X-axis, Y-axis and Z-axis sections of the dialog
- Click **Apply** to check your changes without closing the dialog
- Click **Close** to close the dialog when finished

Note. *To view the coordinates of any point of your contour display, hold your arrow manipulator over this point. Read the coordinates in the bottom left corner of the window right under the dial bars.*

Title and Captions



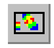
To add a title to your contour plot:

- Select **Plot/Plot Settings/Title**

The **Set Title** dialog appears

- Type your title in the **Label** field and specify the font size in the box below
- Click **Apply** to view the result
- Click **Close** to close the dialog.

Print and Preview

There are two ways of printing your plot: as it is and with an information box added. In the latter case, click the **Plot Surround** button  on the EM Contour toolbar before setting printing parameters. The information box to appear in the bottom right-hand corner of the page will contain information on your data type, receiver, transmitter, separation and channel.

To preview your plot:

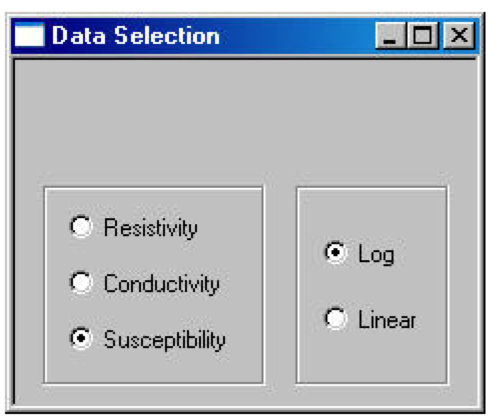
- Select **File/Print Preview** to switch to the preview mode
- Use the **Next Page** and **Previous Page** buttons on the **Print Preview** toolbar to toggle through available pages
- Click the **Two Pages** button to preview two pages at a time, click it again to switch back to the **One Page** mode
- Click the **Zoom In** button to take a closer look at your plot and the **Zoom Out** button to move it away
- To close the **Preview** mode, click **Close**
- To print the plot right from the preview mode, click **Print**. In the **Print** dialog to open, specify the printer and required printing settings. To do this, you can also use the **Print Setup** item on the **File** menu

To print the plot without preview:

- Select **File/Print** or click the **Print** button on the EM Contour toolbar to display the standard **Print** dialog
- Click **OK** in this dialog to start printing.

Viewing Inversion Results

On loading a data set with inversion results, it will be possible to choose from displaying the data as Resistivity, Conductivity or Susceptibility(Density for gravity data). Also, the legend can be toggled between a log and linear scale:

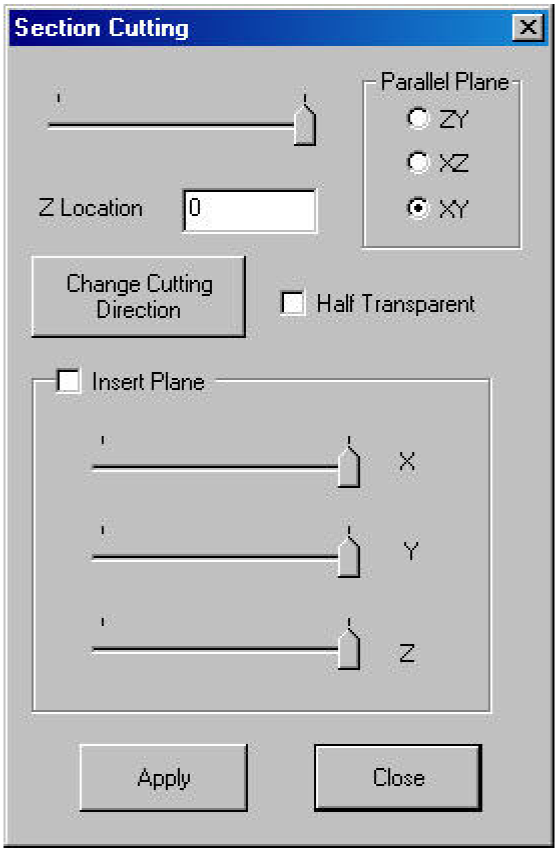


- The display can be further analyzed using the **Section Cutting** tool available by selecting Tools/Section Cutting from the menu.

Switch to gridded data by deselecting the  button on the toolbar.

Using the Section Cutting Tool

The **Section Cutting** tool is available by selecting Tools/Section Cutting from the menu:



To view only a section of the inversion result:
- Select the axis that the desired cutting plane is parallel to in the **Parallel Plane** section.
- In the **Location** box, enter the location where you want your inversion cut.
- Click **Apply** and the inversion volume beyond the cutting plane you have chosen will disappear.

To dynamically change the location of the cutting plane:
- Move the slider located at the top of the dialog box and the size of the volume will change as you move the slider.

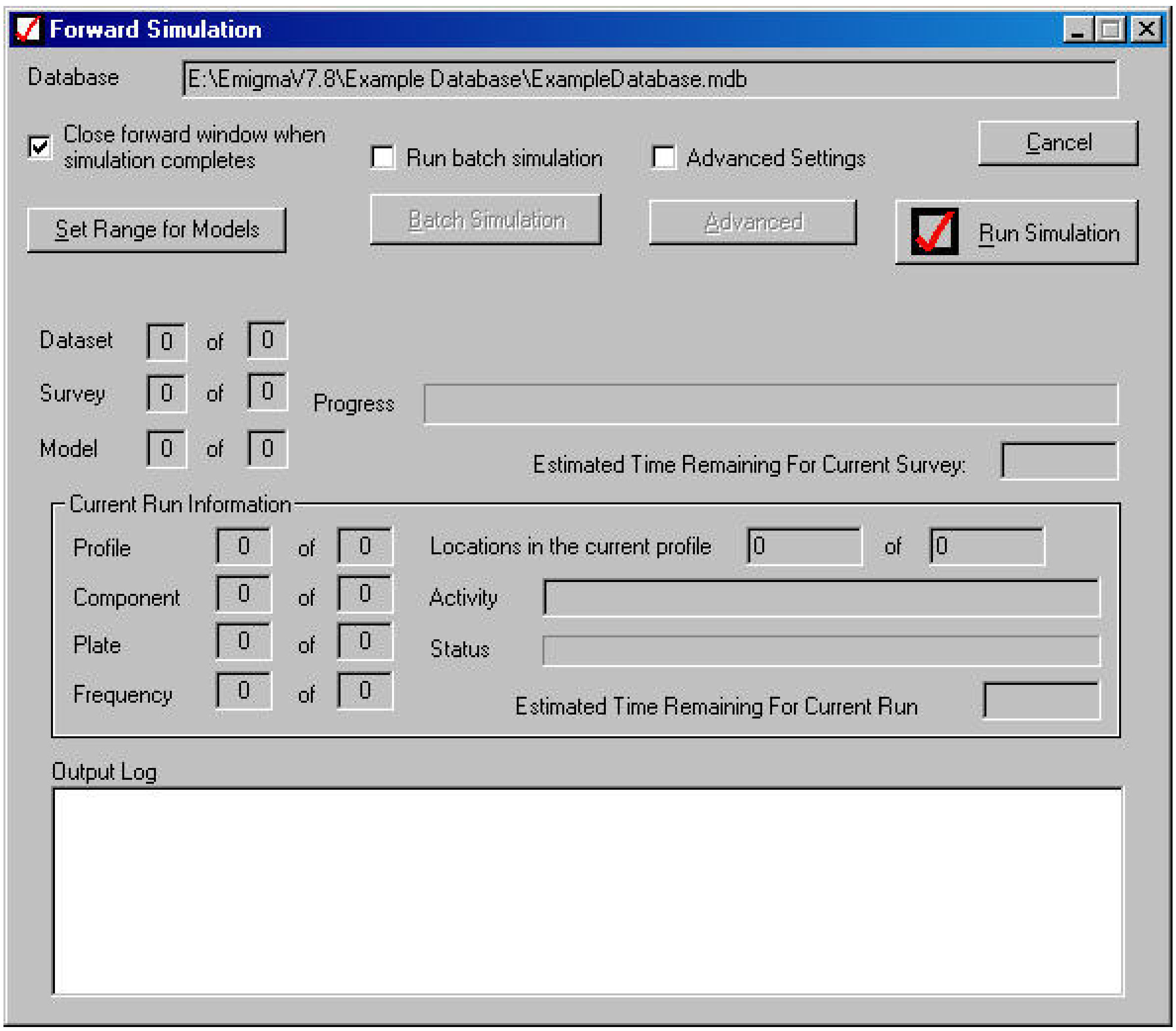
To view the section on the other side of the cutting plane:
- Click **Change Cutting Direction** and then either click **Apply** or move the cutting slider at the top of the dialog.

To make the inversion partially transparent:
- Click the **Half Transparent** checkbox

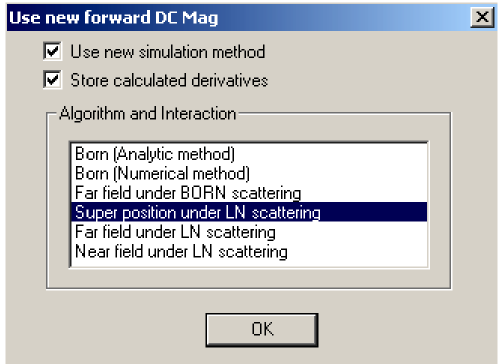
Forward Simulation ( )

Select the data set to subject to simulation in the **Projects in Database** dialog and click the **Forward Simulation** button on the main toolbar of EMIGMA 8.1. A message will ask you whether you want to overwrite the selected data set. Click **Yes** to overwrite it and **No** to create a new data set.

The **Forward Simulation** dialog will appear containing your database in the respective field:

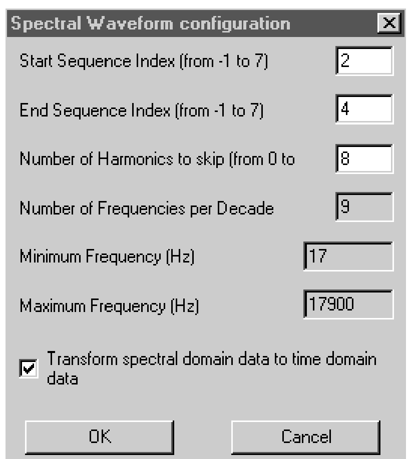


Click **Run Simulation** to perform the forward simulation as is. With FEM data, simulation will start right away. In the case of magnetic, gravity or resistivity system, the **Use New Forward**… dialog (its name to depend on the data type) will open offering you to select the scattering algorithm and the type of interaction to be used in simulation:



- Check the **Use new simulation method** box in the upper part of the dialog to enable the list of methods below and select a method to apply. This option is available only for magnetic or resistivity data
- Check **Store calculated derivatives** to save the derivatives in the same data set. This option becomes active in case you are using a new simulation method
- Click **OK** to start simulation.

In the case of TEM, the **Run Simulation** button will bring up the **Spectral Waveform configuration** dialog:



- Check or change, if necessary, the starting and ending frequency sequence indices and the number of harmonics to skip in the respective boxes. The values in the **Number of Frequencies per Decade**, **Minimum Frequency** and **Maximum Frequency** boxes will change accordingly
- Check the **Transform Spectral Domain Data to Time Domain Data** box to run the respective operation automatically

  This option is useful when you have modified an already available time-domain model and want to subject it to another forward simulation. In this case, you will not need to run the standalone transform to convert your spectral data into time-domain; this option will provide this operation automatically

- Click **OK** to apply changes and close the **Spectral Waveform Configuration** dialog. Click **Cancel** to cancel the changes, if you have made any, and close the dialog.

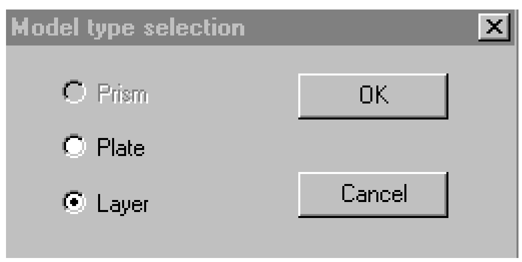
During simulation, which may take a long time, the **Forward Simulation** dialog will keep you updated on the progress, providing information on the status and estimated simulation time of a current survey on the whole and each current run in particular.

***Note**. You can minimize the **Forward Simulation** dialog to do some other things maximizing it from time to time to check on the progress. The **Cancel** button closes the **Forward Simulation** dialog upon completion of a current run.*

Model Suite Generation

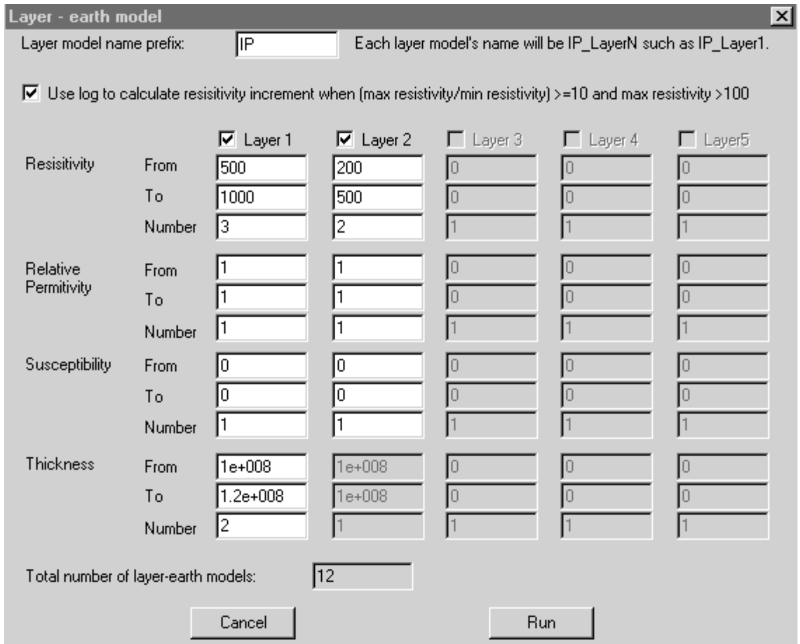
Click **Set Range for Models** if you want to subdivide your model into parts differing from each other in a certain property, such as thickness, resistivity or susceptibility. The objective of this operation is to create a number of "models" to compare your measured data sets with and to select the one that complies best with your survey results

The **Model Type Selection** dialog appears offering you to select between **Plate**, **Prism** and **Layer.** As of now, only the **Plate** and **Layer** options are accessible, with the **Layer** option selected by default:



To specify ranges of a layer-earth model:

- Leave the **Layer** option on and click **OK**. The respective dialog will open:



- In the **Layer Model Name Prefix** field, type in the name of your model so that the resulting data sets ("models") could have the same name prefix followed by the underscore and a subsequent number (e.g. IP_Layer1, IP_Layer2 and so on)
- If max/min resistivity is equal to or more than 10 and maximum resistivity is more than 100, leave the box below the model name prefix checked to use logarithmic function and calculate resistivity increment
- Select the box across the layer you want to subdivide into "models". The column below will become active
- Change any property range in the **From** and **To** boxes across **Resistivity**, **Susceptibility**, **Relative Permittivity** and **Thickness** and specify the number of "models" you want to simulate in the respective boxes

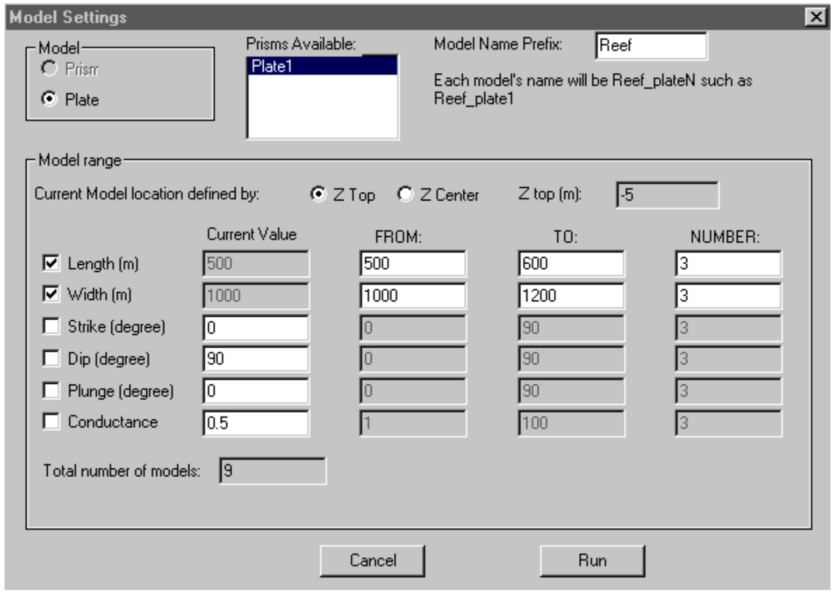
  You can select several properties or several layers at a time. In this case, the number of models to be simulated will be a multiplication product of all **Numbers**. For example, if, in the Layer 1 column, you have 3 in the **Number** field across one property and 2 – across another, the total number of the Layer 1 models will constitute 6. If you add 2 in the **Number** field across any property in the Layer 2 column, the number of models to be simulated will total 12. You can view the total number of layer-earth models in the respective box in the bottom part of the dialog

- All parameters specified, click **Run** to start simulation

To specify ranges of a plate model

- Select the **Plate** option in the **Model Type Selection** dialog and click **OK**

The **Model Settings** dialog will open:

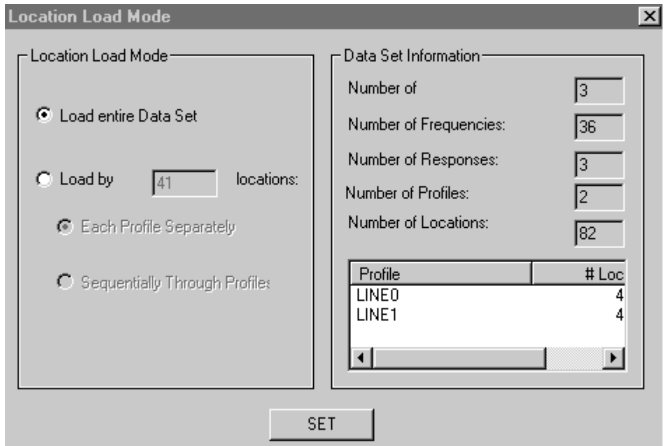


- Select the model you want to specify the parameters of in the **Prisms Available** field
- Type in the model name prefix in the respective box on the right. The resulting data sets ("models") will have the same name followed by the underscore and a subsequent number as shown in the example (Reef_plate1)
- In the **Model Range** section, select between the center point and top center modes of the plate position in space
- Check the property you want to specify the ranges for. The **From**, **To** and **Number** fields will become active
- Specify the ranges and the number of models and click **Run**

Location Load Mode

Click the **Advanced** checkbox then the **Location Load Mode** button to specify the order for the algorithm to pick up locations during simulation. This option is not needed if you are applying a new simulation method to your magnetic or resistivity data as well as in the case of gravity systems.

The **Location Load Mode** dialog opens:



In the right-hand section, you can see the general information on your data set, the list of profiles and the number of locations per profile.

In the left-hand section:

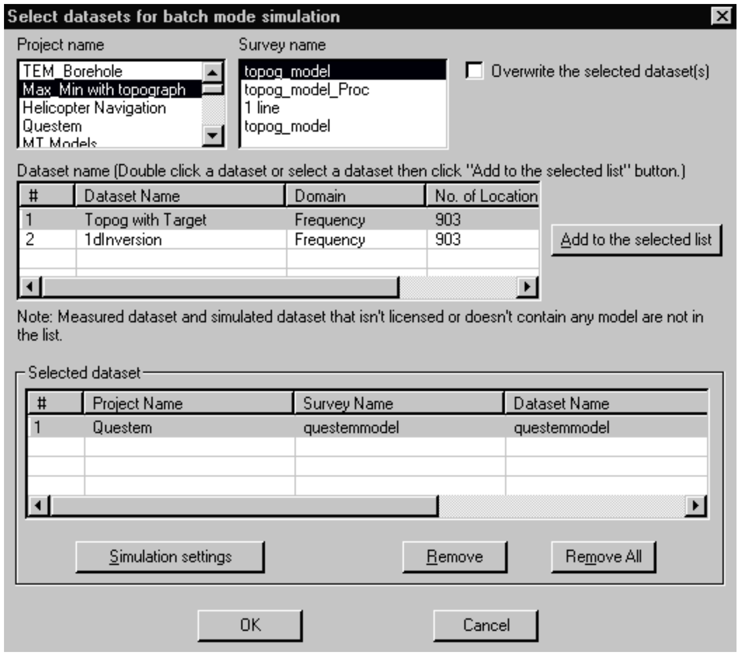
- Select **Load Entire Data Set** if your data set contains not too many locations and you want to subject them to simulation all at once
- Select the **Load by** option if your data set is too big. If it contains more than 4000 locations, this option will be selected for you automatically. Specify the number of locations to be loaded at a time and select between the two ways for the algorithm to go through these locations. If, for example, the number you specified is 20 and your profile contains 41 locations, the **Each Profile Separately** mode will pick up the first 20, then second 20 and then the last 1 location, whereas the **Sequentially Through Profiles** mode will process sets of exactly 20 locations no matter which profile they belong to.

Click **SET** to save your settings and return to the **Forward Simulation** dialog.

Click **Run Simulation** to launch forward simulation.

Batch Simulation

Check the **Run Batch Simulation** box to open the **Select Data Set for Batch Mode Simulation** dialog:



In the **Project Name** and **Survey Name** lists, you will see the current project and survey selected.

- In the **Dataset Name** table below, select the data set you want to subject to simulation and click **Add to the Selected List** button or just double-click on it

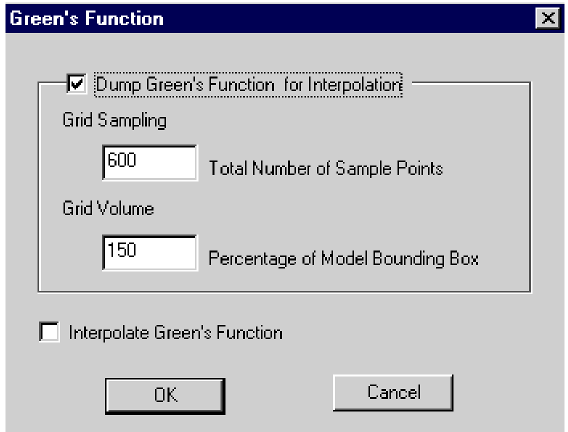
  The data set will appear in the **Selected Dataset** table

- To remove a data set from this table, select it and click **Remove**; to remove all data sets from this table, click **Remove All**
- To overwrite the selected data set by the one obtained through forward simulation, check the respective button in the top right-hand corner of the dialog
- If your data are time domain, gravity, magnetic or resistivity, the **Simulation Settings** button becomes active. Click it to open the **Spectral Waveform Configuration** dialog (in the case of TEM) and the **Use New Forward**… dialog (in the case of gravity, magnetics, resistivity). For details, see the beginning of the **Forward Simulation** section
- Click **OK** in the **Select Data Set for Batch Mode Simulation** dialog to return to the **Forward Simulation** dialog and start simulation.

Green's function

Green's function is an integral part of the forward simulation process. The respective calculation is run automatically as a routine program based on certain default settings. However, in some cases, it may be useful to view the result of this calculation, especially if your models represent "slight" variations of each other differing, for example, in the dip of structures or their electrical properties. The **Set for Green's Function** option will be accessible only if your model is 3D; in layered earth models, i.e. in the absence of scatterers, it will be inapplicable.

- Select the **Advanced** checkbox and select the **Set for Green's Function** box. This will bring up the **Green's Function** dialog:



- Check the **Interpolate Green's Function** box and click **OK** to launch simulation using the settings offered by default and displayed in the **Dump Green's Function for Interpolation** section

Or, you can replace the default settings with your own:

- Check the **Dump Green's Function for Interpolation** box. The respective section will become active

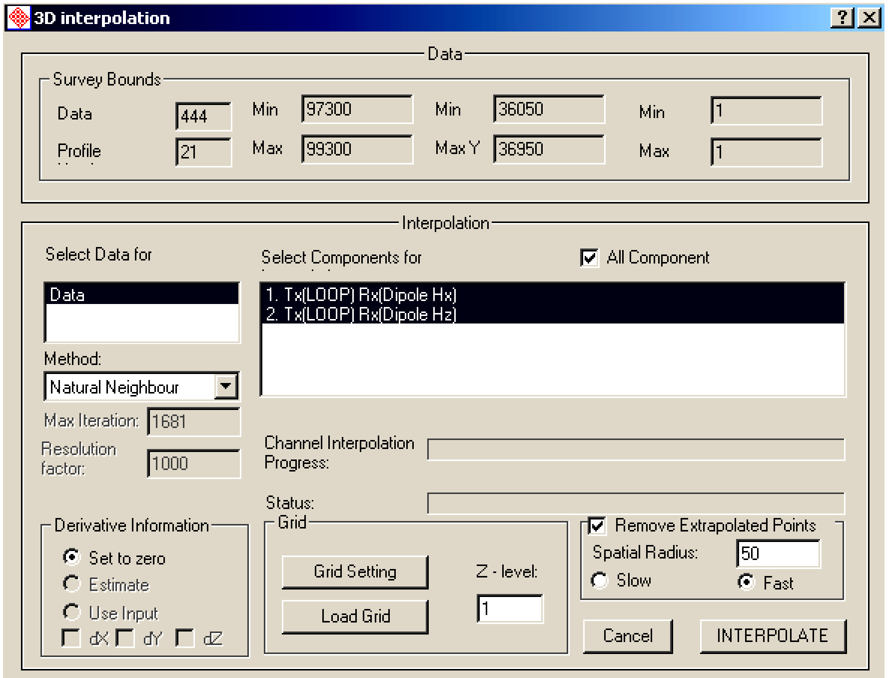
- Specify the values in the **Grid Sampling** and **Grid Volume** fields

  Grid Sampling allows up to 1000 points within the **Grid Volume**, a model region, which may be any multiple of the bounding box around the model, e.g. 150%. The larger the model region, the more points will be required to maintain a fixed accuracy and the more complex (and time-consuming) the resulting interpolation will be

- Check the **Interpolate Green's Function** box and **OK** to start simulation using the settings you specified.

Gridding ( )

Select the data set to interpolate in the **Projects in Database** dialog and click the **Gridding** button on the main toolbar of EMIGMA 8.1. The **3D Interpolation** dialog will open:



In the **Data** section of the dialog, you will see the profile and coordinate information of your data set

To specify interpolation terms:

- Select the type of data to interpolate in the **Select Data** field of the **Interpolation** section

  All data obtained by means of import or simulation in EMIGMA, subjected to normalization, etc., are considered as core data and are referred to as ***Data***; all the rest calculated through various algorithms are considered as optional and referred to in accordance with their type, e.g. ***Apparent Resistivity***, ***Apparent Depth***, ***Voltage***, etc.

- Click on a component in the **Select Components** field to involve it into interpolation or check the **All Components** box to have all components participate in the interpolation process

- Select the method of interpolation in the respective dropdown list.

  There are four choices: **Natural Neighbour**, **Delauney Triangulation**, **Thin Plate Spline** and **Minimum Curvature**, with the first being the most frequently used. If you select **Minimum Curvature**, type the maximum number of iterations to be performed and specify the resolution factor in the respective fields below, which in this case will become activated

- If your data contain derivative information, you can choose to carry out interpolation based on all the three derivatives at a time. In this case, the result will be more accurate in comparison with what is obtained when you use data as is. Turn the **Use Input** button on in the **Derivative Information** section and select the derivatives to participate in interpolation

- Check the **Remove Extrapolated Points** box to activate the respective section and forbid extrapolation into the "no data" locations:

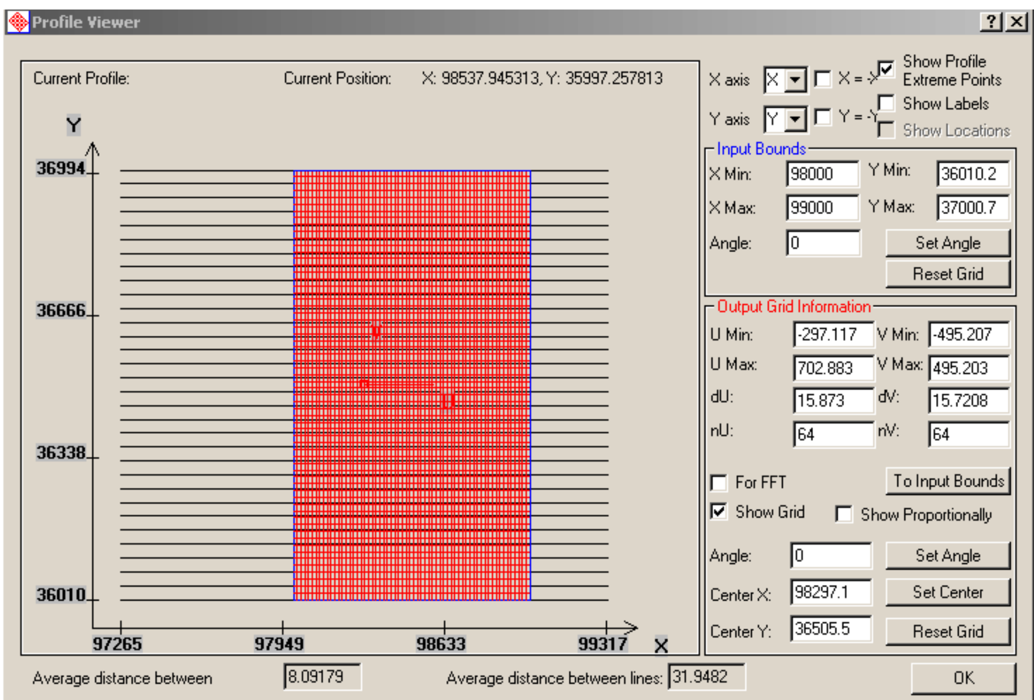
  - Set a required spatial radius to restrict the area of interpolation.

