A guide to using

WinGLink®

By GEOSYSTEM SRL

WinGLink® User's Guide, Release 2.20.02.01

This Manual is produced solely for WinGLink Users Reference and circulation is restricted to registered WinGLink® Users.

Printed in Milan, January 10, 2008

WinGLink ® is a registered Trademark of:

GEOSYSTEM SRL
Via Clericetti 42a, 20133 MILAN, Italy
Tel :++39-02-266279.1, Fax:++39-02-266279.279
www.geosystem.net

Geosystem License Agreement for WinGLink Software

The Agreement

This Geosystem End-User License Agreement is a legal agreement between you (either an individual or a single entity) and Geosystem srl for the software product WinGLink®, which includes computer software, associated media, hardware key and printed materials.

IMPORTANT—READ CAREFULLY

WinGLink® software will only work at full performance when a hardware key (DONGLE) is installed on the parallel port of your PC.

By installing and using WinGLink® in conjunction with its DONGLE, you agree to be bound by the terms of this Agreement. If you do not agree to the terms of this Agreement, return the DONGLE(S) to us for a full refund within 10 days from the date you received them.

WinGLink® LICENSE

WinGLink® software is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The WinGLink® software is licensed, not sold.

GEOSYSTEM grants to you a non-transferable and non-exclusive license to

1. use the WinGLink® Program for your own purposes but only in machine readable form by no-more than the number of users stipulated in Schedule 1. Any exceptions are noted on Schedule 1.

2. utilize the Support Materials and test data in support of the authorized use of the WinGLink® Program;

You may not assign this agreement or any part thereof or sub-license the rights granted herein without the prior consent of GEOSYSTEM.

Title and all intellectual property rights in and to the WinGLink® Program, Support Materials and test data, including, without limitation, copyright, trade secrets, and trade marks, shall remain with GEOSYSTEM. You agree to refrain from raising any objection or challenge to such intellectual property rights, or from assisting or causing or permitting other(s) to do so, during the term of the Agreement and thereafter.

Services provided by GEOSYSTEM

GEOSYSTEM shall supply you with the WinGLink® Program listed in Schedule 1 at all sites listed in Schedule 1, along with Support Materials (as hereinafter defined), hardware keys (dongles) and test data.

GEOSYSTEM shall perform the services, and grant you a non-transferable, non-exclusive license to use the WinGLink® Program, Support Materials, and test data, subject to the Terms and Conditions herein contained. You agree to accept responsibility for the selection, installation, and use of the appropriate programs to achieve your intended results, and for the results obtained from use of the WinGLink® Program. You therefore accept complete responsibility for any decision made based on your use of the WinGLink® Program.

Definitions

In this Agreement the term:	means:
“Agreement”	this Agreement including Schedule 1, included in the shipment of each dongle;
“Effective Date”	The date when you can legaly start to use the WinGLink® program
“Fees”	(i) The one-time initial User(s) License Fee, (ii) The User(s) Maintenance Fee,
“WinGLink® Program”	the version of the licensed data processing program listed in Schedule 1, and any modifications, improvements, or updates provided to you by GEOSYSTEM, collectively of any of them;
“User License(s)”	the right to use the WinGLink® Program by one user. This includes the use on field computers at temporary sites for projects directly subordinate to the Licensed Office.
“Dongle”	the hardware , needed to run the WinGLink® Program.
“Service charge(s)”	GEOSYSTEM’s service charges relating to the services, in accordance with GEOSYSTEM’s current fee schedule;
“Services”	the Services described in Section 7 and Schedule 1, collectively or any of them;
“Support Materials”	any machine-readable or printed material describing or embodying the WinGLink® Program provided to you by GEOSYSTEM under this agreement, including flow charts and applications manuals
“Termination”	means the occurrences contemplated by Section Termination.

Fees and payment

You agree to pay GEOSYSTEM the Fees as follows:

1. The initial User License(s) Fees are to be paid upon receipt of GEOSYSTEM’s invoice and subsequent Maintenance Fees are to be paid within 30 days of GEOSYSTEM’s invoice

2. The Fees are subject to change without prior notice. In addition to the Fees due under this Agreement, you agree to pay amounts equal to any taxes resulting from this Agreement or any activities hereunder. Interest of 1.5% per month shall be paid on any Fees overdue. All payments are to be made payable in US Dollars, or in the currency shown in GEOSYSTEM’s invoices, at the rate of exchange to the US Dollars effective at the date of the invoices.

Term

The Term of this Agreement shall commence on the Effective Date and terminate upon Termination.

Maintenance Services

3. During the first 6 months of this Agreement (from the Effective Date), or up to any later expiry date specified in Schedule 1, GEOSYSTEM shall make available to you, without additional Fees, such corrections and improvements to the WinGLink® Program as may be generally incorporated into the WinGLink® Program by GEOSYSTEM.

4. After expiry of this 6 month period, or after any later expiry date specified in Schedule 1, GEOSYSTEM shall make available all generally available updates of the WinGLink® Program at the then applicable updated subscription(s) charges. A quotation of the update subscription will be provided prior to the expiry date at the then applicable update subscription charge(s). GEOSYSTEM‘s current policy is to maintain the cost of a 12-months subscription at about 15% of the software License Fee;

5. GEOSYSTEM is committed to offer you, subject to the applicable Service Charge(s), continuing support in the form of telephone advice and other assistance in problem diagnosis and the correction of errors or faults in the WinGLink® Program during the life of this License. When a problem occurs, which you believe is related to the errors or faults in the WinGLink® Program, you may contact GEOSYSTEM at the above address and GEOSYSTEM will make an honest effort to solve your problem. However, GEOSYSTEM cannot guarantee service results or represent or warrant that all errors or program defects will be corrected.

6. Further, if your request for service relates to modifications of the WinGLink® Program, to meet your particular needs or conform with your particular operating environment, GEOSYSTEM may, at its discretion, modify the WinGLink® Program to meet your particular needs at your request, subject to Applicable Service charge. However all intellectual property or other rights which may arise from such modification, shall reside with GEOSYSTEM.

Protection and security of the WinGLink® Program and Support Materials

You acknowledge that all copies of the WinGLink® Program, and the Support Materials provided by GEOSYSTEM or made by you pursuant to this Agreement, are proprietary, and the property of GEOSYSTEM, and may not be distributed by you to any other person, without GEOSYSTEM's prior written consent. You will not provide or otherwise make the WinGLink® Program or the Support Materials available to anyone in any form without GEOSYSTEM's prior written consent, except to your employees, or to other persons on your premises for purposes specifically related to your authorized use of the WinGLink® Program.

Termination

This Agreement may be terminated by you only upon thirty-days prior written notice to GEOSYSTEM. GEOSYSTEM may terminate this Agreement upon prior written notice, effective immediately if you fail to comply with any of the terms and conditions of this Agreement.

Events upon termination

You shall forthwith discontinue use of the WinGLink® Program, Support Materials and test data on the day Termination shall occur and you agree not to resume such use in the future without written authorization from GEOSYSTEM.

Within 30 days after termination, you shall return to GEOSYSTEM, at your expense, the WinGLink® Program, Dongles, Support Materials, test data and other material received from GEOSYSTEM or copies permitted to be made in connection with this Agreement. This obligation relates, without limitation, to all copies in any form, including translation, compilation, derivatives and updated materials, whether partial or complete, and whether or not modified or merged into other materials, as authorized herein.

Warranty

GEOSYSTEM does not warrant that the functions contained in the WinGLink® Program will meet your requirements or that the operations of the WinGLink® Program will be uninterrupted or error free, or that all program defects will be corrected. Each WinGLink® Program shall be furnished to you in accordance with the terms of this Agreement.

Limitation of remedies

In no event shall GEOSYSTEM be liable for any damages arising from performance or non-performance of the WinGLink® Program, or for any lost profits, lost savings or other consequential damages, even if GEOSYSTEM has been advised of the possibility of such damages, or for any claim against you by any other party.

General

You agree that this Agreement is a complete and exclusive statement of the Agreement between you and GEOSYSTEM and can only be modified by written instrument executed by both parties. GEOSYSTEM is not responsible for failure to fulfill its obligations under this Agreement due to causes beyond its control.

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

The provision of this Agreement shall be binding upon you and GEOSYSTEM and your respective successors and permitted assigns.

This Agreement will be governed by the laws of Italy.

Amendments

This Agreement together with Schedule 1 constitutes the entire agreement and it supersedes all prior agreements, representation and understandings between you and GEOSYSTEM.

Table of Contents

1: Introduction

Overview of the Manual Structure

This edition of the WinGLink manual differs considerably from previous editions. Instead of taking a module-oriented approach, the manual now follows the workflow a user might typically follow when working with a specific type of data.

For example, the description for magnetotelluric data guides the user from the various quality control elements of data processing and analysis, e.g. use of the Time Series program and data editing, to data import, map and profile creation, 1D modeling, the creation of pseudo-sections and cross-sections, to 2D modeling, 3D modeling, and finally, provides information on how to prepare data for presentation.

As a result of this approach, some modules appear in the manual multiple times, once for each specific data type. For example, separate chapters are dedicated to MT Soundings and TDEM Soundings.

Graphical overviews of the typical workflow followed for each type of data handled by WinGLink are provided in the “Workflow” chapter of this manual. In addition to illustrating a step-by-step approach to working with the respective data types, the diagrams also include references to the corresponding chapters of the manual.

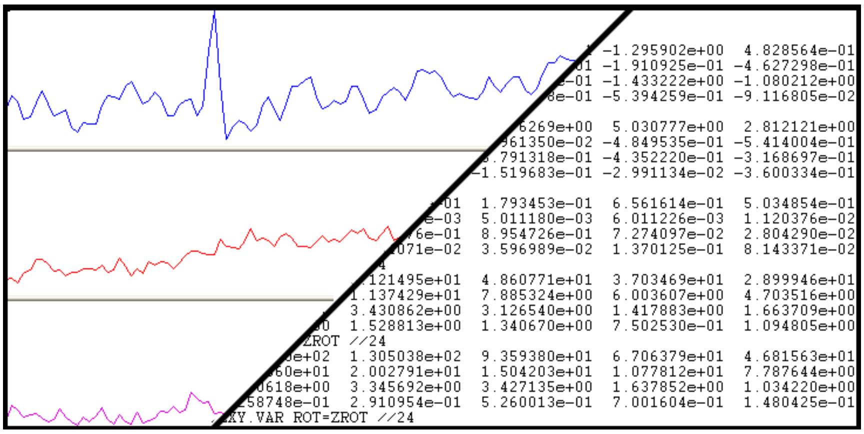
Program features which are identical throughout WinGLink, e.g. printing, gridding, color ranges, are discussed in the “Common Functions” chapter of this manual.

WinGLink: An Overview

This section gives a general overview of the main processing steps you will go through when using the WinGLink program. At the same time, the basic concepts and elements of the application will be introduced.

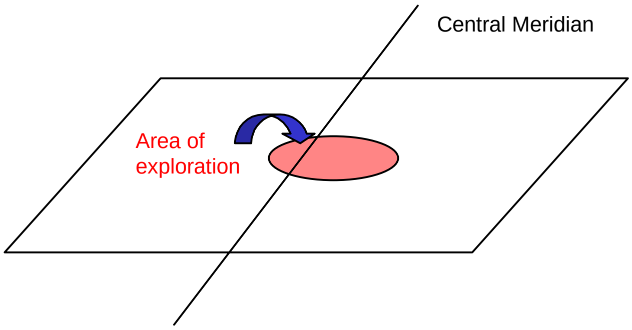
WinGLink tools are used to perform quality control, editing and data processing of the raw data in order to produce data which are suitable for further analysis. MT time series data, for example, can be examined for quality in the Time Series program, processed using WinGLink’s Cascade Decimation tool, and the results edited with the CrossPower and Data Analysis tools

WinGLink tools are used for preliminary data analysis, quality control and data processing.



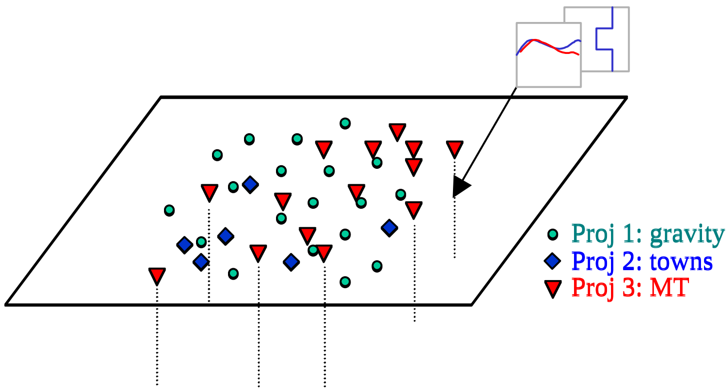
A **WinGLink database** contains the data for all surveys carried out in the area of interest. Information on the central meridian, the projection used for the station coordinates, and the linear units used for distances and depths is stored in the database properties.

The database area can be as wide as desired



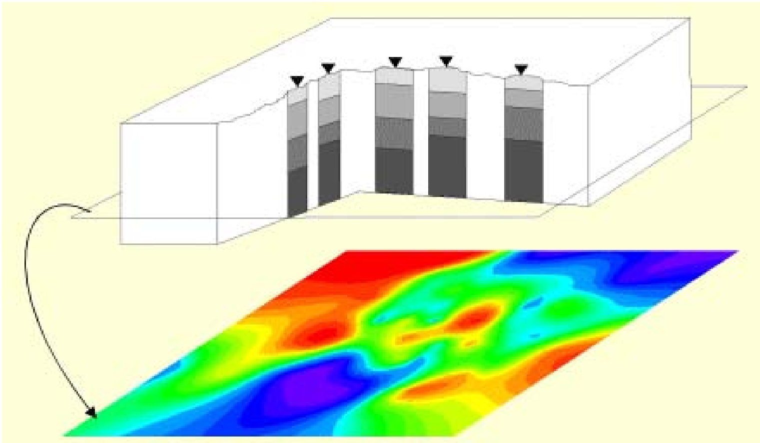
WinGLink projects consist of surveys carried out in the database area. Stations are added to projects by entering the given station's name, coordinates and elevation. This may be performed either manually or by importing station data from external files.

Parameter values, datasets and 1D models are attached to each station for the construction of maps and sections.



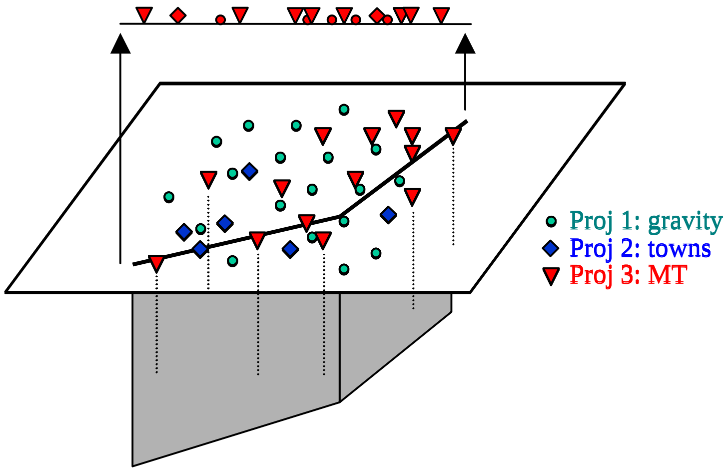
Maps are produced by gridding station values.

Parameter values can be automatically extracted from the station datasets or models

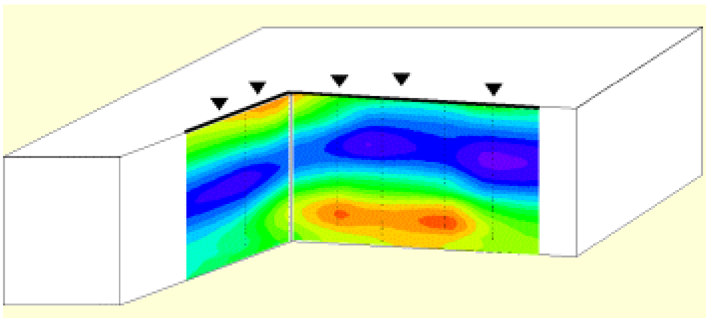


Profile traces are defined as open polygonal lines added to the area. Stations are added to the traces to construct sections.

Profile traces are used to construct sections. The same trace can be used for all types of surveys.



Sections are plotted using station datasets (pseudosections), 1D models, 2D models (cross sections) or extracted from 3D models.



2: Getting Started with WinGLink

Minimum Hardware and Software Requirements

To install and use WinGLink, your system must satisfy the following requirements:

Item	Minimum requirement
Machine type	IBM PC or compatible
CPU type	Pentium processor
RAM	512 MB
Color board	SVGA with at least 256 KB memory
Fixed disk space	100 MB for installation, and an average of 20 MB for each database
Non-fixed disk drive	CD-ROM drive
Operating system	Recommended: Windows XP; Windows 98/2000/ NT 4.0 with latest service pack also supported.
Internet connection	A high-speed Internet connection is needed to download new releases of the software as they are made available.

Note: The above hardware does not represent the best configuration for running WinGLink. Adding more RAM or using a higher speed processor will substantially improve WinGLink performance. Furthermore, a high-speed Internet connection is recommended for downloading new releases as they are made available.

Contents of the Shipping Package

To make sure that your package is complete, check its contents against the list below:

Item	Description
Installation CD-ROM	The WinGLink application and its setup program.
WinGLink dongle (one for each license)	Each license of WinGLink comes with a hardware copy protection dongle. The dongle is needed to run WinGLink with full performance.
User's guide	This manual with instructions on how to use WinGLink programs and its applications.

What's on the CD

The CD-ROM provided with each license contains

Directory	Contents
Winglink\Setup	Installation files and setup program for WinGLink
Winglink\Sentinel	Installation program for the WinGLink dongle driver
Winglink\Manual\Pdfdocs	This manual in Acrobat format
Winglink\Manual\Acroread	Acrobat Reader installation program

:

Installing WinGLink

This section contains instructions for installing and uninstalling WinGLink on a stand-alone computer system. This is the default installation mode for WinGLink programs. Should your machine be connected to a networked environment, we suggest that you copy all of the installation files to a directory on the network server. The same installation procedure can then be followed from each connected peripheral station, specifying the server directory as the source of installation programs.

Installing WinGLink

To begin, start your Windows operating system, then:

1. Close any application still running.
2. Use the Windows Explorer to select the CD-ROM directory, then browse to the WinGLink\Setup folder, which contains the installation files.
3. Double-click the Setup.exe file.
4. Follow the instructions on the screen. Should any messages appear during this process, click the **OK** button to acknowledge.

Uninstalling WinGLink

To remove WinGLink from your computer.

1. Click the 

button on the task bar, point to Settings, and then select **Control Panel**.
2. Double-click the **Add/Remove Programs** button.
3. Select **WinGLink** from the listed programs.
4. Click the **Remove** button**.**

Note: The uninstall program will remove **only** those files placed on your hard disk by the installation program. It will **not** remove the databases and data files you have created using WinGLink. These are stored as permanent files on your computer and can only be removed manually.

Cleaning Up After an Old Version of WinGLink

The WinGLink setup routine automatically removes earlier versions prior to installing a new release.

If, for some reason, you must install a WinGLink release (preceding 1.61.01), you must uninstall WinGLink to remove earlier versions before installing. You can check the directory ***Program Files\WinGLink*** to make sure that all files have been deleted from this directory. If not, delete all files manually and then install the new release of the program.

Note: Should you encounter problems following a new installation, contact us at support@winglinksoftware.com

Installing the WinGLink Dongle

Each WinGLink license comes with a hardware copy-protection dongle which must be plugged into either a USB port or the parallel (printer) port of your PC. When installing a parallel port dongle, connect the dongle between the PC and the printer cable.