    In the present example: a spatial radius of 50 means that if there are no data in the radius of 50 m around a given point – a grid cell center, – this cell will be removed from interpolation
  - Select between the slow and fast interpolation algorithms (slow is more accurate, but fast is almost always sufficient) and click **OK**

To specify grid parameters:

- Click the **Grid Setting** button in the **Grid** section of the **3D Interpolation** dialog

  The **Profile Viewer** dialog will open:



- Customize the coordinate system by selecting required settings in the upper right-hand corner of the dialog

- Select between the **Show Profile Extreme Points** and **Show Locations** options and check the **Show Labels** box to display the profile numbers (names)

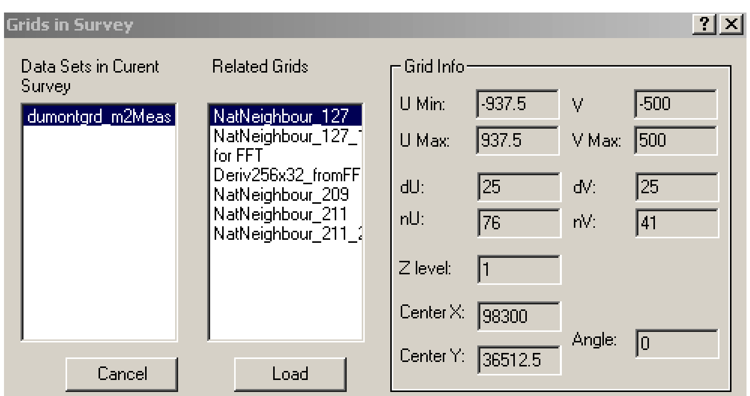
- Specify the input bounds, a blue line enclosing the data to be involved in interpolation, in the X and Y **Min** and **Max** boxes of the respective section. Or, you can simply click and drag the input bounds (blue) right in the grid view field of the dialog. Set the rotation angle of the grid about its local center and click the **To Input Bounds** button

  This will automatically change the **U** and **V** coordinates of the grid in the **Output Grid Information** section, and the input bounds (blue) and output grid bounds (red) will coincide

- If you want the output bounds to cover a different area compared to the input grid, edit the **U** and **V** values and the angle of grid rotation in the **Output Grid Information** section. The output grid (red) will be changed, while the input bounds (blue) will stay the same
- To adjust the grid density, increase or decrease **dU** and **dV** (number of points) or **nU** and **nV** (length of a grid cell side) in the **Output Grid Information** section
- Check the **For FFT** box, if you want to subject your data to FFT. This will automatically change the **nU** and **nV** values to the n power of 2
- To display the grid, check the **Show Grid** box; to provide its proportional view, check the **Show Proportionally** box to the right
- To edit the local center of the grid (U vs V), type your values in the **Center X** and **Center Y** boxes in the bottom of the **Output Grid Information** section and click **Set Center**
- To reset the boundaries of your grid to the ones determined by the initial Input Bounds coordinates, click **Reset Grid** in the **Input Bounds** section if you changed the coordinates here or **Reset Grid** in the **Output Grid Information** section if you changed the values there
- Click **OK** to return to the **3D Interpolation** dialog

To load an existing grid:

- Click the **Load Grid** button in the **Grid** section of the **3D Interpolation** dialog. The **Grids in Survey** dialog will open:



- Select the data set containing the grid you want to load from the **Data Sets in Current Survey** list and the grid itself from the **Related Grids** list

  The settings of the grid to be loaded will be displayed in the **Grid Info** section on the right

- Click **Load** to load the grid

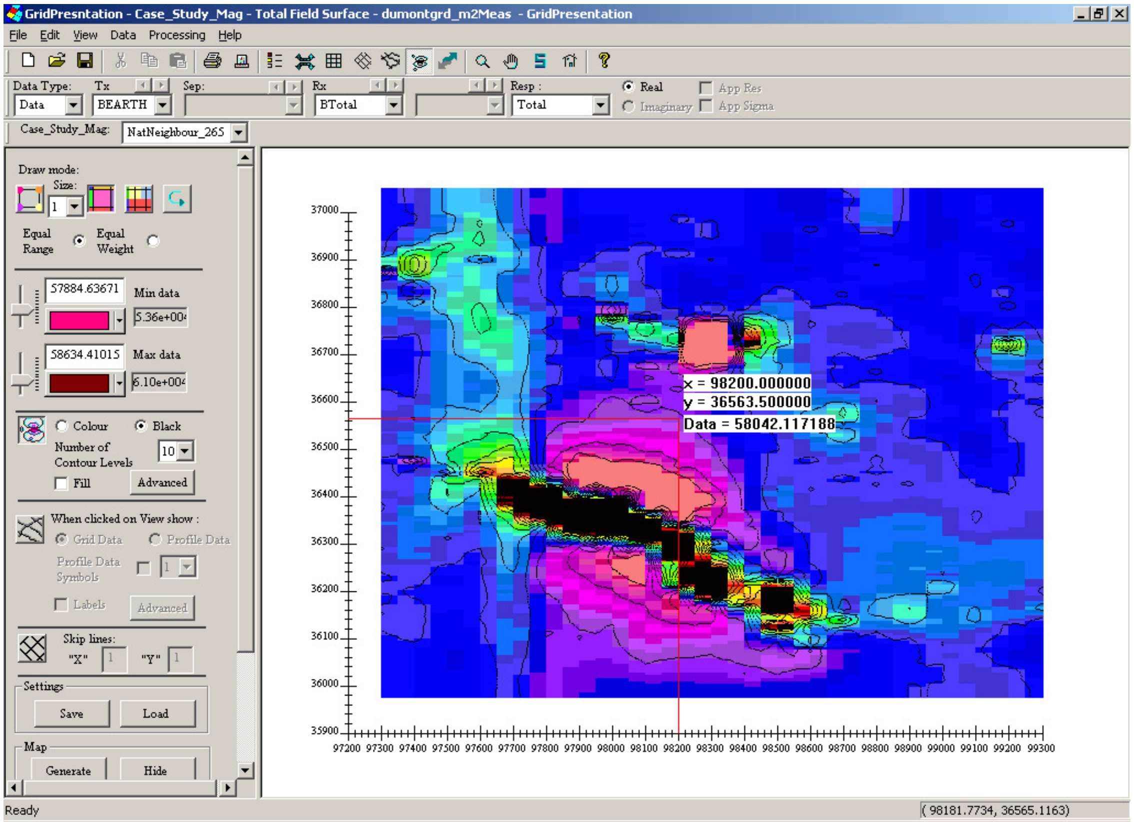
Back in the **3D Interpolation** dialog, click **INTERPOLATE**.

Notes. *To view the interpolation results, click the* ***GridPresentation*** *button*  *on the main toolbar of EMIGMA*

To view grid information, click the ***Has Related Grid*** *button on the* ***Database*** *tab of the main dialog.*

Grid Presentation ( )

If your data set contains a grid, you can view it with the **GridPresentation** tool. Click the button on the main toolbar to open this application:



If there are more than one grid, the grid to open will be the most recent one:

- To switch to another grid, select it from the dropdown list across the name of your data set on the **Info** bar (under the **Data** bar)

On the **Data** bar (under the **GridPresentation** toolbar):

- If you data set contains different data types (data as is, recalculated data such as voltage, apparent resistivity, etc.), the **Data Type** dropdown list will contain the respective number of options, with **Data** selected by default. To switch to another option, select it from the list
- If there are several transmitters, receivers, separations or channels, select the required item from the respective dropdown list or use the arrow buttons above to switch between the items
- If both real and imaginary responses are available, turn one or the other button on to switch between the responses
- Check the **App Res** box to calculate apparent resistivity from data on the fly. Check **App Sigma** to switch to the respective grid

Adjusting Grid Display

The left-hand panel of the **GridPresentation** dialog offers a number of tools to adjust your grid display.

<u>To change the draw mode of your grid</u>

- Click the **Draw Grid Symbols** button in the **Draw Mode** section to display each grid cell as a set of four data (vertices) and to assign a certain color to each data dependently of its value

- Click the **Draw Grid Cells** button to calculate the average of the data located in the vertices of a grid cell. The cell will be filled with a certain color assigned to the average value

- Click the **Draw Cells around Grid Vertices** button to display your grid as a set of cells drawn around each grid vertex and filled with a certain color assigned to the data value in the vertex

- To toggle your grid on and off, use the **Hide Grid Mesh** button .

- Select **Equal Range** to assign different colors to equal ranges independently of the number of points in each range or **Equal Weight** to assign different colors to different ranges covering, however, the same number of points

<u>To adjust the range of data to be displayed</u>

- Insert the minimum and maximum values manually in the active **Min** and **Max** boxes or use the respective sliders on the left. The absolute minimum and maximum values are displayed in the disabled boxes on the right

- Click on the arrow button of the color palette to bring up the palette and change colors to be assigned to your data

- To view the changes, click the **Refresh** button in the bottom of the **GridPresentation** dialog

- To switch to the full data range view, click the button on the **GridPresentation** toolbar

<u>To adjust the contour lines</u>

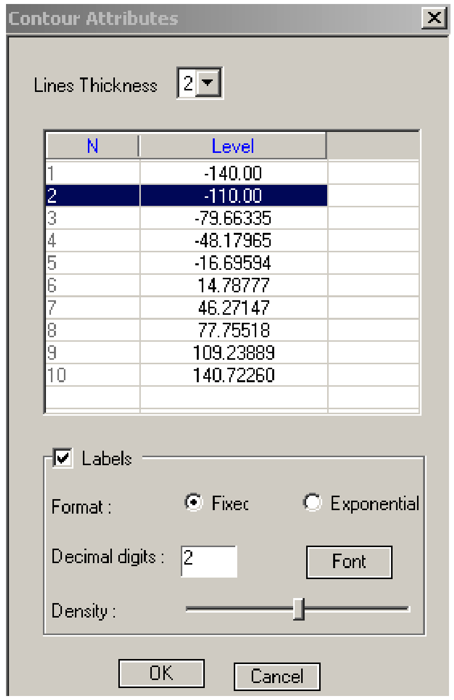
- Click the **Show/Hide Contour Lines** button to toggle the contours on and off (you can also do it from the **GridPresentation** toolbar). When engaged, the button enables the respective section of the dialog

  In this section:

- Select between the **Black** and **Color** options to have your contours black or colored

- Select the contour density from the **Number of Contour Lines** dropdown list

- Click the **Advanced** button to customize the appearance of your contours. In the **Contour Attributes** dialog to open:



  - ➢ Select the thickness of lines to be drawn from the respective dropdown list
  - ➢ Type in contour levels as desired
  - ➢ Check the **Labels** box to specify the format of the contour labels and their density
  - ➢ Click **OK**

- Check the **Fill** box to fill your contours with color.

<u>To view your profiles</u>

- Click the **Show or Hide Profiles** button  to toggle profiles on and off. You can also do it from the **GridPresentation** toolbar. When engaged, this button enables the respective section of the dialog

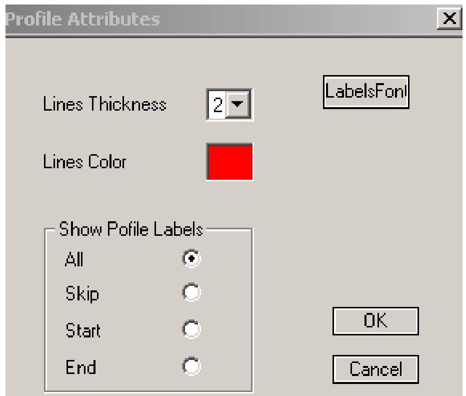
  In this section:

- Select between the **Grid Data** and **Profile Data** options.

  If you leave the **Grid Data** button on and click on a certain point of a profile, the x and y coordinates to appear will refer to the grid cell. With the **Profile Data** button on, the x and y coordinates will refer to the profile location

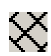
- Specify the style of your profiles to be drawn. Click the **Advanced** button and select the line thickness and color

- Check the **Profile Data Symbols** box to display "stacked" profiles, with a certain color being assigned to each location dependently of the data value in it, and select the size of symbols in the dropdown list to the right. To better view the profiles and data distribution, click the **Hide Grid Mesh** button  in the **Draw Mode** section to switch your grid off and leave only profiles on the screen
- Check the **Labels** box to display the profile labels. Click the **Advanced** button to specify their format, density and location in the **Profile Attributes** dialog to open:



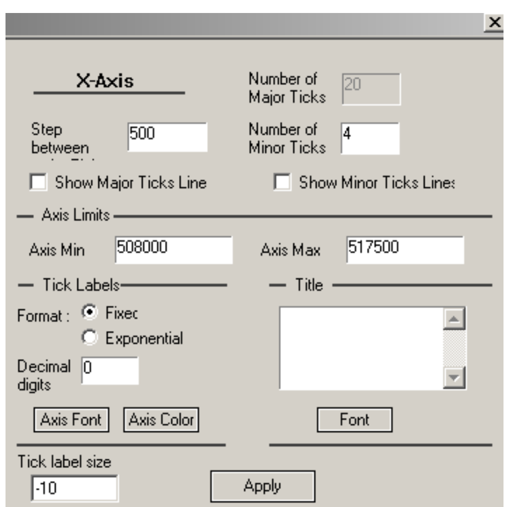
The **All** option will show all labels; the **Skip** option will show each second label. With the **Start** or the **End** option on, the profile labels will appear respectively at the end or at the beginning of the profiles.

To view grid lines

- Click the **Show or Hide the Grid** button  to toggle the grid lines on and off. You can also do it from the **GridPresentation** toolbar. The grid may be very dense; in this case, specify the number of grid lines to be skipped in the X and Y boxes to the right

To adjust axes

- Double-click in the region of the X or Y axis. The respective dialog will open:



- Edit the step between major ticks in the respective box. The number of major ticks will change accordingly
- Check the **Show Major Tick Lines** box and the **Show Minor Tick Lines** box to display the coordinate grid
- Edit the **Axis Min** and **Axis Max** values as desired in the **Axis Limits** section
- Specify the format and color of the tick labels in the respective section
- Type in the title of your X or Y axis in the respective field; use the **Font** button to specify the format of the title
- Click **Apply**

Note. To provide the proportional scaling of the axes, use the **Show Proportionally** button .

<u>To toggle the coordinate grid on and off</u>

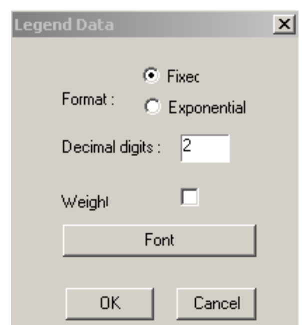
- Use the **Show Coordinate Grid** button  on the toolbar.

<u>To display and customize the legend</u>

- To display the legend in a separate window, use the **Show Legend** button  on the **GridPresentation** toolbar. This option is also available from the **Data** menu
- To display the legend alongside the grid representation, select **Data/Show Legend in the View**. In this case, the legend can be moved to a desired location with the hand manipulator 

In the latter case you can customize your legend

- Double-click anywhere in your legend to bring up the **Legend Data** dialog
- Specify the number of decimal digits
- Check the **Weight** box to display the number of data covered by each color range
- Click the **Font** button to change the font, size and style of the legend labels
- Click **OK**



<u>To provide the full-screen view of your grid</u>

- Use the **Full Screen View** button  on the toolbar. You can also do it from the **View** menu

<u>To toggle off any of the bars and panels</u>

- Select the respective item (**Info**, **Data**, **Toolbar**, **Left Pane**) from the **View** menu. The tick across it will disappear. To bring it back, select this item again

To zoom in on a grid fragment

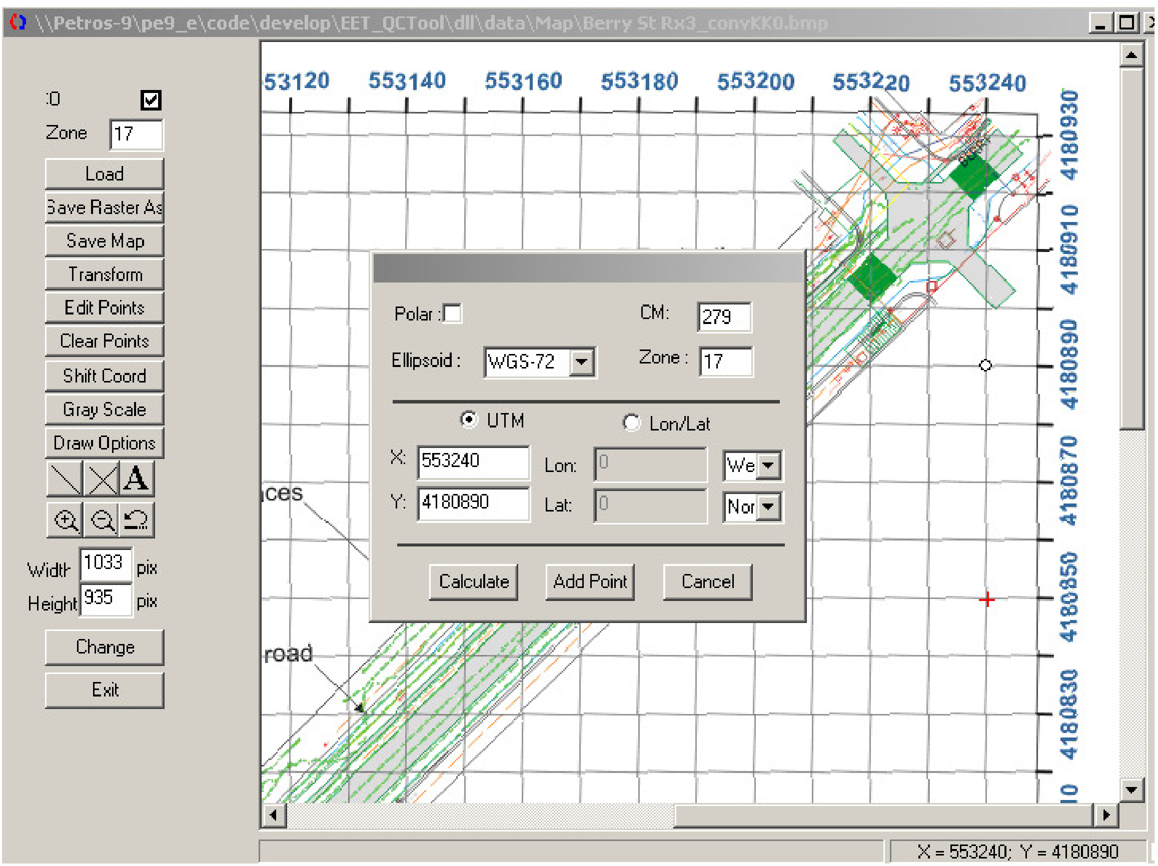
- Engage the **Zoom In** button 
- Click in the grid and without releasing the mouse select the required fragment
- To return to the initial view, use the **Home** button  on the toolbar.

Generating a map

To generate a map to be laid under your grid, click the **Generate** button to bring up the **Calibrate Map** dialog. In this dialog:

- Click **Load**. A Windows-style dialog will appear offering you to select and load a raster format file (bmp, jpg, etc.)
- The map loaded, specify at least three points to calibrate it to
- Double-click on the first point in the map

  The following (smaller) dialog appears:



In this (smaller) dialog:

- Check the **Polar** box if your map comes from a polar region
- Select the required ellipsoid datum from the respective dropdown list
- Specify the central meridian and zone in the respective boxes

- Select between **UTM** and **Lon/Lat** (as a rule, QCTool recognizes this information)

  The respective boxes below will become active

- Type in the coordinates of the point to add and click **Add Point**. Repeat this operation for the rest points
- To convert your UTM into Lat/Lon or vice versa, click **Calculate**

  Lat/Lon are to be recalculated to UTM since raster transformation is based on UTM coordinates

  Back in the **Calibrate Map** dialog:

- To check and, if necessary, change your points, click the **Edit Points** button. The **Edit Entered Points** dialog will open. Select the cell you want to edit and type in a new value
- Click the **Clear Points** button to remove all points
- Use the **Shift Coordinates** button to switch over to the local coordinate system
- Click **Transform** to start raster transformation. Do not forget to save the transformed image (**Save Raster As**). To keep the previous image as well, save it with a new name

  You can customize the appearance of the map:

- To change its color to gray, click the **Gray Scale** button
- To resample, change its width and height in pixels in the respective boxes
- To add a line or any symbol to your map, use the buttons and . If necessary, you can change their style. Click the **Draw Options** button. Specify the line color, style and width as well as the color, shape and size of the symbols using respective sliders and dropdown lists in the **Change Style** dialog to open
- To add an annotation to your map, click the **A** button. In the **Annotations** dialog to open, write your text. Use the **Font** button to change the appearance of the text
- The **Undo** button cancels any previous operation.
- To zoom in on your map, click the standard **Zoom In** button. To zoom out, click the **Zoom Out** button.

  Having finished your work, save the transformed image (**Save Raster As**) as well as a *.map file (**Save Map)** to keep externals, originals and the name of the image. This *.map file will then be available to other applications which can use the created raster image as an underlay.

To load an earlier calibrated map

- Click the **Load** button in the **Map** section of the **GridPresentation** dialog. Browse for the required map in the Windows-style Open dialog to appear.

To toggle the map underlay on and off, use the **Show** and **Hide** buttons in the **Map** section

Saving and Printing

To save your grid settings

- Click **Save** in the **Settings** section of the **GridPresentation** dialog
- In the **Save Settings** dialog to appear, type in the name for your grid to be saved with or select the existing file from the **Settings Name** list and click **Save**. In the latter case, the existing grid will be overwritten with a new one

To save your grid in a different format

You can save your grid as a*.bmp, *.jpg, *.wmf, *.emf, etc.

- Click anywhere in the grid area to activate the view window
- Select **File/Save As** and choose the desired format from the **Save as Type** dropdown list in the standard **Save As** dialog to appear

To load previously saved grid settings

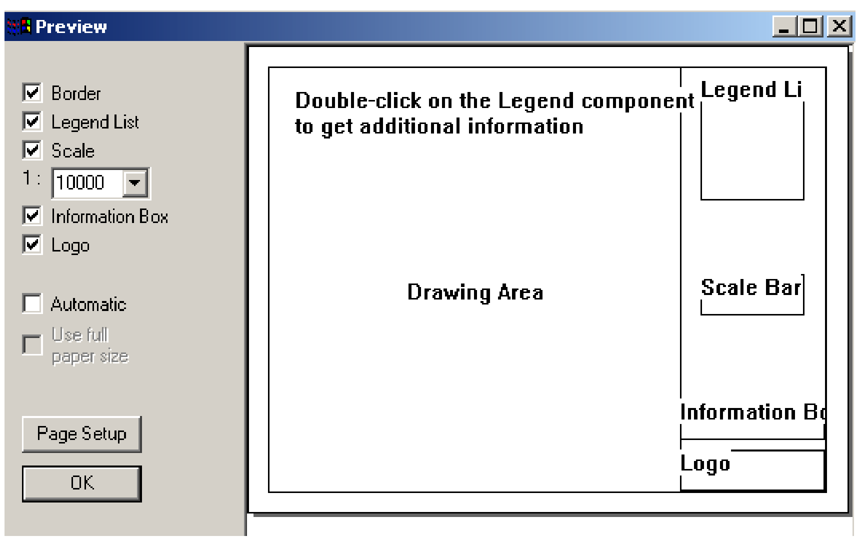
- Click the **Load** button in the **Settings** section of the **GridPresentation** dialog
- In the **Load Settings** dialog to appear, select the file to open from the **Settings Name** list and click **Load**

To print your grid as is

- Select **File/Print** or click the **Print** button on the GridPresentation toolbar to bring up the standard **Print** dialog and specify print setup

To specify the surroundings of your grid

- Click the **Print Settings** button  on the **GridPresentation** toolbar. The **Preview** dialog will open:



To print a grid without any surrounding:

- Leave the **Automatic** option checked (it will contain a flag by default)
- Check **Use full paper size** for your grid to occupy the whole printing area of the page

To print the legend list, scale bar, information box and logo alongside your grid:

- De-select the **Automatic** box
- Check the Border box to draw a border around your grid
- Check the **Legend List** to have the legend printed to the right of the grid. Double-click on the **Legend List** component to bring up and check the legend before printing
- Check the **Scale** box and select a required scale from the dropdown list below. If there is no scale you need, type it in manually
- Check **Information Box** to have it added to your printed grid

Use the standard **Page Setup** button to specify page layout and printer settings. Click **OK** to close the dialog and select **File/Print Preview**.

Analytic Signal (AS) and Horizontal AS

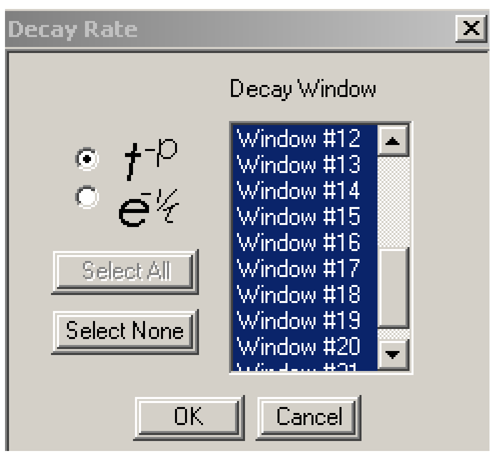
Provided the derivatives have been calculated in your magnetic or gravity survey, you can display the results of processing by the Analytic Signal and Horizontal Analytic Signal techniques

- Select **Processing/Analytic Signal (AS)** or **Horizontal AS** to launch the respective processing and view its results

Decay Rate

For time-domain systems, the decay rate calculation is provided

- Select **Processing/Decay Rate**. The following dialog will open:



- Select the range of decay windows in the respective list on the right to be used in the decay rate calculation

  Note. *Only multiple selections are applicable*

- To select all windows, click the **Select All** button
- To cancel selections and return to initial data, click **Select None**
- Select between the two algorithms on the left

- Click **OK** to close the dialog and view the results.

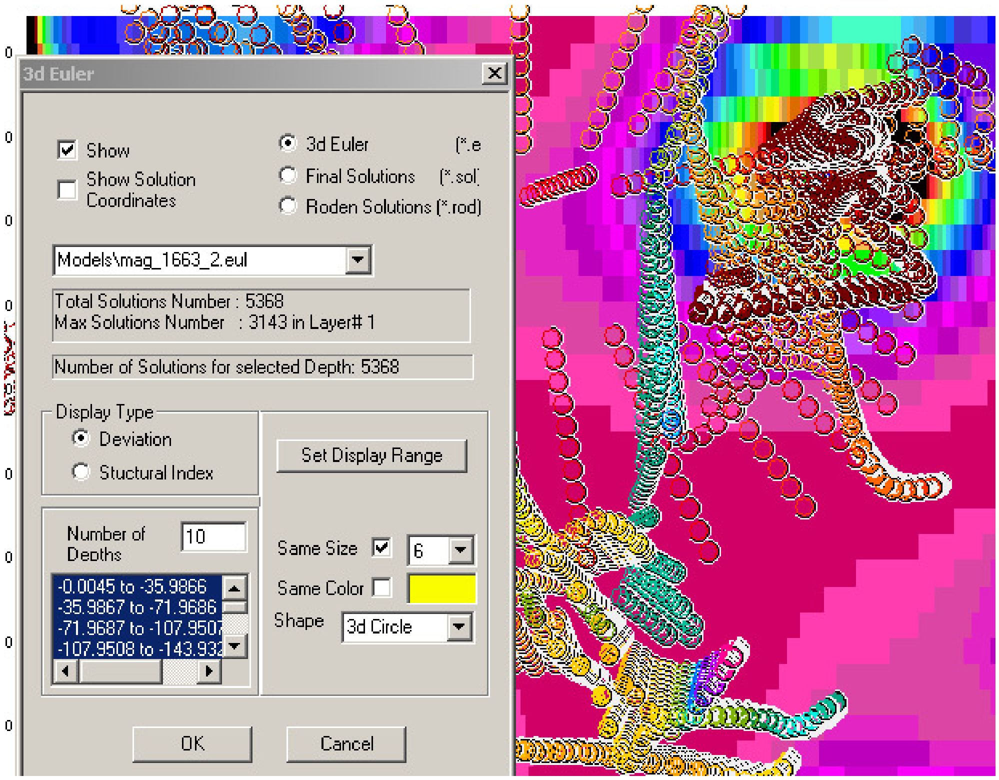
Euler Overlays

Provided your data set contains Euler solutions:

- Select **Processing/3D Euler**. The Euler solutions will appear over your grid

  In the 3D Euler dialog to open:

- Leave the **Show** box checked (if you de-select it, the Euler overlay will be switched off)
- Check the **Show Solution Coordinates** box and click on a certain solution in the grid. The x and y coordinates to appear will refer to the solution and its depth. This box left unchecked, the x and y values to appear will be the coordinates of a grid cell



- In case you data set contains more than one Euler overlay, select the one to view from the dropdown list in the upper part of the dialog. The box below displays information on the total number of solutions and the layer with the maximum number of solutions
- Select between the three kinds of Euler overlays dependently of the post-processing applied in the upper left-hand corner of the dialog

  The 3D Euler option means that the Euler solutions have not passed post-processing. The **Final Solutions** and **Rodin Solutions** options display the results after Final or Rodin post-processing

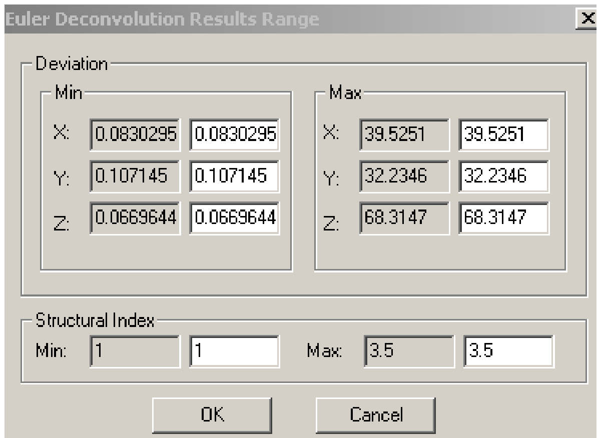
In the bottom left-hand section of the dialog of the **3D Euler** dialog

- Check (or edit) the number of available depths

- Select a particular depth to see Euler solutions for or all depths to display all Euler solutions

  The **Number of Solutions for Selected Depth** will change accordingly with your choice of depths

- Select between the **Deviation** and **Structural Index** options in the **Display Type** section and click the **Set Display Range** button to specify the range of deviation or the range of structural indices to be taken into account:



  The gray fields to the left will show you the initial deviation and structural index minimums and maximums. Adjust the minima and maxima in the active boxes as needed and click **OK** to close the dialog

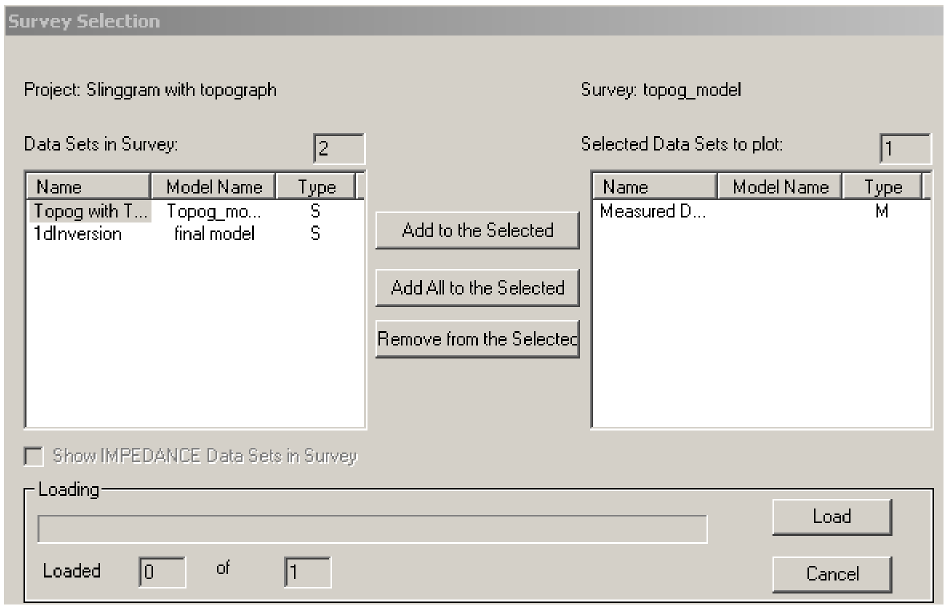
In the bottom right-hand corner of the **3D Euler** dialog, customize the appearance of your Euler overlay

- Select the symbol color, shape and size
- Check the **Same Size** box to assign a common size to all symbols
- Check the **Same Color** box to assign a common color to all symbols.

MultiGrid ()

The MultiGrid tool is designed for comparison between various grids or different components of the same grid. It allows viewing of up to 4 grids at a time and is provided with easy-to-use interface.

Click on the **MultiGrid** button  of EMIGMA's main toolbar. The **Survey Selection** dialog will appear:



If your survey contains only one data set, the latter will be loaded automatically; if it has two or more data sets:

- To view only the current data set, click **Load**
- To compare your current data set with some other data set in the same survey, select this other data set in the left table, click **Add to the Selected** and **Load**
- To compare your current data set with all the data sets available in your survey, click **Add All to the Selected** and **Load**
- To remove a data set from the **Selected Data Sets to plot** list, select it from the list and click the **Remove from the Selected** button

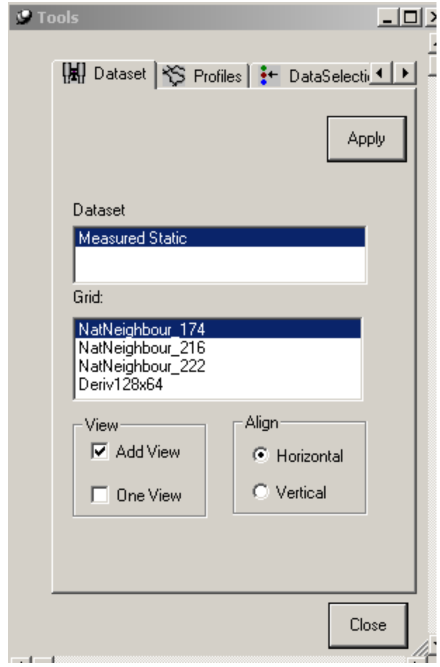
If your data set contains more than one grid, the most recent one will open. Click on any point of your grid to view the grid data.

<u>To switch to another grid</u>

- Click the **Tools** button  on the **MultiGrid** toolbar to appear under EMIGMA's main toolbar or double-click anywhere in your grid. The **Tools** dialog will open. Click on the pin in the left corner of its header to pin the dialog

- Select a required grid from the **Grid** list on the **Dataset** tab and click **Apply**

<u>To display multiple (up to 4) grids at a time</u>



- Open the **Tools** dialog () as described above
- On the **Dataset** tab, de-select the **One View** box in the **View** section. This will automatically activate and check the **Add View** box
- Choose between **Horizontal** and **Vertical** in the now active **Align** section to have your grids arranged horizontally or vertically (to switch from one option to the other in the process of your work, right-click anywhere in your grid and select the required option from the popup menu to appear)

To view grids from the same data set:

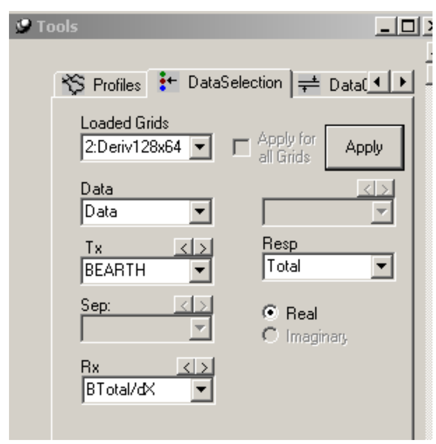
- Select the grid you want to view in addition to the one displayed and click **Apply**
- Repeat this procedure, if necessary, to add a third and a forth grid

To view grids from different data sets:

- Select a required data set from the respective list on the **Dataset** tab
- Select the grid you want to compare to the one displayed and click **Apply**
- Repeat this procedure, if necessary, to add a third and a forth grid

<u>To compare various components of the same grid</u>

- Open the **Tools** dialog () as described above
- On the **Dataset** tab, de-select the **One View** box in the **View** section. This will automatically activate and check the **Add View** box



- Choose between **Horizontal** and **Vertical** in the now active **Align** section to have your grids arranged horizontally or vertically
- Select the same grid to have its duplicate displayed
- If necessary, repeat this procedure two times more. You will have four similar grids displayed at a time
- Go to the **Data Selection** tab of the **Tools** dialog and select the first grid from the **Loaded Grids** dropdown list. You can also select it by clicking in the view

- Select a required component you want to display from the dropdown lists below and click **Apply**
- Repeat the same procedure for three other grids

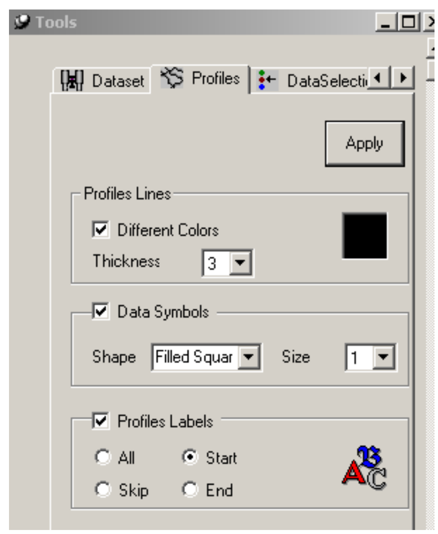
***Note**. To switch back to the single-grid presentation, go to the **Dataset** tab of the **Tools** dialog and select the **One View** box; this will bring up the grid that was loaded by default (i.e. the most recent) at the very beginning.*

<u>To display and customize profiles</u>

- Click on the grid display you want to add profiles to
- Click on the **Show Profiles** button  on the **MultiGrid** toolbar

OR

- Right-click anywhere in your grid and choose **Show Profiles** from the popup menu to appear. The profiles will show over the selected grid
- Open the **Tools** dialog as described above and go to the **Profiles** tab to customize the appearance of the profiles
    - In the **Profile Lines** section, click the color square to specify the color of the profiles using the standard color palette or check the **Different Colors** box to have all the profiles colored differently. Select the thickness of the profiles from the respective dropdown list
    - Check the **Data Symbols** box to display the profiles as a set of locations (symbols), each having its own color. Select the shape and size of the symbols to be used from the respective dropdown lists
    - Check the **Profile Labels** box to display the labels. Select one of the four options below: **All** will display labels for all profiles, **Skip** – for each second profile, **Start** – only at the beginning of the profile and **End** – at its end

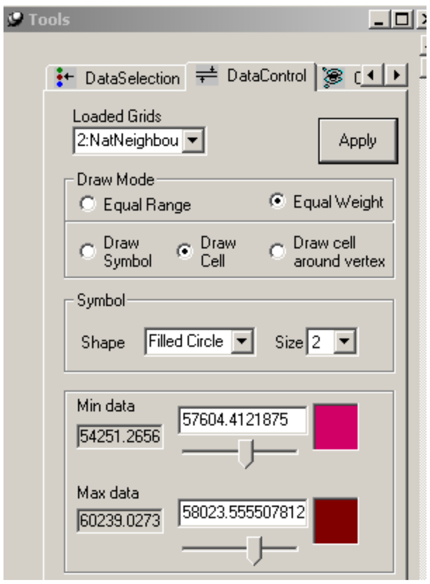


<u>To switch between the Profile Data and Grid Data modes</u>

When you click on a given point in your grid, you will see the x and y coordinates and data value of this point. When profiles are toggled off, this information will automatically be linked to a grid cell (respectively, the **Point on Grid Data** button  of the **MultiGrid** toolbar is engaged). When profiled are toggled on, the **Point on Profile Data** button  will become active and can be engaged to display profile-related data

To specify the draw mode

- Open the **Tools** dialog as described above and go to the **Data Control** tab
- Select the grid display you want to specify a draw mode for from the **Loaded Grids** dropdown list or just click on the display of this grid
- Select **Equal Range** to assign different colors to equal ranges independently of the number of points in each range. Select **Equal Weight** to assign different colors to different ranges covering, however, the same number of points
- Select the way for the color to fill the cells (**Draw Symbol**, **Draw Cells**, **Draw Cell Around Vertex**)

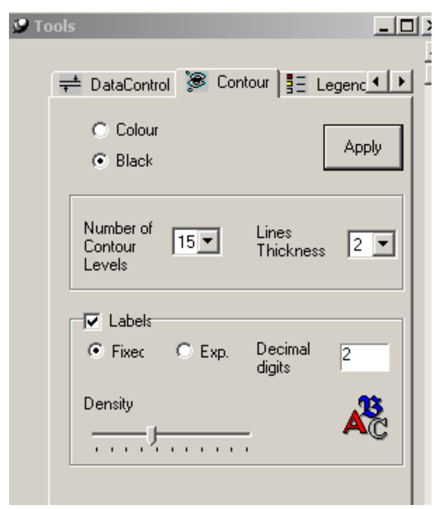


Draw Symbol displays each grid cell as a set of four data (vertices) and assigns a certain color to each data dependently of its value. **Draw Cell** calculates the average of the data located in the vertices of a grid cell. The cell will be filled with a certain color assigned to the average value. **Draw Cell around Vertex** displays your grid as a set of cells drawn around each grid vertex and filled with a certain color assigned to the data value in the vertex

  - If you select **Draw Symbol**, specify the shape and size of the symbols to be used in the **Symbol** section
  - Specify the range of data to be displayed (75% of data are displayed by default) using the sliders to the right of the **Min Data** and **Max Data** fields that show the absolute minima and maxima of your data. Or, type in these values manually. Click **Apply**

- Click on the colored squares to the right to open the standard Windows-style palette and to specify colors for the start and end of the data range

To display and customize contours



- Click the **Show Contour Lines** button on the MultiGrid toolbar to toggle the contours on and off

OR

- Right-click anywhere in your grid and choose **Show Contour** from the popup menu to appear
- Open the **Tools** dialog as described above and go to the **Contour** tab
- Select between the **Black** and **Color** options

to have your contours black or colored

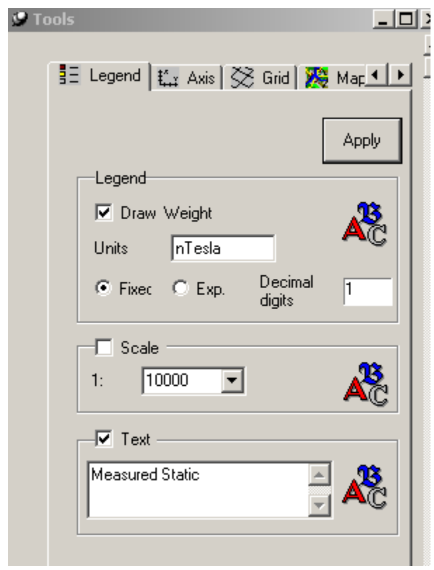
- Select the number of contour levels from the respective dropdown list and the thickness of contour lines to be drawn from the respective list to the right
- Check the **Labels** box to specify the format of the contour labels and their density in the respective section

To display and customize the legend

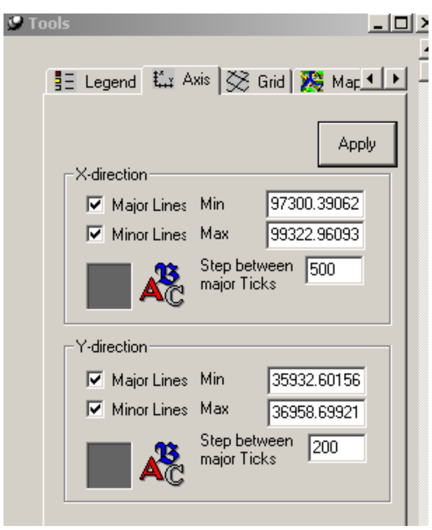
- Click the **Show Legend** button on the **MultiGrid** toolbar

OR

- Right-click anywhere in your grid and choose **Show Legend** from the popup menu to appear. You will see the legend in the left part of your grid
- Move the legend in a desired direction with your pointer cursor
- Open the **Tools** dialog as described above and go to the **Legend** tab



- Select the **Draw Weight** box to display the number of points corresponding to each color in the legend
- Specify the format (fixed or exponential), the number of decimal digits and font ( ) for the values in the legend
- Check the **Scale** box to authorize scale changes and bring up the scale rule
- Choose a required scale from the respective dropdown list and click **Apply** in the upper right-hand corner of the dialog (or click anywhere in your plot) to view the result. The scale rule will change accordingly



- Check the **Text** box to add a title or comments to your display

Type in the field below, press **Ctrl+Enter** to insert a carriage return, click the button to adjust the font and style of your labels and texts in the standard Windows-style **Font** dialog to appear

To display and customize the coordinate grid

- Open the **Tools** dialog as described above and go to the **Axis** tab
- Check the **Major Lines** and **Minor Lines** boxes in the **X-direction** and **Y-direction** sections to show

the coordinate grid

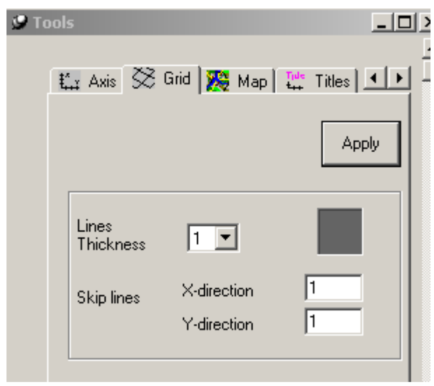
- Adjust the minimum and maximum coordinates in the respective boxes to display the portion of your grid you are primarily interested in
- Change the step between major ticks in the respective boxes to increase or decrease the density of your coordinate grid
- Click the color square in the bottom left-hand corner of the sections to edit the color using the standard Windows-style palette to appear

<u>To display and customize grid locations</u>

- Click the **Show Grid Locations** button on the **MultiGrid** toolbar to toggle grid locations on and off

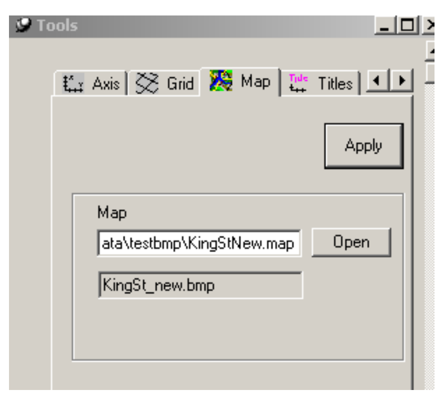
OR

- Right-click anywhere in your grid and choose **Show Grid Locations** from the popup menu to appear
- Open the **Tools** dialog as described above and go to the **Grid** tab



- Select the grid line thickness from the respective dropdown list and click on the colored square on the right to open the standard color palette and to specify the color
- Change the density of your grid as needed both in the **X-direction** and **Y-direction** boxes in the **Skip Lines** section

<u>To load a map underlay</u>



- Open the **Tools** dialog as described above and go to the **Map** tab
- Click the **Open** button to open the **Select a .map File** dialog, a standard Windows-style Open dialog. Select and load a required map

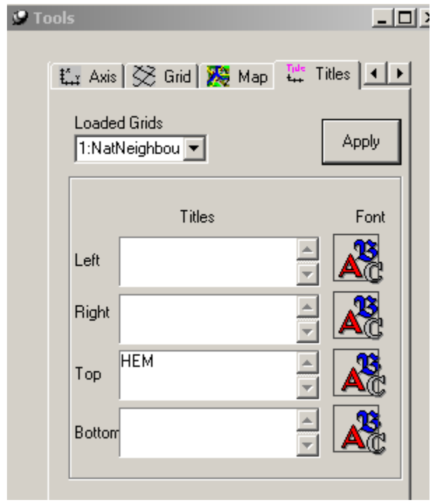
*This map is to be calibrated and saved as a text *.map file containing information on externals, internals, etc., and a bitmap (raster) file.*

- To toggle the map underlay on and off, use the **Show map** button on the **MultiGrid** toolbar

To add and customize titles

- Right-click anywhere in your grid and choose **Show Titles** from the popup menu to appear
- Open the **Tools** dialog as described above and go to the **Titles** tab
- Select the grid you want to add a title to from the **Loaded Grid** dropdown list or just click on the grid display to select it
- Type in your title in the **Left**, **Right**, **Top** or **Bottom** fields wherever you want it to appear (or create several titles)
- Click on the **Font** button to the right of each field to specify the format of your title

  If your title is too big to fit in your display, diminish the grid area using the **Zoom Out** button on the **MultiGrid** toolbar



- Click **Apply**

To switch to proportional axes

- Engage the **Show Proportionally** button on the **MultiGrid** toolbar. Your grid axes will become of the same scale
- Disengage this button to return to the initial view

Note*. If you have multiple grids displayed, all of them will be changed respectively*

To zoom in and out

- To zoom in on the whole grid, click the **Zoom In** button on the **MultiGrid** toolbar. Repeat it as many times as you need
- To zoom out, click on the **Zoom Out** button as many times as needed
- To zoom in on a grid fragment, engage the **Zoom Selected** button . Click and drag to select a grid fragment to be magnified
- To return to the initial scale, click the **Home View** button

Note. *If you have multiple grids displayed, all of them will be zoomed in or out respectively, even if you apply this operation only to one of them*

To move a grid

- Click the **Moving** button on the **MultiGrid** toolbar
- Click and drag your grid in a desired direction

- To return your grid to its initial position, click the **Home View** button 

To hide a grid

- To toggle your grid display off (if you have multiple grids, you have to select the grid to hide at first), click the **Show Grid Data** button  on the **MultiGrid** toolbar

OR

- Click anywhere in your grid and de-select **Show Grid Data** in the popup menu to appear
- To toggle the grid on again, use the same button on the toolbar or the same command from the popup menu

To remove one of the multiple grids displayed:

- Select a required grid
- Right-click anywhere in the grid and select **Hide Grid** in the popup menu to appear
- To return the grid, see the procedure in the **To display multiple grids at a time** section

To print/print preview a grid

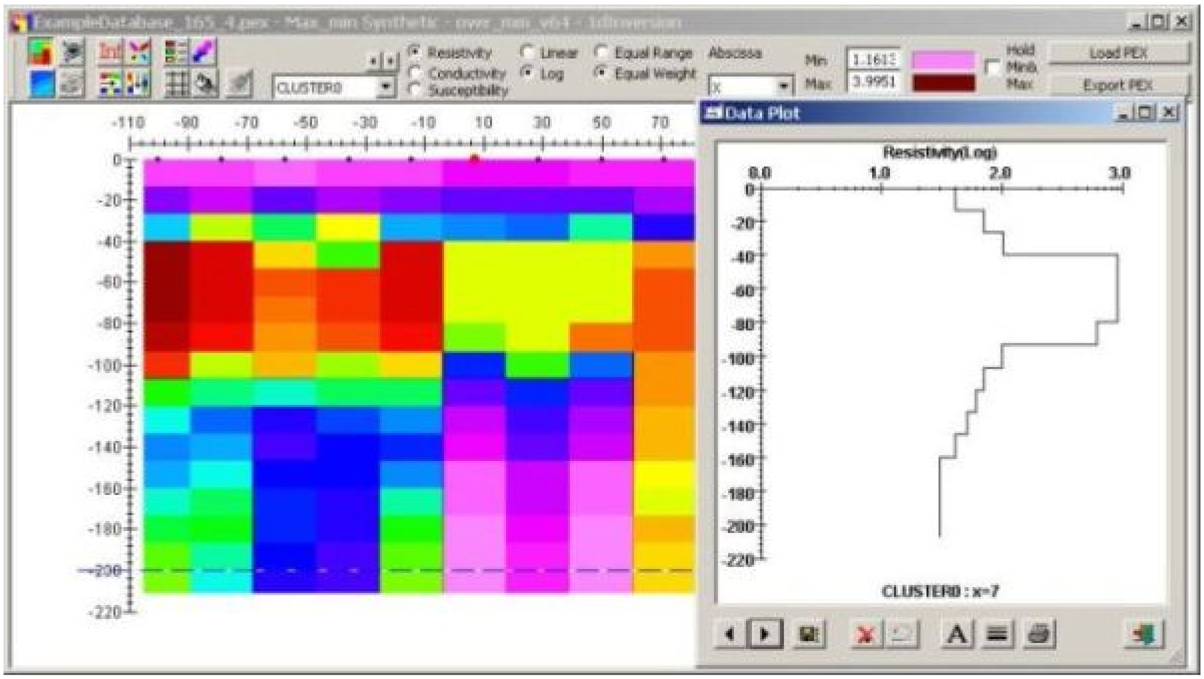
- Select **Print/File Print** and specify printing settings in the standard Windows-style dialog to appear

OR

- Select **Print/Print Preview**. Check the appearance of your grid/grids in the print preview window to open and click the **Print** button. Specify printing settings in the standard Windows-style dialog to appear

PEX Show ( )

The PEX Show tool displays your inverted EM, Resistivity, MT and CSAMT data two-dimensionally. If you have carried out 1D inversion within EMIGMA, the **Data Sets in Survey** list on the **Database** page of the main dialog will contain a *.pex file. Select it and click the now active **PEX Show** button on EMIGMA's main toolbar to open the respective application.



By default, the CDI (Conductivity Data Imaging) to appear will be the resistivity pattern for your first line. The separate **Data** dialog to simultaneously open on the right will contain the resistivity versus depth response for the first location of this line.

<u>To go through all locations of a line</u>

- Use the **Previous** and **Next** buttons in the bottom of the **Data** dialog.

  The plot in the dialog will switch to the previous or next location, respectively. The vertical red bar will simultaneously move across the CDI (in the main PEX Show dialog) indicating the exact location of the plotted data

<u>To view cell information</u>

- Click and hold in a required cell of the CDI.

  The location, depth and data will be displayed, while the **Data** dialog will automatically switch to the response obtained in the corresponding location.

  Note. If the **Data** dialog closes (and it does if you make any changes to your grid display), double-click anywhere in the grid to bring it back. The response it will show on reopening will match your current location.

To better view the CDI pattern:

- Click the button on the **PEX Show** toolbar. Your grid will be divided into equal cells, each containing one depth data

To switch to another line

- Select it from the dropdown list on the **PEX Show** toolbar or toggle through all the lines using the scroll buttons above.

To switch between resistivity, conductivity and susceptibility

- Select the required option on the **PEX Show** toolbar

To switch between the linear and logarithmic scales

- Select between **Linear** and **Log** on the **PEX Show** toolbar

To specify the draw mode

- Select **Equal Range** to assign different colors to different but equal ranges no matter how many data fall within these ranges
- Select **Equal Weight** will assign colors to different ranges which are unequal but covering the same number of data

To switch between the X and Y axes

- Select between the **X** and **Y** buttons on the **PEX Show** toolbar. They both will be active unless the direction of survey lines can be identified unambiguously.

To customize the range of data to be displayed

- Type in new minimum and maximum data in the respective boxes and click on the color squares to the right to bring up the standard color palette and specify the color of the respective data

To carry out interpolation

- Click on the **Interpolate** button Int on the **PEX Show** toolbar

OR

- Use the **Single Row Interpolation (Depths)** and **Single Column Interpolation (Locations)** buttons to specify the kind of linear interpolation. The first interpolates between different depths at the same location, the second – between different locations at the same depth
- To cancel interpolation results and return to the initial view, click the **Reset** button on the **PEX Show** toolbar

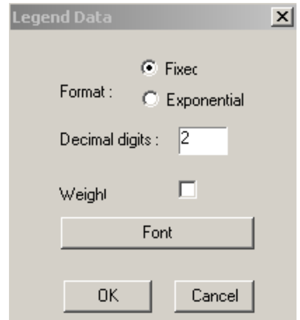
To display and customize the legend

- Click on the **Show Legend** button on the **PEX Show** toolbar or select **Data/Show Legend Window**. The legend will open in a separate window

- To display the legend in the same window with the CDI, select **Data/Show Legend in View**

In the latter case, you can customize the legend:

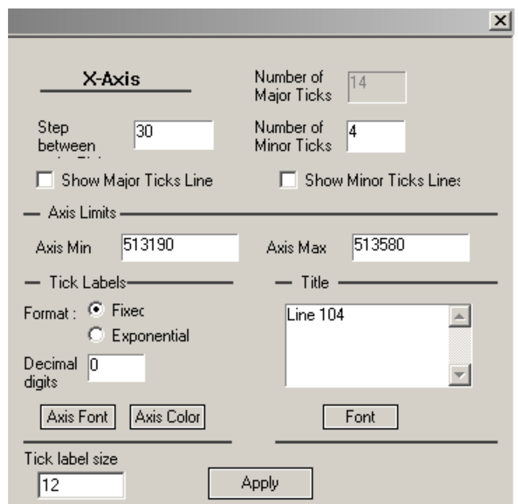
- Double-click anywhere in the legend area to bring up the **Legend Data** dialog



- Specify the number of decimal digits
- Check the **Weight** box to display the number of data covered by each color range
- Click the **Font** button to change the font, size and style of the legend labels in the standard Font dialog to open
- Click **OK**

To customize the axes

- Double-click in the region of the axis you want to adjust. The respective dialog will open:



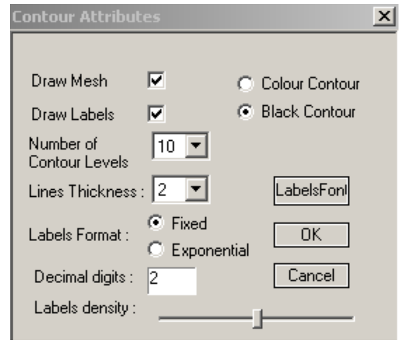
- Edit the step between major ticks in the respective box. The number of major ticks to be displayed will change accordingly
- Check the **Show Major Tick Lines** box and the **Show Minor Tick Lines** box to display the coordinate grid
- Edit the **Axis Min** and **Axis Max** values as desired in the **Axis Limits** section
- Specify the format (fixed or exponential, number of decimal digits), font, size and color of the tick labels in the respective boxes
- Type in the title of your X or Y axis in the **Title** field; use the **Font** button to specify the format of the title
- Click **Apply**

Note. *To provide the proportional scaling of the axes, use the **Proportional View** button*  *on the **PEX Show** toolbar*

To draw and customize contours

- Check the **Contour** box and click the **Show Contour** button . The **Contour Attributes** dialog will open:

- De-select the **Draw Mesh** box checked by default to hide your grid and to leave only contours



- Select the **Draw Labels** box to add the contour labels
- Select between the black and color options to have your contours in black or different colors
- Choose the contour line thickness from the respective dropdown list
- Specify the label format (fixed or exponential, the number of decimal digits, the density of labels and their font)
- Click **OK** to close the dialog and view the results

<u>To save your grid settings</u>

- Click the **Save** button on the PEX Show toolbar
- In the **Save Settings** dialog to appear, type in the name for your grid to be saved with or select the existing file from the **Settings Name** list and click **Save**. In the latter case, the existing grid will be overwritten with a new one.

<u>To load previously saved grid settings</u>

- Click the **Load** button on the PEX Show toolbar
- In the **Load Settings** dialog to appear, select the file to open from the **Settings Name** list and click **Load**

<u>To print your grid</u>

The **File** menu offers the standard **Print**, **Print Preview** and **Print Setup** functionalities.

SECTION IV - BEHIND THE USER'S INTERFACE

CHAPTER I - INSIDE THE SCATTERING ALGORITHMS

Numerical simulation code is just that: physical phenomena are simulated by a series of numerical approximations. The accuracy of the simulation, and indeed the applicability of the simulation can change significantly from one scenario to the next. Using simulation code effectively is benefited by some knowledge on the part of the user as to how it actually works.

HOW THE LN SCATTERING ALGORITHM WORKS

The LN approximation used in **EMIGMA** is an integral equation solution, so has elements in common with other integral equation solutions such as those developed at the University of Utah (Hohmann, 1975). In all such solutions, the integral equation represents the volume of ground with anomalous conductivity, that is, having a conductivity separate from the background conductivity structure. The background conductivity structure is chosen to be simple enough that incident fields inside it can be calculated either analytically or quasi-analytically. In practise, this has limited the background to either a whole space, two half-spaces (e.g., air and ground), or two half-spaces with layers.

The integral equation specifies how the anomalous conductivity alters the current flow in the ground from the current distribution that would have flowed naturally in the background medium. To do this, the electric field at select points inside the anomalous conductivity is calculated and fed into the integral equation solution. The output of the integral equation solution is the current pattern associated with the presence of the anomalous conductivity. By integrating over that current pattern, the scattered fields (magnetic and electric) associated with the presence of the anomalous conductivity can be calculated.

Like all integral equations known to the authors, the scattered fields from the LN scattering algorithm are calculated by first assigning points in each prism (representing the anomalous conductivity) where the scattered currents are to be found. This approach has the disadvantage that the modeller is often unsure where these points should be selected and how densely distributed the points should be. If one is interested in generating accurate numerical results, the usual procedure is to run the model several times, each with progressively denser sampling. The solutions so obtained may then be assured to be accurate if the results are constant with increases in sampling. A stable value appears not to be changing significantly as the anomalous body is sampled with a greater and greater density of points. Note how we say stable value rather than correct. Correct sampling of the conductor is only one of many requirements for a correct solution.

The theoretical breakthrough embodied in the LN scattering algorithm is quite significant because it embodies a radical departure about how the integral equation is solved, and therefore the scattered current distribution is found. In conventional approaches, the unknown scattering current density is represented by a set of basis functions, each interacting with the other, and each representing a current distribution with a known shape and an unknown amplitude.

In the conventional approach then, solving for this representation of the scattering currents with the integral equation yields a set of amplitudes for the current distribution, and the solution is inherently limited by that choice. The LN scattering algorithm uses no explicit basis to represent the scattering current density. The current density can be calculated at any point inside the prism using a semi-analytic transform which maps the incident field onto the scattered current density. In this regard the solution is not dissimilar to the propagation matrix method for computing the electric field inside a layered structure.

The transform is defined semi-analytically and thus is represented by a 3 by 3 matrix, the solution is extremely rapid compared with conventional techniques, particularly when more than 50 points are used to represent the current inside the scatterer. The time savings delivered by LN scattering algorithm increases dramatically as the number of points increases. Although accuracy of the LN approximation is limited to current channelling responses, the theory has been extended to inductive modes via the ILN scattering algorithm.

Scattering capabilities include single or multiple prisms combined with single or multiple polyhedra. Prisms are fully rotational with arbitrary aspect ratios. These algorithms allow one to model a host of different physical effects including contrasts in conductivity, electrical permittivity and magnetic susceptibility. We recommend this algorithm for DC Magnetics, IP/Resistivity including EM effects, CSAMT and MT surveys.

Note: The LN algorithm utilizes a complex conductivity formed from the conductivity, electrical permittivity and the Cole-Cole parameters. The LN algorithms also allows for variation in the magnetic susceptibility calculating a resulting magnetic current flow analogous to galvanic current flow to add the effects of the susceptibility variations to the resulting scattered field.

HOW THE ILN SCATTERING ALGORITHM WORKS

The ILN approximation used in **EMIGMA** is an integral equation solution, so has elements in common with other integral equation solutions such as those developed at the University of Utah (Hohmann, 1975). In all such solutions, the integral equation represents the volume of ground with anomalous conductivity, that is, having a conductivity separate from the background conductivity structure. The background conductivity structure is chosen to be simple enough that incident fields inside it can be calculated either analytically or quasi-analytically. In practise, this has limited the background to either a whole space, two half-spaces (e.g., air and ground), or two half-spaces with layers.

The integral equation specifies how the anomalous conductivity alters the current flow in the ground from the current distribution that would have flowed naturally in the background medium. To do this, the electric field at select points inside the anomalous conductivity is calculated and fed into the integral equation solution. The output of the integral equation solution is the current pattern associated with the presence of the anomalous conductivity. By integrating over that current pattern, the scattered fields (magnetic and electric) associated with the presence of the anomalous conductivity can be calculated.

Like all integral equations known to the authors, the scattered fields from the ILN scattering algorithm are calculated by first assigning points in each prism (representing the anomalous conductivity) where the scattered currents are to be found. The ILN technique then utilizes point clusters about the target points inside the scatterer. As with the LN technique, this approach has the disadvantage that the modeller is often unsure where these points should be selected and how densely distributed the points should be. If one is interested in generating accurate numerical results, the usual procedure is to run the model several times, each with progressively denser sampling. The solutions so obtained may then be assured to be accurate if the results are constant with increases in sampling. A stable value appears not to be changing significantly as the anomalous body is sampled with a greater and greater density of points. Note how we say stable value rather than correct. Correct sampling of the conductor is only one of many requirements for a correct solution.

The ILN scattering algorithm is quite significant because it again embodies a radical departure about how the integral equation is solved, and therefore the scattered current distribution is found. In conventional approaches, the unknown scattering current density is represented by a set of basis functions, each interacting with the other, and each representing a current distribution with a known shape and an unknown amplitude. In the conventional approach then, solving for this representation of the scattering currents with the integral equation yields a set of amplitudes for the current distribution, and the solution is inherently limited by that choice. The ILN scattering algorithm (like its predecessor, the LN technique) uses no explicit basis to represent the scattering current density. The current density can be calculated at any point inside the prism using a semi-analytic transform which maps the incident field onto the scattered current density. In this regard the solution is not dissimilar to the propagation matrix method for computing the electric field inside a layered structure.