Note: Licenses purchased as of mid-December 2002 may be supplied with a USB dongle. When installing the USB dongle, proceed following the same instructions provided for the parallel port dongle, connecting the dongle to the USB port instead of the parallel port.

The USB dongle is NOT supported for the **Window NT** operating system. If you intend to run WinGLink under Windows NT and you have been provided with a USB dongle, please contact Geosystem for a replacement parallel port dongle.

The USB dongle is supported for the Windows 98, ME, 2000 and XP operating systems.

W/hen installing a USB dongle driver, you must remove the dongle prior to running the driver installation program!

To Install the Parallel Port Dongle:

1. Turn off your computer.
2. If a printer or another device is connected to the parallel port, turn it off and disconnect it from the computer port.
3. If installing a parallel port dongle, connect the dongle. Do not connect the USB dongle prior to running the driver installation program.
4. Reconnect the printer cable to the PC by plugging it into the free end of the dongle.
5. If installing a parallel port dongle, turn on the printer, otherwise the dongle may not be detected.
6. Turn on your computer to reboot the system.

To Install the USB Dongle:

1. Install the dongle driver as described below
2. Plug the USB dongle into a free USB port.

Installing the Dongle Driver

Open the **\Winglink\Sentinel** folder on the installation CD ROM and double-click the file <Sentinel.exe>. This will install the dongle driver and enable your PC to recognize the dongle.

Note: The dongle does not affect any of the standard operations of the system; any device connected to the parallel/USB port via the dongle will work as it did before inserting the dongle.

Downloading and Using the WinGLink Downloader

WinGLink is under continuous development. New releases, as they are made available, are placed on our Web server for downloading. For this purpose, Geosystem provides a downloader utility, the WinGLink Downloader, which automatically establishes a connection with our server, downloads files selected by you to your computer, then gives you the option of installing any downloaded software.

Downloading the WinGLink Downloader:

The WinGLink Downloader is available for downloading from the Geosystem website at:

http://www.geosystem.net -> WinGLink Software -> How to get it

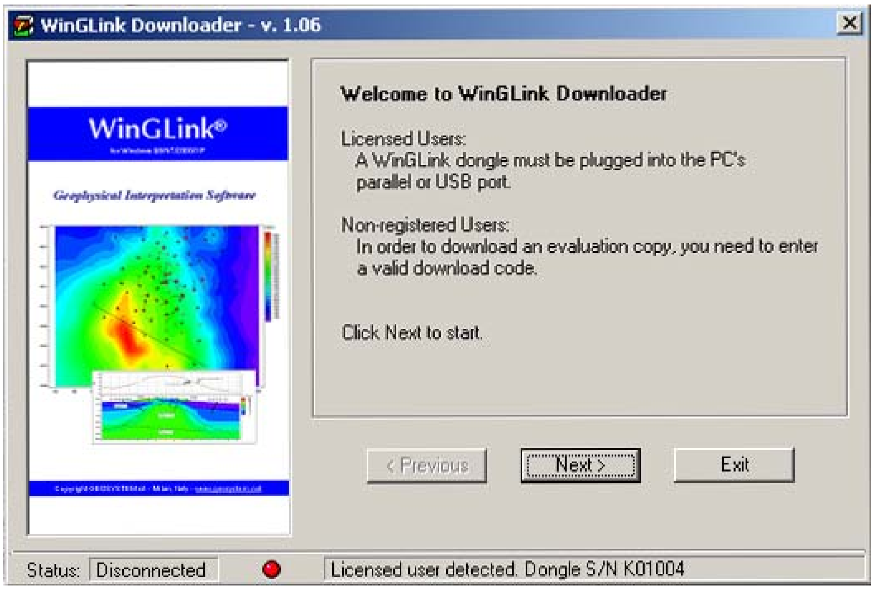
To download Downloader installer to your computer, right-click the “Download the ‘WinGLink installation/update utility’”, and specify the download destination.

After downloaded to your computer, execute the installer to initiate the installation process.

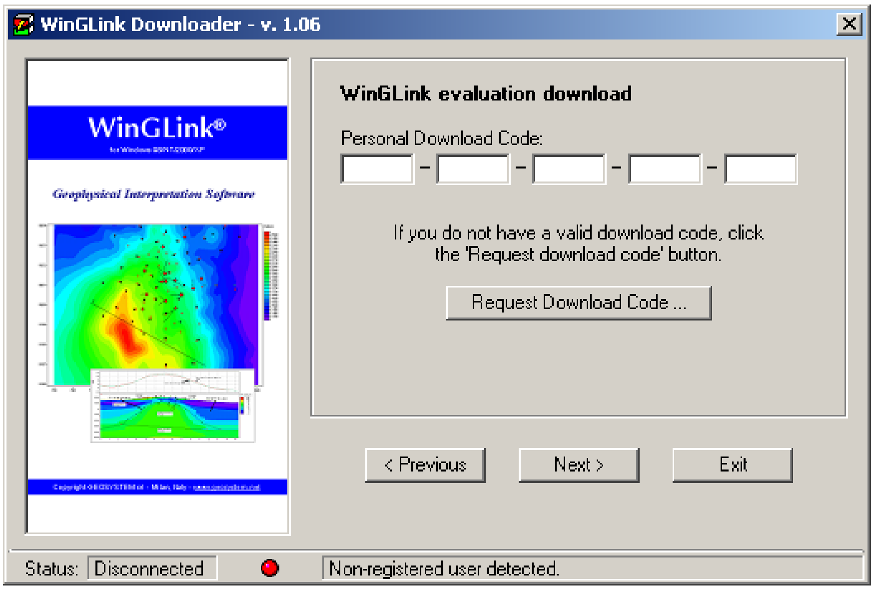
Using the WinGLink Downloader

If you are a licensed user, make certain that a dongle is plugged into the computer's parallel or USB port before using the WinGLink Downloader to download and install the current release. If you are a non-licensed user, you will need to enter a valid download code, which can be requested from Geosystem at winglink_support@geosystem.net.

Start the program from the WinGLink entry in the Windows Start menu:

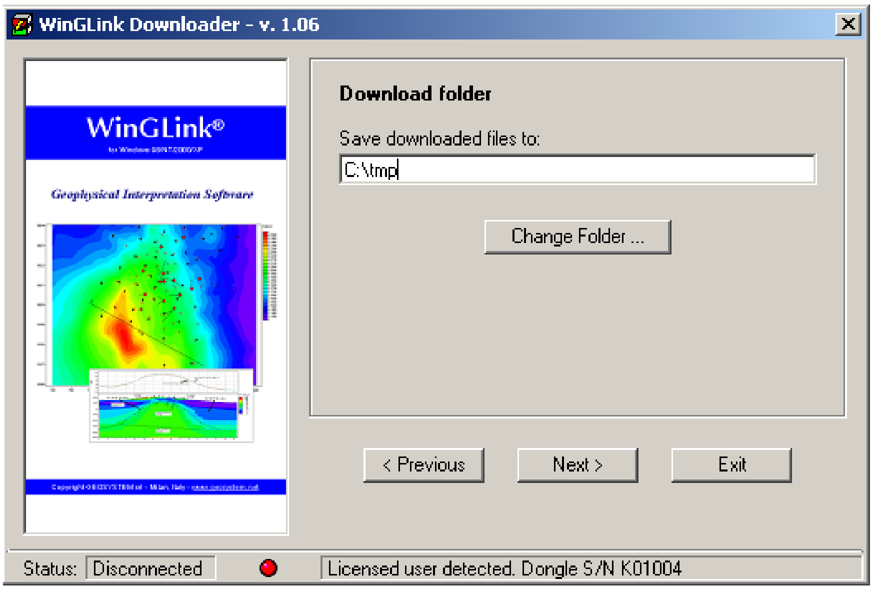


If no dongle is connected or detected, a screen which prompts the entry of a download code is displayed:

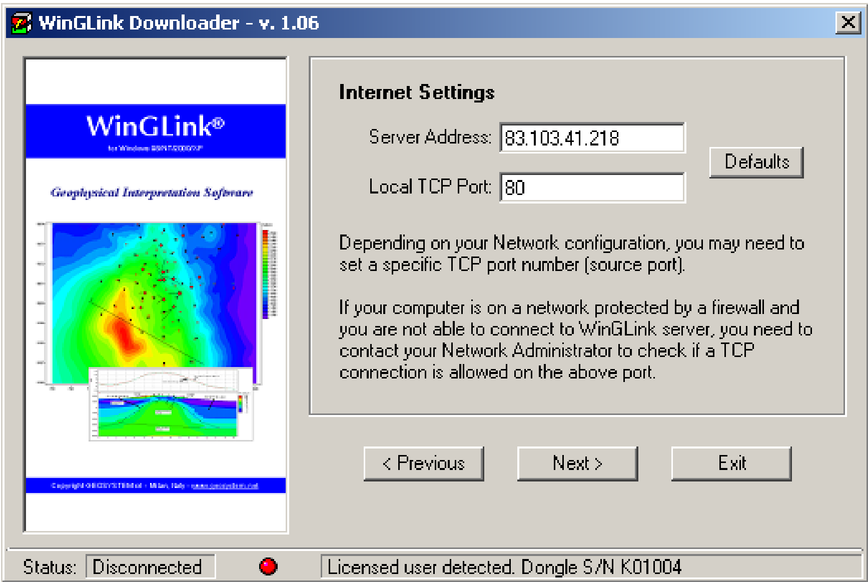


If you are not a licensed user and have requested a download code in order to evaluate the WinGLink software, enter the supplied code in the provided fields.

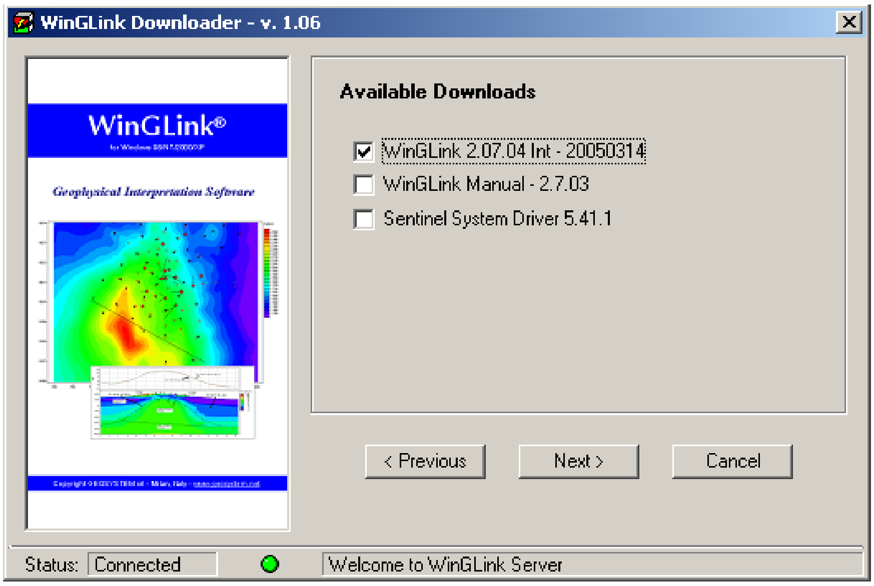
Otherwise, the program advances to the Download Location screen. Here, specify the folder to which the files are to be downloaded:



On the next screen, you are given the option of adjusting Internet settings. Unless given specific instructions, do not change the default server. If your network is protected by a firewall and you are not able to connect to the WinGLink server, you will need to contact your network administrator to determine whether the default port allows for a TCP connection. If not, the **Local TCP Port** value must be changed to a port which permits this type of communication:

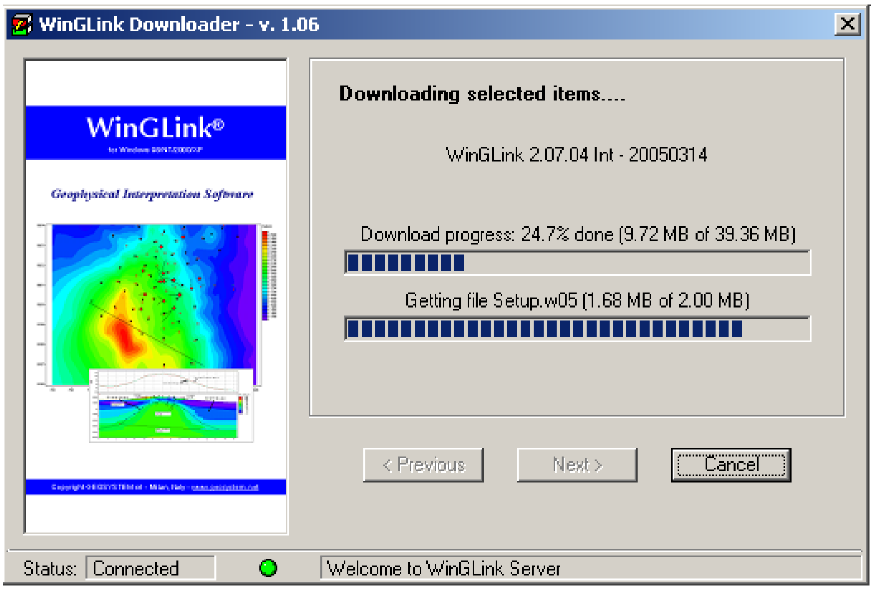


Click the **Next** button to establish a connection with the server and display a list of available downloads:

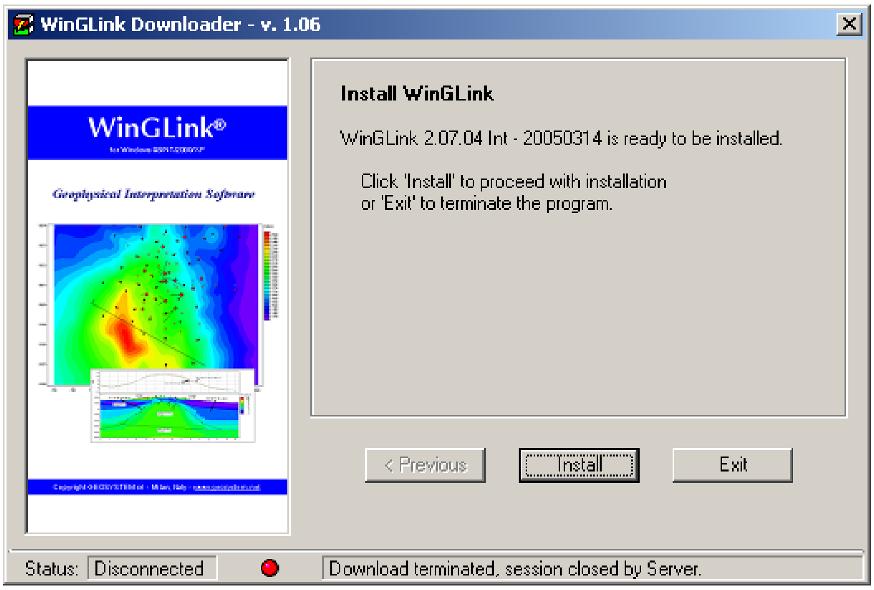


Note that new releases of the WinGLink Downloader are made available for downloading here as well.

Select the files to be downloaded and click the **Next** button to begin the download process:



After downloading has completed, click the **Next** button to advance to the next screen. Here, you can opt to either automatically install the downloaded software or exit the Downloader. Clicking the **Install** button starts the WinGLink installation process. If the installer detects a WinGLink installation on the computer, the old version is uninstalled before the new release is installed:



Installation Troubleshooting

Common Warning Messages

Warning messages may pop-up during the installation depending on the system settings of your PC. Here are the most common messages and the correct actions to undertake:

Message	Action
"Setup cannot continue because some system files are out of date on your system. Click OK if you would like to update these files for you now. You will need to restart Windows before you can run setup again. Click Cancel to exit setup without updating system files"	Click 'OK' to proceed with the installation of WinGLink. This also means that the system will have to be rebooted. A new message will be displayed:
"Do you want to restart Windows now ? If you choose 'No', you will not be able to run setup again until after the system is rebooted at a later time"	Click 'YES'; the setup process will be terminated and the system will be automatically rebooted. You will need to restart the setup procedure
"A file being copied is older than the file currently on your system. It is recommended that you keep your existing file"	Click 'YES' to proceed
When starting WinGLink, a dialog box displaying "You are running WinGLink in Demo mode" is displayed.	Make certain that the WinGLink dongle is correctly connected and that the dongle driver has been installed. If the dongle is connected and the driver has been installed, your computer has not detected the dongle. It is possible that the parallel port on your computer has not been enabled (a problem frequently reported by owners of Toshiba laptops). To correct, please refer to section "What to do if program will only run in demo mode" of this chapter.

Optimizing the Graphic System Settings

These operations are needed to optimize your system for the use of WinGLink.

1. Click the 

 button, point to Settings, and then click Control Panel.
2. Double-click the Display button; select the **Settings** tab.
3. In Color Palette, select **High Color (16-bit)** or higher.
4. In **Font Size**, select **Small fonts**.

What To Do If Program Will Only Run in Demo Mode

If a message is displayed when launching WinGLink indicating that the program is being run in Demo mode, despite having installed the dongle driver and having correctly connected the dongle, it is quite likely that your computer has not detected the dongle. A common cause of this problem is that the parallel port has not been enabled (when using a parallel port dongle). This may be the case even if the printer functions!

To Check Whether Your Parallel Port Dongle Is Enabled:

1. Launch C:\Program Files\Rainbow Technologies\Sentinel System Driver\SetupSysDriver.exe
2. Click "Configure Driver" and make sure you see a physical address and port type for the parallel port. If you do not see one, the parallel port is not enabled! Follow the instructions below to enable the parallel port. Note that Windows XP will still allow the printer to work even though the port is not enabled.
3. Uninstall the Sentinel System Driver using the Add/Remove Programs wizard in the Windows Control Panel.
4. Reboot your computer and hold down the "ESC" key while tapping the "F1" button to enter BIOS.
5. Find the parallel port mode setting and enable it.
6. Save changes and exit BIOS.
7. Reinstall the Sentinel System Driver.

If, after enabling the parallel port, WinGLink continues to run in demo mode only, please contact WinGLink technical support at support@winglinksoftware.com.

What To Do If Using a USB Dongle

If using a USB dongle, you may need to uninstall the dongle driver if the dongle was connected when the driver was installed. In this case, remove the dongle, uninstall the driver, then reinstall the driver.

Starting WinGLink

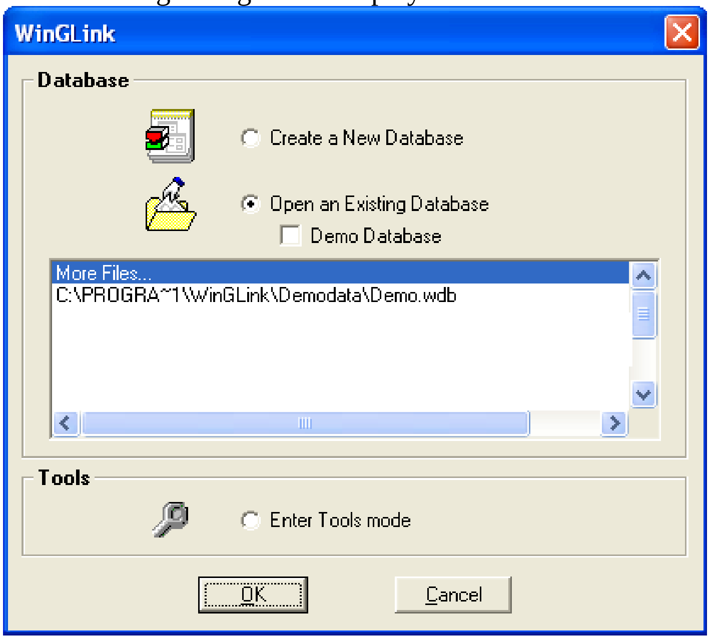
After successfully installing WinGLink on your PC, installing the dongle driver and plug-in in the dongle, you are ready to start the program.