The transform is defined semi-analytically and thus is represented by a 12 by 12 matrix, the solution is extremely rapid compared with conventional techniques, particularly when more than 50 points are used to represent the current inside the scatterer. The time savings delivered by ILN scattering algorithm increases dramatically as the number of points increases. The LN approximation has been demonstrated to be effective for modelling both current channelling and highly inductive responses.

This algorithm is also available for single or multiple prisms combined with single or multiple polyhedra but we recommend it for systems which have inductive type sources such magnetic dipoles or loops and magnetic receivers which are measuring dB/dt.

HOW THE VH SCATTERING ALGORITHM WORKS

The VHPLATE, like the LN scattering algorithm is an integral equation solution, and so has elements in common with other integral equation solutions such as those developed at the

University of Utah (Hohmann, 1975). In all such solutions, the integral equation represents the volume of ground with anomalous conductivity, that is, having a conductivity separate from the background conductivity structure. The background conductivity structure is chosen to be simple enough that incident fields inside it can be calculated either analytically or quasi-analytically. In practise, this has limited the background to a whole space, two half spaces, or two half-spaces with layers.

The integral equation specifies how the anomalous conductivity alters the current flow in the ground from the current distribution that would have flowed naturally in the background medium. To do this, the electric field, at select points inside the anomalous conductivity, is calculated and fed into the integral equation solution. The output of the integral equation solution is the current pattern associated with the presence of the anomalous conductivity. By integrating over that current pattern, the scattered fields associated with the presence of the anomalous conductivity can be calculated.

Like all integral equations known to the authors, the scattered fields in the VHPLATE scattering algorithm are calculated by first assigning points in the plate where the incident electric field to be found. In the current version, the incident electric field is found at 441 points (a mesh of 21 by 21 points) equally distributed in each of the two larger dimensions of the plate. These input electric fields are then used to compute the scalar potential, magnetic field and then the scattering current distribution on the plate using a basis of polynomials ranging continuously from order 0 through 5 inclusively.

CHAPTER II - MODEL SPECIFICATION AND PROGRAM CONTROL

This chapter deals with specific issues in regard to operating ***EMIGMA-Forward V7.x.*** For a more complete discussion of how to use these inputs to represent a particular geophysical system and geological structure, refer to Chapter III on Effective Modelling, Inversion and Data Analyses

PRISM INPUTS

To simulate the complex conductivity structure of the ground, a series of right rectangular prisms, each with its own set of physical properties, can be used. The prisms may be combined to build up irregular conductivity structures which may represent overburden thickness variations, fractures, lithology changes and a host of other geological structures. These would have conductivity, permittivity and permeability contrasts in relation to the background layer conductivity.

Scattering calculations are based on the Localized Non-Linear approximation (Habashy, Groom and Spies). This method is extremely rapid and is of particular significance for calculating scattering from complicated geometrical shapes. For such shapes, execution times 1000 fold less than alternate solution methods can be achieved. In this version, a new algorithm called the Inductive Localized Non-Linear (ILN) Approximation (Murray) is also available.

THEORETICAL LIMITATIONS: The Localized Non-Linear approximation, as is currently implemented, performs best when the scatterers are energized in a current channelling mode or when energization is weakly inductive. When the energization is strongly inductive, the response tends to be underestimated and the ILN or VH technique should be used. As a rule of thumb, the method tends to work the best when the scatterers are located in a conductive background layer. For magnetic (loop) transmitters, best results are obtained when loops are are off to the side of the conductor. For grounded electrical transmitters and receivers, and for natural source fields, the location of the prism relative to the receiver is not as important.

BUILDING THE MODEL: The prism model is built by specifying the number of prisms desired, and then by assigning geometric (locations of the faces of the prisms) and physical (conductivity, magnetic permeability, electrical permittivity) properties. Also, 3 rotation angles (essentially strike, dip and plunge) may be assigned plus the algorithm to be used should be indicated. At the user's option, the prisms can be assigned Cole-Cole polarization parameters. Each prism is assigned its particular characteristics in accordance with a unique index. The prism is located as to its top point or center point in the absolute frame. The user should view the prism location in the Visualizer prior to modelling. This can be done by saving the file after the user is satisfied with the model (it is not necessary to execute the model) and then opening the file into the Visualizer. The model can then be modified or run directly from the Visualizer.

CONVERTING TO POLYHEDRA: Prisms and plates can be selected and converted to polyhedra in the Visualizer. They can then be modified into more complex shapes by selecting and either pulling a corner, or by specifying the XYZ location of the corner through an interface). Tetrahedran primitives, of a specified height, can also be added onto a face of the body.

PLATE INPUTS

The plate model is a simple model used to simulate the effect of dipping tabular structures. Such structures can be as varied as massive sulphide ore lenses, graphitic sheets, alteration zones associated with a contact, shear zones or even lenses of ground water. In the model used here, the plate must be more conductive than the ground surrounding it. Resistive plates which would impede the flow of current are not supported. Also, current flow (TM mode) across the face of the plate has no response.

Scattering calculations are based on the VH algorithm (Walker and West). The method is extremely effective for modelling electromagnetic scattering from thin bodies having both current channelling and inductive responses, and so provides a robust solution when the nature of the scattered response is not known.

__THEORETICAL LIMITATIONS:__ In the current implementation of the *VH* solution, the source fields and the scattering currents are expressed with polynomials ranging in order from 0 through to 5 inclusively. Thus if the energizing fields over the scattering currents on the plate require higher orders of polynomials to be represented effectively, the solution will be inaccurate. Such situations can occur for example when the plate is energized by a dipole, and the dipole is located less than ½ of a plate dimension from the plate. The range of the polynomials used in the solution also effectively limits the induction number of plates that can be accurately simulated to approximately 10,000.

The plate is assumed to be infinitely thin. The thin edges of the plate therefore present an infinite impedance to currents, and all currents must therefore exit or enter the plate along its broad faces. This assumption can cause subtle differences when results are compared with algorithms where the conductor is assumed to have finite thickness, and where current can both enter and leave through the edges. Note that because the plate is thin, and no current can enter or leave through the edges, so the side of the plate cannot be in contact with a layer of different conductivity. Additionally, the theoretical formulation restricts the plate to lie within a single layer.

Although the plate is infinitely thin, it does have a finite conductance or conductivity-thickness product. Conductivity thickness refers to the relative conductivity between the plate and the background conductive layer, and must be positive. Assumptions made when deriving the theory limit the upper bound of the relative conductivity of the plate. The limit is a value such that the skin depth of the fields within the plate must be much greater than the thickness of the plate.

__BUILDING THE MODEL:__ The electrical parameters defining the plate are its conductance and Cole-Cole parameters. Two different modes can be used to locate the plate within space: a

Center Point mode and ***Top Point*** mode. Three angles of rotation are allowed (strike, dip and plunge) and multiple plates may be utilized.

RESTRICTIONS:

The plate must lie totally within a single layer and should have an induction number less than 10,000 and a channelling number less than 1,000,000. It may not intersect any other anomalous structure, such as a prism. The ratio of the long dimension of the plate to the short dimension should not exceed 9:1. Plates are available only to users who have acquired plate licenses.

AUXILLIARY FILES: In order to compute the response efficiently, files are built and maintained in the \PREIKON directory. These files are not essential to the calculation, and if deleted will be automatically rebuilt. Files have three classes NRSS*, XRSS* and YRSS*. The * takes on a value diagnostic of particular ratios of strike extent to dip extent.

GEOMETRIC MODES

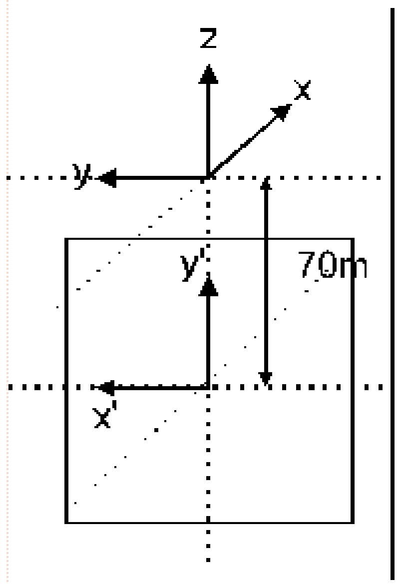
In the following discussion, the primed variables denote the axis system on the plate and the unprimed coordinates the absolute system. The plate axis system is oriented such that the *x'* axis points towards the strike of the plate and the *y'* axis is parallel to its dip extent. For a plate with 0 strike and dip, the unprimed axis system is parallel to the primed axis system.

In ***(C)enter Point*** mode, the plate is located in space by defining the location of its center point in the absolute frame. This mode is most useful for modelling plates which are horizontal or sub-horizontal. If the plate has neither strike nor dip, then its *x'* and *y'* axis are parallel to the absolute *x* and *y* axes respectively. The dimension parallel to the *x'* axis corrseponds to the strike extent and that parallel to the *y'* axis the dip extent. A positive strike angle rotates the plates *x'* towards the absolute *y* axis, so that with a 90 degree strike for example, the *x'* axis of the plate will rotate around absolute *z* axis and will be aligned with the absolute *y* axis. A positive dip angle pushes the plates *y'* axis downwards rotating around the *x'* axis, so that a plate with a 90 degree dip will have is *y'* axis parallel to the negative *z* axis in the absolute system. Note that this dip angle convention is opposite to the top point dip angle convention.

In ***(T)op Point*** mode, the plate is located in space by defining the location of its top centre point in the absolute frame. If the plate has no strike or dip, then the *x'* and *y'* axes of the plate lie parallel to the *x* and *y* axes of the absolute frame, and the plate extends equally in either *x* direction from the *x-y* origin and has its full dip extent along the negative *y* axis. A positive strike angle will move the *x'* axis of the plate towards the absolute *y* axis. A positive dip angle will cause the bottom of the plate to drop: that is, when viewing towards the direction of the positive *x'* axis of the plate, the plate will dip to the right by the specified amount. Note that this dip angle convention is opposite to the center point dip angle convention. A 90^0 dip aligns the plate's *y'* axis with the absolute -*z* axis.

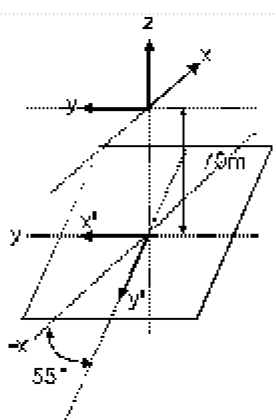
PLATE EXAMPLES

In the example shown, center point input was selected, and the three step-by-step examples following will illustrate center point inputs of increasing complexity.



EAST EXTENT 100	
North EXTENT 100	
<X> Center Location	0
<Y> Center Location	0
<Z> Center Location	-70
STRIKE ANGLE	90
DIP ANGLE	-90
PLUNGE ANGLE	0

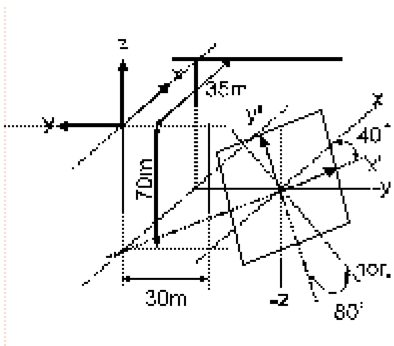
In the afore mentioned example, the plate has dimensions of 100 meters, and its top is buried at a depth of 20 meters. The plate is oriented vertically and with the axis normal to it. Note that the *x*' and *y*' axes lie in the plane of the plate, and represent the original position of the plate. Before the strike and dip rotation of the plate, the *x*' and *y*' axes were coincident with the *x* and *y* axes. The following 2 examples show more complex geometries and the inputs necessary to produce them.



STRIKE (East) EXTENT	100
DIP (North) EXTENT	100
<X> Center Location	0
<Y> Center Location	0
<Z> Center Location	-70
STRIKE ANGLE	90
DIP ANGLE	55
Plunge ANGLE	0

This second example is identical to the first except for the dip angle. This has been entered as +55 and thus the *y*' plate axis was dropped into the ground and rotated 55 degrees away from the *z* absolute axis.

In this third and final example, the strike angle is negative since the *x*' plate axis is rotated 40 degrees towards the negative *y* absolute axis. Alternatively, we could have entered 140 degrees. The dip angle is -80 degrees since the *y*' plate axis is rotated towards the +*z* absolute axis.



STRIKE (East) EXTENT	50
DIP (north) EXTENT	50
<X> Center Location	35
<Y> Center Location	-30
<Z> Center Location	-70
STRIKE ANGLE	-40
DIP ANGLE	-80
Plunge ANGLE	0

PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

COLE-COLE INPUTS

The Cole-Cole model can be used in conjunction with the layered earth model, and with either plate, prism or polyhedra scatterers. It allows the modeller to account for frequency dependent conductivity effects which are often associated with low frequency induced polarization phenomena. It should be also noted that at higher frequencies, electric permittivity is also responsible for frequency dependent phenomena. In the Cole-Cole model, the conductivity is complex, so currents are shifted in phase from the electric fields that produce them. The Cole-Cole model is discussed in further detail in Section III under the heading "Choosing Physical Parameters". An internal report is also available on IP effects which the user may find helpful.

The Cole-Cole model requires three inputs, a chargeability m, a time constant t and a frequency dependance c. These parameters augment the conductivity/resistivity, magnetic permeability and electric permittivity parameters which are used to characterize the properties of an electrical structure within ***EMIGMA***.

MT STATION INPUTS

For an MT station, between four and six receivers are simulated. Three of the receivers measure the magnetic field intensity (rather than its derivative) with respect to the MT survey coordinate system; another one measures the vertical electric field; and two measure the potential that would be measured in a pair of orthogonal cables laid out in the MT coordinate system *x* and *y* directions. Using finite length E field receivers allows the effects of local structure on an MT measurement to be modelled.

The sensors are oriented in the MT survey measurement coordinate system. The *z* axis of this coordinate system is up, and the *x* axis of the coordinate system is specified as a declination counterclockwise from the absolute *y* axis. (If *y* corresponds to north, declination is measured east of north.) North is also assumed to be the *y* direction in the absolute coordinate system of the model. It is suggested that for ***EMIGMA***, that the user utilize the strike direction of the conductivity structure as the absolute *y* direction. In ***EMIGMA*** *Version 3*, the prisms must be aligned so that the absolute coordinate axes are normal to the faces (i.e.: no strike rotation, dip, etc. of the prisms is allowed). However, if you are utilizing Version 6 and above, you are allowed a strike rotation, dip and a plunge angle.

To simulate a MT measurement, 3 inputs are required: The declination angle of the electric field polarization, the strike angle of the local MT coordinate system x axis, and the length of the electric field receiver cables. The declination angle is a relative angle with respect to the absolute *y* axis. Usually, the *y* axis will be coincident with the strike axis of your conductivity structure. The polarization angle and the strike of the local MT *x* axis are entered in degrees east of the absolute *y* axis, so in the case where the absolute *y* axis points northwards, northeast would have a strike of +45 degrees while east would be represented by 90 degrees. The MT *y* axis direction in these examples is north 45 degrees west and due north, while *z* is assumed to point away from the ground.

The magnetic field sensor, the vertical electric field sensor, and the center points of the horizontal electric field sensors are assumed to be at the local station origin. The absolute position of the local station origin is located at a point determined by the station location on the selected profile, as determined in the profile input section.

The survey site parameters are given with respect to the absolute coordinate system. In this manner, the strike of the structure, the orientation of the MT survey system and the profile can all be independent. This is appropriate if the site definition and the survey profile are not perpendicular to the strike direction of the structure.

Impedance Tensors:

To create impedance tensors, the user must compute the model twice for 2 orthogonal polarizations.

Note:

In V6.0 and above, both polarizations can be computed automatically by utilizing the OUTPUT page and the impedance tensors are created to the output ***gsflds.dat*** file. This file may then be read into **EIKPLOT** for viewing either the fields or the impedance tensors as apparent resistivity and phase. All elements of the tensor may be viewed either in the measurement orientation or in any rotated position. If the user wishes to export the ***gsflds.dat*** file to .EDI or .G format then they may pass the file through Gbformat_win.exe. Tipper vectors are also computed in this process.

RESTRICTIONS: MT inputs are available only to users who have acquired MT licenses.

CONTROLLED SOURCE INPUTS

In the controlled source input section of ***EMIGMA*****,** there is facility to enter loops, bipoles and dipoles as either transmitters or receivers. Two modes of a survey are supported, either fixed or moving source surveys, and a great deal of flexibility is provided to allow transmitters and receiver to be oriented in a number of possible coordinate systems. This flexibility means that virtually any commercially available electromagnetic system can be simulated with ***EMIGMA.*** To the geophysicist, this means that the same software can be used to simulate the results of an airborne survey, through the ground follow-up phase and into detailed borehole interpretation.

RESTRICTIONS: Controlled sources are available to licensed users only. In the current release, loops must lie in a single plane.

ANTENNAE :

LOOP : A ***loop*** is an antenna that is inductively coupled to the earth. There is no direct electrical contact with the ground, either along the antenna or at its ends if the loop is open. All coupling with the ground is radiative. Loops could be specified as transmitting antennae for systems such as UTEM, EM37, Crone PEM, Sirotem and Zonge systems, and as receiving antennae for dual-loop or Slingram type Sirotem or Zonge surveys. Simulated output from a loop receiver is a voltage.

Loops are input by defining a list of vertices, between which straight segments of wire are assumed to run. The sense of current flow in a loop is positive when proceeding through the list of vertices. Note, that to simulate a closed loop, the last vertex must have the same location as the first vertex. The elevation of a loop should not coincide with an interface in the layered earth model, since the ability of a wire to radiate depends on the physical properties of the medium in which it is located. If the antenna is located on an interface, the physical properties of the medium are not unique.

BIPOLE : A ***bipole*** is an antenna that is both inductively and galvanically coupled to the earth. In this sense, the fundamental difference between a loop and a bipole is the electrical contact made by the bipole at its ends with the ground. Bipoles would be used as transmitting antennae for IP-resistivity, LOTEM and CSAMT surveys. It should be noted that the bipole must be in electrical contact with the ground, so it should not be placed in the air layer. Simulated output from a bipole receiver is a voltage.

Bipoles are input by defining a list of vertices, between which straight segments of wire are assumed to run. The sense of current flow is positive when proceeding through the list of vertices. Note, that a closed bipole should be the same as a closed loop. Physically it is, but there are numerical problems associated with cancelling the galvanic terms properly for a closed bipole, **so bipoles should not be used to simulate inductively coupled closed loops**. The elevation of a bipole should not coincide with an interface in the layered earth model.

☞ *NOTE: The offset of the antenna from an interface should be at least 0.1 m. Bipoles should be placed below the air level.*

<u>DIPOLES</u> : A dipole is a useful representation for sources that are small enough to be treated as points. A useful rule of thumb for deciding whether or not to represent an antenna with a dipole is that the dimensions of the source should be no more than 1/10th of the other dimensions in the survey. The elevation of a dipole should not coincide with an interface in the layered earth model.

There are two types of dipoles available in ***EMIGMA.*** An ***electric dipole*** represents a short wire which is grounded to the earth at each ends. In this respect, the electric dipole is similar to a bipole. However, simulated output from an electric dipole is an electric field in volts/meter. The electric dipole is calibrated to a 1m bipole in the far-field locations.

The second type of dipole available in ***EMIGMA*** is a ***magnetic dipole***. A magnetic dipole simulates a single turn loop of wire with 1 ampere of current through it, and is similar to a loop source. The orientation of the dipole is however, orthogonal to the plane of the loop of wire it represents, with the sense of the orientation defined by the right-hand rule. Realizing that the magnetic dipole represents a circular loop of wire is important. Subtle differences will be noted if the radiation pattern of a magnetic dipole is compared with the radiation pattern of a square loop with unit area. (It is for this reason that fields from loops in ***EMIGMA*** are not computed using the equivalent moment of a magnetic dipole.) Simulated output from a magnetic dipole receiver is different from a loop in that units returned are Amperes/meter, that is, magnetic field intensity.

x, y and z directions are relative to the germane coordinate system (see coordinate systems below).

Survey Options:

T*wo options are available for modelling a controlled source survey: the fixed transmitter and the moving transmitter modes. The fixed transmitter mode is useful for simulating CSAMT, LOTEM and large loop - moving receiver surveys such as those commonly used in MELIS, UTEM and EM-37 surveys. The moving transmitter mode is useful for simulating Max-Min, EM34, dipole-dipole resistivity, Sirotem Slingram and inloop geometries, HLEM and SAMPO, and airborne surveys such as HEM and GEOTEM. Also included is the facility to compute more than one component of the field at any point if dipole receivers are selected. This for example, is useful for simulating three component borehole surveys. One could also use the moving transmitter mode to simulate the response of an in-hole logging tool.

FIXED TRANSMITTER : In a fixed transmitter survey, the transmitting antenna is specified in absolute coordinates, and is assumed to be fixed in one location. When receivers are defined, their antennae are defined in coordinates relative to the point on the profile where a measurement is being taken, and not necessarily in absolute coordinates. ***EMIGMA*** will compute the absolute location of the receiver required to simulate the response at a point on the profile from

a) the definition of the profile,

b) the selected relative coordinate system (absolute, profile, borehole or horizontal; see the following section) which has been selected for specifying the orientation of the receiver and

c) the coordinates of the receiver relative to the point on the profile.

MOVING TRANSMITTER : In a moving transmitter survey, both the transmitter and the receiver are specified in relative coordinates with respect to a point on a profile. In this respect, transmitting and receiving antennae are treated in exactly the same fashion as receivers are in fixed transmitter surveys.

There is one additional parameter to take note of in moving transmitter surveys, however, and that is the transmitter-receiver ***separation***. The presence of this parameter is a fundamental difference between the fixed transmitter and the moving transmitter modes. Like the transmitter and receiver geometries, the separation is defined in terms of the current relative coordinate system. The transmitter will have its reference point displaced from the reference point of the receiver by the separation. When ***EMIGMA*** locates the transmitters and the receivers to carry out the field calculations, they will be displaced from one another by the separation. Multiple separations are supported in **EMIGMA V5.**

☞ *NOTE: The separation is specified with respect to the same germane coordinate system used to specify the orientation and coordinates of the antennae (see coordinate systems below).*

The location of the transmitter and the receiver are fixed relative to each other by the separation, but they must also be fixed to the currently active profile position (plot point). To do this, there is a ***moving system reference point*** parameter. Selecting a centered reference point will fix the midpoint of the separation at the plot point; the reference point for the transmitter will then be displaced +½ the separation from the currently active profile position and the receiver reference point will be displaced -½ the separation from the profile position. (This is useful for simulating HEM and HLEM surveys.) However, selecting the transmitter as the reference point will locate the transmitter reference point at the currently active profile position. (This is useful for simulating GEOTEM surveys).

Take for example, a Wenner resistivity survey which would naturally be modelled with a centered reference point. This survey can be defined either with 0 separation, or with a separation of *na*. The following table tells the story of how the bipoles can be located at identical positions with two different separation parameters.

Using the separation parameter: An example using a Wenner resistivity survey

Separation	Tx electrode 1	Tx electrode 2	Rx electrode 1	Rx electrode 2
0	(n-1)a/2	(n+1)a/2	-(n+1)a/2	-(n-1)a/2
na	-a/2	a/2	-a/2	a/2

MULTIPLE COMPONENT : Several systems on the market have the facility to measure more than one orientation of the field at every point on the profile. The modeller can specify the number of field components needed, rather than having to rerun ***EMIGMA*** for every field component, and then specify the orientation codes for each component to be calculated.

COORDINATE SYSTEMS:

If a fixed transmitter geometry is being used, the transmitter is always located in the absolute coordinate system. The coordinate system which is germane to the receiver in a fixed transmitter geometry, and to the transmitter, receiver and separation in moving transmitter geometries can be specified by the modeller. In the current version of ***EMIGMA***, four choices of coordinate system are available for specification by the modeller: absolute, borehole, profile and horizontal. The orientation of these coordinate systems is constant for all points along a profile.

In the ***absolute*** system, orientation is parallel to absolute model coordinates. This is the simplest coordinate system to visualize and is useful for model studies, but is not very practical for modelling actual field data.

In the ***profile*** coordinate system, the *x*-axis is located parallel to the direction of the profile, the *y*-axis is horizontal, and on the left-hand side when viewing down the profile and the *z*-axis is perpendicular to the *x*- and *y*-axes and is pointing up. Profile coordinates are useful when the orientation of system components is relative to the profile, even when the profile has an elevation change associated with it. For example, the profile coordinate system would be useful for simulating the response of an HLEM/MaxMin survey when the profile is not at a constant elevation (so as to have the transmitting and receiving loops occupy the same plane).

The ***borehole*** coordinate system is similar to the profile coordinate system. In the borehole system however, the *z*-axis always point upwards in the line of the profile, the *x*-axis is horizontal, and the *y*-axis is orthogonal to the two in a right-handed system. In this sense, the projection of the y-axis on the horizontal is the azimuthal direction of the borehole. The borehole system is useful for modelling, what else but *borehole surveys!*

In the ***horizontal*** coordinate system, the *x*-axis is the horizontal projection of the profile, the *y*-axis is also horizontal and on the left-hand side when viewing down the horizontal *x*-axis and the *z*-axis is parallel to the absolute *z*-axis. Thus, the horizontal system is just a rotation of the absolute system about the absolute *z*-axis by an angle equal to the strike angle of the profile. The horizontal system is useful for modelling systems which are oriented along a profile, but are levelled with respect to true vertical and horizontal, such as is done either with level bubbles or when the sensor is assumed to be horizontal (such as in a HEM survey), or in large loop surface surveys.

CONTROLLED SOURCE EXAMPLES

TRANSMITTER/RECEIVER SECTION

The transmitter/receiver input section is headed with a banner, following which, input is required to define the transmitter as fixed or moving, to define the type of antenna to use for the transmitter and receiver, and the coordinate system to use for orienting the receiver.

Following this set of preliminary inputs, control will branch to various routines depending on the type of antenna specified. Examples dealing with how to respond to those inputs are considered in separate sections. The two examples to follow illustrate how to model (F)ixed and(M)oving transmitter surveys. The example specifying the fixed transmitter is illustrative of a borehole survey with a loop source while the example with the moving transmitter is illustrative of a resistivity survey.

☞ *NOTE: Inputs may vary depending on the nature of the license. License holders who, for example, have only purchased dipole licenses will be prompted only for the transmitter and receiver dipole types.*

LOOP ANTENNA

Fixed loops are input in absolute coordinates, moving loops are input in relative using the coordinate system specified by the user. The required inputs are the number of points in the loop, its fixed elevation, and the x and y coordinates of all the vertices. Note that to close the loop, one more point than the number of sides in the loop is required. Also, the loop should be placed in the air layer if it is used to simulate a loop which has been laid out on the ground.

The fundamental difference between a fixed loop and a moving loop is that the elevation of a moving loop is defined by the elevation of the current point on the profile and the *z* separation between the transmitter and the receiver. The elevation of a fixed loop is defined by an input parameter.

Note: For a Moving system, loop corners are relative to the measuring point and separation distance.

BIPOLE ANTENNA

Fixed bipoles are input in absolute coordinates; moving bipoles are input in relative coordinates. The required inputs are the number of points in the bipole, its elevation if a fixed transmitter, and then the x and y coordinates of all the vertices. Bipole inputs are essentially the same as loop inputs.

The fundamental difference between a fixed bipole and a moving bipole is that the elevation of a moving bipole is defined by the elevation of the current point on the profile and the *z* separation. In the example below, the fixed bipole is typical of a cable laid out along a road, while the moving bipole is typical of a 50-meter cable centered at the receiver reference point (or of a similar moving source transmitter). Note: For grounded bipoles, ensure that the Z co-ordinates of the bipole are in the ground.

Note: In V6.3 bipole antennae are not restricted to a fixed elevation.

DIPOLE ANTENNA

Dipole orientations and types are specified through their codes (Mj or Jj for the Tx, Hj or Ej for the Rx), and are oriented according to the germane coordinate system, as with loop and bipole transmitters and receivers.

PROFILE INPUTS

Profiles are used to define the location of a measurement in space. They are defined by a series of line segments, each having a start point, an end point and a number of sample positions. A profile which, for example, starts just below the surface of the earth and extends downwards can be used to simulate a borehole. Conversely, profiles that start just above the surface of the earth and rise to a different altitude can be used to provide a first order estimate of the effects of topography (ignoring the electrical effect of the ground above z=0). Specific profile or borehole trajectories can be specified by importing an ASCII file such as a borehole log file.

NORMALIZATION INPUTS

The normalization inputs control which fields are outputted by ***EMIGMA*** and how they are normalized. In some systems, such as Sirotem or in DC resistivity, no normalization is used, and outputs are typically in voltages. In other systems, such as HEM, HLEM, GEOTEM and UTEM, the fields are normalized to equivalent fields in free space. Finally, in a third class of systems such as MELIS, signals are measured with respect to a fixed reference coil. In such cases normalization should be done with respect to the layered earth model used in the field calculations.

NOTE: For time domain systems, normalization is performed in FSEMTRS.

If one does not want to apply any normalization, ***absolute fields*** should be specified. If normalization is desired, a selection of multiplicative factors are available for the normalized output. There is also an option to normalize to the ***complex*** field, its ***magnitude*** or either its ***in-phase*** or ***quadrature*** components. One should be aware that when normalizing to the magnitude of the field, the sign is lost, and outputs may be reversed from what was expected. Normalized HEM data, for example, are often presented in parts per million (***PPM***) where as HLEM data are usually normalized as ***percent***.

It is often useful to normalize to a different spatial field ***component*** than the component of the field measured in the survey. This is particularly true for "null coupled" transmitter and receiver configurations such as the Dighem "whaletail" system. However, it is also useful for borehole surveys, where the receiver may be null coupled or poorly coupled with the primary field over a part of the hole. In such cases, three-component normalization is often useful. In three-component normalization the magnitude of the total field is used as the normalizing field. ***EMIGMA*** offers the full range of one, two or three component normalization.

OUTPUT FIELDS:

Scattered fields (secondary fields) are generated from currents flowing inside prisms, plates or other anomalous structures which would not have been present had those structures been absent. Scattered fields can be thought of as the “anomalies” which are sought after in exploration geophysics.

Host/Incident fields are reflected from the electrical property contrasts in the layered earth model. To simulate the signal from surface or airborne prospecting systems which back out the primary field for example, computing the sum of the scattered and host fields is often useful. ***EMIGMA*** offers the choice of outputting the sum of the scattered and the host field.

When the transmitter and receiver are in different layers, i.e. in surface-to-borehole surveys, the ***total fields*** include the scattered and host fields, but also the field transmitted directly from the transmitter to the receiver through the layer interfaces. (The total field represents the entire signal sensed by the receiver.) When the transmitter and the receiver are in the same layer, the total field again includes the scattered and reflected fields, but also the fields that would be measured if the transmitter and receiver were in a uniform whole space. Total fields are often useful for simulating crosshole and surface-to-borehole surveys.

Freespace fields are generated as the response of the system to an absolutely resistive environment or vacuum. In EMIGMA this is approximated as 10e8 Ohm-m. Air is also given this resistivity so in effect, the freespace field is the response to the system in a whole-space air model.

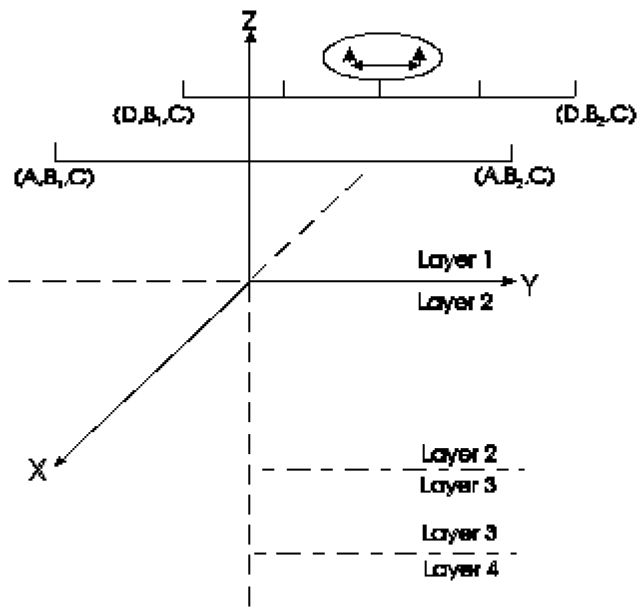
A final choice for field output is the ***total field minus the free space field***. This output is useful for simulating some borehole surveys. For surface and airborne surveys, one would expect that the total field minus the free space field should be equivalent to the sum of the reflected field and the scattered field. In principle this is so, but numerical round off errors can make a difference, and calculation of the sum of the scattered and reflected fields from the difference of the total and the free space field can yield poor results.

DIRECTORY OF FIGURES

Figure	2-1	Earth Coordinate System
Figure	2-2	Prism Coordinate System
Figure	2-3	Specifying Plate in Geological Mode
Figure	2-4	Plate Reference Points
Figure	2-5	Magnetotelluric Inputs
Figure	2-6	Moving System Plot Points
Figure	2-7	Loops, Bipoles and Dipoles
Figure	2-8	Fixed Loops and Extended Bipoles Are Defined in Absolute Coordinates
Figure	2-9	Moving Loops Can Be Defined in Absolute Ir Profile Coordinates
Figure	2-10	Coordinate Systems

Figure 2-1 Earth Coordinate System

Earth Coordinate System



The earth coordinate system is illustrated with two profiles, one running from (A,B_1,C) to (A,B_2,C) and the second running from (D,B_1,C) to (D,B_2,C).

Profile sampling positions are illustrated with the tick marks. In the first profile, 2 locations are selected, while in the second, 5 are selected.

Figure 2-1

A schematic transmitter, receiver configuration is shown in the ellipse. The horizontal arrow represents the separation, and the vertical arrows the dipoles (each having code 6 in this case).

Figure 2-2 Prism Coordinate System

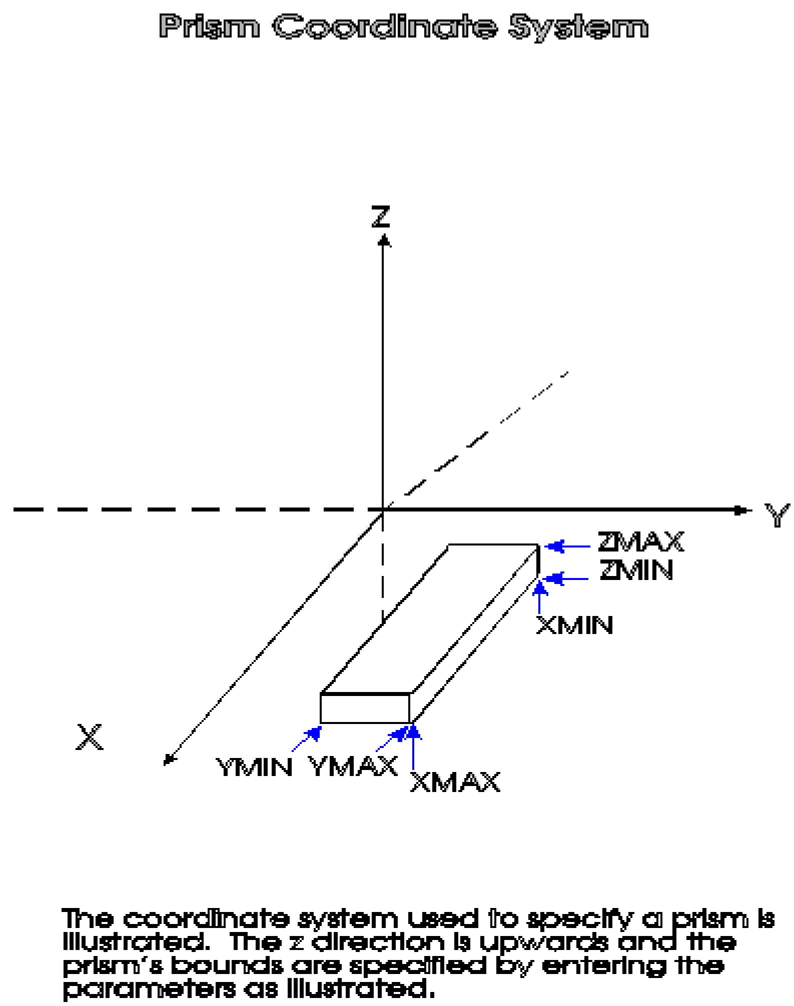


Figure 2-2

NOTE: For users using V6.3, afterwards you are able to strike, dip and plunge the prism.

Figure 2-3 Specifying Plate in Geological Mode

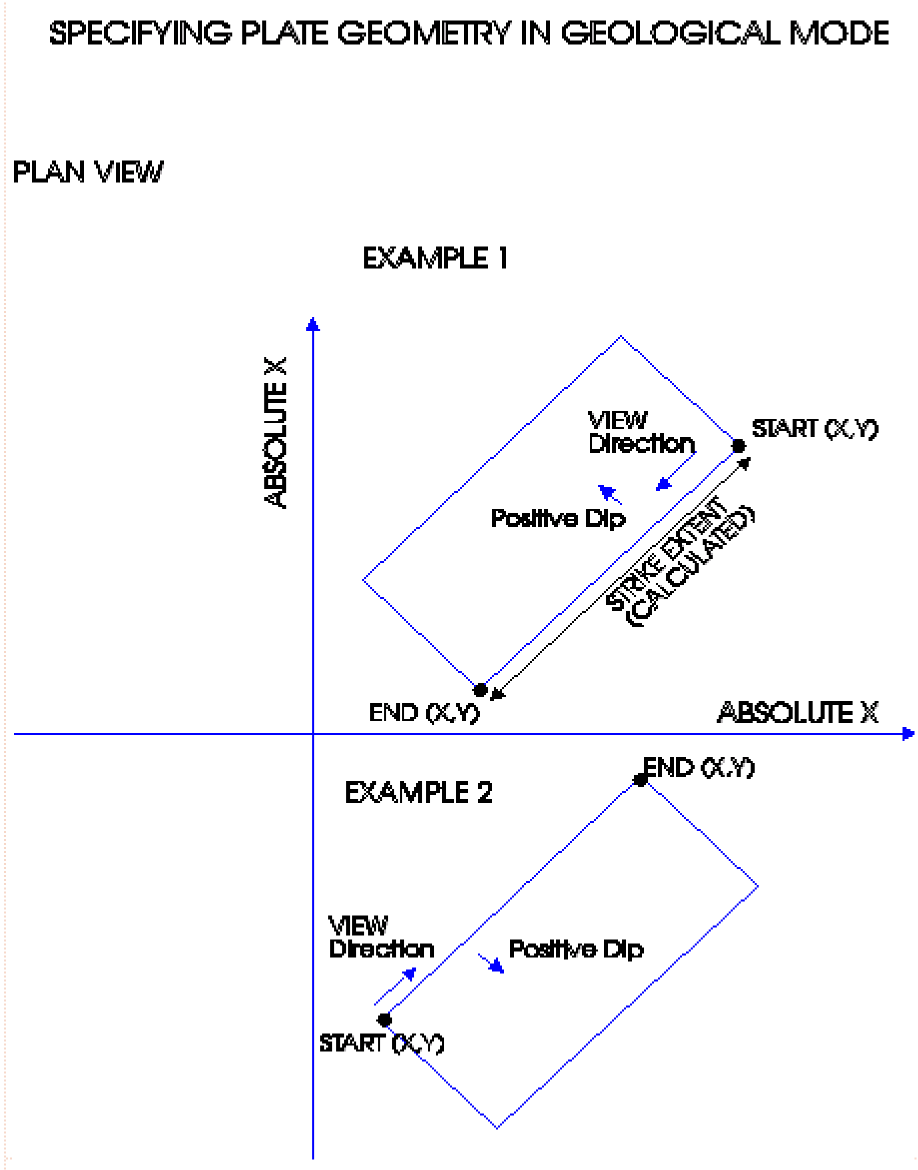


Figure 2-4 Plate Reference Points

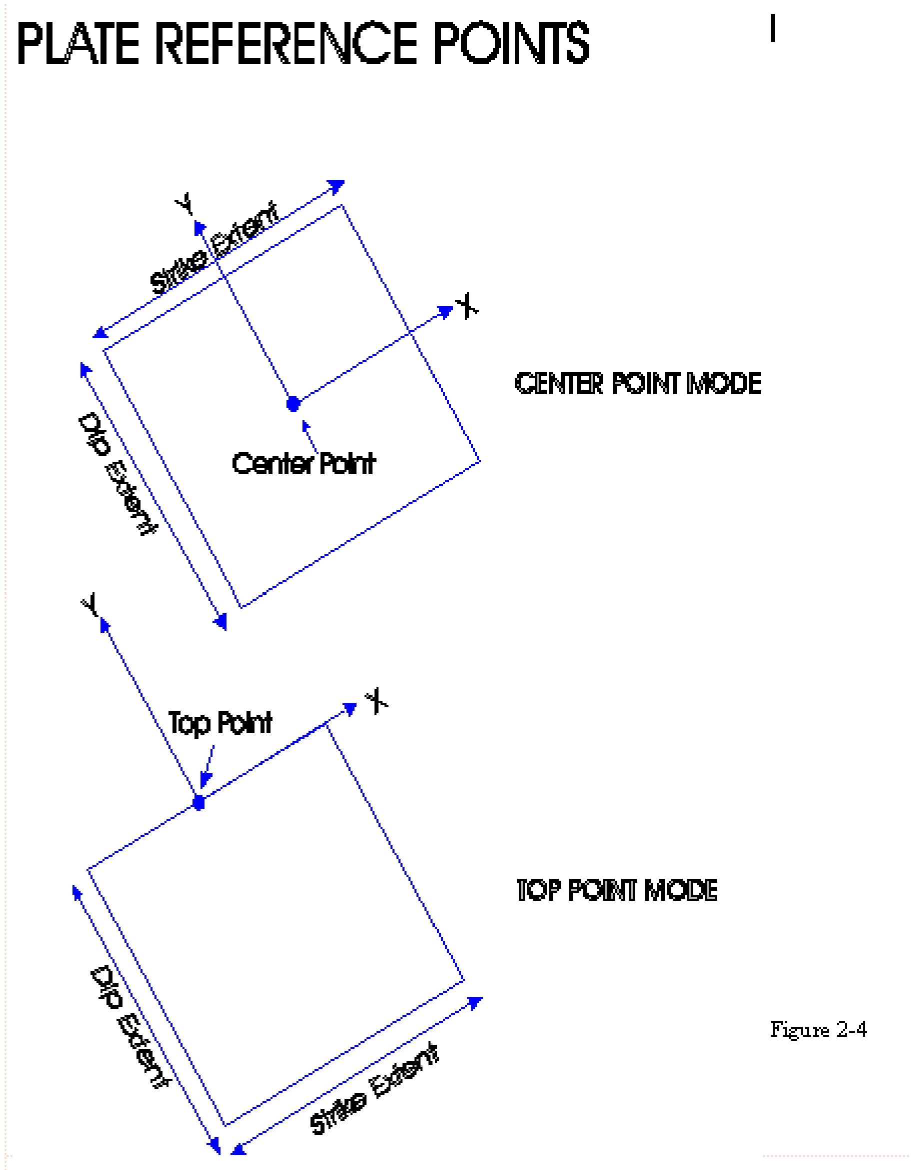


Figure 2-5 Magnetotelluric Inputs

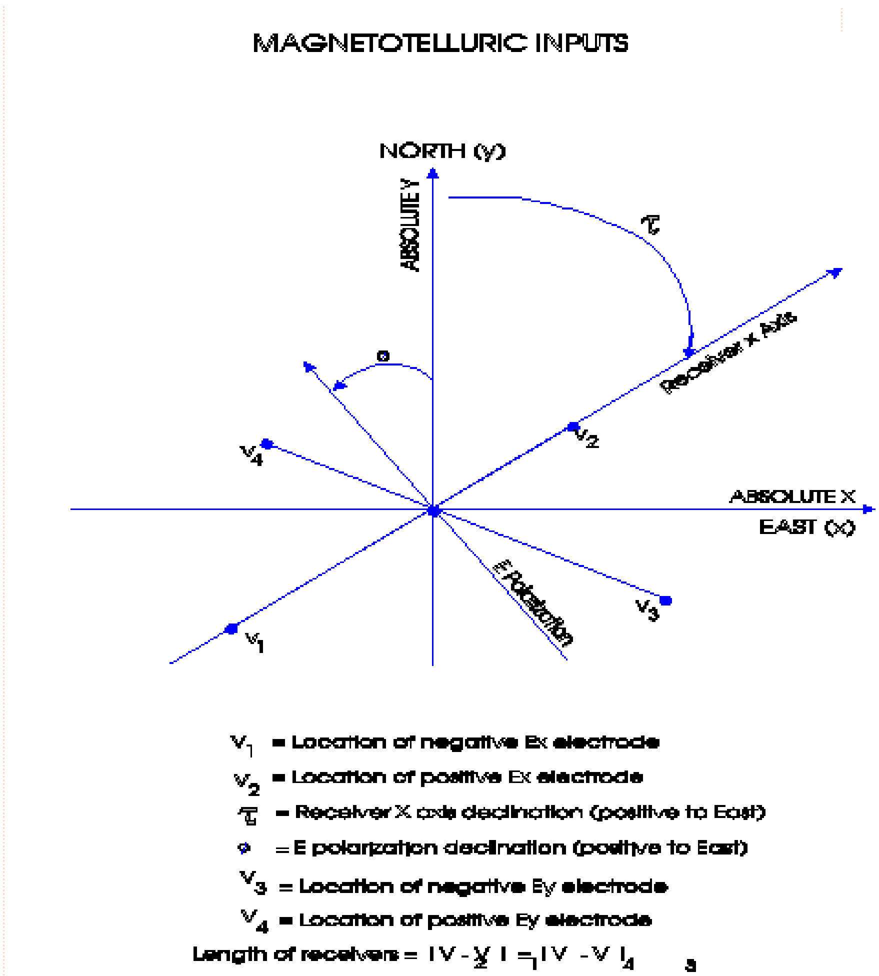

MAGNETOTELLURIC INPUTS
NORTH (y)
ABSOLUTE Y
Receiver x Axis
ABSOLUTE X
EAST (X)
E Polarization
V1
V2
V3
V4
V1 = Location of negative Ex electrode
V2 = Location of positive Ex electrode
τ = Receiver X axis declination (positive to East)
φ = E polarization declination (positive to East)
V3 = Location of negative Ey electrode
V4 = Location of positive Ey electrode
Length of receivers = I V2 - V1 I = I V4 - V3 I

Figure 2-5

Figure 2-6 Moving System Plot Points

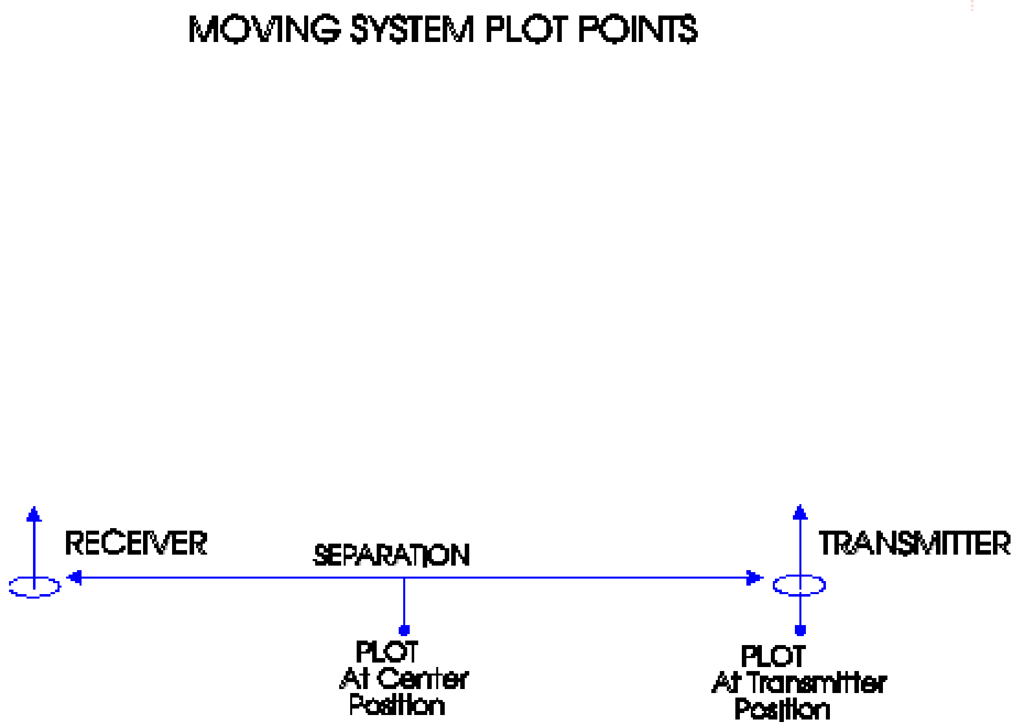


Figure 2-6

Figure 2-7 Loops, Bipoles and Dipoles

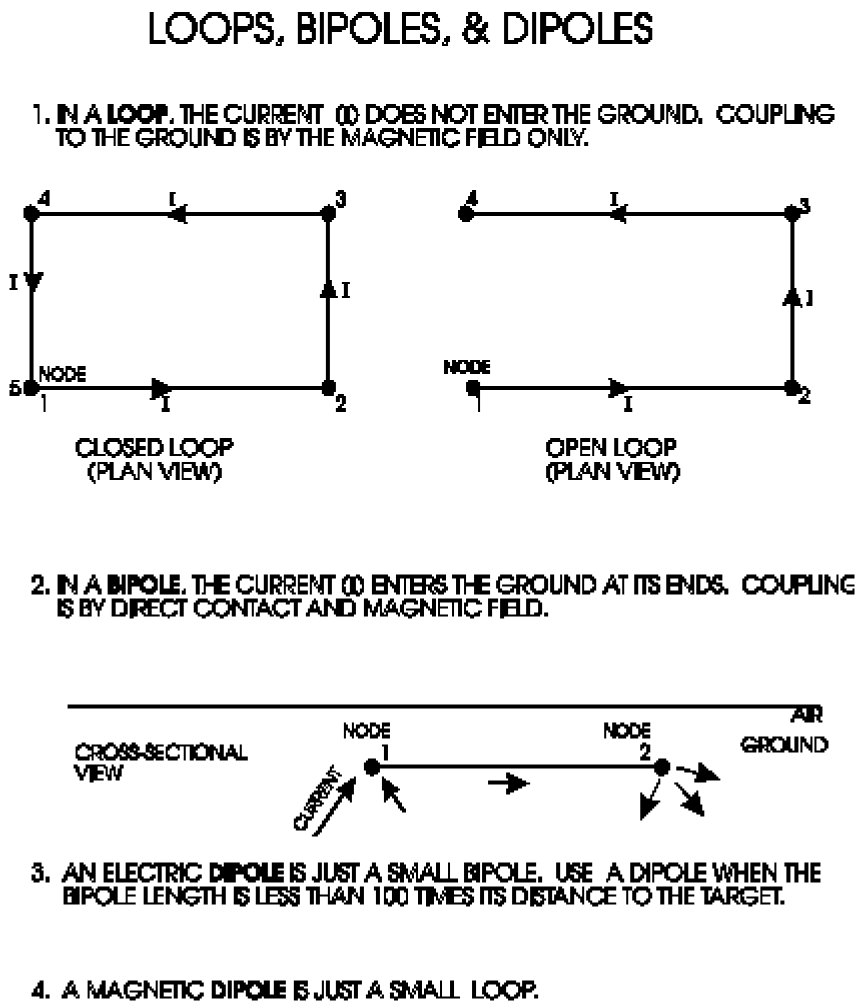


Figure 2-7

Figure 2-8 Fixed Loops and Extended Bipoles

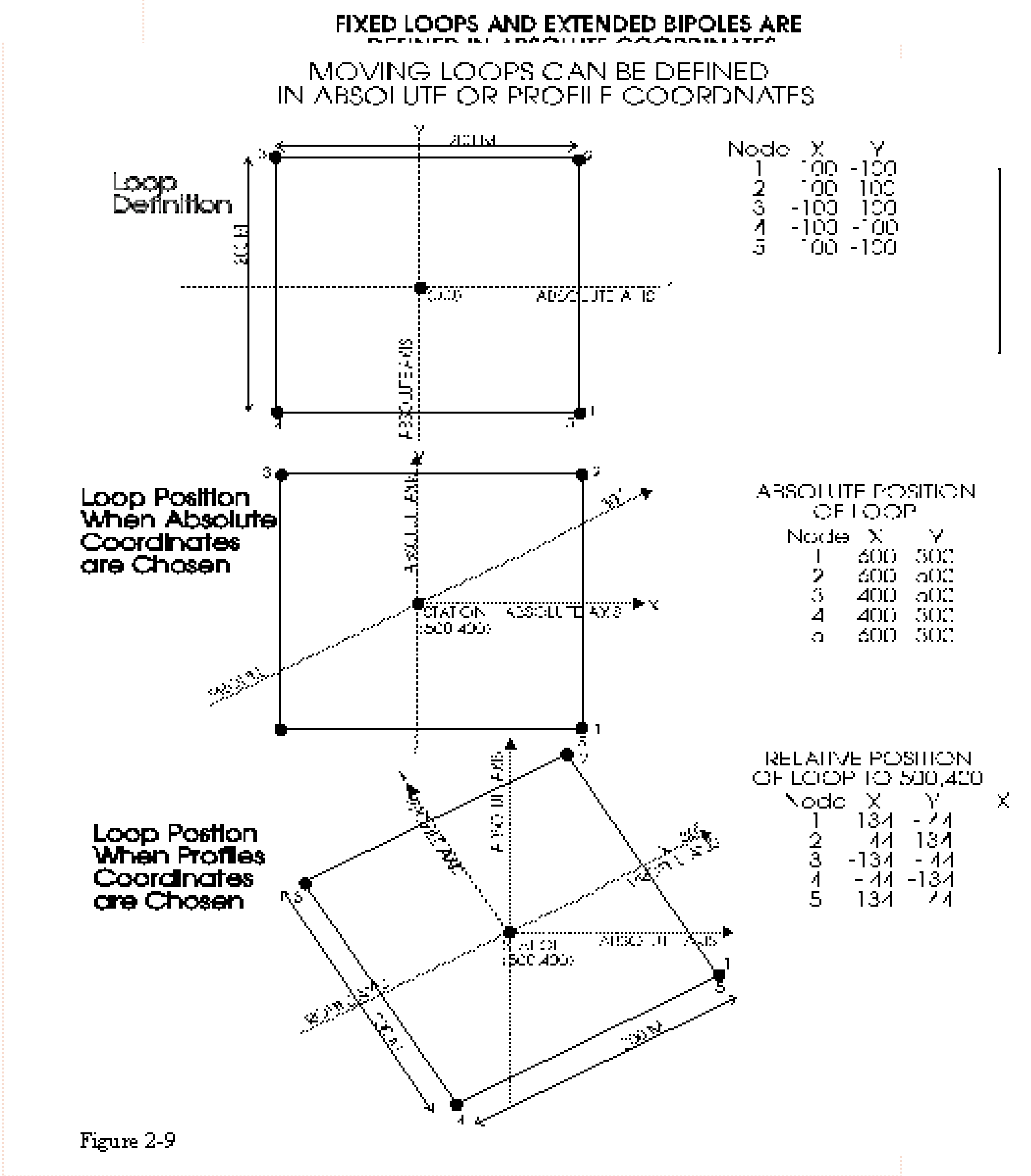


Figure 2-9

Are Defined in Absolute Coordinates

Figure 2-9 Moving Loops Can Be Defined in Absolute Ir Profile Coordinates

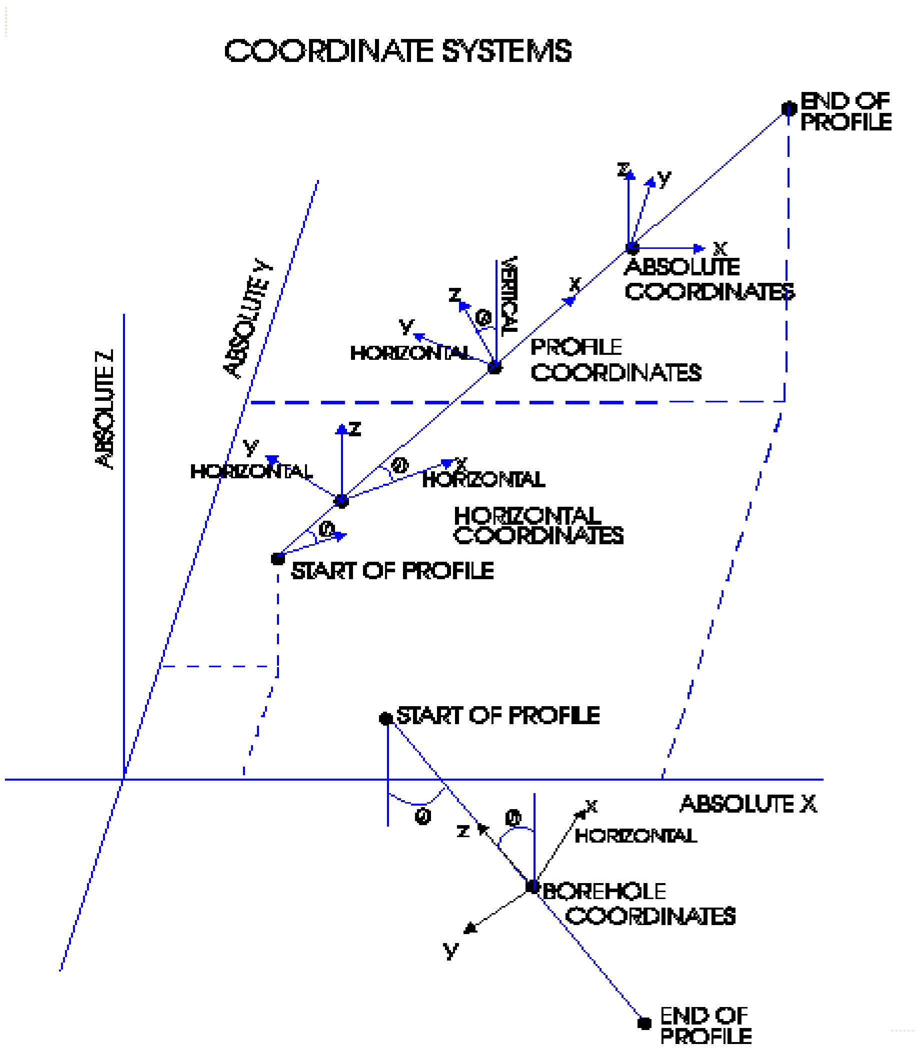


Figure 2-10 Coordinate Systems

CHAPTER III - USERS GUIDE TO EFFECTIVE MODELLING

In this chapter, we explain how to use ***EMIGMA*** to model some practical situations. Some background knowledge of the software is assumed.

This chapter discusses which physical parameter values should be used to build a representative geological model; how to model the response of different systems; how to set up coordinate systems and how to sample the prism model effectively under the *LN* approximation and ILN approximations.

CHOOSING PHYSICAL PARAMETERS

EMIGMA is an electromagnetic simulation platform designed to simulate the response of a variety of geophysical systems to geological structure. However, ***EMIGMA***'s input is in terms of physical parameter values, and not in terms of geologic lithologies or structure. This section is intended to help bridge the gap, so that reasonable physical parameter values can be assigned to represent the geology being modelled.

HETEROGENEITY

When using ***EMIGMA***, remember that geological bodies are intrinsically heterogeneous, but the physical parameters assigned to the body in ***EMIGMA*** assume homogeneity. The assumption of homogeneity in the presence of heterogeneity can be the source of complications, some of which are discussed here.

One complication introduced as a result of heterogeneity is that the appropriate homogenous physical parameter which best represents a body may be a function of scale size. That is, a body may have *bulk parameter* values that represent it when energized with large-scale systems but may change when energized with small-scale systems. Heterogeneities which effectively blend together to produce an effective bulk property parameter value at large scales may appear as distinct anomalies at smaller scale. The modeller must be willing to accept that the parameter values which best represent a geological body may not be independent of the system used to measure it. However, the nature of this variation with scale size may be important, and should not be viewed as an annoyance.

A second complication which results from heterogeneity is that the physical parameter value used to model a given structure of lithology in one area may be completely different in another area. The reason for this is that geological classification is not typically based on physical parameters that characterize a body. Thus in the following sections where individual parameters are discussed and typical values and ranges are listed, remember that these values serve only as a guide, and should not be assumed to be rigorous limits.

RELATIVE MAGNETIC PERMEABILITY

When a material is placed in a magnetic field, it may acquire a magnetization in the same direction as the field. This *induced magnetization*, or *magnetic polarization*, results from the alignment of the ions or molecules with the magnetic field. The intensity of the induced magnetization, **I**, is related to the magnetizing force, **H**, of the field as

$$\mathbf{I}=k\mathbf{H},$$

where k is the magnetic susceptibility of the magnetic material.

In a vacuum, the relationship between the magnetic field **B**, and the magnetizing force **H**, is

$$\mathbf{B}=\mu_o\mathbf{H},$$

where μ_o is the magnetic permeability of free space. Since air and water have magnetic permeabilities close to μ_o, the above equation represents the magnetic field of the earth in the absence of magnetic materials. When a magnetic material is introduced, the induced magnetization of the magnetic object adds to the field of the earth and the total field becomes

$$\mathbf{B}=\mu_o\mathbf{H} + \mu_o\mathbf{I} = \mu_o(\mathbf{H}+k\mathbf{H}) = \mu_o(1+k)\mathbf{H} = \mu_o\,\mu_R\mathbf{H}.$$

μ_R is the dimensionless *relative magnetic permeability* which corrects for the additional magnetic field of the region due to the magnetic material. In general, $\mathbf{B}=\mu\mathbf{H}$, where $\mu=\mu_o\,\mu_R$. Table 1 lists magnetic permeabilities of some common materials.

Diamagnetic and paramagnetic materials have weak magnetic susceptibility, so their relative permeability is approximately that of a vacuum. Ferromagnetic materials exhibit strong spontaneous magnetization which may exist in the absence of a magnetic field. Antiferromagnetic materials have no external magnetic field, and parasitic antiferromagnetism creates a negligible field. Ferrimagnetic materials have strong spontaneous magnetization and a high susceptibility.

RELATIVE ELECTRIC PERMITTIVITY

Poor conductors and insulators conduct current by dielectric conductivity. Dielectric conduction occurs when atomic electrons are slightly displaced from their nuclei in the presence of a varying electric field. This polarization may occur with ions or molecules, and is the means for dielectric conduction. The parameter of interest in dielectric conduction is the dielectric constant, *k*.

Like magnetic fields and magnetic polarization, electric fields can induce electric polarization in materials. The intensity of the electrical polarization, **P**, is related to the electric field, **E**, by

$$\mathbf{P}=\eta\mathbf{E},$$

where η is the electric susceptibility of the material.