To Start WinGLink:

1. Click the 

button on the Windows Task Bar.
2. Select **Programs**, and then click the **WinGLink** entry.

3. The following dialog box is displayed:



Select the options listed in the **Database** frame if you would like to use WinGLink in Database Mode, or click "**Enter Tools Mode**" to run WinGLink in Tools Mode. See below for information on WinGLink's two operating modes.

Using WinGLink in Tools Mode

The field data collected during a survey often require preliminary treatment and must undergo a certain amount of quality control in order to produce processed data which are suitable for further analysis. This is the case, for instance, of meter readings for gravity surveys or time series for MT soundings.

WinGLink is equipped with a suite of programs (Tools) which handles data reduction and quality control (QC). These tools produce output files that can be imported into a WinGLink database for further processing and interpretation.

These include programs for

MT tools	Function
MT field processing	Time series display (MT) Cascade decimation processing Cross-power editing Data analysis
EDI files conversion/manipulation	Conversion to EDI format of Stratagem/Imagem z*.* files Splitting multi-site EDI files Combining EDI files

:

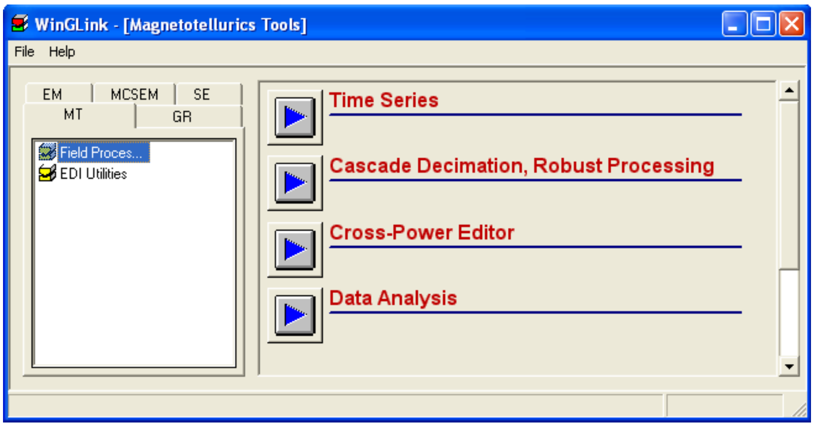
GR tools	Function
Gravity reduction	Data reduction for Scintrex and Lacoste-Romberg g-meters
Tide corrections	Plots of tide corrections and output of values at user-assigned time intervals

EM tools	Function
TemMerge	Import, edit and merge raw data from field dump files (Geonics, Zonge, Sirotem and text files).

To Use WinGLink in Tools Mode:

- Select the **Enter Tools Mode** option in the starting dialog box or
- When in Database Mode, select the **File | Tools menu** command on the main menu.

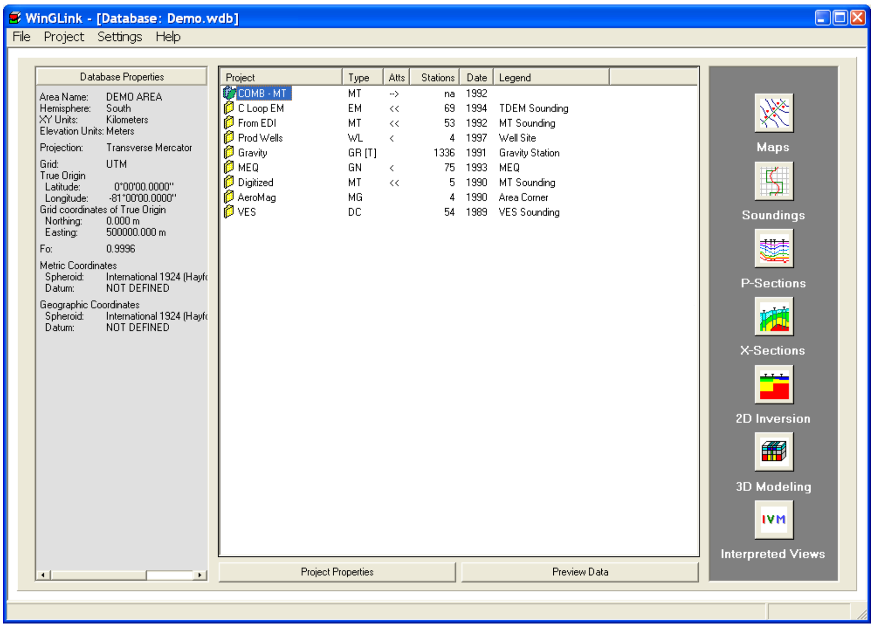
WinGLink then opens in a view from which all tools supplied with your license can be selected:



Detailed instructions on the use of the individual Tools programs can be found in the respective chapters of this manual.

Using WinGLink in Database Mode

The bulk of WinGLink's power is accessed while in Database Mode. Here, databases and projects can be created and managed, data imported and the individual program modules opened:



To Use WinGLink in Database Mode

To use the program in Database Mode, select one of the options in the **Database** frame of the initial dialog box, i.e. select either an existing databases, the demo database or create a new database**.** Detailed information on working with WinGLink in Database Mode is provided in the “WinGLink Shell” chapter of this manual.

The User’s Guide

This user’s guide is provided in printed form and as a PDF (Portable Documents Format) file which can be viewed and printed using Acrobat Reader version 3.0 or later. This is supplied with the installation CD-ROM.

Where To Find the Manual

The Acrobat PDF file with the WinGLink user’s guide can be found in the **\Winglink\Manual\Pdfdocs** directory on the installation CD-ROM. Using Acrobat Reader, you can load the file from this directory or you can copy it to a directory on a hard disk and load it from there for faster access.

How To Read and Print the Manual

Use Acrobat Reader 3.0 or higher to open the **manual.pdf** file. Refer to Acrobat’s online help to learn how to view and print the manual.

Acrobat Reader Installation

To install Acrobat Reader 3.0 on your Windows 95/98 or NT4.0/2000/XP computer:

1. Move to the \Winglink\Manual\Acroread directory on the WinGLink installation CD-ROM.

2. Double-click the Setup.exe program and follow the instructions.

3: Workflow

Overview

Presented on the following pages are typical workflows for each of the data types supported by WinGLink. There are provided, in particular, for users new to WinGLink. Each of the typical work steps includes a reference to the corresponding WinGLink module and the chapter in this manual where detailed instructions on the given module can be found.

The information is presented in tabular form, with the flowchart information in the left-hand column, the program module in the center and the manual reference at the right.

For most data types, the workflow is divided into three main steps:

- Quality Control
- Interpretation
- Presentation

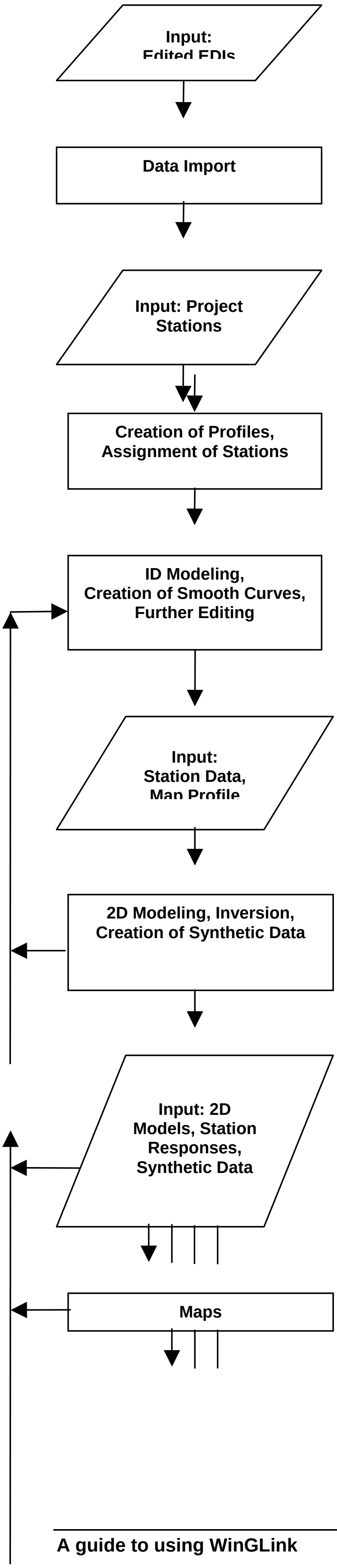
Within each of these steps, two different flowchart elements are used:

- Parallelogram: Used to indicate the type of data used in the next operation, e.g. EDI file or well course.
- Rectangle: A description of the given operation, e.g. data import or MT 2D inversion.

:

MT DATA

Operation	Module	Chapter
Input: Time Series		
Quality Control	Time Series Tool	Chapter 4
Input: Edited Time Series		
Data Processing	Cascade Decimation	Chapter 5
Input: X-Powers, EDI		
Data Editing	Cross Power Editor Data Analysis Tool	Chapter 6 Chapter 7
Input: Edited EDI		
Interpretation		
Database Creation	WinGLink Shell	Chapter 11

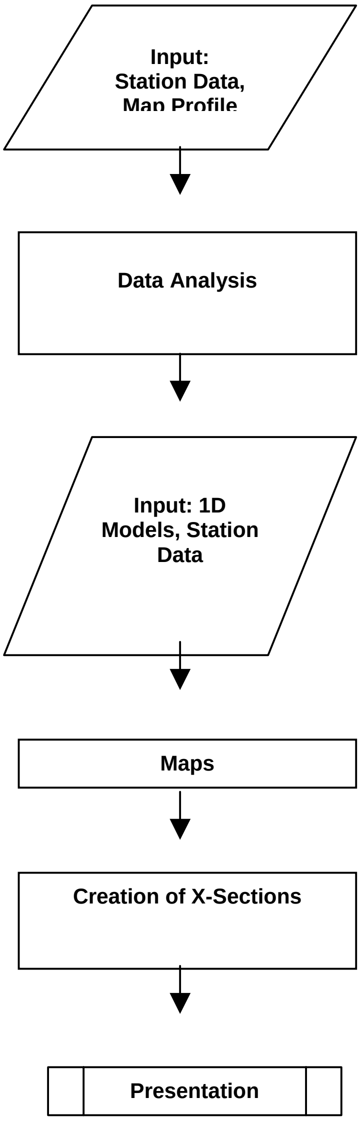


WinGLink Shell	Chapter 11
Maps	Chapter 14
MT Soundings	Chapter 16
MT 2D Inversion	Chapter 18
Maps	Chapter 14

Creation of Pseudosections	MT Pseudo-Sections	Chapter 17
Creation of X-Sections	X-Sections	Chapter 23
3D Modeling, 3D Forward Calculation	MT 3D Modeling	Chapter 19
Presentation	Interpreted Views and Montage	Chapter 25

TDEM DATA

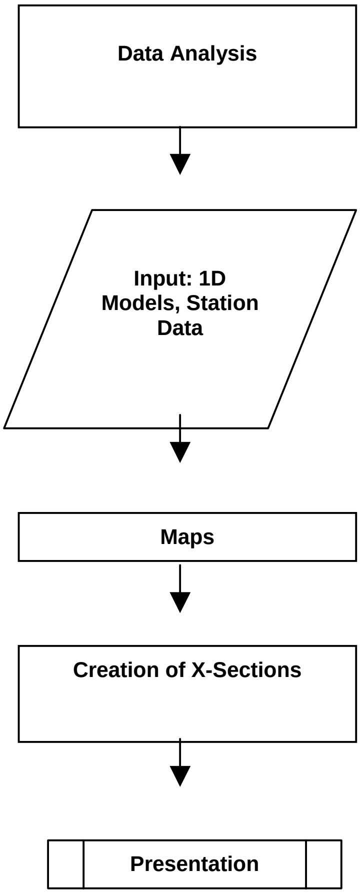
Operation	Module	Chapter
Raw System Files		
Quality Control, Editing, Merging	TemMerge	Chapter 9
Interpretation		
Database Creation	WinGLink Shell	Chapter 11
Input: TemMerge files		
Data Import	WinGLink Shell	Chapter 11
Input: Project Stations		
Creation of Profiles, Assignment of Stations	Maps	Chapter 14
Further Editing, ID Modeling	EM Soundings	Chapter 20



EM Pseudo-Sections	Chapter 22
Maps	Chapter 14
X-Sections	Chapter 23
Interpreted Views and Montage	Chapter 25

DC DATA

Operation	Module	Chapter
Input: Schlumberger Soundings		
Interpretation		
Database Creation	WinGLink Shell	Chapter 11
Input: TemMerge files		
Data Import	WinGLink Shell	Chapter 11
Input: Project Stations		
Creation of Profiles, Assignment of Stations	Maps	Chapter 14
Further Editing, ID Modeling	DC Soundings	Chapter 21
Input: Station Data, Map Profile		



DC Pseudo-Sections	Chapter 22
Maps	Chapter 14
X-Sections	Chapter 23
Interpreted Views and Montage	Chapter 25

GRAVITY | MAGNETIC DATA

Operation	Module	Chapter
Input: Gravity Values		
Gravity Reduction	Gravity Reduction Tool	Chapter 10
Interpretation		
Database Creation	WinGLink Shell	Chapter 11
Input: Station Coords, Grav/Mag. Data		
Data Import	WinGLink Shell	Chapter 11
Input: Project Stations		
Creation of Profiles, Assignment of Stations	Maps	Chapter 14
Input: Station Data, Map Profile		

Modeling	2.5D Modeling	Chapter 24
Presentation	Interpreted Views and Montage	Chapter 25

WELLS

Operation	Module	Chapter
Database Creation	WinGLink Shell	Chapter 11
Input: Well Coords, Well Traces		
Data Import	WinGLink Shell	Chapter 11
Input: Wells Courses		
Creation of Profiles, Assignment of Wells	Maps	Chapter 14
Input: Well Courses		
Edit Well Courses, Import Layer Data	Wells Editor	Chapter 15
Presentation	Interpreted Views and Montage	Chapter 25

4: Time Series^{MT}

Overview

WinGLink’s Time Series tool is generally used as an initial quality control step during an MT survey in order to analyze the acquired time series. This tool provides a variety of functions which are useful for evaluating the data quality in both time and frequency domains as well as a number of powerful tools for handling time series such as filtering, spectrographs, time shifting, extraction and others.

Supported Formats

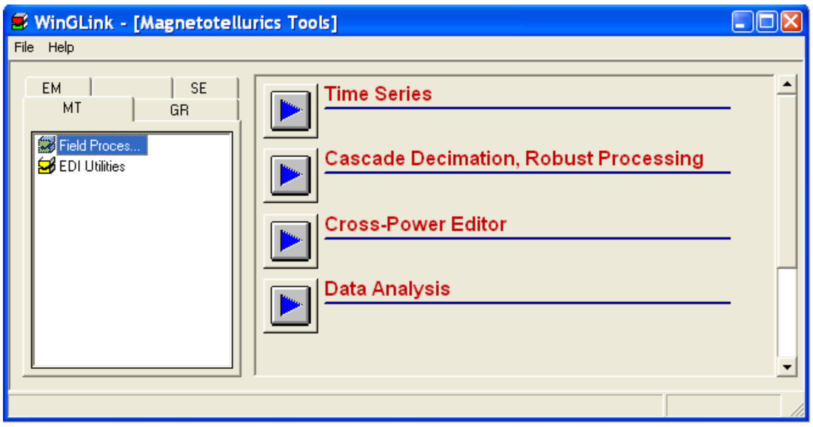
The Time Series tool provides support for most of the leading acquisition systems including:

- EMI MT-1
- EMI MT-24
- EMI MMT-24
- Metronix ADU-06
- Phoenix V5
- Phoenix V5-2000
- Phoenix V8

Getting Started with the Time Series Tool

Opening the Tool

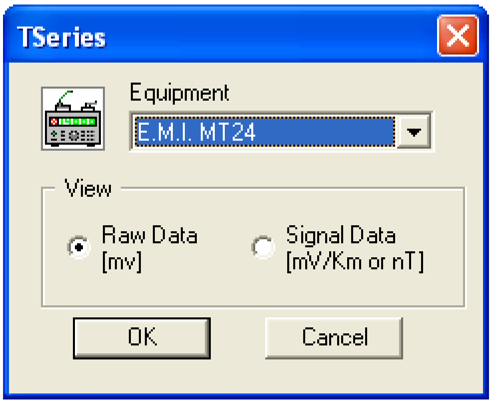
After opening WinGLink in Tool Mode, select **MT** in the left panel of the tool form, select **Field Processing** in the left frame, and click the arrow to the left of Time Series.



Note: For information on starting WinGLink in Tool Mode, refer to the "Getting Started With WinGLink" chapter of this manual.

Equipment Selection

The Time Series tool opens with a dialog box prompting you to select a type of equipment, e.g. E.M.I. MT24. Make the appropriate selection and click **OK**.

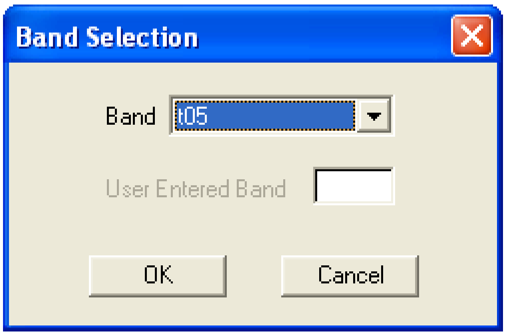


This window is opened automatically when the Time Series tool is started and can also be opened by selecting the **File | New**… menu command.

Note: It is possible to view time series either as raw data, where only the AD conversion factor (mV/digitalized-unit) is applied, or as signal data.

Band Selection

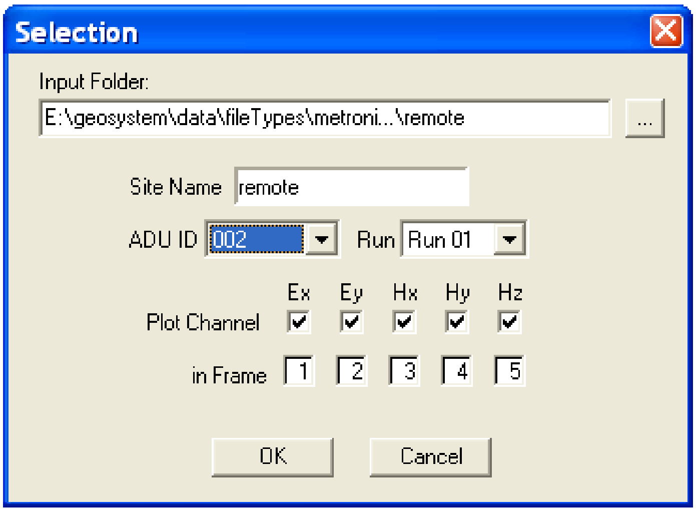
After selecting the data type, the program prompts you to select the band extension of the acquired time series, i.e. *t05* for 50Hz sampling rate. It is also possible to select decimated time series by entering the appropriate band extension.



This window can also be opened with the **Settings | Band Selection…** menu command

Time Series Selection

In the Time Series Selection form, use the left combo boxes to select the drive and folder containing the time series files to be opened. In the middle window, select the check box associated with a specific file. After selecting, a brief description (receiver ID, channel type, sampling frequency) of the file's contents appears in the right window. Channels for which the check box is selected in the panel at the right are displayed after the **OK** button is pressed.

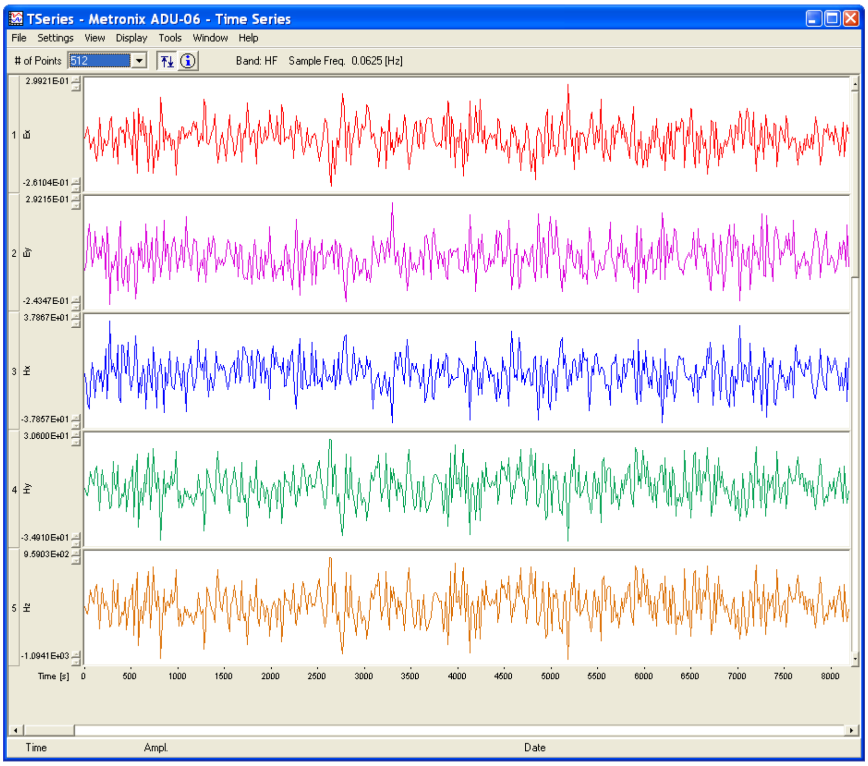


This window can also be opened with the **Settings | Plot Selection…** menu command

Displaying Time Series

At this point, the selected time series are displayed in the main window. One time series is plotted in each frame. The plotted series reflect the number of samples specified by the **# of Points** drop-down list in the upper left part of the main window. By changing the value in this menu, it is possible to change the number of plotted samples from a minimum of 64 to a maximum of 131,072.

Click the arrow buttons located in the scroll bar  at the bottom of the main window to scroll through the time series, i.e. move forwards and backwards along the time axis.



The status bar, located at the bottom of the window, is used to display information corresponding to the current cursor location:

Time: Number of seconds from the acquisition start time

Amp: Amplitude value and specific unit of measure

Site: Station/receiver ID

Channel: Type(e.g. Ex, Ey,…)

Date: Absolute date and time (e.g. 01/May/2006 22:30:05), if available.

The information bar at the top of the window is used to display two general pieces of information regarding the time series: the sampling frequency and the band ID.

Changing the Channels to be Plotted

Use the mouse to click the frame number at the far left edge of the window:

The **Plot Selection** window opens. Enter the necessary information and click **OK** to apply the new selections.

Working with Frames

Band & Plot Selection

The Band & Plot Selection windows, familiar from Time Series startup, can be accessed directly with the **Settings | Band Selections…** and **Settings Plot Selections…** menu commands, respectively. These two functions can be used to add time series to the current display.

Clearing Frames

Use the **Settings | Clear Selections** menu command to clear all currently displayed time series from the main window. Prior to clearing the display, the Time Series tool displays a confirmation window which must be acknowledged before the operation is performed.

Adding or Removing Frames

By default, the program makes seven frames available. It is possible, however, to add additional frames to the display and compare many channels of different time series in the same plot. Likewise, frames can also be deleted from the display.

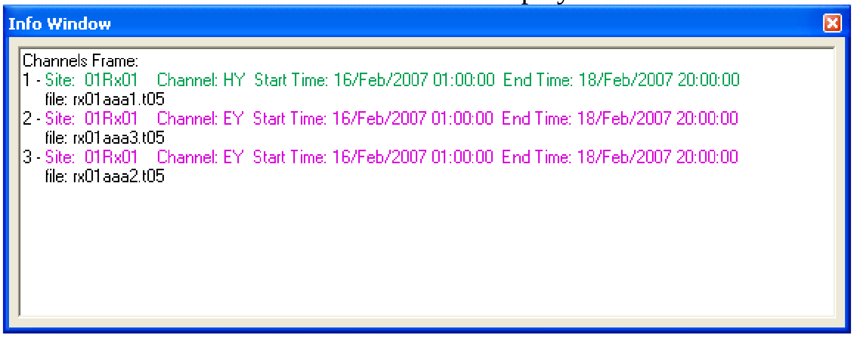
On the **Settings** menu, choose **Add Frame** or **Remove Frame**.

All channels must be of the same band and must be synchronized.

Note: Frames are added or deleted at the bottom of the plot.

Info Window

Click the  icon to open an info window in which the following information for each of the time series is displayed:



Site: Station/receiver ID

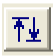
Channel: Data component

Start Time: Date and time of the first sample

End Time: Date and time of the last sample

File: Time-series file name

AutoScale Function

Click the Autoscale icon  to enable or disable the autoscale function, which automatically sets the range of the vertical scale of each frame to include all points from the minimum to the maximum value for the displayed samples.

Note: When auto-scale is off, the vertical scale of each frame can be adjusted manually by clicking the two buttons  at the top and bottom left of each frame. Use the up and down buttons to increase or decrease the value of the scale endpoints.

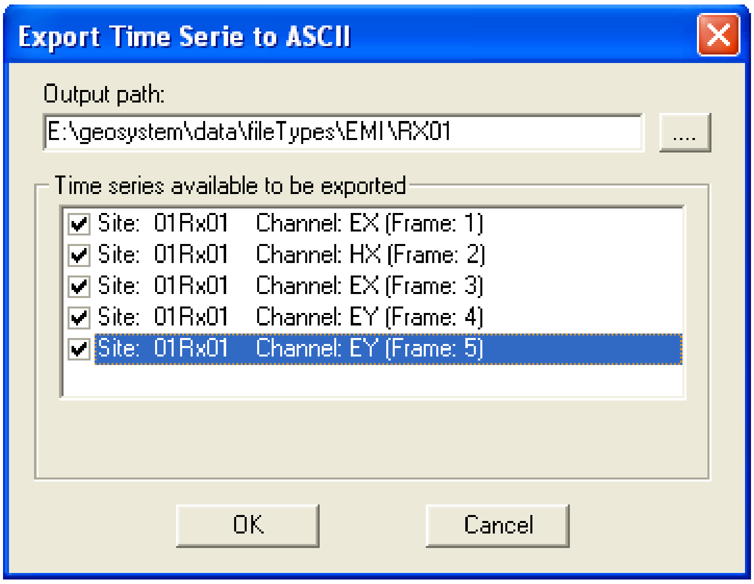
---

of Points

Use this drop-down list to select the number of samples currently displayed. The number of samples is also used in the spectra and correlation computations.

Exporting Time Series

Use the **File | Export** menu command to open the Export Time Series to ASCII window, which is used to export the selected files to external text files:



Printing Time Series

Select the **File | Print…** menu command to open WinGLink's familiar Print interface. Details on using this window can be found in the "Common Functions" chapter of this manual.

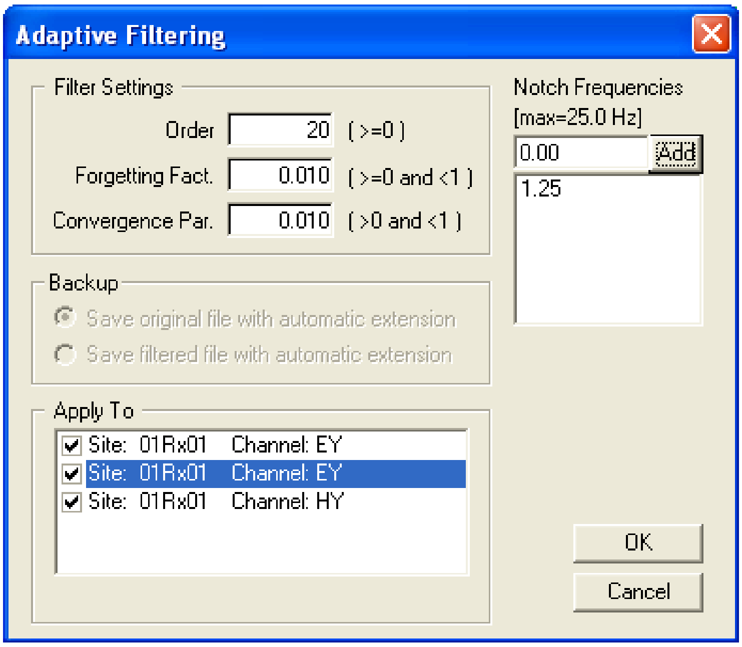
Time Series Tools

Filtering

As described below, the Time Series Tool provides a number of filtering functions, including adaptive filtering, wavelet filtering and notch filtering.

Adaptive Filtering

This tool, accessed with the **Tools | Adaptive Filtering…** menu command, applies an adaptive filter (normalized least mean square filter) to the selected signals in order to notch a subset of desired frequencies, adopting an approach similar to that of echo cancellation.



In the Adaptive Filtering dialog box, shown above, set the parameter values in the **Filters Settings** frame for:

Filter Order: The number of coefficient used for filtering (>0).

Forgetting Factor: Weighting factor used in signal power estimation between new and current estimate (0< x <1).

Convergence Parameter: The adaptation step, which controls the convergence time and adaptation quality
(0<= x <1).

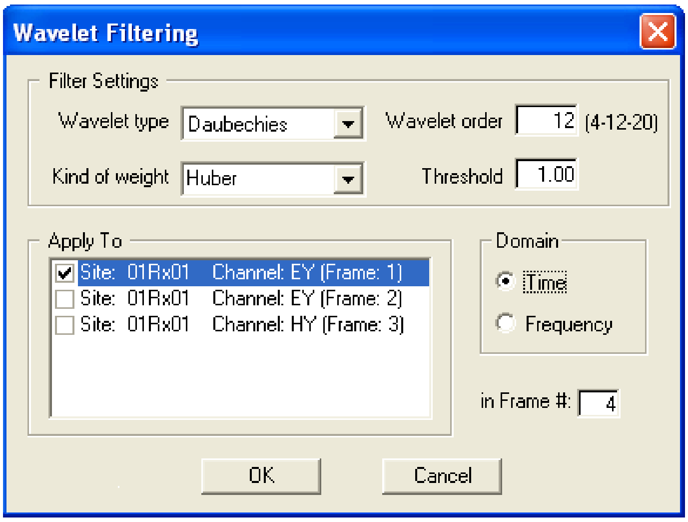
Enter the frequencies to be notched in the upper right frame and click the **Add** button.

In the **Apply To** frame, select the signals to be filtered by clicking the appropriate check boxes. To run the filter, click the **OK** button.

Note: A backup of each original file is created for each filtered channel. These backup files are stored in the same folder as the original. A popup message appears following the filtering operation indicating the new file name.

Wavelet Filtering

This tool, accessed with the **Tools | Wavelet Filtering…** menu command, provides a filter that operates in the wavelet domain leaving good data sections unchanged. Data in the wavelet domain are analyzed through scale levels, allowing separation of noise from signals. Noisy elements of impulsive nature, either in time or frequency domain, are filtered out.



In the **Filter Settings** frame, select the parameters to be used in the filtering:

Wavelet type: The wavelet prototype function (e.g. Daubechies)

Wavelet order: Number of coefficients used to specify the set of wavelets

Kind of weight: Weight function for anomalous data (i.e.Huber/Thomson)

Threshold: Threshold level for identifying anomalous data

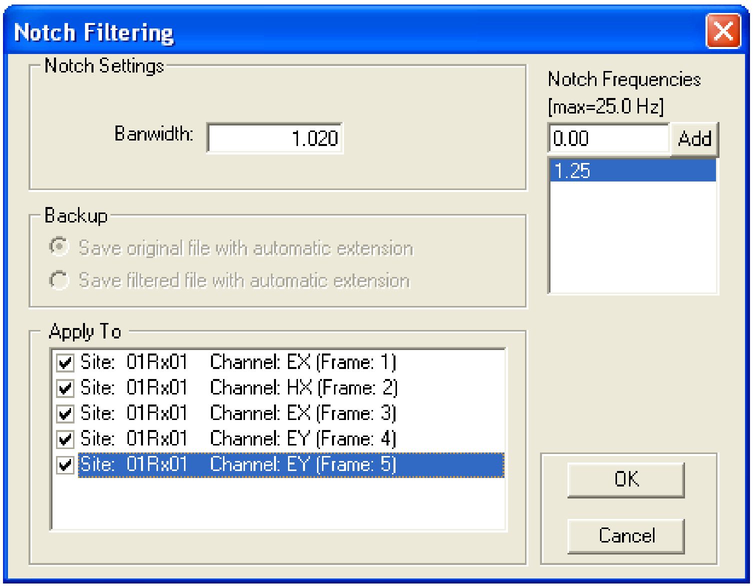
In the **Domain** frame, specify the domain for the wavelet filtering by clicking the corresponding checkbox.

In the **Apply To** frame in the lower left, select the signal to be filtered by clicking the corresponding check box. To run the filter, click the **OK** button. Only one channel can be filtered at a time.

Note: The filtered data are displayed in a new frame as a new time series. Select the **File | Save** menu command to save this time series into a new file. The suffix _sav is appended to the name of the new time series file, but before the file extension.

Notch Filtering

This tool, accessed with the **Tool | Notch Filtering…** menu command, provides a recursive (IIR) filter for notch filtering a subset of frequency components from the observed data. The filter is applied in the forward and reverse direction in order to avoid the application of a phase shift to the data.



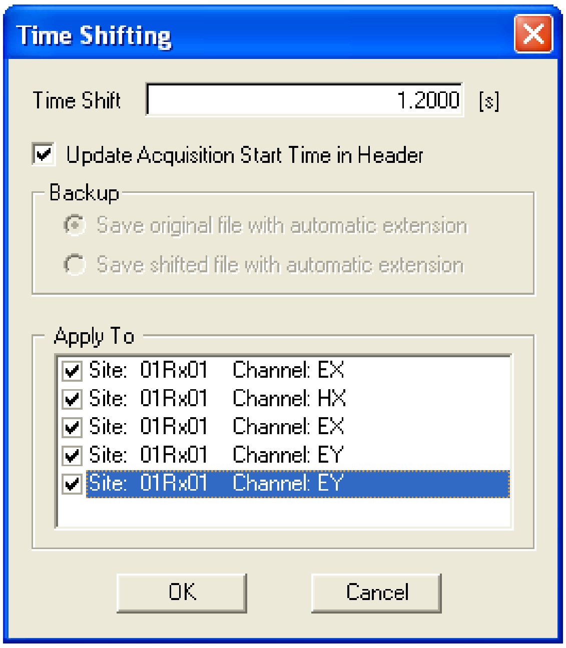
In the **Notch Settings** frame, enter the bandwidth value of the notch. In the right frame, enter a subset of frequency values to be notched. Enter the value and then click the **Add** button.

In the **Apply To** frame, select the signals to be filtered by clicking the appropriate check box. To run the filter, click the **OK** button.

Note: A backup of each original file is created for each filtered channel. These backup files are stored in the same folder as the original. A popup message appears following the filtering operation indicating the new file name.

Time Shifting

This tool is useful for synchronizing different signals. To open the Time Shifting window, select the **Tools | Time Shifting…** menu command:



In the **Time Shift** field, enter a value in seconds (decimal) for the shift which is to be applied.

Note: The shift value can also be negative; in this case, a random signal is added to the beginning of the time series. The random signal samples are calculated by adding to the first sample of the time series its own value times a pseudo-random number which is uniformly distributed between -0.001 and 0.001.

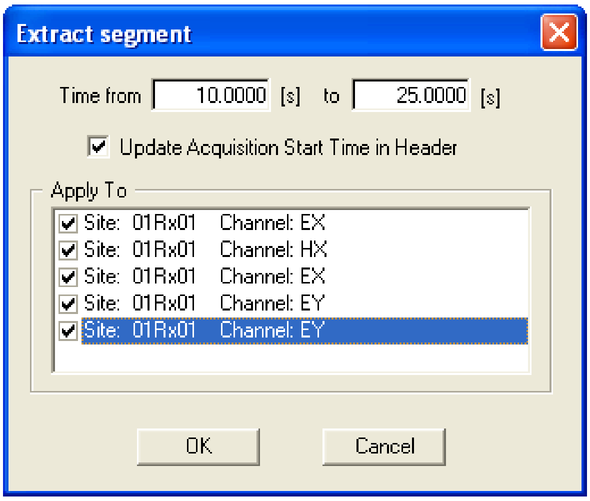
In the **Apply To** frame, select the signals to be shifted by clicking the appropriate check box. To shift the series, click the **OK** button.

Note: A backup of each original file is created for each shifted channel. These backup files are stored in the same folder as the original. A popup message appears following the shifting operation indicating the new file name.

Extracting Segments

Extracting Segments from Time Series

The **Extract Segments** function, accessed with the **Tools | Extract Segment…** menu command, is used to extract a segment from a time series using the number of seconds from the first sample as the limit for the extraction:



Enter the start and end times of in the respective fields at the top of the window.

To update the header with the new acquisition times, select the checkbox provided for this purpose.

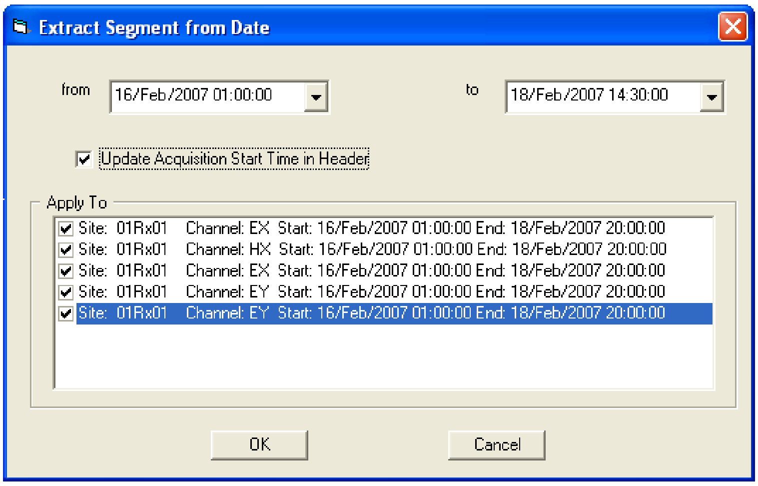
In the **Apply To** frame, select the signals from which time series are to be extracted by clicking the appropriate check boxes. Click the **OK** button to extract the time series.

Note: The extracted data are displayed in new frames as new time series. Select the **File | Save** menu command to save the new time series to a new file. The suffix _sav is appended to the name of the new time series file, but before the file extension.

Extracting Segment from Date

This tool differs from the previous one in the format of the limits used in the extraction. In this case, the absolute date and time are used as the start time and end time.

To open the tool, select the **Tools | Extract Segment from Date…** menu command:



Enter the start and end dates and times of in the respective fields at the top of the window.