In a vacuum, **D** is related to the electric field by

$$\mathbf{D}=\varepsilon_o\mathbf{E},$$

where ε_o is the electric permittivity of free space. When a dielectric material is present, the electric displacement is altered due to the electric polarization of the material introduced. The overall electric displacement becomes

$$\mathbf{D}=\varepsilon_o\mathbf{E}+\mathbf{P} = \varepsilon_o\mathbf{E}+\eta\mathbf{E} = \mathbf{E}(\varepsilon_o+\eta) = \varepsilon_o k\mathbf{E} = \varepsilon\mathbf{E},$$

where the dielectric constant is defined as

$$k=1+\eta/\varepsilon_{o,}$$

and the *relative electric permittivity* is defined as

$$\varepsilon=\varepsilon_o k.$$

Table 1 lists some examples of relative electric permittivity.

PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

ELECTRICAL CONDUCTIVITY

Electrical conductivity is defined as the inverse of resistivity. It describes a material's ability to transmit conduction current. There are three ways that electric current can propagate through rocks and minerals; dielectric (discussed above), electronic, and electrolytic. The conductivities of common materials may be calculated from the resistivities in Table 1.

ELECTRONIC CONDUCTIVITY:

In metals, electrons are the charge carriers, so the conductivity is said to be electronic. The electrons in a metal are not tightly bound to their atoms, so the conductivity of metals is high. For a metal rod of cross-sectional area *A*, length *L* and resistance *R*, the resistivity ρ, is given by

$$\rho = \frac{RA,}{L}$$

where the resistance is the ratio of the electrical potential across the rod to the current through the rod. The conductivity of the rod, σ, is simply

$$\sigma = 1/\rho.$$

Electronic conductivity is a relatively rare phenomenon in the earth, and its occurrence is usually confined to metals and certain massive sulphide ores.

ELECTROLYTIC CONDUCTIVITY:

Electrolytic conductivity is common in the ground in comparison to electronic conductivity, and is associated with the presence of water where ions are the charge carriers. Much of the conductivity encountered in geophysics is electrolytic and because of this, is difficult to characterize conductivities which are associated with particular lithologies since water content is such a dominating factor when determining conductivity. Electrical resistivity can thus be quite variable depending on mobility, concentration, and the degree of dissociation of the ions. Electrolytic conduction is a slow process because the movement is actually a transfer of material and perhaps even a chemical change, and so polarization phenomena often occur at low frequencies.

The effective resistivity of a porous rock, ρ_a, was determined empirically by Archie (1942) to be

$\rho_e = a\phi^{-m} s^{-n} \rho_\omega$, where φ is the porosity, s is the fraction of the pores containing water, ρ_ω is the resistivity of the water, $n \approx 2$, and a and m are constants ($0.5 \leq a \leq 2.5$, $1.3 \leq m \leq 2.5$).

ANISOTROPY OF RESISTANCE

The earth is neither uniform nor homogenous. Furthermore, it is not anisotropic. Small fractures associated with strain, bedding planes and preferential crystalline alignments due to schistosity and gneissosity all contribute to presenting paths of conduction which are preferential to current flow in one direction over another.

To understand anisotropy, consider a layered material where the layers are stacked vertically. If a current is passed through the stack from top to bottom, the system becomes a circuit of resistors in series. The equivalent resistance of such a circuit is equal to the sum of the individual resistances. To summarize,

$$R_{series} = R_{layer1} + R_{layer2} + \ldots$$

Now consider the same stack of layers, but with current being passed through all the layers from side to side. The circuit that would represent this situation is one with the resistors in parallel.

$$\frac{1}{R_{\_}} = \frac{1}{R_{layer}} 1 + \frac{1}{R_{layer}} 2 + \ldots$$

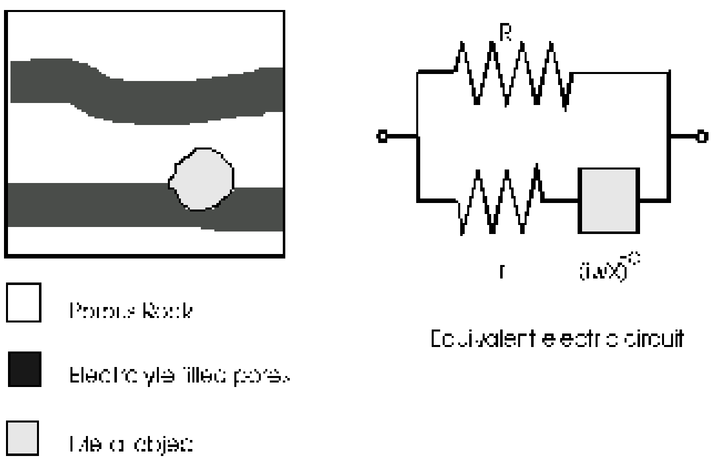
The equivalent resistance for this circuit is
Since the equivalent series resistance is not equal to the equivalent parallel resistance, seeing why the resistance in a layered material is anisotropic is easy. The *anisotropy coefficient* is the ratio of the maximum resistivity to the minimum resistivity. For graphitic slate, this coefficient may be as high as 2, but is in the range of 1 to 1.2 for limestone, shale and rhyolite. Anisotropy can be caused by many effects: preferential crystal orientation such as happens with olivine in the mantle, banding of minerals, preferential orientation of water-filled fissures and fractures, and micro-layering of sediments.

In general the anisotropy is best described by a tensor, since the coefficient of anisotropy can vary in three directions. The effect of this is that current is not necessarily parallel to the electric field as the scalar Ohm□s Law would suggest.

THE COLE-COLE RELAXATION MODEL:

It is first important to understand that normal field procedures for specifying IP responses in msec or degrees are not related to the actual material characteristics but are in fact related to the observed responses. In a time domain case, if a geophysicist comments on an IP response as 20msec, he refers to the integrated V/V_0 response through some intermediate window normally the Newmont window. If the geophysicist comments on a 50mradian response, then she means that a 50mradian variation in the phase response was observed between 2 measured frequencies. On the other hand, the Cole-Cole parameters attempt to explain the target or hosts entire response at an arbitrary frequency or time.

The Cole-Cole relaxation model is useful for representing the electrical conductivity or resistivity of a polarizable material. This model simulates the resistivity of a material in the ground with the electrical circuit shown in the figure below. Resistor *R* represents the resistance of an unblocked electrolyte filled passage in a porous rock. Resistor *r* represents the resistance of a blocked electrolyte filled passage. The complex impedance $(i\omega X)^{-c}$ represents the ionic-metallic interface of an embedded metal object (it should be noted that this is an empirically deduced impedance). The total impedance of this circuit is given by



The Cole-Cole Model

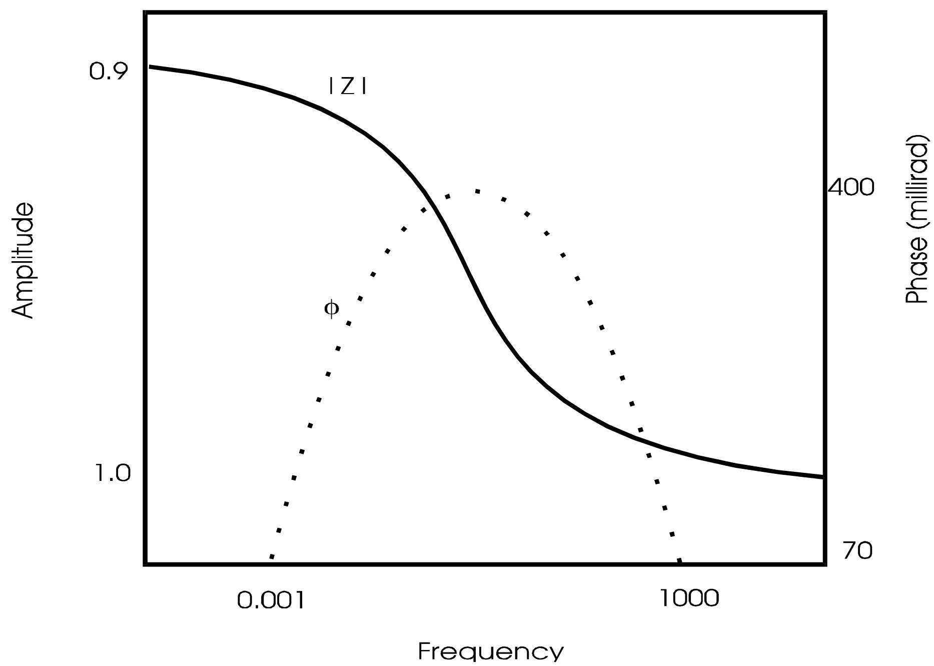
where $m=(1+r/R)^{-1}$ is the chargeability as defined by Seigel

$$Z(\omega) = R\left[1 - m\left(1 - \frac{1}{(1+(i\omega\tau)^{c})}\right)\right],$$

(1959), $\tau=X(R/m)^{-c}$ is the time constant, and ω is the angular frequency of the ac current.

Materials analysed using the Cole-Cole model are described by 4 parameters; *R*, the dc resistivity, *m*, the chargeability, τ, the time constant, and *c*, the frequency dependence. *R* is related to the resistivity measured by conventional static (dc) resistivity techniques. The figure below is a frequency domain plot of the phase φ, and amplitude |Z| of the total impedance. Changing *R* changes the vertical position of the amplitude, but does not affect the phase angle. The chargeability (defined by Seigel (1959)) is the ratio of the potential immediately after, to the potential immediately before an infinitely long charging current ceases to flow. Increasing the chargeability increases the polarizability or phase angle. The time constant is a measure of the time it takes for the signal to decay. If the time constant is increased, the phase and amplitude plot will be shifted to the left. Increasing the frequency dependence, results in higher slopes on the amplitude curves, and clearer peaks on the phase angle curves. For data analysis, the Cole-Cole parameters are manipulated to fit the measured curves.

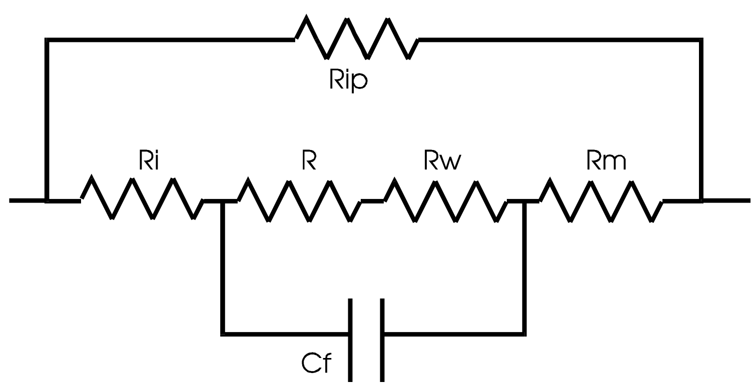
A Typical Cole-Cole Model Phase and Amplitude Plot



PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

OTHER IP MODELLING METHODS:

The Cole Cole model is merely one of a number of models which can be used to represent IP phenomena. Resistor and capacitor circuits are also used to represent IP phenomena and the figure following represents the same situation as the Cole-Cole model above. In this circuit R_i, and R_{ip}, are similar in function to R and r in the Cole-Cole model, but the complex impedance due to the metallic object is represented by R, R_{ω}, C_f, and R_m. As with the Cole-Cole model, time and frequency domain responses can be studied, however, the disadvantage is that each domain has its own set of parameters.

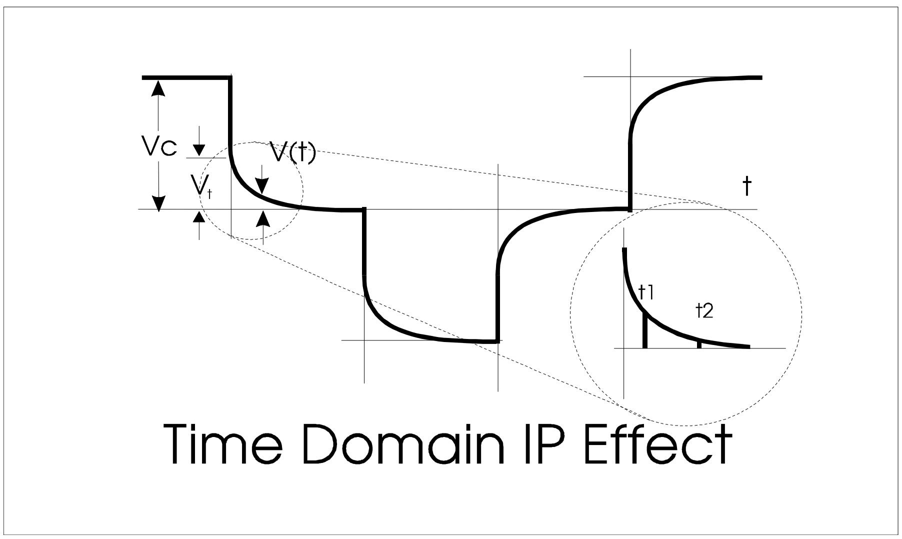


Electric Circuit to
Simulate the IP Effect

The Time Domain

In time domain studies, current is pulsed through the rock long enough to reach the equilibrium voltage V_c. When the current ceases to flow, the voltage decays with time (see below) in a manner that varies with rock properties. The time varying decay voltage V(t) may be measured at intervals and the ratio of $V(t)/V_c$ gives the shape of the decay curve. If V(t) and V_c are measured in millivolts, then 100 $V(t)/V_c$ is called the *IP percent*. An extension of instantaneous V(t) measurements is measurement of the *decay time integral* from t1 to t2. Some workers define *chargeability* as

$$M = \frac{1}{V_{c\,t}} 1\int^{t} 2V(t)dt,$$



where the unit is milliseconds, and t1 and t2 are the gate times over which the measurement was taken. This definition should be distinguished from the definition of chargeability proposed by Seigel which is m=V_t /V_c (see the diagram above).

The Frequency Domain

In the frequency domain, the apparent resistivity is measured at different frequencies. The frequency effect *fe* is a measure of the percentage change in field with change in operating frequency. It is defined as

$$fe = (\rho_{dc} - \rho_{ac}) / \rho_{ac}.$$

The percent frequency effect is

$$PFE = 100\, fe.$$

In order to account for variable IP effects due to the resistivity of the host rock, a metal factor *MF* was introduced by Marshall and Madden (1959). The Marshall and Madden metal factor has the form

$$MF = 2\pi \text{x} 10^5 fe/\rho_{dc.}$$

The metal factor ranges from 0 to $2 \text{x} 10^4$ mhos/m for surface materials, and 0 to $1 \text{x} 10^4$ mhos/m for competent rocks. For massive sulfides, the metal factor is $1 \text{x} 10^6$ mhos/m, for fracture filling sulfides the range is $1 \text{x} 10^5$ to $1 \text{x} 10^6$ mhos/m. Shale, sandstone, and finely disseminated sulfides have metal factors in the range from $1 \text{x} 10^3$ to $3 \text{x} 10^4$ mhos/m.

Table 1: Typical Values of Electrical and Magnetic Constants of Rocks and Minerals						
	Resistivity(Ohm.Meter)				Relative Electric Permittivity	Relative Magnetic Permeability
	Range			Average		
Minerals						
Graphite	0	-	10	1000	-	-
Chalcopyrite	0.00001	-	0.3	0.004	-	-
Bornite	0.00003	-	0.5	0.003	-	-
Pyrite	0.00003	-	1.5	0.3	-	1.0015
Pyrrhotite	0	-	0.05	0.0001	-	2.55
Galena	0.00003	-	300	0.002	18	-
Sphalerite	1.5	-	10000000	100	-	-
Bauxite	200	-	6000	-	-	-
Hematite	0.0035	-	10000000	-	25	1.05
Limonite	1000	-	10000000	-	-	-
Magnetite	0.00005	-	5700	-	-	5
Quartz	40000000000	-	2.000e+1 4	-	4.2-5	-
Hornblende	200	-	1000000	-	-	-
Bitumen	0.6	-	100000	-	-	-
Coals (various)	10	-	1.000e+1 1	-	-	-
Anthracite	0.001	-	200000	-	5.6-6.3	-
Lignite	9	-	200	-	-	-
Meteoric Waters	30	-	1000	-	80	1
Surface waters:				-		
(ign. rocks)	0.1	-	3000		80	1
(sediments)	10	-	100	-	80	1
Soil waters	-		-	100	-	-
Sea water	-		-	0.2	-	-
Saline water, 3%	-		-	0.15	-	-
Igneous and Metamorphic Rocks						
Granite	300	-	1000000	-	4.8-18.9	-
Diorite	10000	-	100000	-	6	-
Dacite	2.0E+04			-	6.8-8.2	-
Andesite	45000	-	170	-	-	-
Diabase porphyry	1000	-	170000	-	-	-
Gabbro	1000	-	1000000	-	8.5-40	-
Basalt	10	-	13000000	-	12	-
Olivine norite	1000	-	60000	-	-	-
Gneiss (various)	68000	-	3000000	-	8.5	-
Quartzites (various)	10	-	20000000	-	-	

SETTING UP COORDINATE SYSTEMS

EMIGMA has a number of coordinate systems available to the modeller. The coordinate systems are standard mathematical coordinate systems with an x, y and z axis. Thus the coordinate system is somewhat different from systems used in geological representation, where plan and section views are often preferred.

The fundamental coordinate system used in ***EMIGMA*** is the ***absolute coordinate*** system. In this system, the x and y axes are horizontal and the z axis pointing vertically upwards. When viewed from above, the y axis lies counterclockwise at a 90-degree angle from the x axis. The z origin of the coordinate system is fixed at the lower interface of the upper half-space in the layered earth model. The x and y origin of the absolute coordinate may be anywhere.

We recommend that the absolute coordinate system be used to represent the geographic coordinate system in which your survey data has been collected. This for example, could be ***UTM Northing and Easting***, where the absolute x and y axes would correspond to the easting and northing respectively. Alternately, the absolute axes could correspond to a locally cut, picketed and chained grid, again where grid east and north would correspond to the absolute x and y coordinates in *EMIGMA*.

If the user is using Version 6+ of **EMIGMA**, the user places the prism in the absolute system but is then allowed 3 rotation angles.

☛*WARNING: In the current version, prisms must be defined in the absolute coordinate system, although this restriction will be removed in later versions. This restriction may influence selection of the geographic coordinate system that the absolute coordinate system represents.*

All ***profile*** and ***conductivity structure*** coordinate inputs in *EMIGMA* are with respect to the ***absolute coordinate*** system, as are the coordinates used to establish the location of a ***fixed loop***. In this way, as conductivity models are built up to represent a project area, all can be referred to in the same coordinate system. Coordinates used to locate a profile can then be the actual survey coordinates of that profile, and not the coordinates with respect to some imaginary and unrepresentative system.

In many cases, profiles and boreholes will not be aligned parallel to the absolute coordinate system, and moving sensors will be aligned parallel to the profiles or borehole. To allow for this, a number of relative coordinate systems have been introduced so that sensors can be aligned parallel to the profile or borehole, rather than being restricted to the absolute system. More information on this aspect of modelling is given in Chapter II, under Controlled Source Inputs.

SAMPLING FIELDS IN PRISMS

In the *LN* and *ILN* scattering algorithm in ***EMIGMA***, as with any numerical modelling program, to successfully compute a realistic model response, sampling must be considered. In the current version, sampling must be specified manually. Automated sampling algorithms require a sophisticated knowledge of electromagnetic scattering and we know of no algorithms that use such a technique. For the immediate future then, manual sampling must be used.

Sampling for the LN Techniques (LN &ILN) are not part of the 3D Solutions for internal fields but simply a means for integrating over the anomalies to obtain the scattered (secondary) fields.

The *VH* algorithm uses a fixed number of points to represent the incident fields on the conductor (a 21 by 21 point grid), and so is not subject to the discussion here, although similar considerations do apply to that algorithm as well.

THEORETICAL CONSIDERATIONS:

When we consider the *LN* approximation, the spatial variation of the total internal electric fields in any prism must be considered. Internal electric field sampling requires the modeller to consider such effects as skin depth and the proximity of the conductor to the source and to the receiver. When the skin depth of the layer in which a prism sits is small compared with a dimension of the prism, the field will be strongly attenuated inside the conductor. The electric field should thus be sampled densely enough near the edges of the prism to reflect this variation. Similarly, when the prism is located close to either a transmitter or a receiver, sampling of the fields near those points becomes critical if the scattered fields are to be properly calculated.

The second issue to be considered is the sampling of the internal transfer functions, namely, how the incident primary field from the transmitter determines the scattering currents inside the conductor. These transfer functions can be broken into two parts: a part due to charges on the surface of the prism and a part due to currents flowing inside it (induction). In as far as charge effects are concerned, the LN algorithms compensate well for them. However, the LN only partially compensates for the inductive component of the transfer function. The use of the ILN algorithm would be recommended in this case.

PRACTICAL CONSIDERATIONS:

When deciding if sampling is adequate, the best course of action is to test convergence. Convergence testing is typically done by recursively running the model and recording the change in the model response with the number of sample points. A correctly sampled model's response will show no change with the number of sample points. Unfortunately, this can only be detected by running models that in retrospect have had too many sample points included in it. But sampling density criteria is quickly learned when using **EMIGMA**.

The *LN* algorithms' capability to accommodate many points, yet still execute rapidly, allows convergence to be easily checked. This is a feature of numerical modelling that has often been overlooked until now, simply because computing proper convergence checks was virtually impossible.

INTERNAL ELECTRIC FIELD SAMPLING:

Once the model parameters have been inputted, ***EMIGMA*** tests them and if they are found satisfactory, examines the internal sampling points in **each** prism. Sampling points are located on a regular rectangular grid inside each prism, so specifying the location of each point individually is not necessary. EMIGMA is default for a fixed number of points per body. VHPLATE defaults the number of sampling points to 441. LNPRISM defaults the number of sampling points to 100, but up to 600 may be used per prism. LNFDPrism calculates 7 positions in a cluster around each sampling position. The default is set to 40 sampling points, but allows up to 85 per ILN Prism.

In V6.3 the user can control the sampling rate and distribution. The number of points used affects the total program memory requirements, and the suggested sampling in the x, y and z directions. This suggested sampling is based on an algorithm which makes the distances between the sample points in the x, y and z directions approximately equi-dimensional subject to the condition that the total number of points is approximately the pre-set limit. In V6.3, the user can chose to accept the suggested sampling, or modify it interactively.

☛*NOTE:* ***EMIGMA*** *allows a maximum number of points per prism. The total allowed points for the entire model are the number of prisms used times the number of points per prism.*

EMIGMA differs from conventional electric field integral equation solutions because the computation of the electric fields inside the prisms is so rapid. In many practical problems then, this part of the calculation is insignificant compared with the actual computation of the incident fields. Thus, the part of the computation that was relatively fast in other solutions, the incident field calculation is now relatively slow. For optimal performance with **EMIGMA**, it is wise to reduce the number of points used to sample the scattered fields.

As stated previously, the scattered field must be sampled densely where it exhibits rapid variation, such as near the edges of the conductor if the skin depth inside the conductor is large, or near transmitter and receiver dipoles. In such cases, it is often useful to compose a single prism out of two or more prisms, so that prisms where the field varies rapidly can be less densely sampled.

MODELLING TIME DOMAIN SYSTEMS

To model a time domain response, ***EMIGMA*** must be used together with the time series transformation software, ***FSEMTRS***. The two routines communicate with one another either directly through memory or through PetRos EiKon's *.pev or .mdb formatted data files which are generated by ***EMIGMA*** and then imported into ***FSEMTRS*** where the transform is computed. The operation of ***FSEMTRS*** is dealt with in a separate manual.

EMIGMA simulates the electromagnetic response of the earth in the frequency domain. To simulate a time domain response, the frequency domain response must be computed over a broad band which includes frequencies which are low enough to include the base frequency of the time domain system. A discussion of how to select the upper frequency limit that should be computed is covered in the ***FSEMTRS*** manual, but for many commercially available time domain EM systems, 100 Khz is often sufficient.

CHAPTER III-A - EXTENDED TRANSMITTER SYSTEMS

DIPOLE-DIPOLE IP/RESISTIVITY

Dipole-dipole IP surveys can be modelled using the ***moving transmitter*** or separations option. For modelling surveys in line along a flat profile, use ***horizontal coordinates*** and use a separation in the ***X direction*** only with a centred reference point (as opposed to the reference point at the transmitter). Both the transmitter and the receiver should be specified as ***bipoles***.

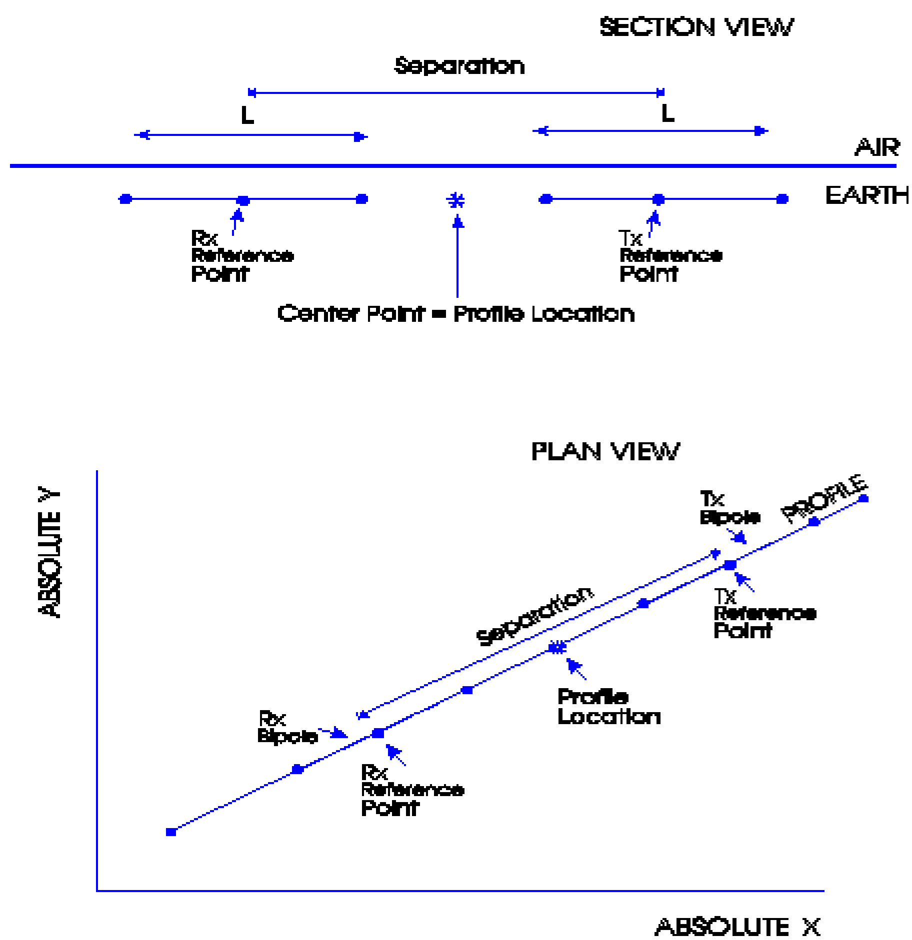
If a ***Wenner*** array is being modelled with a bipole length of "*L*", then when specifying the bipoles, both the transmitter and the receiver should have a start and end point relative to their local origins at (*-L,0,0*) and (*0,0,0*) or (*0,0,0*) and (*-L,0,0*). The ***separation*** parameter specifies the separation of the local bipole origins and therefore the "*n*" factor, so choose a separation of '*nL*' in the '*X*' direction. The ***reference point*** should be 'C' to calculate the data at the mid point of the array.

The ***z coordinat***e for the start and end coordinates of the profiles is recommended to be less than zero (say -0.1) so the bipoles are electrically grounded.

For frequency domain calculations, choose the ***frequency domain mode***. To compute the time domain response, use the ***spectral mode***. The base frequency should be contained within the lower frequency list, so for a 5-second cycle time, choose a 0.17 Hz starting frequency. The upper frequency used should be sufficient to represent the earliest and latest sample time accurately. For more information on this aspect of modelling, see the *FSEMTRS* manual. ***Resistivity*** surveys can be simulated with frequency domain mode by using a low frequency (say 0.1 Hz) and taking the in-phase component of the output only.

For best results, particularly in soundings, keep the transmitter and receiver wire segments a distance of more than two meters from each other. The reason for this is that the numerical techniques used for integrating along the path of the wires are more accurate when separation is at least 2 meters. At low frequencies, this path of the wires will not be important since the electric field is governed by its scalar potential, and results of the integration is therefore path independent. At high frequencies, the transmitter and receiver cables would interact, and this coupling would lead to spurious signals and is poor survey design.

DIPOLE-DIPOLE IP CONFIGURATION



PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

LOOP - BOREHOLE SYSTEMS

Borehole surveys can be modelled using the ***fixed transmitter*** option. For modelling surveys down a borehole, use ***borehole coordinates*** to resolve the fields such that the 'Z' component points up the borehole. The transmitter antenna should be represented by a ***loop*** and the receiver should be specified as a ***dipole***. For a single component axial system, choose Hx as the receiver. For a three component magnetic measurement, chose Hx, Hy and Hz.

The loop is fixed, so it is specified in absolute coordinates. Enter the ***loop coordinates*** in the sense that the direction of positive current to be flowing was considered. The number of nodes used to specify the loop should be 1 greater than the number of wire segments used to represent the loop, since to close the loop, the last node must be the same as the first node.

To compute the time domain response, use the ***spectral mode***. The base frequency should be contained within the lower frequency list. The upper frequency used should be sufficient to represent the earliest and latest sample time accurately. For more information on this aspect of modelling, see the *FSEMTRS* manual.

CENTRAL LOOP SOUNDING

Central loop sounding surveys can be modelled using the ***moving transmitter*** option. For modelling surveys in a line along a profile, where the loop is laid out relative to the profile, use ***horizontal coordinates.*** (Absolute coordinates could be used if the loop is laid out in the absolute system such as would occur if GPS or a compass were used to set the wires due north-south.) Since the receiver is a dipole and the transmitter is a loop, employ a ***loop*** transmitter and a dipole ***receiver***.

If the receiver is in the centre of the loop, set the ***separation*** between the transmitter and the receiver to (0,0,0) in the x, y, and z directions. The loop coordinates entered should be a distance relative to the point on the profile where the measurement is to be simulated. The profiles should be in the air since both the transmitter and the receiver are above the ground.

To compute the time domain response, use the ***spectral mode***. The base frequency should be contained within the lower frequency list. The upper frequency used should be sufficient to represent the earliest and latest sample time accurately. For more information on this aspect of modelling, see the *FSEMTRS* manual.

FIXED LOOP - MOVING DIPOLE RECEIVER

Large loop transmitter - moving receiver surveys can be modelled using the ***fixed transmitter*** option. For modelling surveys where the receiver is oriented in the direction of the profile, but horizontally, use ***horizontal coordinates***. (Where the receiver is oriented parallel to absolute north/south, use absolute coordinates). The transmitter antenna should be represented by a ***loop*** and the receiver should be specified as ***dipoles***.

The loop is fixed, so it is specified in absolute coordinates. Enter the ***loop coordinates*** in the sense that the direction of positive current is considered to be flowing. The number of nodes used to specify the loop should be 1 greater than the number of wire segments used to represent the loop, since to close the loop, the last node must be the same as the first node.

To compute the time domain response, use the ***spectral mode***. The base frequency should be contained within the lower frequency list. The upper frequency used should be sufficient to represent the earliest and latest sample time accurately. For more information on this aspect of modelling, see the *FSEMTRS* manual.

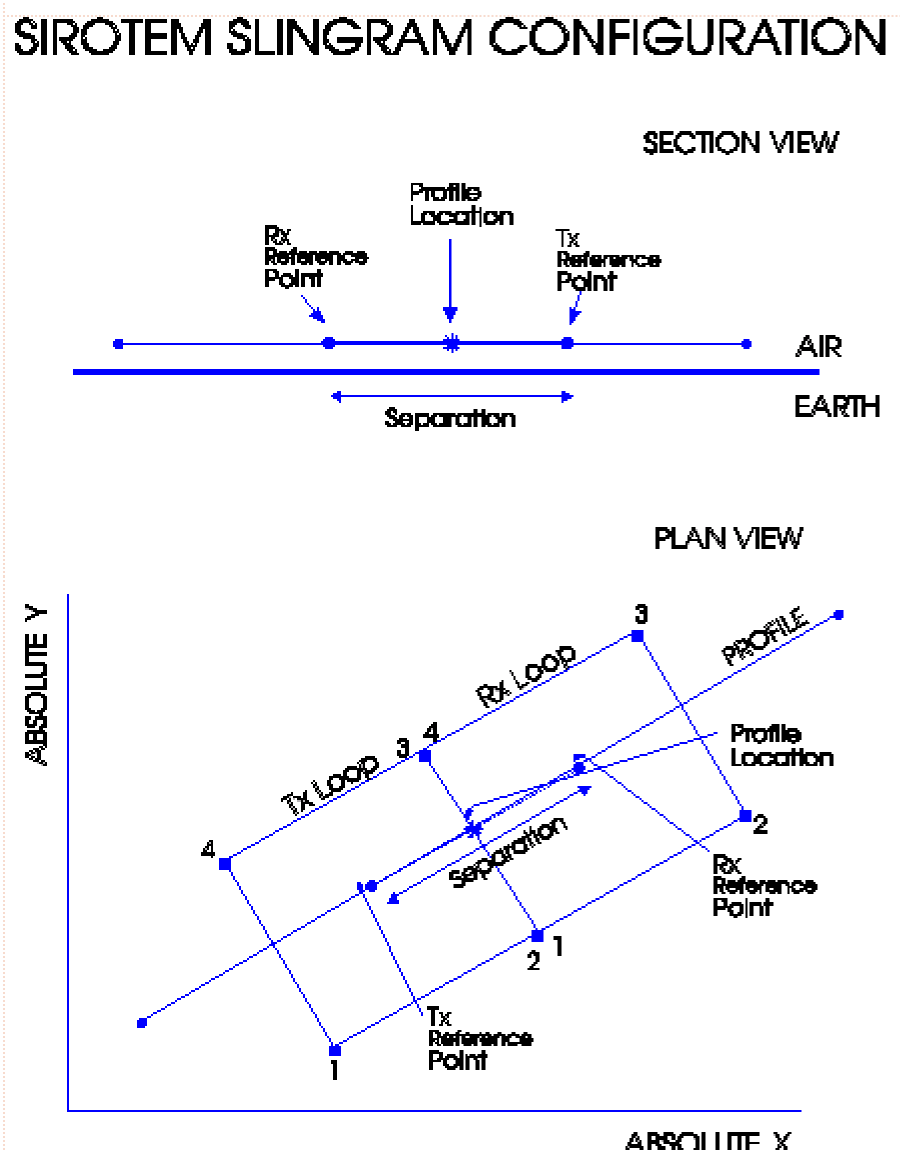
SIROTEM DUAL LOOP CONFIGURATION

Dual loop surveys can be modelled using the ***moving transmitter*** option. For modelling surveys in line along a profile, where the loops are laid out relative to the profile, use ***horizontal coordinates.*** (Absolute coordinates could be used if the loop is laid out in the absolute system such as would occur if GPS or a compass were used to set the wires due north-south.) In this case, both the transmitter and receiver antennae are ***loops***.

The ***separation*** of the transmitter and the receiver should be 0 in the z direction. If the receiver is ***coincident*** with the centre of the loop, set the separation between the transmitter and the receiver to (0,0) in the x and y directions. For ***Slingram*** configurations, where the transmitter and receiver loops are the same dimension and the loops share a common side, choose the separation in the x direction to be equal to the loop dimension. For ***dual loop*** configurations, where the transmitter and the receiver are slightly offset from each other, apply small values to the x and y separation parameters to achieve the required offsets.

The loop coordinates entered should be relative to the local origin of the loop. For a square loop, the coordinates specifying the loop nodes should be +/- ½ of a loop dimension. If ***reference point*** is the transmitter ('T'), then the transmitter reference point will be coincident with the profile position, whereas if a centred reference point is used, the transmitter local origin will be located ½ a separation ahead of the profile point and the receiver local origin will be located ½ a separation behind the profile point. The profiles should be in the air since both the transmitter and the receiver are above the ground.

To compute the time domain response, use the ***spectral mode***. The base frequency should be contained within the lower frequency list. The upper frequency used should be sufficient to represent the earliest and latest sample time accurately. For more information on this aspect of modelling, see the *FSEMTRS* manual.



CSAMT

CSAMT, or Controlled Source Audio Magnetotellurics uses a long bipole to energize the earth. Measurements are of the horizontal electric field parallel to the transmitter and horizontal magnetic field perpendicular to the transmitter. Measurements have traditionally been made in the far field regime where the incident field can be treated as a plane wave. EMIGMA makes no far-field assumption but rather models a 3D source. Tensor CSAMT surveys measure both horizontal components of E and H.

CSAMT is a ***frequency domain*** system, and the response can be modelled for a number of frequencies in one run simply by specifying the required frequency list using frequency domain (F) mode. The usual frequencies of operation range from a range of 1-10,000 Hz.

For H polarization measurements, the transmitting bipole laid out perpendicular to the geological strike, whereas for E polarization measurements, the transmitting bipole is parallel to strike. To model a CSAMT survey, select a ***bipole*** transmitter. The receivers should be aligned, parallel to or perpendicular to, the transmitter bipole. ☞The bipole should be specified as, being either parallel to or perpendicular to, the absolute coordinate axes, and choose ***horizontal*** coordinate system for the receiver orientations. Assuming the transmitter bipole is parallel to the absolute *x* axis, a CSAMT survey can be modelled with ***two receiver dipoles*** per transmitter.

MELIS

In the MELIS system, a large loop is laid out on the ground and magnetic field measurements are taken relative to a reference coil. In this sense, the system is similar to the original TURAM systems.

The MELIS system can measure 3 magnetic field components in a borehole**.** Since data are output in absolute coordinates, the absolute system should be used for the receiver. To model a MELIS system, use a ***fixed loop transmitter***, and ***receiver dipoles*** Hx, Hy and Hz in the absolute system. However, many data processing characteristics of MELIS systems assume point dipole sources.

Normalize the output, again using the transmitter loop as the normalizing source and the ***reference coil*** as the normalizing receiver. If the reference coil is not close to any significant anomalous structures, such a normalizing model should be sufficiently accurate.

MMR

Magnetometric resistivity surveys are typically carried out at low frequencies with a grounded source. The intention of the method is to measure magnetic field distortions which result from the current channelling in the ground. In this sense, it is similar to the DC resistivity method. The method has an advantage over conventional resistivity techniques because it avoids problems due to small inhomogeneities with cause local distortions of the electric field. From a practical point of view, the method also has an advantage over conventional techniques when electrode contact is either difficult or impossible.

MMR is a ***frequency domain*** technique, and the response can be modelled for many frequencies in one run simply by specifying the required frequency list using frequency domain mode. The usual frequencies of operation range from 1 - 10 Hz.

The transmitting antenna should be represented by a ***bipole***, with the ***receiver*** being a magnetic field dipole. As MMR represents a technique rather than a specific commercial system, any number of survey configurations are possible with it. However, in MMR surveys the source is generally fixed rather than moving.

CHAPTER III-B - DIPOLE TRANSMITTER SYSTEMS

HLEM (MAXMIN/SLINGRAM)

Horizontal loop EM surveys (i.e.: Max-Min or Slingram) are moving loop - moving receiver frequency domain systems**.** Frequencies typically range from approximately 100 Hz to approximately 15 KHz. A number of different loop separations and transmitter-receiver orientation configurations are possible. The method is a ***frequency domain*** technique, and the response can be modelled for a number of frequencies in one run by specifying the required frequency list using frequency domain mode.

Both the transmitter and the receiver should be specified as ***dipoles***, and ***horizontal coordinates*** should be used if the coils are level at each station. If an inclined profile is being modelled, and no in-phase coupling between the coils is desired, select profile coordinates and a *z* separation parameter of 0. In the horizontal loop mode of operation, dipole codes of z should be used for the transmitter and receiver, although other configurations are possible with MaxMin and Slingram systems.

Data should be plotted using the ***centre point*** convention. In this convention, the loops are offset from the centre point by ½ the ***separation*** parameter (the transmitter is located +½ the separation parameter from the plot point). To ensure the coils lie on the profile the separation should in the *x* direction only, the *y* and *z* separation parameters should be zero. ***Profiles*** should be approximately 1 meter above the ground.

The output should be ***normalized*** in percent using a free-space model. Normalization geometry is usually the same as for the geometries used in the model.

GEOTEM: Note a detailed tutorial is provided.

GEOTEM is a time domain airborne system in which a horizontal coil mounted around an aircraft is used to transmit a signal which is measured in a coil mounted in a towed bird which trails the aircraft. GEOTEM is a descendant of the INPUT system. Thus, GEOTEM is modelled as a ***moving source*** system.

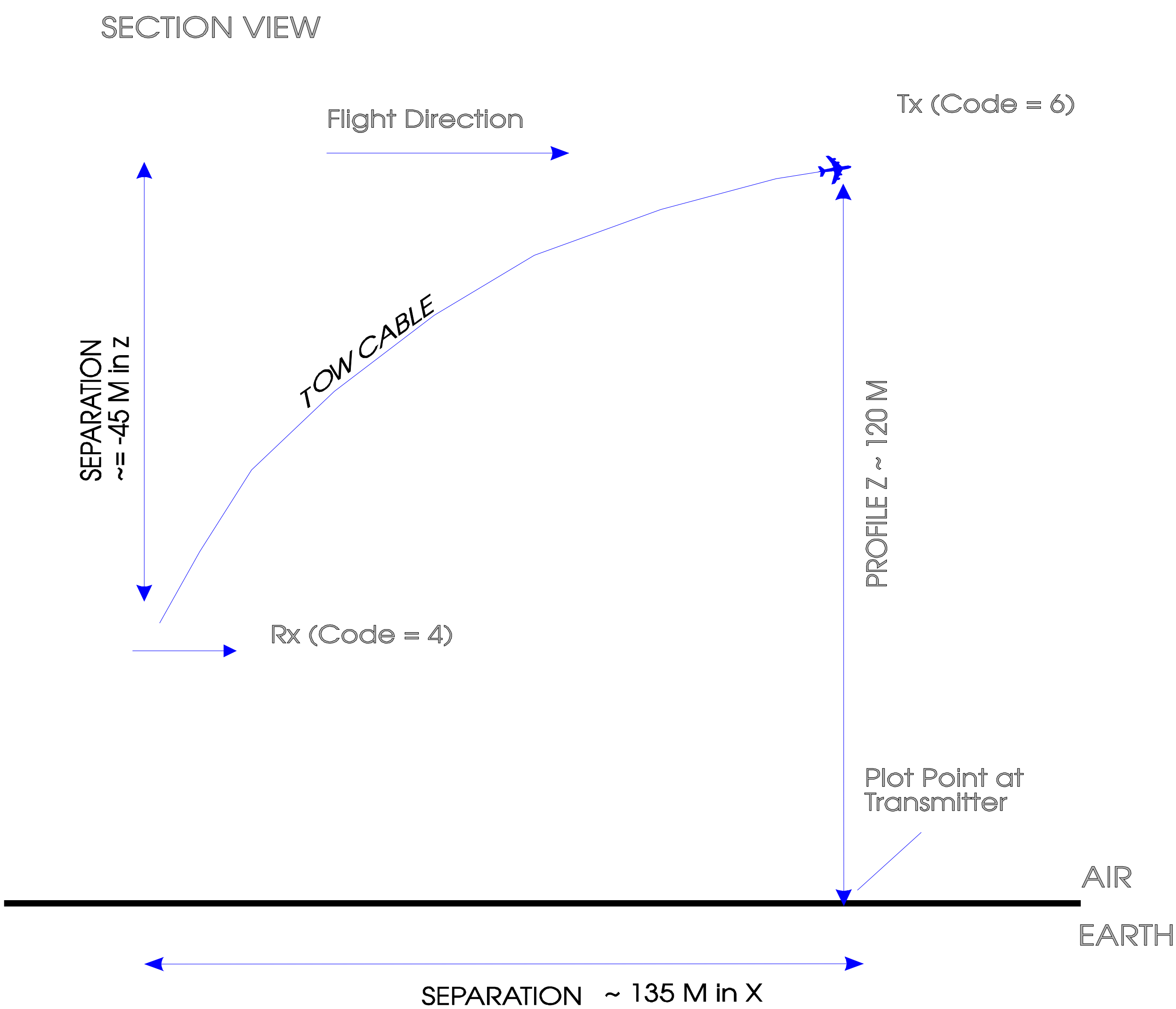
To compute the time domain response, use the ***spectral mode***. The GEOTEM base frequency ranges from 25 to 150 Hz, and should be contained within the lower frequency list. The upper frequency used should be sufficient to represent the earliest and latest sample time accurately. For more information on this aspect of modelling, see the *FSEMTRS* manual.

To model GEOTEM responses, select either ***profile or horizontal coordinates***. Profile coordinates would be more applicable when trying to simulate the effect of change in aircraft altitude due to changes in profile altitude. Data often is plotted using the ***transmitter point*** convention. By using the transmitter point convention, the plot point and the transmitter occupy the same position, while
the receiver is displaced from the transmitter by the negative separation in the selected coordinate system. Typical ***separation*** parameters are 135 meters in the ***x*** (profile/horizontal) direction and -45 meters in the *z* direction. Profiles should be at aircraft altitudes, which is typical 120 meters. Note that profiles should run in the direction of flight.

The ***transmitter*** can adequately be represented by a dipole with code Mz, while for standard GEOTEM surveys, the ***receiver*** should be modelled by a horizontal dipole (Hx and Hz). Newer ***multi-component*** systems can be modelled by assigning more than one receiver to each transmitter.

The output should be ***normalized*** in PPM using a free-space model. Normalization geometry is usually the same as for the geometries used in the model. Note this normalization is done within the transform software.

GEOTEM INPUTS



Base Frequency ~ 75 to 150 Hz

PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

HEM

Helicopter electromagnetic systems are fixed geometry, moving source frequency domain systems. Operation is typically in the frequency range between 500 and 60,000 Hz, although specialized systems have been built which operated at frequencies as low as 40 Hz and as high as several hundred kilohertz.

A number of different loop separations and transmitter-receiver configurations are possible with the system. The method is a ***frequency domain*** technique, and the response can be modelled for many frequencies in one run by specifying the required frequency list using frequency domain mode.

Both the transmitter and the receiver should be specified as ***dipoles***, and either ***horizontal or profile coordinates*** should be used. In the ***horizontal coplanar*** configuration, dipole codes of Z should be used for the transmitter and receiver, while the ***vertical coplanar*** mode should be modelled with dipoles codes of Y. The ***vertical coaxial*** configuration should be modelled with codes of X.

Data should be plotted using the ***centre point*** convention. In this convention, the loops are offset from the centre point by ½ the ***separation*** parameter (the transmitter is located +½ the separation parameter from the plot point). To ensure the coils lie on the profile the separation should in the *x* direction only, so the y and z separation parameters should be zero. ***Profiles*** should be approximately 30 to 40 meters above the ground, but can be significantly higher in tropical areas.

The output should be ***normalized*** in PPM using a free-space model. Normalization geometry is usually the same for the geometrics used in the model.

SAMPO

Sampo is a Finnish electromagnetic system which consists of a transmitter loop roughly 5 - 50 metres across and a three component magnetic receiver and an optional electric field (y directed) receiver.
The method is a ***frequency domain*** technique, and the response can be modelled for many frequencies in one run simply by specifying the required frequency list using frequency domain (F) mode. SAMPO operates at 81 frequencies in the range of 2 - 20,000 Hz.

Since the separation between the transmitter and receiver is usually large, (50-1500 m) with respect to the size of the transmitter loop, both the transmitter and the receiver should be specified as ***dipoles***, and either ***horizontal or profile coordinates*** should be used. If the dimensions of the transmitter are small relative to the distance to the receiver, a dipole code of Z should be used for the transmitter and codes X, Y and Z should be used for the three magnetic receiver components. A dipole code Y should be used for the electric field receiver.

Data should be plotted using the ***centre point*** convention. In this convention, the loops are offset from the centre point by ½ the **Separation** parameter (the transmitter is located +1/2 the separation parameter from the plot point). To ensure the coils lie on the profile the separation should be in the x direction only, so that the y and z separation parameters should be zero. ***Profiles should*** be approximately 0.1 metres above the ground except if z field is used.

CROSSHOLE SURVEYS

Although no specific provision is made in the main interface **EMIGMA** to model crosshole surveys, they can nevertheless be effectively simulated. *Note: **EiKon Technologies** does provide a specific tool for modelling crosshole surveys. In this tool, full crosshole panels can easily be simulated with a variety of different antennae. From this tool, one can access all of the simulation algorithms including EMSPHERE.*

To do so, we recommend using only two halfspaces (layers) in the layered earth component of the input. The layers should each have the same electrical parameters, and the sources and receivers should both be located in the lower layer only. In this way, the numerical components of the layered earth simulation code will not come into play, and the simulation code will be accurate for much higher frequencies than had the layered earth component of the code been used. In this mode of operation, field calculations are based on analytical solutions for a whole space and should be accurate to very high frequencies (GigaHertz).

Crosshole surveys are often operated in a mode where the transmitter is held stationary while the receiver is moved through its own borehole. Once a receiver borehole is completely read for one transmitter position, the transmitter is moved and the process is duplicated again. Such surveys can be simulated in **EMIGMA** using the **fixed transmitter** option, and running a new model for each transmitter to modify the location of the fixed transmitter.

As the transmitter is fixed, it will be specified in terms of the absolute system. Electric sources can be modelled with code Ex and magnetic sources with code Hz but extended electric sources can be used. Note that code Ez assumes the electric source is grounded to the background medium while in the crosshole tool non-grounded radiating transmitters can be utilized. **Absolute, profile or borehole coordinates** can be used to specify the orientation of the receiver. The borehole, in which the receiver is positioned, can be modelled as a profile.

NOTE: In Version 6, the movement of the transmitter down a transmitting borehole will be automated for users who are so licensed.

Note: Crosshole imports, simulation, ray path representation, contouring and tomograph production has been developed by PetRos EiKon and are available as an add-on to EMIGMA. Please contact PetRos EiKon for more information.

CHAPTER III-C - NATURAL FIELD AND PLANE WAVE SYSTEMS

MAGNETOTELLURICS

Please refer to the MT Tutorial

Magnetotelluric survey models can be modelled provided the dongle is licensed for MT. To simulate a magnetotelluric measurement, the ***frequency domain*** mode of operation should be used. Generation of impedances and tippers is done through a post-simulation processing which is normally automated.

Each magnetotelluric station usually consists of 5 receivers, clustered at a common point on a profile. The horizontal electric field receivers are simulated by extended grounded sources which return voltages rather than electric field values. The reason for using the extended receivers instead of simple point dipoles is so the effect of variations in local conductivity on extended electric sensors could be included in the simulation, if these effects are not considered important, merely select dipole lengths of 1 to generate outputs in electric field units. The shorter the dipole length, the faster the run time is.

There are two directions to be established before a magnetotelluric model can be run. The first is the ***polarization direction*** of the incident electric field: ***EMIGMA*** is three-dimensional simulation software and the response will vary with a polarization angle. The second is the direction of the ***x-axis*** of the receivers. Both these directions are input as angles in degrees east of the ***absolute*** y axis. We suggest for the ease of building a model, that the absolute *y* axis be assigned to northwards and the absolute *x* axis to eastwards.

The normal setup automatically causes the model to be run twice with two right angles source polarizations to determine impedances.

☞*NOTE: This option is available to Magnetotelluric system licence holders only.*

Refer to section **MT Station Inputs.**

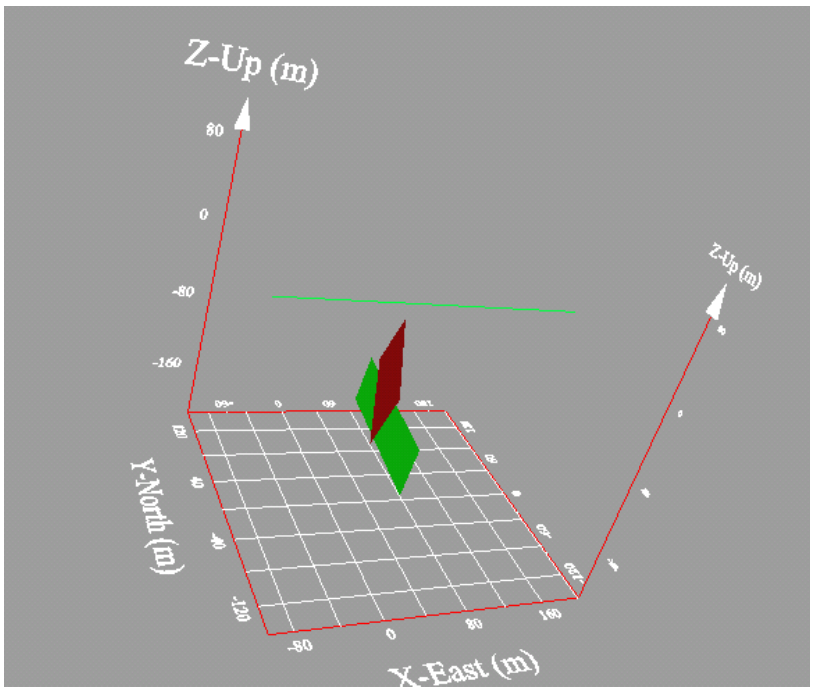
SECTION V - EXAMPLE CALCULATIONS

The User will find on their installation CD example databases which contain example projects for many systems and applications. You may install the example databases or copy then to your hard drive or refer to the CD whenever you need. You can open both database files (*.mdb) into EMIGMA V7 to view the models and simulated data. Remember you will need to change the attributes from "Read Only" to "Archive" if you wish to modify these example databases.

INTRODUCTION

EXAMPLE 1: Max Min - Multiple PLATES

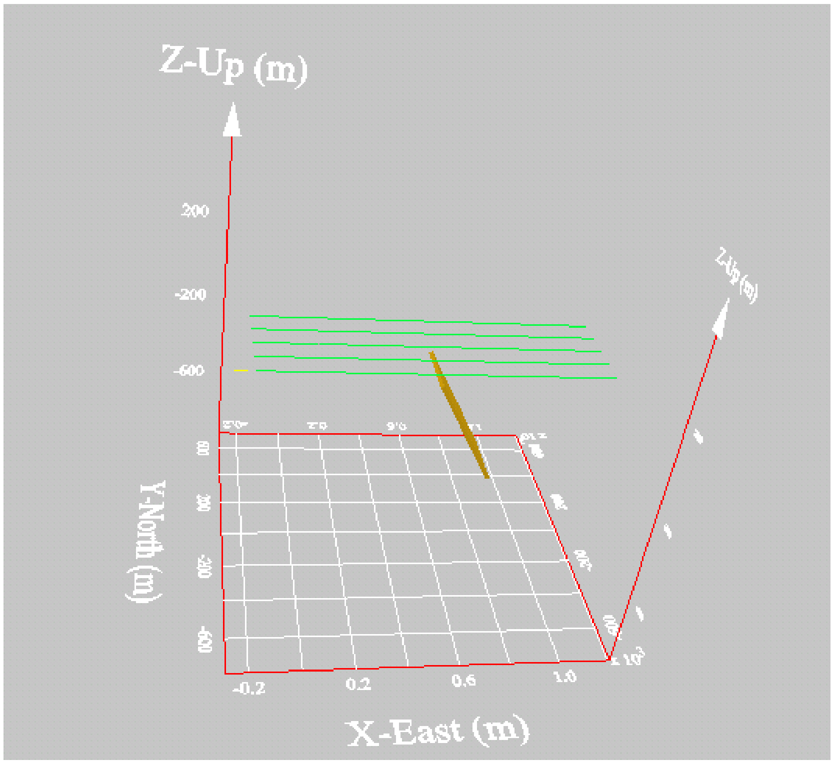
For this example, in the model, there are 2 plates and 3 layers (air, overburden, basement)



PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

EXAMPLE 2: Induced Polarization - IP/Resistivity

Some simple examples of a 3D IP/Resistivity modelling project. Here spectral frequency responses are used allowing transformation to time domain if required. The bipole (extended dipoles) are 50m long in this example for both TX and RX and 6 separations are used - 50m to 300m.



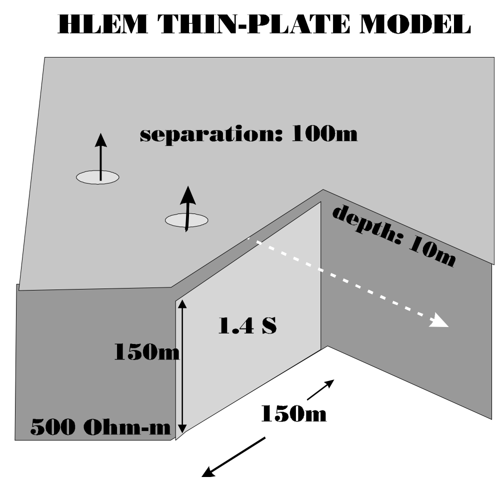
The model consists of a single dipping prisms and 3 layers.

PetRos EiKon 10A Bram Court, Unit 10, Brampton, Ontario Canada L6W 3R6
Tel: 905-796-0324 Fax: 905-796-5214 E-Mail: support@petroseikon.com
Web site: www.petroseikon.com

EXAMPLE 3: Thin-Sheet Models - HLEMsystem

This is a simple comparison between the use of the LN and the VH algorithms to model a thin lens anomaly. The VH algorithm explicitly uses what is termed, in EM, a thin-sheet which has no thickness but a conductivity thickness. To model this type of anomaly, LN utilizes a very thin prism. In this example, we have used a model in which the thin aspect ratio is only 1m. LN is very stable with aspect ratio and one could actually reduce this thickness much further if so desired.

The system simulated is a standard HLEM type system with a 100m dipole separation and a frequency of excitation of 1777Hz. The plate is square (150m) and is buried with a depth to top of 10m in a 500 Ohm-m halfspace. The structure is vertical and strikes perpendicular to the survey direction. The integrated conductivity of the target in the thin direction is 1.4 S. The model is illustrated above. (**Figure 5-1**)



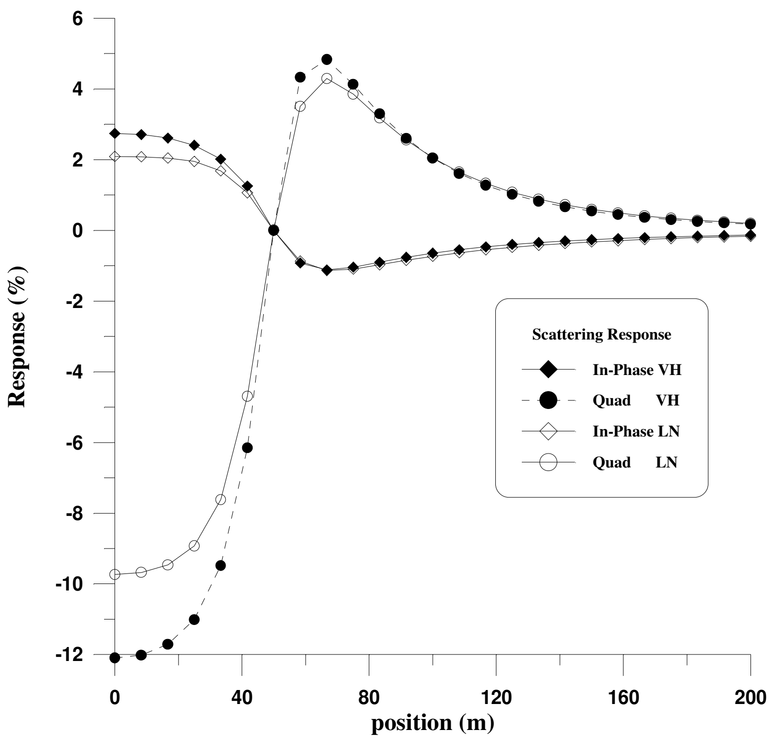


Figure 5-2 shows 4 responses, 2 In-Phase and 2 Quadrature. EMIGMA is used in both cases, once using the LN scattering model and once using the VH scattering model.

This example is used for several reasons, the most obvious being to show the user how to set up a model. The second purpose is to illustrate that there are several algorithms within **EMIGMA**. The user has the choice as to which algorithm to use (when so licensed.)

From a physical perspective, however, this example is used to illustrate a limitation of the LN. If the user notes the response on the flanks of the survey line, it will be noticed that the two algorithms compare extremely well. However, over the top of the plate there is a significant difference between the LN and the VH responses both in the IN-Phase and the Quadrature. This is due to the limited ability of the LN algorithm to reproduce induction effects in the anomaly. In Version 5, a new algorithm termed the ILN algorithm (Induction LN) was introduced, which produces induction effects much better.