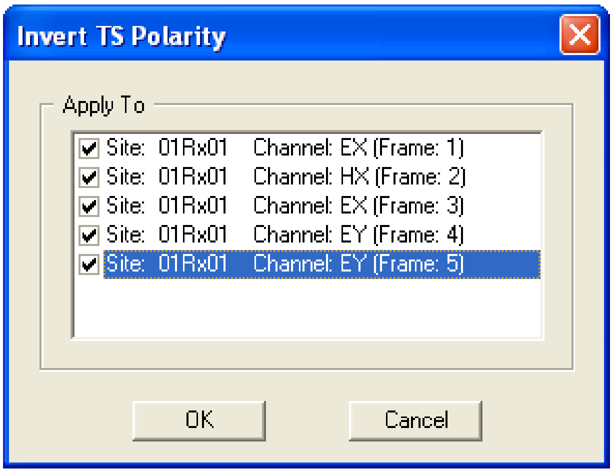
To update the header with the new acquisition times, select the checkbox provided for this purpose.

In the **Apply To** frame, select the signals from which time series are to be extracted by clicking the appropriate check boxes. Click the **OK** button to extract the time series.

Note: The extracted data are displayed in new frames as new time series. Select the **File | Save** menu command to save the new time series to a new file. The suffix _sav is appended to the name of the new time series file, but before the file extension.

Inverting Polarity

The **Invert Polarity** tool, accessed with the **Tools | Invert Polarity…** menu command, changes the sign of the selected data signals, i.e. the signal is rotated by 180 degrees:



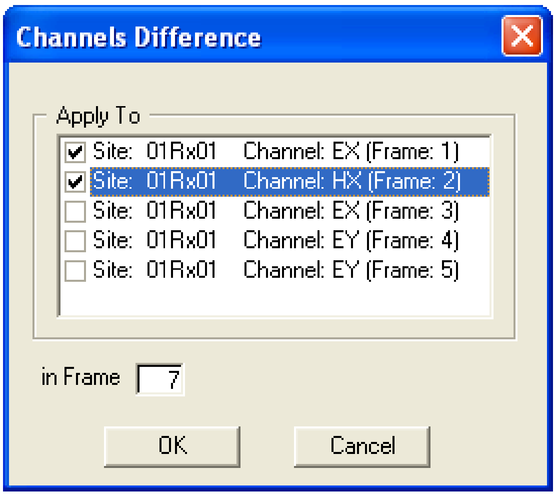
In the **Apply To** frame, select the signals to be inverted by selecting the appropriate check boxes. To invert the time series, click the **OK** button.

Note: A backup of each original file is created for each inverted channel. These backup files are stored in the same folder as the original. A popup message appears following the shifting operation indicating the new file name.

Computing Differences

Channel Difference

This tool, accessed with the **Tools | Channel Difference…** menu command, provides a useful function for data quality control for homogeneous signals. It is used to compute the difference between two channels. The result can be treated like a new time series.

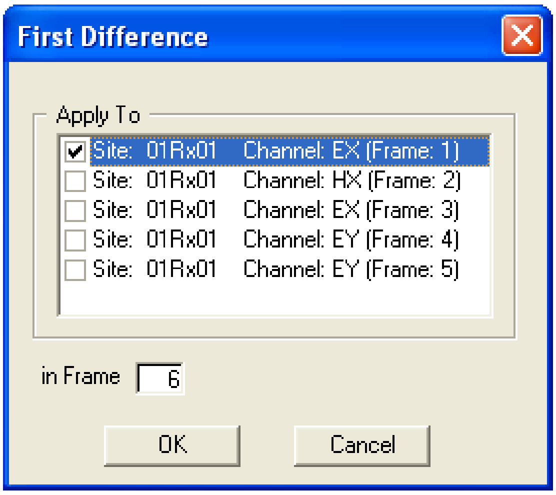


In the **Apply To** frame, select the *two* signals for which the difference is to be calculated by selecting the desired check boxes. To compute the difference between the two selected time series, click the **OK** button.

Note: The new time series generated with the tool is displayed in the first free frame and can be saved by selecting the **File | Save** menu command.

First Difference

This tool, accessed with the **Tools | First Difference…** menu command, computes the first difference of a channel, subtracting each sample from the previous one.



In the **Apply To** frame, select the signal to be processed by clicking the appropriate check boxes. To run the tool, click the **OK** button

Note: The new time series generated with the tool is displayed in the first free frame and can be saved by selecting the **File | Save** menu command.

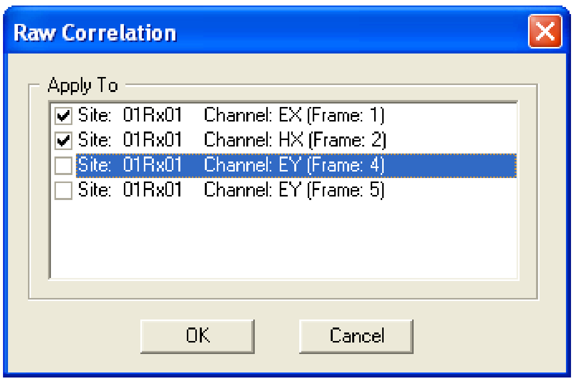
Correlation

Raw Correlation

The **Raw Correlation** tool provides a powerful function for evaluating the information shared by two channels (e.g. it can be used to test the synchronization of signals). The correlation function can be applied to a single channel (auto-correlation) or to a pair (cross-correlation).

If the maximum in the cross-correlation between two signals occurs at sample number 0, the two signals are perfectly synchronized; the shape of the cross-correlation is dependent on their spectral content (e.g. the auto-correlation of a white random signal is an impulse).

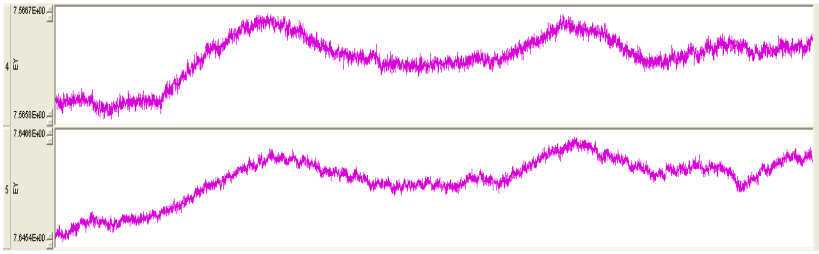
Access the Raw Correlation tool with the **Tools | Raw Correlation…** menu command:

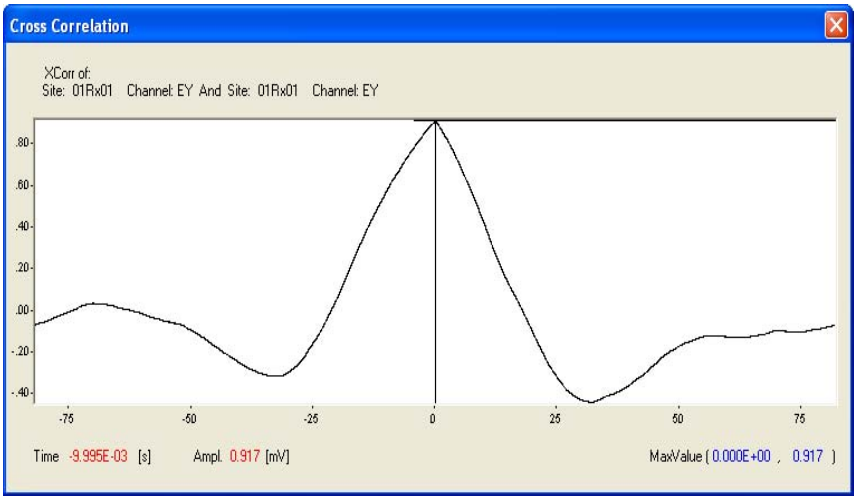


The **Raw Correlation** tool computes the auto- or cross-correlation of the selected signals using the number of samples displayed in the main window. In order to compute the raw correlation over a wider range of samples, change the value in the **# of Points** drop-down list in the upper left area of the main window.

In the **Apply To** frame, select one (auto-correlation) or two (cross-correlation) signals to be processed by selecting the appropriate check boxes. To perform the correlation calculation, click the **OK** button.

A new window containing the auto- or cross-correlation curve now opens. The value and position of the maximum are displayed in the lower right; the value and position at the current mouse position are displayed in the lower left.

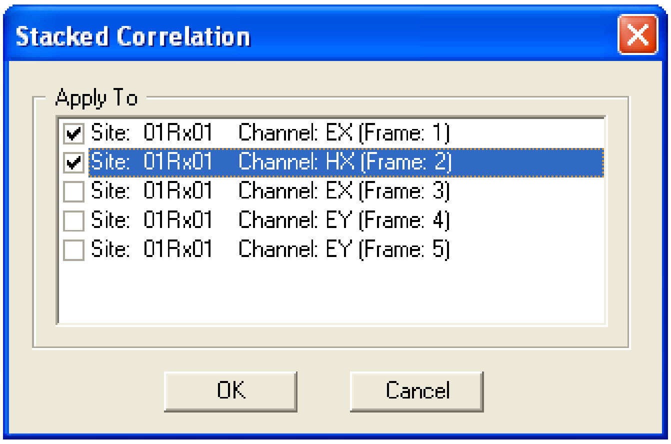




Stacked Correlation

This tool differs from Raw Correlation in that it computes auto- and cross-correlation over the entire signals. The signals are divided into windows of size equal to the number of samples set in the **# of points** drop-down list. The correlation is computed for each window and the results averaged to output a stacked correlation.

Access the tool with the **Tools | Stacked Correlation** function:



In the **Apply To** frame, select one or two signals to be processed by enabling the appropriate check boxes. To compute the stacked correlation, click the **OK** button.

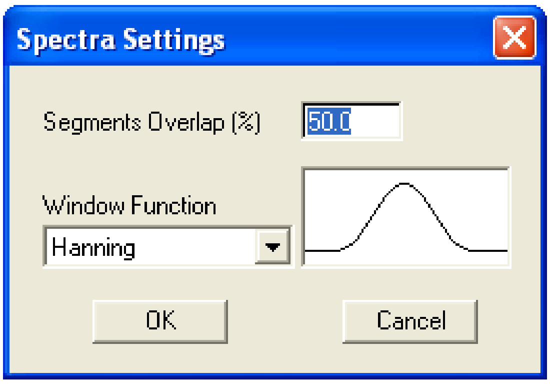
A new window containing the auto- or cross-correlation curve opens. The value and position of the maximum is displayed in the lower right area of the window; the value and position at the current mouse position are displayed in the lower left area of the window.

Spectra

Raw Spectra

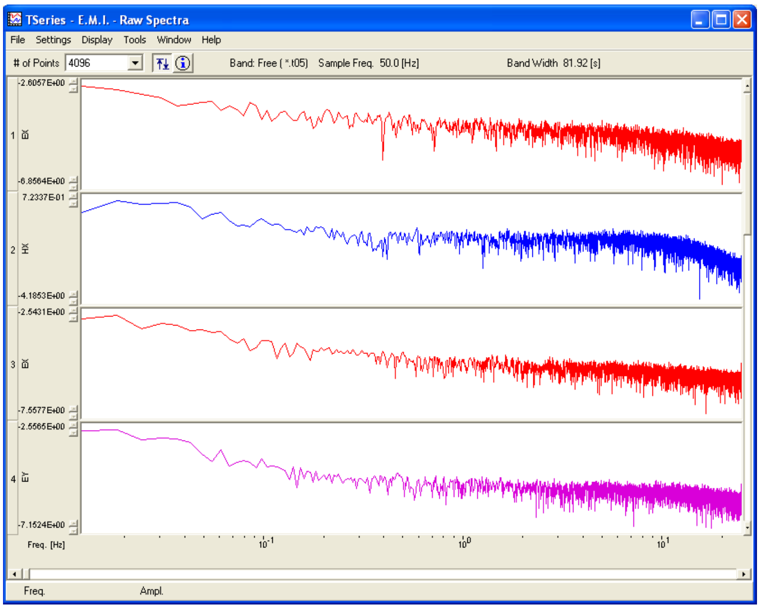
Select the **Display | Raw Spectra** menu command to compute and display the frequency domain spectra of the displayed channel samples based on the currently displayed data.

The spectra are calculated using the settings set with the **Settings | Spectra Options…** menu command:



The default spectra settings are 50% overlap and Hanning window.

Click the scroll buttons  at the of the main window to scroll through the time series in the frequency domain in steps equal to the value set in the **# of Points** drop-down list:



This tool provides a useful instrument for locating frequency domain outliers in MT time series.

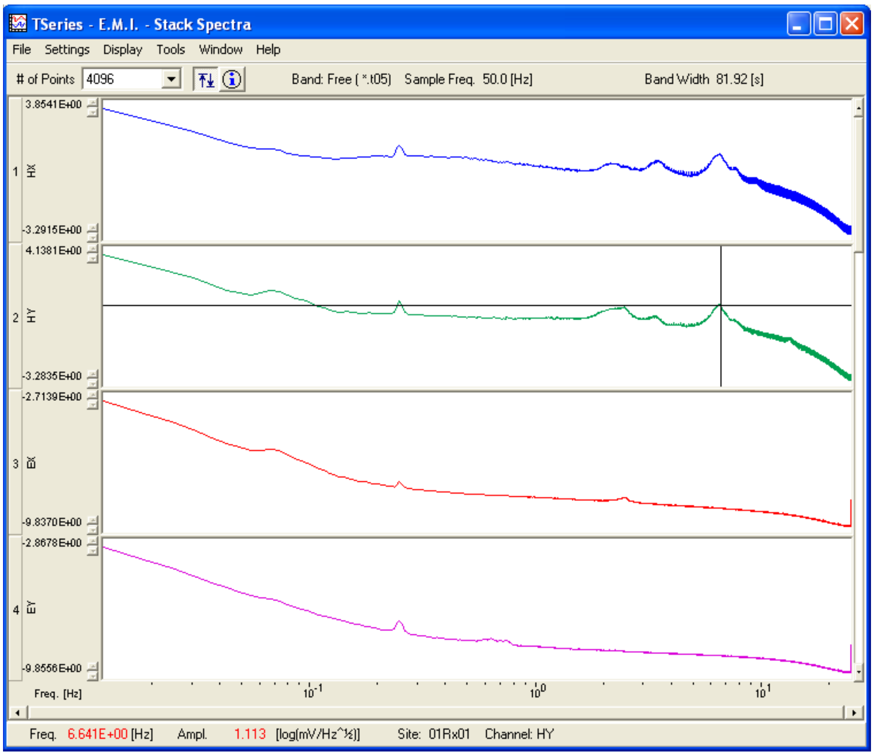
Select the **Display | Time Series** menu command to visualize the time series segment which corresponds to the current raw spectra.

Note: The spectra are computed using the number of points set in the **# of Points** drop-down list.

Stacked Spectra

Select the **Display | Stacked Spectra** menu command to compute and display spectra over the entire time series using the spectra parameters set using the **Settings | Spectra Settings…** menu command:

To find spectral peaks, the user can move the cursor over the spectra curves: Current frequency and amplitude values are displayed in the lower left.

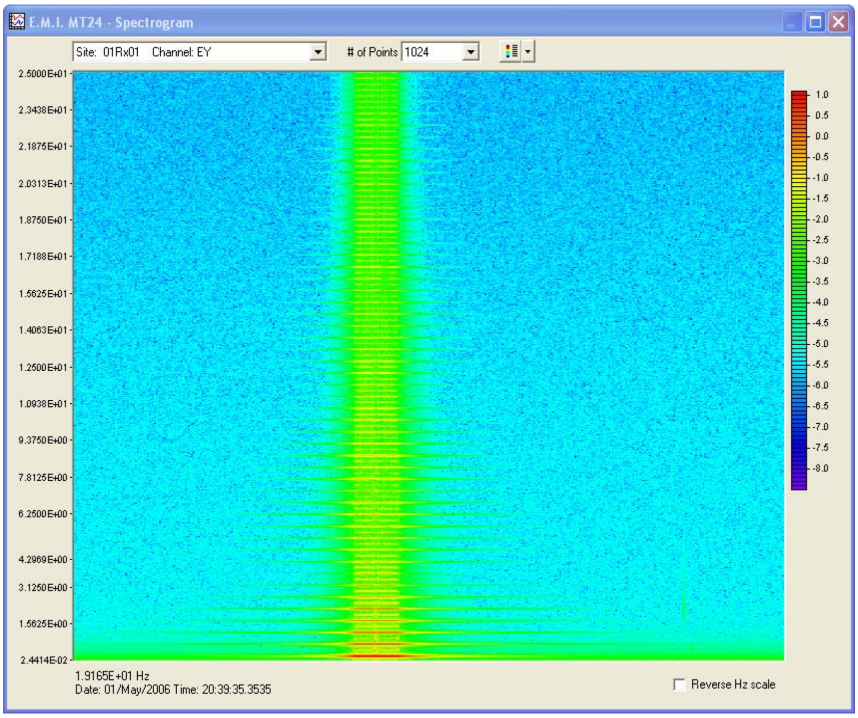


Spectrogram

The **Spectrogram** tool is a powerful utility for checking the temporal variations of the spectral content of a signal because, in a spectrogram, the horizontal scale is the time axis and the vertical scale is the frequency axis.

The time domain signal is divided into n windows of sample length equal to the value set in the **# of Points** drop-down list; the spectra of all windows are computed and displayed side by side.

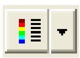
Select **Tools | Spectrogram** to open a new window which displays the spectrogram:



As explained in the following, the number of points used in each spectra computation is, in the frequency domain, equal to one half the number of values set in the **# of Points** drop-down list. This is due to the following: Because the time series contains real values, the transformation is even (symmetric with respect to the y axis) for the amplitude and odd (symmetric with respect to the origin) in phase. Because the spectrogram is a representation of the power spectra (i.e. the power of the amplitude), only half the number of points are needed - those for the positive axis of the frequencies.

Select the channel to be displayed in the upper left menu of the spectrogram window.

To change the number of points used for each spectrum, select a different value in the **# of Points** drop-down list (the number of samples used in the FFT is two times this value).

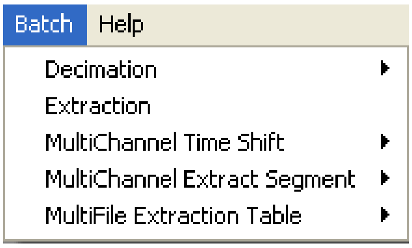
Click the  button to edit the color range used in the display. Select the check box in the bottom right to reverse the frequency scale.

The frequency value, and date and time at the current mouse position are displayed in the lower left part of the window.

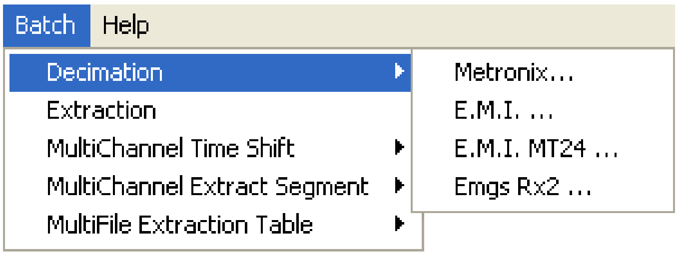
As with the other spectra options, the window function and overlap used in the spectra calculation can be set in the Spectra Settings window, opened with the **Settings | Spectra Options…** menu command.