EXAMPLE 4: Comparison with a 3-D Code - Coplanar Helicopter System

Next we turn to a full three-dimensional model with some more realistic physical characteristics. In this case, it is a hydrographic model of a shoal of width 32 m with the conductivity of the basement (10 Ohm-metres) rising into seawater 16 metres. The shoal is essentially two-dimensional because the coplanar airborne system flies at 20 metres above the seawater across its centre. This example illustrates that the **LN** approximator generates solutions of comparable accuracy to traditional IE techniques with much greater computational speeds. In addition, the approximator allows much greater flexibility in sampling the scattering currents to produce even greater increases in computation speed. The model is given in **Figure 5-3.**

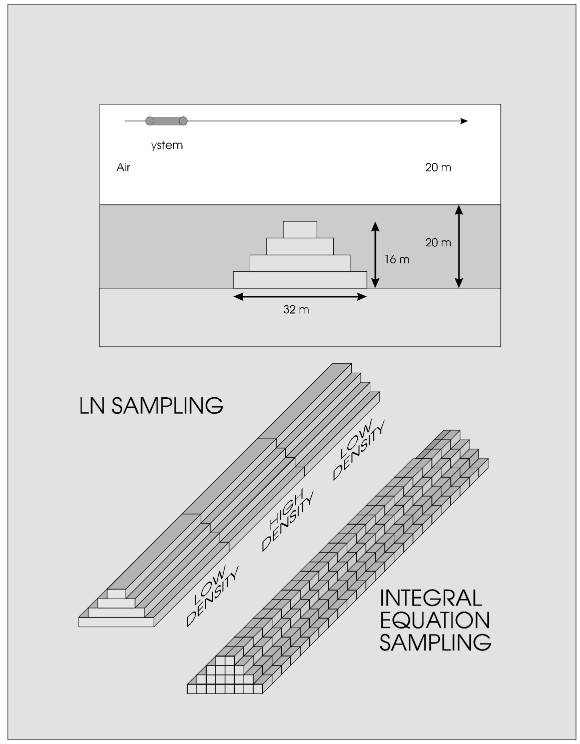
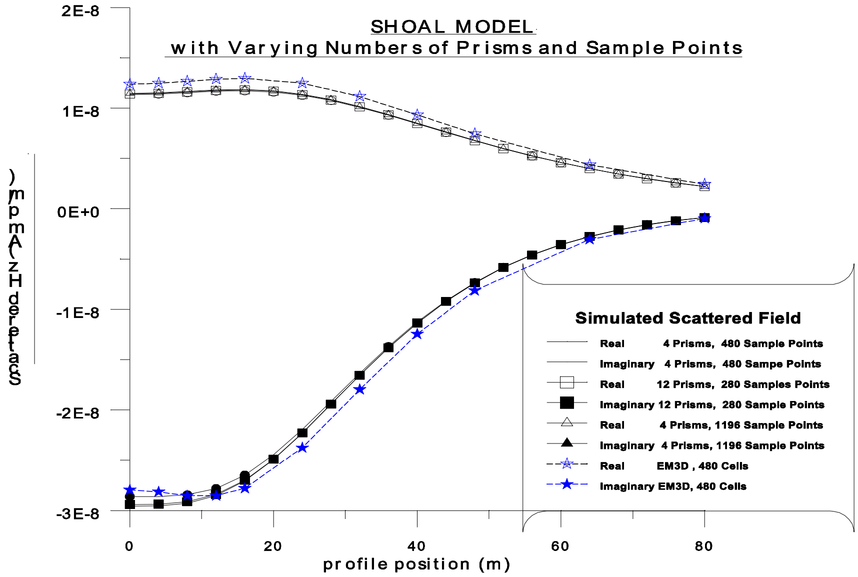


Figure 5-3

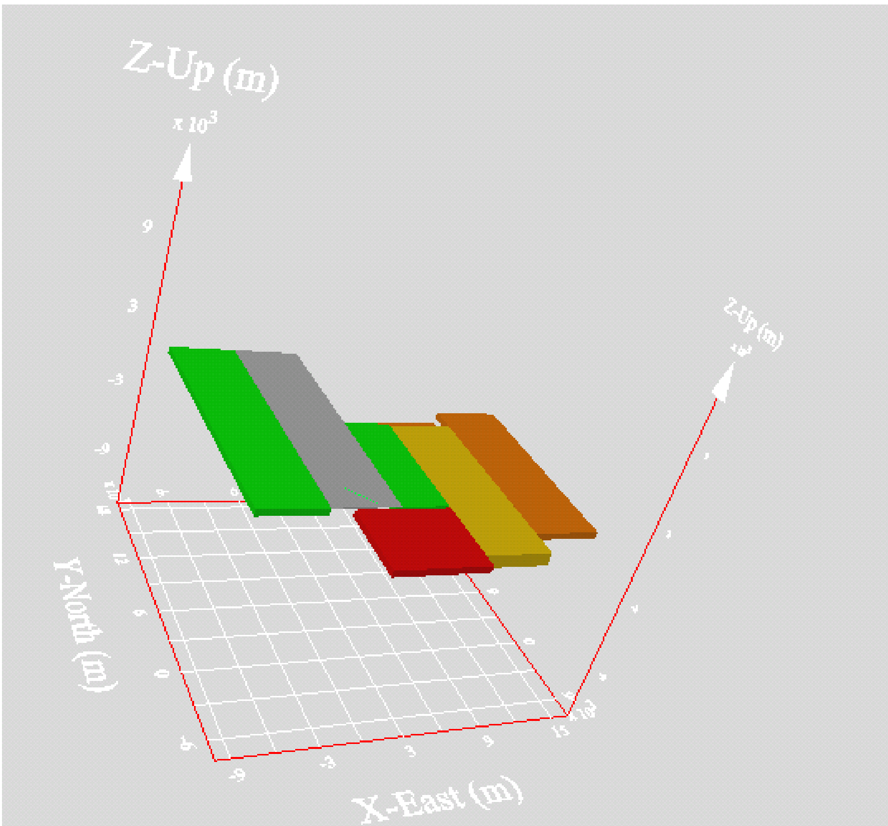


In the **Figure 5-4**, we compare a 280,480 and a 1196 sample point solution for the **LN** scattering model to a 480 cell (double symmetry) solution from the University of Utah code. The solutions are acceptably close although not identical. The computation time for the University of Utah code was about 1.1 hours on a 85 MHz SPARC 5 for 11 stations using the fortuitous symmetry in the model to reduce the computation time by a factor of 64. The calculation time for the **LN** solution using 1196 sample points was about 10 minutes on a PENTIUM 90 for 21 stations though no use was made of symmetry in the **LN** approximator case. More importantly, if the desired structure were not symmetric then 480 cells would be required for the University of Utah Code resulting in an increase in computation time to more than 128 hours for 21 stations assuming the machine had sufficient memory not to require swapping.

This is a fairer comparison to the **LN** computation time since no use was made of symmetry and it is rare in realistic complex geological solutions for such twofold symmetry to so conveniently exist.

Dramatic as these improvements in computation time are, they still do not adequately express the true computational efficiency of the **LN** approximator. To realize this, one must consider the problem of the convergence as a function of the internal scattering current sampling. Analyses of the University of Utah solution as a function of cell size indicated that one could not be sure of an accurate answer (within 5%) with a 4-metre sampling. However, to reduce the sampling one needs to reduce the cells to 2 metres thus increasing the number of cells to 960 using the twofold symmetry. This would increase the computation time from about 6 minutes of CPU per station to about 50 hours/station or 1000 hours for the 21 stations. Clearly, such solutions are not the means of putting rapid EM modelling on the desktop of the practising geophysicist.

EXAMPLE 5: Natural Field Modelling - Magnetotellurics System



EMIGMA can also simulate natural field models and generate, for example, MT impedance simulations. A complex example is given here of an overthrust bounding gravels. The user will note that the only differences in generating simulations are with regard to the MT settings. These settings require defining the polarization angle of the E-field of the source, the orientations of the survey site coordinates with respect to the absolute coordinate system and the lengths of the e-bipole receivers. Refer to the MT section in this manual for more details.

In EMIGMA, the user can create impedances and tippers from both of the orthogonal source polarizations in one model. 1D inverses are available in the 1D tool. Simulation can be provided directly from field data.

SECTION VI – TROUBLE SHOOTING

If you get a TGS License Check Warning - *Product "Open Inventor" is not licensed for this host* - or - *Product "GraphMaster" is not licensed for this host* when you execute EMIGMA V8 (and/or open a file into Viz and/or Contour) - Click **OK**. This is a warning only and you will be able to continue using EMIGMA with full functionality.

SECTION VII - TUTORIALS

The User will find in their installed directory of EMIGMA a subdirectory Examples which contains in subdirectories example files for many systems and applications.

Detailed Tutorials for a variety of purposes are provided within the *Tutorials* sub-directory.

SECTION VI: GENERIC LICENSE

This generic license is included as an example only. Consult your particular licence for special details which may pertain to you.

THIS SOFTWARE LICENSE AGREEMENT made as of the XXX day of XXX , 19XX.
B E T W E E N:

PETROS EIKON INCORPORATED, a corporation incorporated under the laws of Canada

(hereinafter the "Licensor")

- and -

XXXXXXX , a corporation incorporated pursuant to the laws of XXXXXXX

(hereinafter the "User")

WHEREAS the Licensor has agreed to grant to the User a license to use certain computer software.

NOW THEREFORE WITNESSETH that in consideration of the terms of this Agreement and other good and valuable consideration, the parties agree as follows:

1. Definitions

1(1) **"Affiliate"** as used in this Agreement shall mean a company which directly or indirectly controls, or is controlled by or is under common control with a party to this Agreement. The term "control" as used herein means the rights to the exercise, directly or indirectly, of more than 50% of the voting rights attributable to the shares of the controlled company.

1(2) **"Agreement"** shall mean this License Agreement and all schedules hereto;

1(3) **"Licensed Materials"** shall mean user manuals and such other documentation as may be provided by the Licensor to the User for use with the Licensed Program and shall include the Product Definition;

1(4) **"Licensed Program"** shall mean the computer program in a machine readable version of all executable deliverables, excluding the source code, which computer program is owned by the Licensor and also includes any updates, modifications and enhancements thereto provided by the Licensor, unless otherwise agreed by the parties;

1(5) **"Licensed Program and Materials"** shall mean the Licensed Program and Licensed Materials as defined above;

1(6) **"Product Definition"** shall mean a written description of the functions of the Licensed Program, which description is contained in Schedule 1(5).

2. Grant of a License

2(1) Subject to the terms and conditions of this Agreement, the Licensor hereby grants to the User a non-exclusive, non-transferable perpetual license to use the Licensed Program and Materials in machine readable form on one (1) computer at each of the sites specified in Schedule 2(1) for the conduct of the business of the User, other than the purposes specified in Schedule 2(1). The Licensed Program and Materials shall not be kept at any sites other than those specified in Schedule 2(1) with the exception of laptop or other field portable computers owned and operated by the User. This licence is valid for the period and subject to the conditions described in Schedule 2(1).

2(2) The **Licensed Program and Materials** granted herein are restricted to use by the User solely for its own internal operation to process its own data, and not for processing the data of others for hire. The User shall also have the right to use the Licensed Materials to instruct its staff in the operation of the Licensed Program. The User shall have the right to make archival copies of the Licensed Program for the purpose of retrieval in the event of hardware failure or inadvertent destruction of the Licensed Program.

2(3) The User shall have no right to sub-license the use of the Licensed Program and the User shall not assign its license to use the Licensed Program and Materials hereunder, whether voluntarily or by the operation of the law or otherwise, without the prior written approval of the Licensor, which approval may be unreasonably or arbitrarily withheld. Without limiting the generality of the foregoing the User shall not enter any arrangement with any person, firm or corporation whereby use of the Licensed Program and Materials is shared by the User with said person, firm or corporation, with the exception of an employee of a company involved in joint venture operations with a licensed User, or an employee of a contracting company hired by a licensed User to collect and interpret data for the licensed User, in which case

i) Said employee may use licensed software only under the supervision of an employee of the User, who shall ensure the licensed software is used in the case of joint ventures exclusively for the purposes of the joint venture activity or in the case of an employed contracting company, for interpretation of the data sets collected by the contractor for the User.

ii) Should any individual who is not an employee of the User who is involved in the joint venture or who is an employee of a contracting company be granted access to the software under this exception, the User undertakes to make said individual aware of the license restrictions.

iii) The licensed software may only be installed and operated at a normal place of business of the User in the case where the Licensed software is to be operated under these joint vemture or contracting company provisions.

iv) The User agrees to accept responsibility for any misuse of the Licensed Program and Materials by third parties, their employees and agents, using such Licensed Program and Materials pursuant to a contract with the User and to indemnify the Licensor for any damages arising as a result of such misuse which a court of competent jurisdiction has ordered should be paid to the Licensor.

2(4) All results generated by the User, wholly or partially, by means of the use of the Licensed Program and Materials shall contain the appropriate acknowledgement of the use of the Licensed Program and Materials by the User whenever said results are published or could be made available to any person or group except the Licensed User.

3. Copies of Licensed Materials

3(1) The Licensor shall provide the User with the number of copies specified in Schedule 6(1) of the Licensed Materials. Any additional copies of the Licensed Materials as are reasonably necessary for the use of the Licensed Program by the User shall be provided promptly by the Licensor to the User following a request from the User for these to be provided at the then current charges therefore.

4. Data Conversion and Operation

4(1) The Licensor shall provide the User with specifications for data formats for use with the Licensed Program. The User is responsible for conversion of its current data to the data format required for use with the Licensed Program.

4(2) The Licensed Program is designed to operate on the operating system with compatible hardware as specified in Schedule 4(2). If the User requests modifications of the Licensed Program to meet particular requirements, the Licensor may at its discretion modify the Licensed Program to operate under such conditions subject to applicable service charges to be negotiated prior to the start of the work. This Agreement shall apply to all such modifications to the Licensed Program.

5. Services

5(1) In accordance with the conditions specified in Schedule 5(1), the User may purchase from the Licensor, support and corrections and improvements to the Licensed Program.

5(2) Upon expiration of any support period specified in Schedule 5(1) the User may purchase from the Licensor, support and corrections and improvements to the Licensed Program.

5(3) The User acknowledges that the Licensor, upon expiration of any support period, may elect not to offer continued support for the Licensed Program if the Licensed Program has been withdrawn from the market.

6. License Fee

6(1) In consideration of the grant of the Licensed Program and Materials by the Licensor to the User hereunder, the User has agreed to pay the amount specified in schedule 6(1), and in so doing has acquired an option to acquire additional licenses in accordance with the provisions of Schedule 6(1).

6(2) The license fee does not include any provincial sales taxes, or any other taxes or duties whether presently in force or imposed in the future, and any such taxes or duties shall be assumed and paid by the User. In the event that further taxes are determined to be payable, the User shall promptly pay them upon demand by the Licensor.

6(3) If the User fails or neglects to make any payment for the Licensed Program when due, the Licensor may at its own option, subject to the terms of Clause 10(2) and in addition to any other right which it has under this Agreement or at law:

(i) enter upon the User's premises and take possession of the Licensed Program and Materials during normal business hours;

(ii) delay delivery of any computer equipment or other materials, or the performance of any services under this or any other agreement with the User until such payment is made; and/or

(iii) delay delivery of any part of the Licensed Program and Materials until such payment is made.

7. Delivery

7(1) The Licensed Program and Materials will be delivered to the User on the date agreed to by the parties and specified in Schedule 7(1) hereto. The Licensor shall use all reasonable efforts to meet such delivery dates but does not represent or warrant that such delivery dates will be met.

7(2) If the Licensed Program or Licensed Materials are lost or damaged during shipment from Licensor to User, the Licensor shall replace same at no additional cost to the User.

8. Warranty

8(1) The Licensor does not warrant that: (1) the functions contained in the Licensed Program will meet the requirements of the User; (2) the Licensed Program will operate in the combinations which may be selected by the User; (3) that the operation of the Licensed Program will be uninterrupted or error free; or (4) that all defects in the Licensed Program will be corrected.

8(2) No warranties, either express or implied, are made to the User by the Licensor regarding the Licensed Program and Materials.

8(3) In the event of breach of this Agreement by the Licensor its sole obligation to the User shall be to replace the Licensed Program and Materials free of charge to the User. Without limiting the generality of the foregoing, the Licensor shall not be liable to the User for any loss, cost, damages, claims, actions, suits suffered by the User resulting consequent upon a breach by the Licensor of any of its obligations herein.

9. Ownership of Licensed Program and Confidentiality

9(1) The User acknowledges that the Licensed Program and Materials are the property of the Licensor and that the only right that the User obtains to the Licensed Program and Materials is the right of use in accordance with the terms of this Agreement. The parties acknowledge that any machine readable software products created by the User but embedding the Licensed Program shall be the property of the Licensor.

9(2) The User will ensure that all copyright, proprietary and trade secret notices of the Licensor will remain on the Licensed Program in any form, and on all Licensed Materials. The use of a copyright notice on the Licensed Program and Materials shall not be taken to indicate that they have been published.

9(3) The User acknowledges that the Licensed Program and Materials and any archival copies of the Licensed Program contain proprietary and confidential information of the Licensor. The User will take the same care to safeguard the Licensed Program and Materials as it takes to safeguard its own confidential information and such care shall not be any less than would be taken by a reasonable company to safeguard its information, but so that the User shall be free to disclose information to an Affiliate provided that the Affiliate agrees to keep such information confidential and not to disclose it to any third party.

Without limiting the generality of the foregoing, the Licensed Program shall be accessible only to those employees of the User and employees of Affiliate companies of the User who, in User's sole opinion, have a need for such access to perform their duties, and Licensed Materials other than operator's manuals shall be stored in a secure place and shall be accessible only to those employees of the User with a need for such access in order to perform their duties. Employees of the User or of Affiliate companies of the User, having such access shall be specifically advised of the confidentiality of Licensed Program and Materials. Operator's manuals shall prominently bear a legend stating that they are the property of the Licensor and they contain confidential information, and operators shall not be permitted to take them from their workplace. In the event the Licensed Program and Materials have been stolen, the User shall not be held liable for injury caused by such theft to the Licensor provided

i) The User notifies the Licensor of the theft upon its discovery
ii) The User has acted reasonably in safeguarding the Licensed Program and Materials and has provided the Licensed Program and Materials with the same level of security as is provided to its own business assets.

9(4) No copies of any portions of the Licensed Program or Licensed Materials shall be made by the User or the User's employees save that the User may make copies of the Licensed Program and updates or enhancements (if any) provided by the Licensor for archival and backup purposes. Said copy shall be and shall remain the property of the Licensor.

9(5) In order to assist the Licensor in the protection of its proprietary rights with respect to the Licensed Program and Materials, the User shall permit the Licensor to inspect, during the normal business hours, the facility at which the Licensed Program is used and any facility at which the Licensed Program or Licensed Materials are stored at a mutually agreed time provided the Licensor has reasonable cause to believe that a breach of the licensing provisions by the User has occurred.

9(6) The Licensor shall not disclose to any third parties, other than the employees of the Licensor, any business or financial information of the User provided to the Licensor in confidence, except to the extent such disclosure is required by the Licensor to enforce its rights under this Agreement.

10. Default

10(1) Failure of the Licensor or the User to comply with any term or condition of this
Agreement shall entitle the other party to give the party in default written notice requiring it to make good such default.

10(2) If the default complained of has not been cured within thirty (30) days following receipt of such notice, the notifying party shall be entitled, in addition to any other rights it may have under this Agreement or otherwise at law, to terminate this Agreement by giving notice to the other to take effect immediately.

10(3) It is recognized by the parties that the confidentiality of the Licensed Program and Materials is of great and central importance to the business of the Licensor. The parties therefore agree, that if the User shall breach any term of Article 9 of this

Agreement entitled "Ownership of Licensed Program and Confidentiality," then the Licensor shall have the right to terminate provided that (i) the User provides the Licensor with written notice of the material breach, which notice shall specify the nature of such breach and for thirty (30) days to correct such alleged breach and that notice has expired and the failure has not been corrected.

10(4) This Agreement shall terminate immediately and automatically if the User enters or is placed into receivership or if the User is petitioned into bankruptcy or makes a proposal under the Bankruptcy and Insolvency Act for the benefit of its creditors, or ceases to carry on business or is wound up.

10(5) The User may terminate this Agreement by thirty (30) days notice in writing to the Licensor to do so, whereupon the User shall have no further obligations to the Licensor other than those referred to in Article 11.

11. Procedure on Termination

11(1) Upon any termination of this Agreement, the User shall return the Licensed Program and Materials and any full or partial copies thereof to the Licensor and shall certify, under the hand of a duly authorized officer of the User, that the original and all copies of the Licensed Program and Materials have been given up to the Licensor, all records or copies of the Licensed Program or Materials on all media have been destroyed, and that no copies of any part of the Licensed Program and Materials, in any form, remain in the possession or control of the User.

11(2) Termination of this Agreement shall not affect any right of action of either party arising from anything which was done or not done, as the case may be, prior to such termination taking effect.

12. General

12(1) Neither party shall be liable for delay or failure in performance resulting from acts beyond the control of such party, including, but not limited to Acts of God, acts of war, riot, fire, flood, or other disaster, acts of government, strike, lockout, communication line or power failures, or failure or inoperability of any software other than the Licensed Program. Provided that no party shall be excused from the obligation to pay any amount due pursuant to this Agreement due to any of the above causes.

12(2) This Agreement shall be governed by and construed in accordance with the laws of the Province of Ontario, Canada.

12(3) The User shall have no right to assign this Agreement without the express written consent of the Licensor, which consent may be unreasonably or arbitrarily withheld. If the User merges with another entity or sells or assigns its entire business to another entity, the Licensor will not withhold its permission to assign this Agreement to such other entity, provided that the Licensor is reasonably satisfied that the confidentiality of the Licensed Program and Materials will be maintained.

12(4) Any notice, demand or request that any party shall give to the other shall be in writing and shall be deemed to have been validly given if delivered at or mailed by registered mail or sent by telecopier to the parties at the addresses specified below or to such other address as such party shall have given written notice. Any notice, demand or request shall be deemed to have been received at the time it was delivered; or in the case of those given by registered mail on the third business day following the date of mailing. Any notice, demand or request given by telecopier shall be deemed to have been received on the date the transmission is received, provided if the date of receipt of the transmission is not a business day, notice shall be deemed to have been received on the first business day following the date of receipt of the transmission. In the event of threatened or actual postal disruption due to strike, lockout or labour dispute, any notice, demand or request shall be delivered personally or sent by telecopier to the addressee.

Licensor: PetRos EiKon Incorporated.
917 Nipissing Road
Milton, Ontario
L9T 5E3

User: XXXXXXX
XXXXXXX
XXXXXXX
XXXXXXX

12(5) This Agreement and any schedule attached hereto and initialled or signed by both parties contains the complete and exclusive statement of the Agreement between the parties with respect to the subject matter hereof and supersedes all prior and contemporaneous agreements, understandings, proposals, negotiations, representations or warranties of any kind, whether oral or written. No oral or written representation that is not expressly contained in this Agreement is binding on the Licensor.

12(6) If any provision of this Agreement is declared by a Court of competent jurisdiction to be invalid, illegal or unenforceable, such provision shall be severed from the Agreement and the other provisions shall remain in full force and effect.

12(7) A term or condition of this Agreement can be waived or modified only by written consent of both parties. Forbearance or indulgence by either party in any regard shall not constitute a waiver of the term or condition to be performed, and either party may invoke any remedy available under the Agreement or by law despite such forbearance or indulgence.

12(8) In the event of the termination of this Agreement, the provisions of section 8(3) and Articles 9, 10, 11 and 12 hereof shall remain in full force and effect, until such time as the parties mutually agree to the release of the terms thereof.

12(9) This Agreement may be executed by the parties in counterparts, all of which, when taken together shall constitute a fully executed version of this Agreement. A copy of this Agreement executed by a party and sent by telecopier to the other party (or to that Other party's respective solicitors) shall constitute delivery of an executed counterpart of this Agreement.

IN WITNESS WHEREOF the parties have executed this Agreement as of XXXXXXXXXX

PETROS EIKON INCORPORATED

Per:

A.S.O.

Date

XXXXXXXXXX

Per:

A.S.O.

Date

SCHEDULE 1(5)

Product Definition

The Licensed Programme, **EMIGMA** models electromagnetic scattering from a scatterer or scatterers in a horizontally stratified ground.

The Licensed Programme consists of a number of licensable components each of which can be licensed either separately or together

1. A license to execute machine readable code on an XXXXX computer located at the licensed sites of

XXXXXXXXX
XXXXXXXXX

2. One copy of an operating manual in English for modelling programs described in section (1) of this Schedule.

3. One software key device.

4. The User has the option to purchase additional licenses until XXXXXXX for XXXXXXX
per license in XXXXXXX funds.

SCHEDULE 2(1)

Number of Computers, Sites and Use of Licensed Program and Materials

1. The Licensed Programs and Materials may be used on 1a XXXXXX computer owned or leased by the User which is used for the purposes of conducting his regular business at each of the sites referred to in this Schedule 2(1).

2. The Licensed Program and Materials may be used at any location in the world for the normal business of the User. The Licensed Program and Materials may be transported from site to site, and may be installed upon a laptop or other field portable computer owned or leased by the User for the purposes of his business, provided such use and installation is in conformity with the provisions of the License. The User agrees not to run the same copy of the Licensed Program on more than one computer at the same time.

3. The parties acknowledge that the business of the User is mineral exploration and related data processing and research services. The User shall not use the Licensed Program and Materials for any of the following purposes:

(a) commercial consulting;

(b) preparation of nomograms, typed curves and/or tables for use by interpretation software distributed to third parties;

(c) interpretation of data sets on a commercial consulting basis or for inclusion, as a minor component, in the report of work carried out on a commercial consulting basis.

For greater certainty, the following use of the Licensed Program and Materials shall not be considered to be a commercial consulting use:

(a) demonstrations to third parties for promotion of the services and products provided by the User in the ordinary course of business;

(b) preparation of typed curves, nomograms and/or tables not otherwise restricted by this Agreement;

(c) basic geophysical research;

(d) equipment or survey design;

(e) use of the software by a partner engaged in a joint venture, or by a commercial contractor who has acquired data and shall be interpreting that data for the User, as specified in section 2(3) , provided that in the case where the Licensed Software is used in work performed for/with joint venture partners, the partners are not charged for the use of the Licensed Software, and the use of the Licensed Software forms a small component of the total contribution of the User toward the joint venture activity.

PRIMARY SITE

CONTACT:

ADDITIONAL SUPPORTED SITE LICENSES

CONTACT:

SCHEDULE 4(2)

Operation

The Licensed Software shall execute on an Intel based computer under one of the DOS, WINDOWS[1], OS/2 or UNIX operating systems, with the selection of operating system to be WINDOWS unless otherwise specified by the User.

OPERATING SYSTEM: ____

[1] **The precise choice of WINDOWS system is still as yet undefined, but the Licensor will endeavour to provide support for those WINDOWS operating systems for which stable compilers become available.**

SCHEDULE 5(1)

Support

1. The Licensor shall provide the following support to the User for the period of the first year of the licence at no cost to the User, through an individual specified by the User to the Licensor**:**

(a) 12 telephone service calls per year as required by the User.

(b) modifications and upgrades to the Licensed Programs and Materials as they are released by the Licensor**;**

provided the cost of all long distance telephone charges incurred by the Licensor in providing support to the User shall be paid by the User.

2. The User may elect to purchase extended support of the Licensed Program and Materials granted under this license at the then currently applicable rates (usually 15% of the purchase price). This will entitle the user to

(a) 12 telephone service calls per year as required by the user ;

(b) modifications and upgrades to the Licensed Programs and Materials as they are released by the Licensor ;

provided the cost of all long distance telephone service charges incurred by the Licensor shall be paid by the User. Telephone calls in excess of the supported amount will be billed at $50.00 per call.

SCHEDULE 6(1)

License Fee and Operation

1. License Fees are payable in Canadian funds as follows

a) XXXXXXX in XXXXX funds for a single perpetual XXXXXX based computer license.

b) XXXXXXX in XXXXX funds per additional unsupported "perpetual user" upto XXXXX licenses

c) XXXXXXX in XXXXX funds per additional supported "perpetual user" upto XXXXX licenses, where support conditions are specified in Schedule 5(1)

SCHEDULE 7(1)

Delivery

1. The Licensed Program and Materials, upgrades and additional licensed copies of the software shall be shipped by mail to the User at the address specified below. Requested installations shall be billable to the User at a consulting rate of XXXXXX per day in XXXX funds, plus travel expenses.

USER: ____________________________________

Attn:______________________________________

Address:

____
____
____
____

SECTION VIII - REFERENCES

REFERENCES

Some of these reference documents are provided in the *Technical* sub-directory.

Papers

1. Archie, G.E. (1942),
The electrical resistivity log as an aid in determining some reservoir characteristics: *Trans. A.I.M.E.*, **146**, 52-62.

2. Marshall, D.J., Madden, T.R. (1959).
Induced polarization: A study of its causes. *Geophysics*, **24**, 790-816.

3. Seigel, H.O. (1959).
Mathematical formulation and type curves for induced polarization. *Geophysics*, **24**, 547-65.

4. Hohmann, G.W. (1975)
Three-dimensional inducted polarization and electromagnetic modelling. *Geophysics* **40**

5. Hanneson, J.E. and West G.F. (1984)
The horizontal loop electromagnetic response of a thin plate in a conductive earth: part 1 - Computational method: *Geophysics*, **49**

6. Telford, W.M., Geldart, L.P., Sheriff, R.E., Keys, D.A., (1988).
Applied Geophysics: Cambridge University Press, 451-457.

7. Walker, P.W. and West, G.F. (1991)
A robust integral equation solution for electromagnetic scattering by a thin plate in conductive media, *Geophysics* **56**, 1140-1152

8. Habashy T. M, Groom, R.W. and Spies. B.R. (1993)
Beyond the Born and Rytov Approximations: A Nonlinear Approach to Electromagnetic Scattering, *J. Geophys Res*., **98**

9. Groom, R.W., Hyde, C.H. and Lajoie, J. (1996)
A case study of the application of the EMIGMA modelling package in interpretation of UTEM data over the Cominco Cerattepe deposit in Turkey, Extended Abstracts, 66th SEG Conference, Denver, Colorado
On your CD: 1996_Cerattepe.pdf

10. Parker, D.H, Boivin, M., Murray, I. R and Groom, R.W. (1997).

A case study on the application of the EMIGMA modelling package to Crone data over the SOQUEM Lac Volant Region, Sept-Îles, Quebec, Extended Abstracts, 67th SEG Conference, Dallas, Texas.
On your CD: 1997_CaseStudyCrone.pdf

11. Murray, I.R. (1997)
On Extending the Localized Non-Linear Approximation to Inductive Modes Extended Abstracts, **F004**, 59th EAGE Conference, Geneva, Switzerland.
On your CD: 1997_ExtLocLN_EAGE.pdf

12. Murray, I. R., Alvarez, C. and Groom, R.W. (1999).
Modelling of complex electromagnetic targets using advanced non-linear approximator techniques, Extended Abstracts, 69th SEG Conference, Houston, Texas.
On your CD: 1999_ModelComplEMTarg.pdf

13. Groom, R.W., Murray, Alvarez, C., North, J. (1999).
An Interpretation Study of Pulse EM Reconnaissance Data from the Raglan Belt of Northern Canada, Extended Abstracts, 3D Symposium, Utah, Colorado.
On your CD: 1999_IntStudyPulseEMRaglan.pdf

14. R.W. Groom, C. Candy (2001)
Development in a normal mode helical electrical antennae crosshole instrumentation and integrated interpretation system
On your CD: 2001_DevelopHelicalElAnt.pdf

15. R. Groom, C. Alvarez (SAGEEP 2002).
3D EM Modelling – Application of the Localized Non-Linear Approximator to Near Surface Applications
On your CD: 2002_3DEMModelling.pdf

16. Ruizhong Jia (2002)
An Inverse Magnetic Problem
On your CD: 2002_InverseMag.pdf

17. R.W. Groom, Ruizhong Jia, C. Alvrez (2003)
Investigations into Inversion of Magnetic and Gradient Magnetic Data for Detection and Discrimination of Metallic Objects
On your CD: 2003_InvMagGradSAGEEP.pdf

18. Ruizhong Jia, R.W. Groom (SAGEEP 2004)
On Inversion of Gradient Magnetic Data for Detection of Multiple Buried Metallic Objects
On your CD: 2004_EulerVectorInvUXO.pdf

19. R.W. Groom, Ruizhong Jia, Bob Lo (2004)
Magnetic Compensation of magnetic noise related to aircraft's maneuvers in airborne survey. Extended Abstract SAGEEP 2004
On your CD: 2004_MagCompSAGEEP.pdf

20. Ruizhong Jia, R.W. Groom, Bob Lo (2004)
The Use of GPS Sensors and Numerical Improvements in Aeromagnetic Compensation
On your CD: 2004_MagCompwGPS.pdf

21. FSEMTRS (2004). Frequency To Time Transform (Internal paper)
On your CD: 2004_FSEMTRS.pdf

Technical Talks In PowerPoint format

1. R.W. Groom, Bob Lo (2002)
Some Pitfalls in Gradient Magnetic Processing or Why Rotate Gradients
On your CD: 2002_PitfallsGradMagProcess.pdf

2. Bob Lo, R.W. Groom
Improved Depth Distributions by Inversion of Magnetic Surveys Collected at Different Survey Heights
On your CD: ImprDepthDistrInvMag.pdf

3. Exploring for Groundwater with EM Methods. R. Groom
On your CD: ExplGroundwaterEM.pdf

4. Magnetic 3D Inversion
On your CD: Mag3DInv.pdf

5. 3D Modelling of Near Surface Problems
On your CD: 3DModelNearSurfProblems.pdf

6. UXO Applications for Geophysics: TEM Modelling for UXO's using EMSphere (2002)
On your CD: UXOApplGeoph.pdf

7. X-hole Tomography. A New Frontier in Equipment and Software
On your CD: XholeTomogr.pdf

8. Developments in a Normal Model Helical Electrical Antennae Crosshole Instrumentation and Integrated Interpretation System
On your CD: 2001_HelicaAntXhole.pdf

9. Inversion of Gradient Magnetic Data for Detection and Discrimination of Metallic Objects
On your CD: 2003_InvGradMagData.pdf
2004_InvGradMagUXO.pdf

10. R.W. Groom, Ruizhong Jia, Bob Lo
Magnetic Compensation of magnetic noise related to aircraft's maneuvers in airborne survey.
PowerPoint Presentation at SAGEEP 2004
On your CD: 2004_MagCompSAGEEP.pdf

11. 3D Modelling – Application of the Localized Non-Linear Approximator to Near Surface Applications. R. Groom, C. Alvarez
On your CD: 2002_LocLNApproxNearSurf.pdf