Batch Functions

The Time Series tool offers a number of batch functions which can be used to simultaneously perform any of a number of different functions on multiple time series. To access the batch functions, close any open times series with the **File | Close** menu command. The **Batch** menu item then appears in the main menu:



The arrows to the right of each menu item open a drop-down list from which the desired data type can be selected:

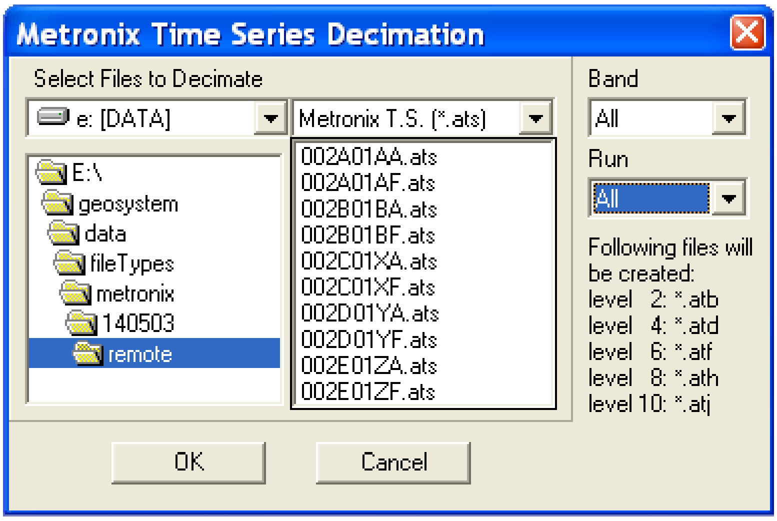


Note: Not all functions are available for all data types.

Decimation

This tool generates new decimated time series files from the original one. A decimation factor of 2 is repeatedly applied to the signal until the 10th level of decimation is reached.

In the left part of the window, select the drive and folder that contains the time series files. To the right of the drop-down list for the drive, select the time series format of the files that are to be selected in the field in the center of the window. Click the **OK** button to start decimation of the selected files.



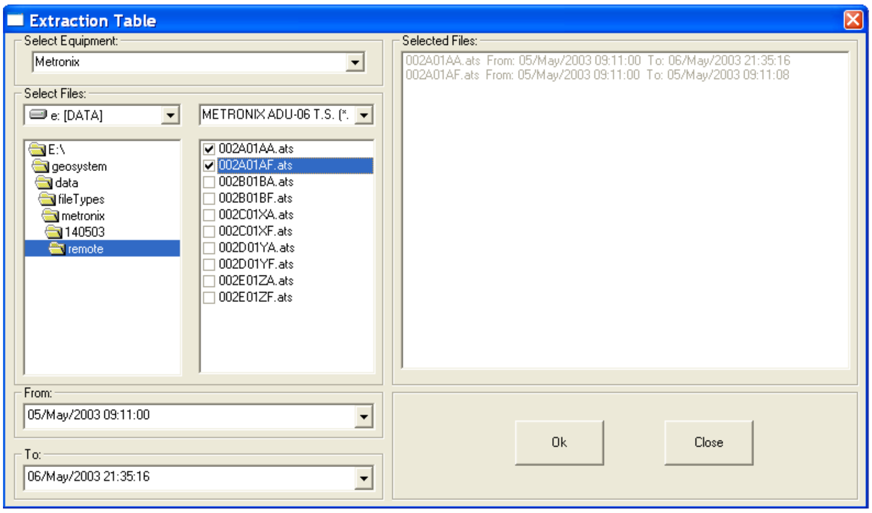
Extraction

Use this tool to extract time series segments which cover a given time interval from a subset of time series. The limits used for the extraction are the absolute date and time. The extracted time series are written to new files.

Select the appropriate equipment type in the **Select Equipment** frame.

In the **Select Files** frame, choose the drive and folder, then select the desired file extension and the list of original files by clicking the appropriate check boxes. A brief description (file name, start date and time, end date and time) of the selected files is displayed in the upper left window.

In the lower left fields, enter the desired values for start and end times. Click the **OK** button to extract the data.



MultiChannel Time Shift

Use the **Batch File | MultiChannel Time Shift** menu command to perform a shift operation on selected EMI MT1 and Phoenix V8 time series files.

MultiChannel Extract Segment

Use the **Batch File | MultiChannel Time Shift** menu command to perform a shift operation on selected EMI MT1 and Phoenix V8 time series files.

MultiChannel Extraction Table

5: Cascade Decimation MT

Overview

WinGLink’s Cascade Decimation program uses the cascade decimation processing method to generate cross-powers from MT time series files. The program also provides an option for using coherence robust processing to further improve data quality, though this option considerably lengthens processing times.

Execution of the actual processing routines occurs in batch mode using parameter files which are created using a Wizard interface, which is an integral part of the Cascade Decimation program.

Input Data

The program supports the following time series formats:

- EMI MT-1, EMI MT-24
- Metronix (ADU-06)
- Phoenix (V5)

Output Data

Processed data are output both as binary weighted cross-power files (*.bxp), which can then be edited using WinGLink’s Cross Power tool, as well as standard EDI files, which can be directly imported into WinGLink or opened in WinGLink’s Data Analysis tool.

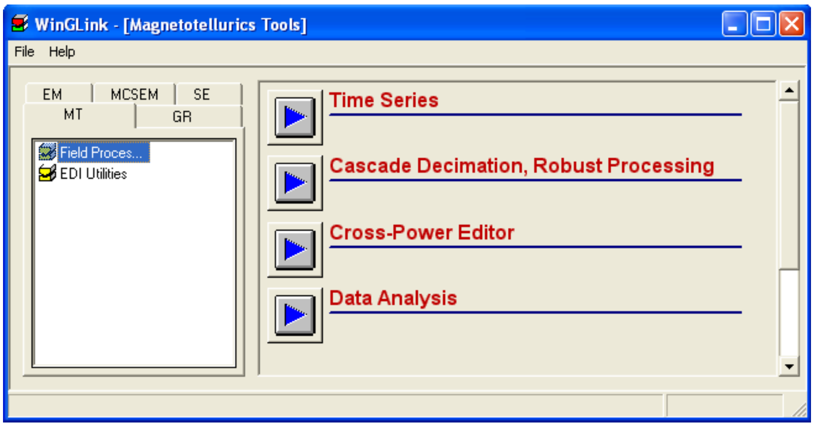
About Command Files

A single RCD command file may be used to process a maximum of two sites, i.e. local and remote and, depending on the data source, up to three different frequency bands. The names of the sites, the corresponding time series paths, and the relevant parameters to use are all saved in the text-based command file. The processing routine, which is implemented as an external executable file, accesses the command files during program execution.

Getting Started with Cascade Decimation

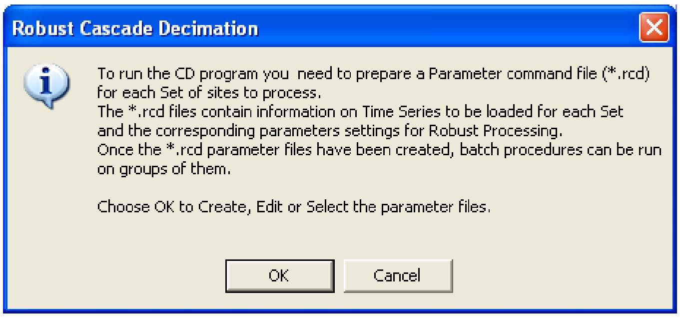
Opening Cascade Decimation

After opening WinGLink in tool mode, select **MT** in the left panel of the tool form, select **Field Processing** in the left frame, and click the arrow to the left of Cascade Decimation, Robust Processing.

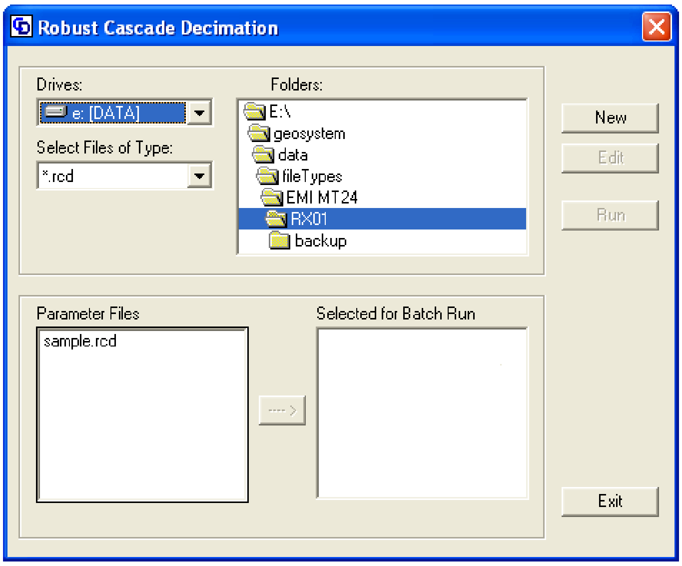


Note: For information on starting WinGLink in Tool Mode, refer to the "Getting Started With WinGLink" chapter of this manual.

The program opens with a message alerting you to the fact that an RCD command must be created for each set of sites:



Confirm with **OK** to acknowledge the info box and open the main program interface:



This main program interface is used to select the command files to be used for data processing. Use the path navigation controls provided in the upper part of the window to select the folders which contain the desired command files. Any RCD files found in a given folder are listed in the **Parameter Files** field. Use the arrow button to select a file for processing. Files can be removed from the **Selected for Batch Run** field with the Delete key.

Note: It is not necessary that all RCD files to be processed be located in the same folders. To select files from different folders, after selecting one RCD file, use the navigation controls to navigate to the folder which contains the next.

Creating a New Command File

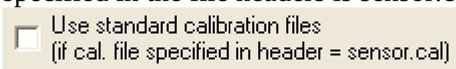
If working with the Cascade Decimation program for the first time or to create a new RCD command file, first use the path navigation controls provided in the upper part of the window to specify the folder in which the file is to be created. Then click the **New** button to open the RCD File Wizard, which will guide you through the file creation process:

- Wizard window 1: File type and processing parameters
- Wizard window 2: Weighting and processing types
- Wizard window 3: Time series selection
- Wizard window 4: Channel mapping

Wizard Window 1: File Type and Processing Parameters

In the **Parameters File** field, enter the name of the RCD file which is to be created. The selection made in the **Time Series Type** drop-down list determines what other options are displayed on this first window:

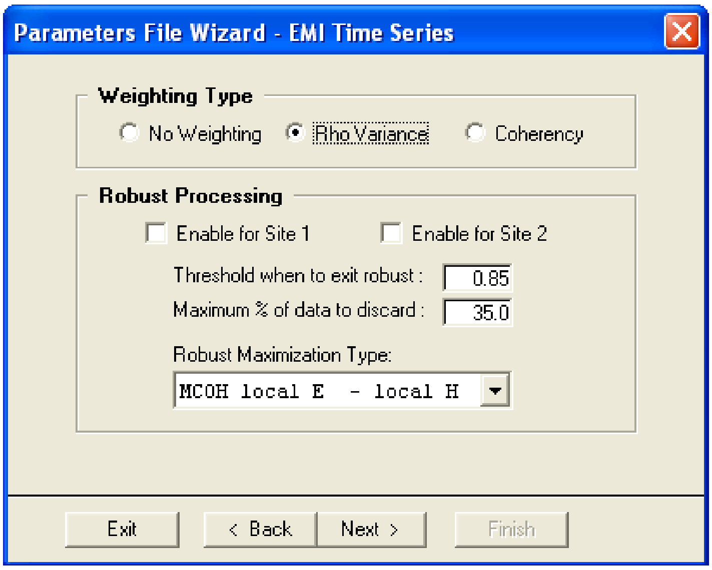
- For E.M.I. MT-1 and MT-24 data, options are provided for specifying the number of sites to be processed as well as the number of frequency bands as shown above.
- For Phoenix V5 data, only the number of sites must be specified.
- For Metronix ADU06 data, in addition to the number of sites to be processed and the number of frequency bands, specify whether a default set of calibration files should be used if the calibration file specified in the file headers is sensor.cal:

  According to the system manufacturer, there is generally very little difference between sensors. The standard calibration files, which were provided by the manufacturer, generally provide accurate results for most coils. For maximum accuracy, however, we recommend using the calibration files provided with the coils.

After making the desired selections, click the **Next** button to advance to the second window of the Parameters File Wizard:

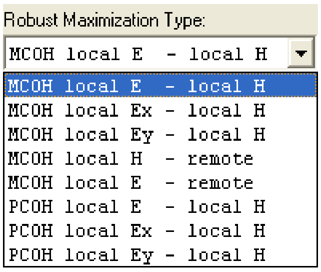
Wizard Window 2: Weighting and Processing Types



In the **Weighting Type** frame, specify the type of weighting to be performed.

In the **Robust Processing** frame, enable or disable robust processing for the site(s) being processed.

The **Robust Maximization Type** drop-down list offers a number of different maximization types for selection:



These maximization types differ as follows:

Type	Method
1)-MCOH local E-local H	Combine the weighting factor of scheme 2 with that of scheme 3
2)-MCOH local Ex-local H	Use MULTIPLE COHERENCE of Ex with the total MAGNETIC field
3)-MCOH local Ex-local H	Use MULTIPLE COHERENCE of Ey with the total MAGNETIC field
4)-MCOH local H-remote	Use the combined weighting factor of the MULTIPLE COHERENCE of Rx with the total MAGNETIC field with the MULTIPLE COHERENCE of Ry with the total MAGNETIC field.
5)-MCOH local E-remote	Use the combined weighting factor of the MULTIPLE COHERENCE of Rx with the total ELECTRIC field with the MULTIPLE COHERENCE of Ry with the total ELECTRIC field.

6)-PCOH local E-local H	Combine the weighting factor of scheme 7 with that of scheme 8

Note: If robust processing is enabled for either of the two sites, the time series of both sites must be synchronized.

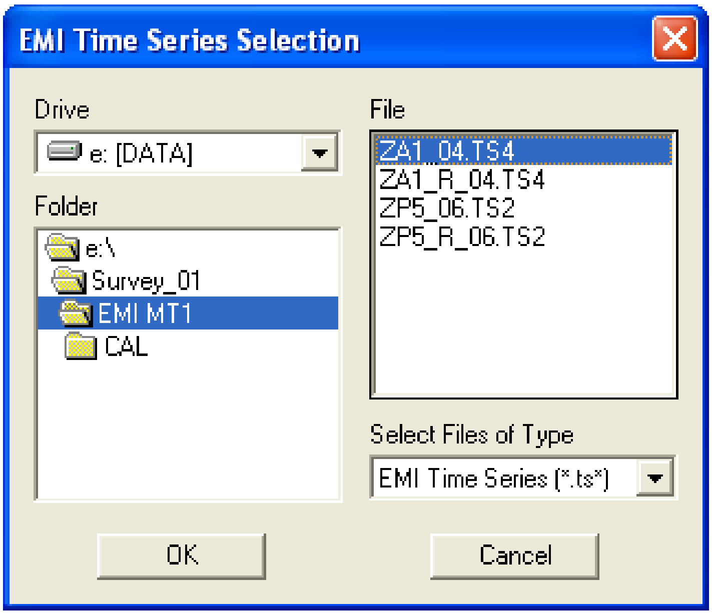
After setting the weighting type and robust processing options, click the **Next** button to advance to the next window of the wizard:

Wizard Window 3: Time Series Selection

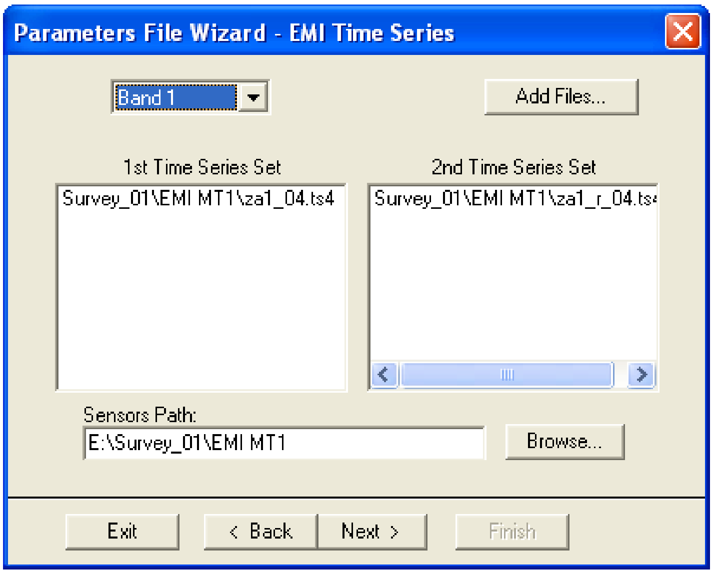
The third window of the wizard, which is used to select the data to be processed, varies depending on the type of equipment selected in the first window. The windows associated with each of the four equipment types currently supported are described below:

EMI MT-1

Click the **Add Files…** button to open the EMI MT-1 time series selection window. Note that when clicking the **Add Files…** button, you must select from the drop-down list which appears the time series set to which the data are to be added:



Select the time series for each data set and for each band. Select the band using the drop-down list in the upper left corner of the time series selection window. The path to the sensor calibration files for all coils used in the sounding must be specified in the **Sensors Path** field at the bottom of the window.



EMI MT-24

The main window of the EMI-24 time series selection interface is nearly identical to that for EMI-MT1 data shown above. Unlike EMI-MT1 time series files, which contain all data associated with a given sounding, each component of an MT24 sounding is stored in a separate file. Thus, instead of selecting a single time series file, it is necessary to select an entire set. This is accomplished using an .SRE (survey) file. An SRE file is nothing more than a list of file names with their respective absolute file paths. Shown below are the contents of a sample SRE file:

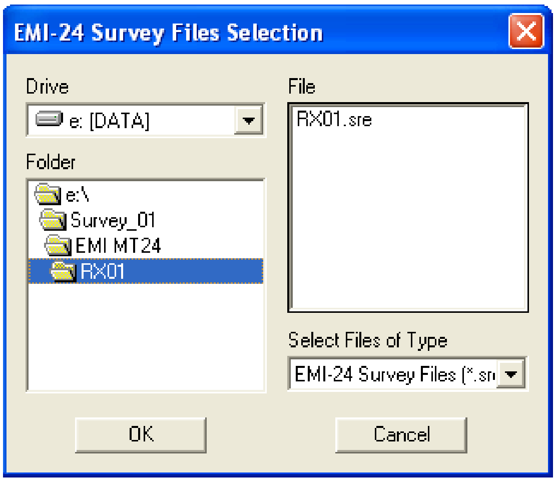
```
E:\Survey_01\RX01\rx01_1.t05
E:\Survey_01\RX01\rx01_2.t05
E:\Survey_01\RX01\rx01_3.t05
```

`E:\Survey_01\RX01\rx01_4.t05`

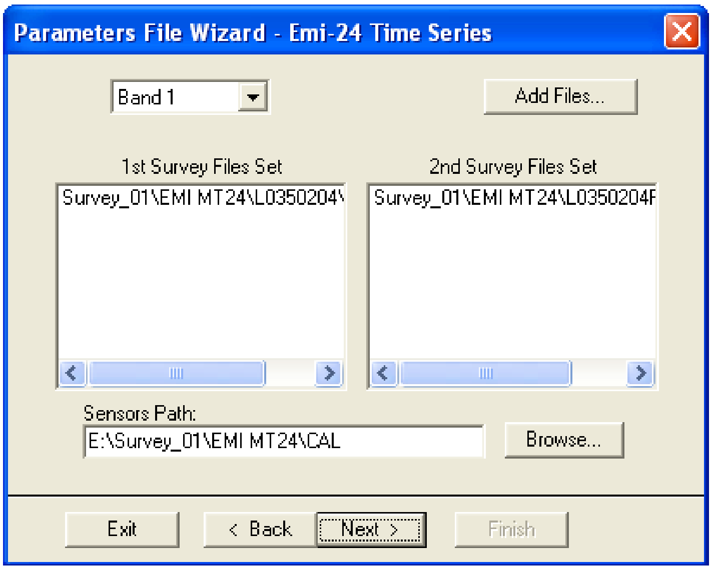
`E:\Survey_01\RX01\rx01_5.t05`

Note: SRE files must be manually created for each set of time series data.

To select the SRE file for a given time series set and band, select the desired band from the **Band** drop-down list and then click the **Add Files…** button, selecting the desired time series set in the process:

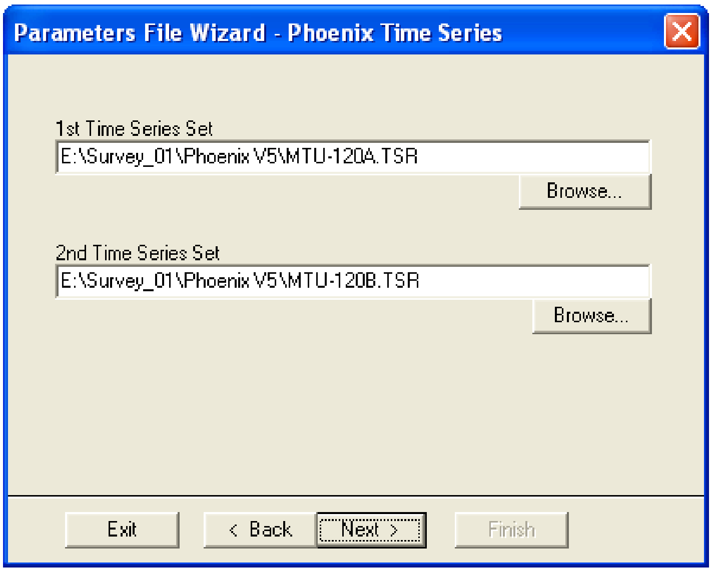


Repeat this process for each band and data set. Specify the sensor path to complete the file selection process:



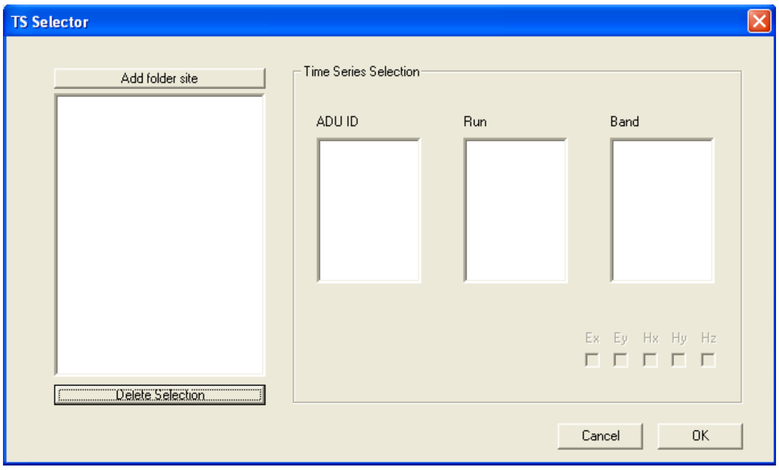
Phoenix V5

As it is not necessary to specify a frequency band when processing Phoenix V5 data, time series selection is limited to the selection of the time series files, one file for each site:



Metronix ADU06

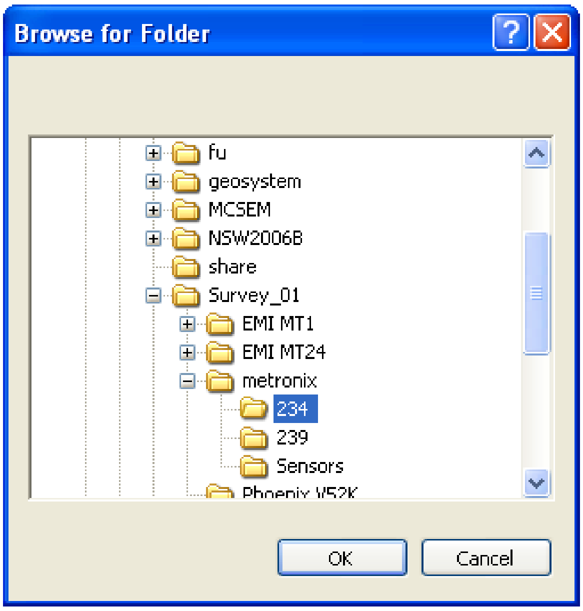
Metronix files, like EMI MT24 files, each contain data associated with a separate channel. For Metronix files, however, the Cascade Decimation program automatically generates a survey file which contains the paths of the respective files. As with the other file types, click the **Add Files…** button to open the TS Selector window:



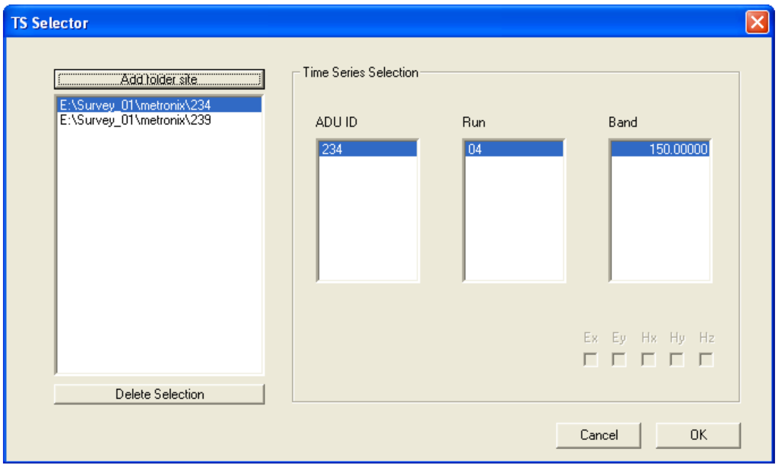
This window contains four frames:

- Folders: Contains all sites which have been added by the user
- ADU ID: Station ID of the selected site
- Run: Run number of the selected site
- Band: Frequency band of the selected site

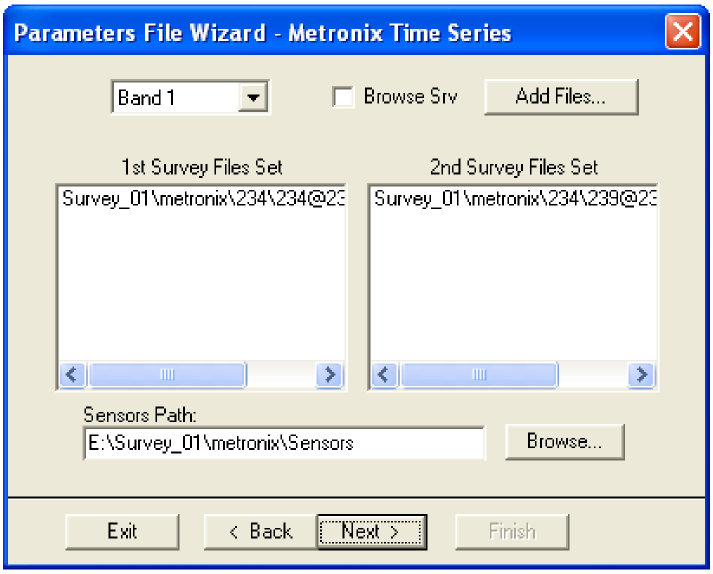
Next, click the **Add folder site** button to add time series files to the selector window:



Navigate to the folder which contains the desired time series files. Click the **OK** button to accept the selection and close the window. In the TS Selector window, now click the folder which was just added: all time series contained in the folder are now listed by station ID, run number and frequency band:



Select time series for each site and band. Note that the time series added to the TS Selector window are retained for future use until cleared with the **Delete Selection** button.

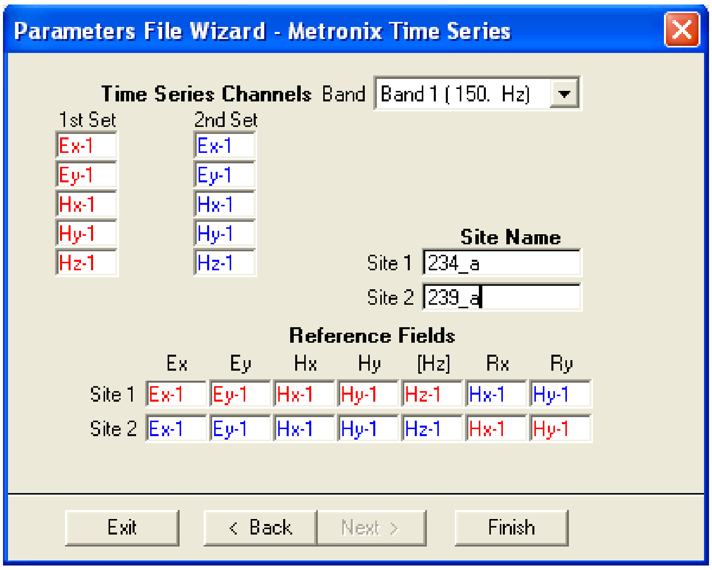


File sets can be removed from the **Survey File Set** fields by clicking the respective field with the right mouse button and selecting the **Clear Time Series Set** shortcut menu.

After selecting the time series, click the Next button to advance to the fourth Wizard window:

Wizard Window 4: Channel Mapping

The fourth and final window of the wizard is used define station mapping and specify the site names. In the example shown below, the program has automatically used the magnetic channels from the 2nd site as the remote channels for the 1st site and vice versa.



Channel mapping can be changed by clicking a component name and dragging it to the new position in the **Reference Fields** section of the window. Channels can either be selected from the lists located under the **Time Series Channels** heading, or channels which have already been assigned a field in the **Reference Fields** area of the window can be moved from one field to another.

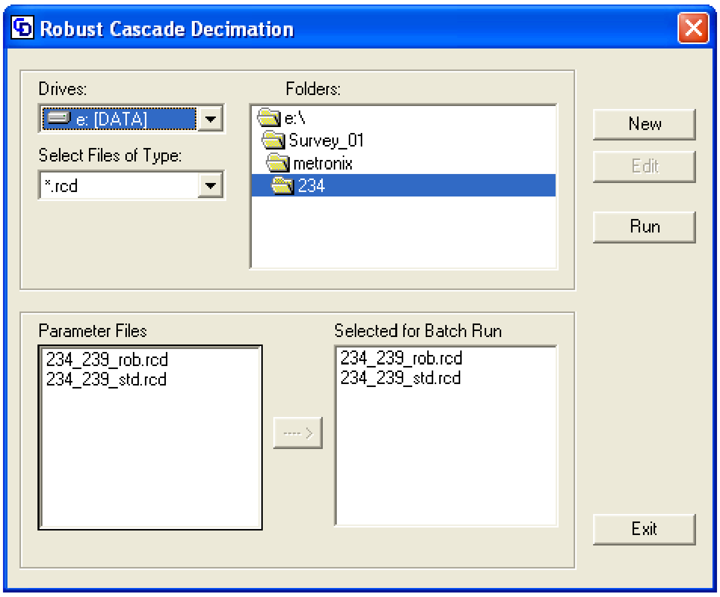
The program responds to unexpected settings, e.g. placing an Hy component on an Hz field with, a warming message:

Such reference field assignments may be necessary under certain circumstances, e.g. when using non-standard field configurations.

Use the **Band** drop-down list to switch between the bands which have been selected for processing. After assigning the reference fields for each band, click the **Finish** button to complete the Wizard.

Processing Data

Upon completion of the Wizard, the Cascade Decimation program returns to the main window. The just-created RCD file, as well as any other RCD files present in the current folder, are available for selection. Use the → button to move the RCD files to be used to process data to the **Selected for Batch Run** frame of the window:



Click the **Run** button to begin processing of all selected RCD files. Depending on the set processing parameters, the size of the time series files and the number of bands to be processed, processing time may be considerable. During processing, a DOS command window will appear on the screen; once processing has completed, the program window closes. Closing this window with the mouse will prematurely terminate data processing.

Upon successful completion of processing, an EDI file and a BXP (cross-power) file are output to the same folder as the RCD file. BXP files can be opened in WinGLink's Cross-Power Editor program; EDI files can either be edited in WinGLink's Data Analysis program or imported directly into a WinGLink database.

6: Cross-Power Editor MT

Overview

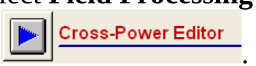
WinGLink's Cross-Power Editor provides an easy-to-use interface for editing the cross powers generated by WinGLink's Cascade Decimation program. The resistivity and phase values at a given frequency typically consist of a number of samples. The Cross-Power Editor is used to edit the individual samples. Upon completion of editing, the average values at each frequency are used to generate an EDI file which can then be opening in the Data Analysis tool for MT data or imported directly into WinGLink as station data.

Supported Formats

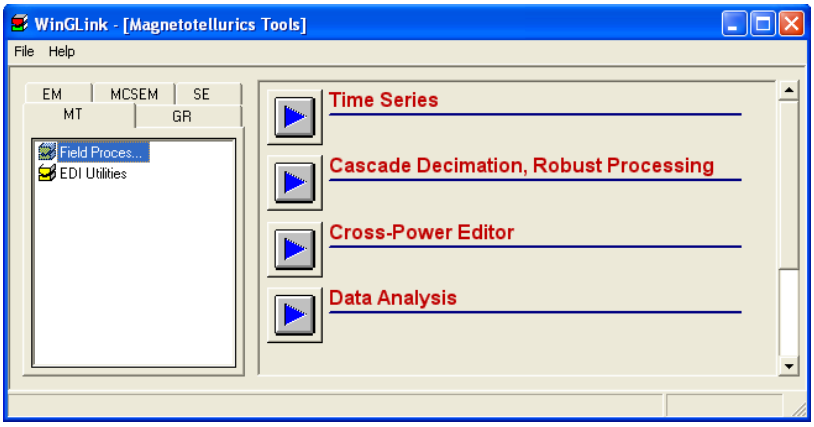
The Cross-Power Editor can be used to edit the BXP files generated by the Cascade Decimation program. In addition, the program can also edit .MT and .AMT files generated by the Phoenix V5 processing software.

Getting Started with the Cross-Power Editor

Starting the Cross-Power Editor

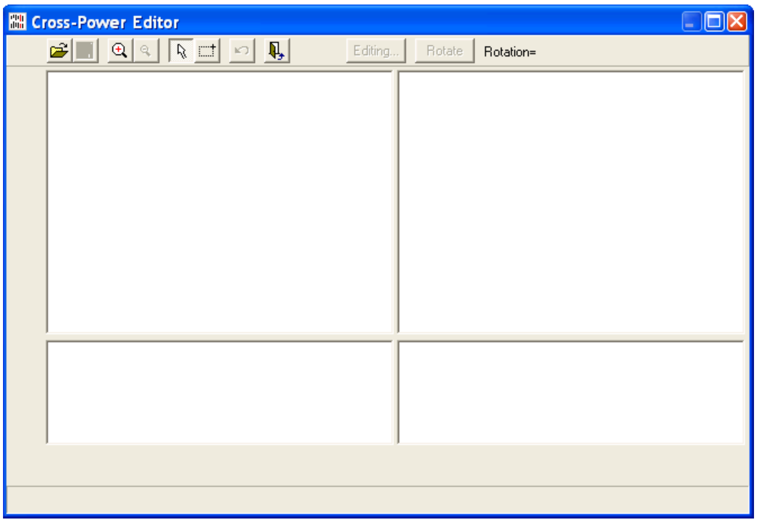
After opening WinGLink in tool mode, select **MT** in the left panel of the tool form, select **Field Processing** in the left frame, and click the arrow to the left of 

.



Note: For information on starting WinGLink in Tool Mode, refer to the "Getting Started With WinGLink" chapter of this manual.

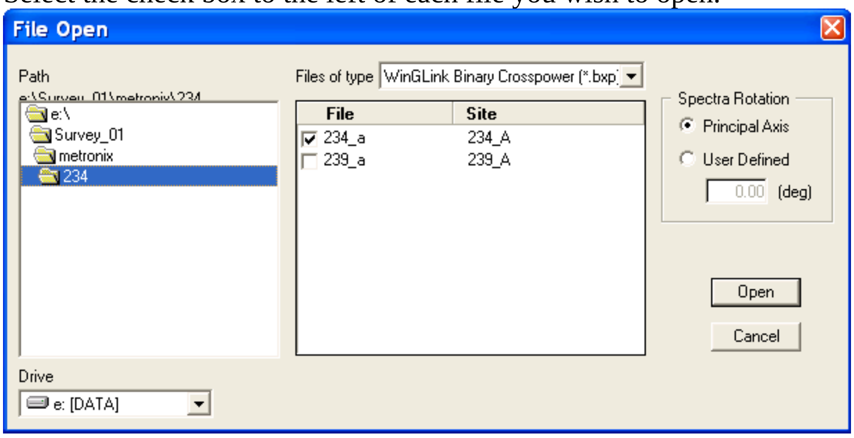
The Cross-Power Editor opens:



The program window is divided into four panels. The two panels on the left side are used to display the average rho and phase values at each frequency; the two on the right are used to display all samples at a given frequency.

Loading a Cross-Power File

1. Click the  button to open the dialog box for selecting the cross-power files to be opened.
2. Click the  button in the **Files of Type** dropdown list to display the supported input file formats. Make the appropriate selection:
3. Use the file navigation controls in the left part of the window to browse the directory structure until the files you wish to edit are displayed in the **File | Site** frame of the window.
4. Select the check box to the left of each file you wish to open.



Note: The Cross-Power Editor can be used to open more than one file at the same time. If you do so, the opened files will be merged into a single file when the results are saved. This option should be used only to merge cross-powers for the same station contained in different input files.

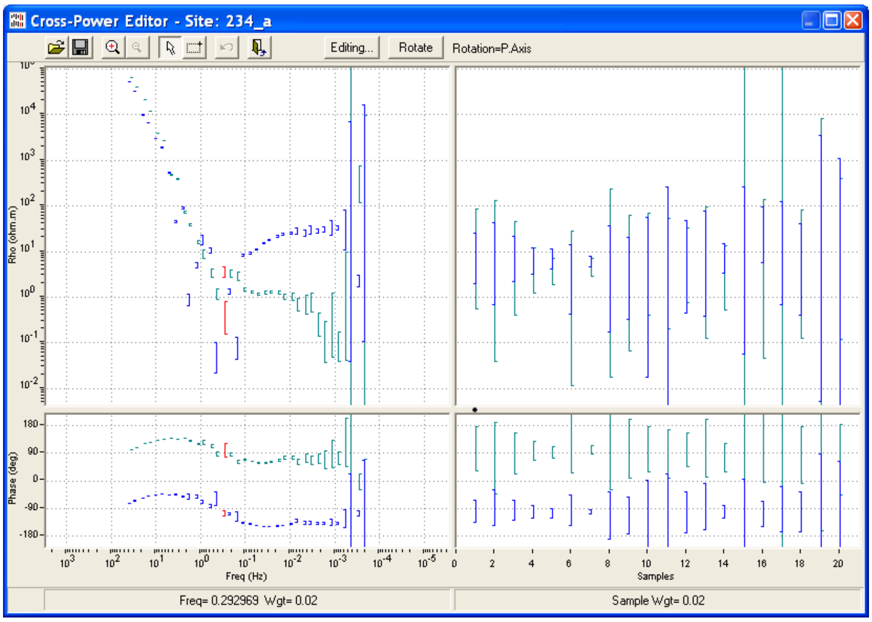
5. If desired, specify an angle for spectra rotation in the appropriate field.

   Note that the rotation is returned to the principle axis when the file is saved regardless of any setting made here. Rotation options are provided in the Data Analysis tool, the Soundings module and when importing EDI files into a WinGLink database.

6. Click the **Open** button to open the cross-power file for editing.

Editing a Cross-Power File

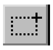
To begin editing, select a frequency from the left side of the Cross-Power Editor by clicking the mouse anywhere in the average resistivity panel of the editor, i.e. the upper left panel. The data sample at the frequency closest to the mouse location is selected. The currently selected frequency is displayed in red in both the resistivity as well as the phase panels; the blue bars correspond to the yx component, the green bars to the xy component:



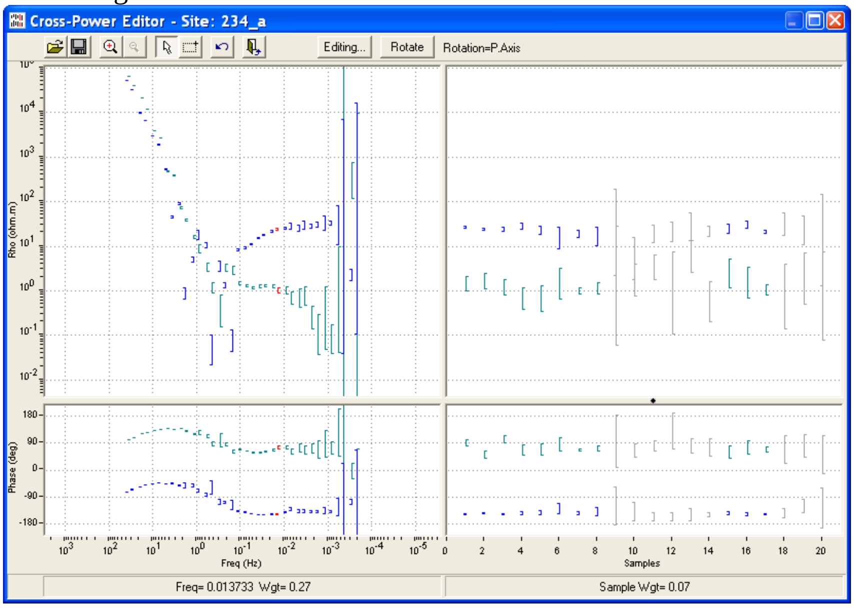
Masking/Unmasking Components

1. In the left window, use the mouse to click the resistivity or phase of the frequency to edit. The color of the error bars for the selected frequency changes to red. All components associated with a given frequency, i.e. resistivity and phase of both xy and yx components, are masked/unmasked together.

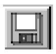
2. Click the  button on the toolbar to mask/unmask individual component in the right window, **or**

3. Click the  button to mask/unmask a group of adjacent individual components in the right window.

4. The disabled components are displayed in gray. The results of the editing are shown in real time in the left window:



Saving the Edited Results

Click the  button to save the edits.

Files can be saved either as BXP or EDI files:

- BXP files
  Contain all original file information, i.e. information about which samples have been disabled. These files can be read into the Cross-Powers Editor at a later time for further editing.

- EDI files
  Contain the final averaged results in EDI format. These files are used as the input source for the "Data Analysis" program, an MT Tools program, and can also be imported directly into WinGLink as station data.

7: Data Analysis MT

Overview

The Data Analysis program is used to edit and analyze individual stations as standalone datasets. The user interface is very similar to that of the MT Soundings module, which can be accessed while in Database Mode, and offers many of the same functions. Because this program is intended for performing quality control and editing operations, however, the menu commands available in Tools Mode are somewhat limited, i.e. this tool does not include 1D modeling.

Supported Formats

The Data Analysis program can only be used to import standard EDI files.

Getting Started With the Data Analysis Program

Opening the Program

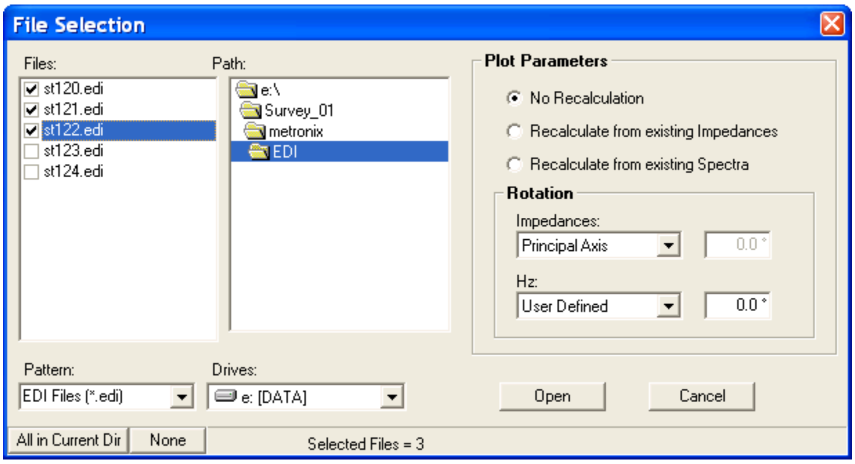
After opening WinGLink in Tool Mode, select **MT** in the left panel of the tool form, select **Field Processing** in the left frame, and click the arrow to the left of Data Analysis.

Note: For information on starting WinGLink in Tool Mode, refer to the "Getting Started With WinGLink" chapter of this manual.

File Selection

On program startup, the Data Analysis program opens with a dialog box prompting you to choose either individual station mode or batch mode. This second option, as will be described later, can be used to perform a specific function on multiple stations. To get started wit the program, select the **Open stations** option and click the **OK** button:

The program opens the File Selection window. This can be used to select any EDI files accessible from the computer:



In the **Plot Parameter** area of the window, you can specify whether the plot parameters should be recalculated, and, if so, whether impedances or spectra, if present, are to be used. The options function as follows:

Plot Param. Option	When to use
No recalculation	To import the plot parameters saved in the EDI file without recalculating them from existing impedances or spectra If plot parameters are missing, the program calculates them from existing impedances. If impedances also are missing, plot parameters and impedances are calculated from existing spectra.
Recalculate from existing Impedances	To compute plot parameters from existing impedances without recalculating impedances from spectra.
Recalculate from existing spectra	To recalculate plot parameters and impedances from spectra.

Rotation

In the **Rotation** area of the window, specify how impedances and tipper are to be rotated upon import. In doing so, note the following:

If there is a ZROT block in the EDI file, the data are first “back rotated” to 0 degrees. Thus, to match the rotation in the EDI file, you must use the **User Defined** field to specify the angle to which the impedances are to be rotated. The angle specified here is used for all frequencies for the given file.

When rotating to the principle axis, the program rotates the impedance matrix to the maximum and minimum values of Zxy and Zyx, respectively.

If there are tipper data present in the EDI file, these data are likewise “back rotated” to 0 degrees from the angle specified in the TROT block. If there is no TROT block, it is assumed that the data are already at 0 degrees and no back rotation is performed. The ZROT block is NOT used to specify the tipper rotation in the case that no TROT block is present.

The EDI files of stations which were rotated contain ZROT and TROT fields with values corresponding to the station rotations.

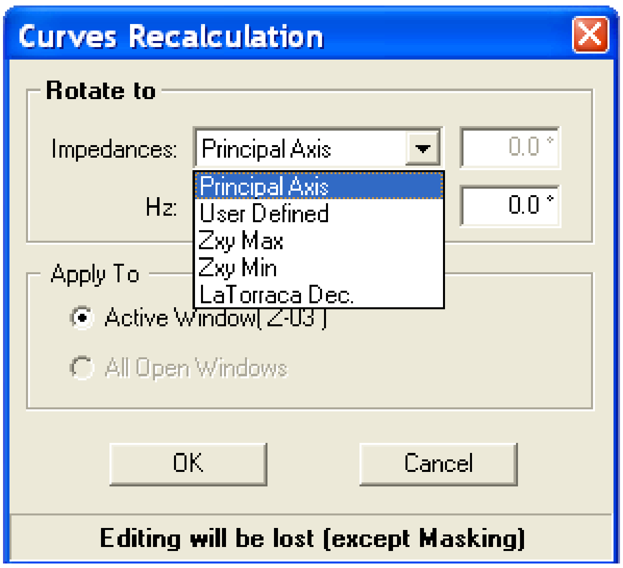
Note: When using the Data Analysis program, tipper should be rotated to strike.

Saving Data

Edited station data can be saved to EDI files from the main window of the Data Analysis program with the **File | Save As** or **File | Save All** menu commands. To prevent the accidental overwriting of the original data files, the **Save All** command saves files with the extension EDE.

Data Rotation & Decomposition

Should it be necessary to rotate a station after the EDI file has been opened in the Data Analysis program, the Curves Recalculation window, which contains the same rotation options as the File Selection window described above, can be opened by selecting the **Rotate** menu command on the main menu:



In addition to standard rotation, this window provides an option for performing LaTorraca decomposition.

Correcting Spectra

WinGLink reads and writes stacked spectra from/to EDI files using a [Real+Imaginary] spectral conjugation structure. If stacked spectra are packed in an EDI file using a different structure, the MT parameters may be incorrectly displayed.

When spectra are packed properly, the phase components are displayed by WinGLink in the correct quadrants, i.e. +45° for one component, and -135° for the other, and predicted coherencies will be consistent. This may not be the case when the spectra packing is made using a different conjugation structure.

The **Fix Spectra** function, accessed with the menu command by the same name, allows the user to correct the spectral set by swapping or conjugating the spectra packing:

The options in the upper part of the window serve the following function:

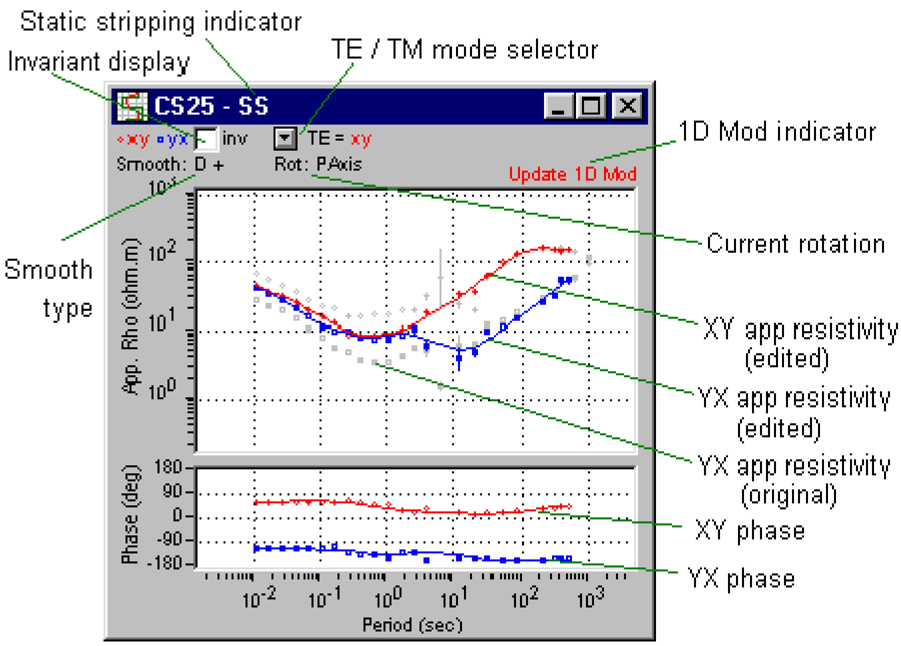
Option	Result
Swap	changes (A+B) into (-B+A)
Conjugation	changes (A+B) into (A-B)

The proper spectra packing is achieved by combining these corrections in an appropriate sequence.

Viewing Data and Plot Parameters

The Data Analysis View Form

The following figure shows the Data Analysis view form:



TE/TM mode selector	Click this button to change which curve corresponds to TE mode. In the above example, the xy curve is selected for TE mode (therefore, the yx is for TM mode)
1D model validation	This text is displayed whenever the 1D model of the station needs to be recalculated because a change was made on the curve.
Invariant display	If this option is checked, the invariant curves for resistivity and phase are shown. The invariant resistivity is calculated as the geometric mean of rhoXY and rhoYX; the invariant phase is calculated as the arithmetic mean of the two.
Smooth type	Indicates the algorithm used to calculate the smoothed curves.
Current rotation	Indicates the rotation (if any) applied to the impedances.
XY, YX curves	The original curves are displayed in gray, the edited curves in red (the XY) or blue (YX); the continuous curves represent the smoothed data.
Static stripping	If present, this text indicates that the edited curves have been static stripped.

Data Display Options

The options available under the **View** menu can be used to select from various displays:

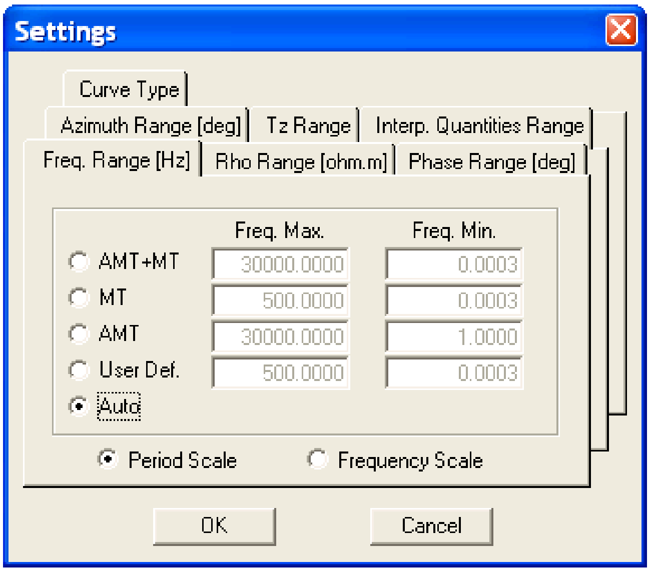
- Resistivity and phase
- Resistivity, phase and strike
- Interpretational quantities
- Polar diagrams
- Resistivity, phase and polar diagrams

- Resistivity, phase and induction arrows
- Data quality
- Tzx and Tzy

Each of the above selections made under the **View** menu applies to the current window. The selection can be applied to all open windows with the **View | Update All View** menu command.

View Options

Display parameters, e.g. ranges, axes, curve types, can be set in the Settings window, opened with the **View | View Options…** menu command:

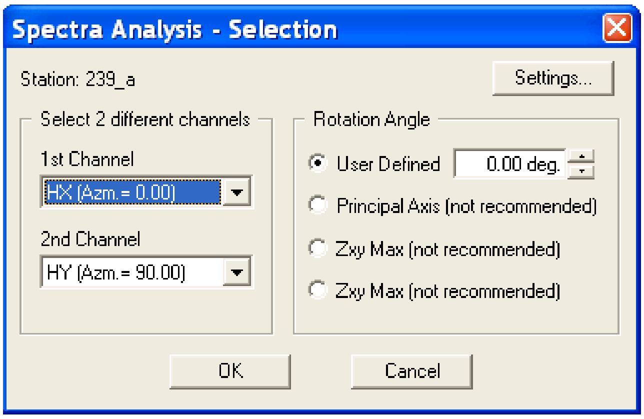


Settings made here are applied not only to all open windows, but are also used the next time the Data Analysis program is started.

Spectral Analysis

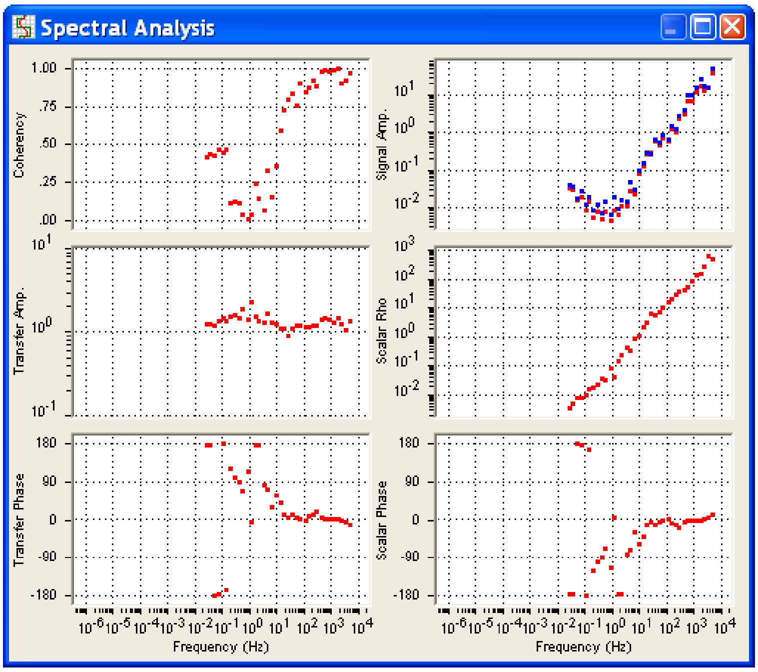
The **Spectral Analysis** function, which is only available for soundings which have spectral data, is opened with the **View | Spectral Analysis** menu command. This function, used to compare a number of spectral characteristics between two channels, serves as a tool for gathering insight on data quality.

Upon selection of the menu command, the Spectral Analysis Selection window opens. Select the channels to be evaluated in the provided drop-down lists. A rotation angle, if desired can be selected in the **Rotation Angle** frame of the window.



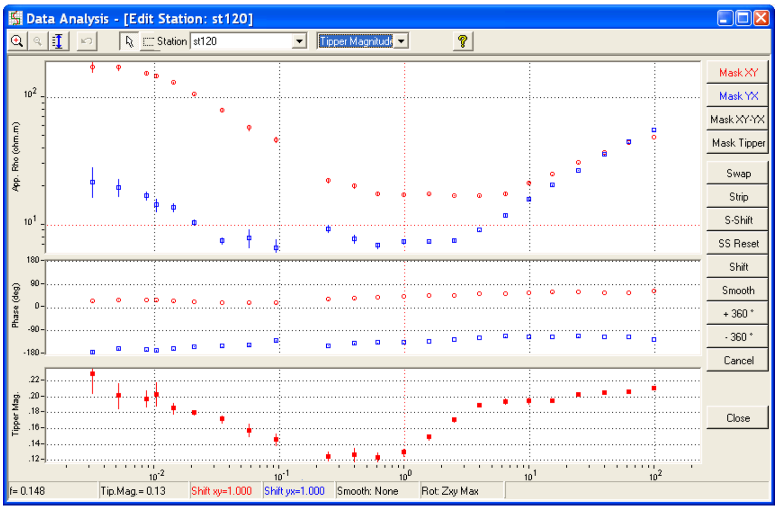
Transfer function ranges can be set for both amplitude and phase by clicking the **Settings** button.

Click the **OK** button to open the Spectral Analysis window for the selected channels:



Editing Data

The interactive Sounding Editing form is opened with the **Edit** menu command while a sounding window is selected:



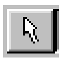
The form includes commands for performing a number of different editing functions. After clicking any of the function buttons at the right edge of the Edit window, the remaining buttons, with the exception of the **Cancel** button, are disabled. The selected function remains active until either the respective function button is pressed again or until the **Cancel** button is pressed.

Changes made to the curves with the edit functions available here are not actually applied to the data until the **Close** button is used to close the Edit window and the user acknowledges the changes in the Exit Editing window.

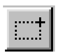
In addition to the actual curves, the status bar at the bottom of the Edit window provides additional information on the curves as well as with respect to the current mouse position:

- f = frequency at current cursor position
- Rho = resistivity value at current cursor position
- Shift xy = static shift of xy curve
- Shift yx = static shift of yx curve
- Smooth = indicates type of curve smoothing, if any
- Rot: rotation type and/or value

Selecting Data Points

 Single point selection

Click to select/unselect for editing single data points on the curve.

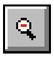
 Multiple points selection

Click to select/unselect for editing all data points included in a user-defined range: click and drag the mouse to define the area.

Zooming in on the Curve

 Zoom in

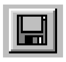
1. Click a point in the curve diagram and release the button.
2. Move the mouse to define the zoom area.
3. Click and release the left mouse button.

 Zoom out

Click to reset the previous view.

Note: this button is available only after the **Zoom in** button has been used. It remains available until the original curve size is restored.

Saving Editing Changes

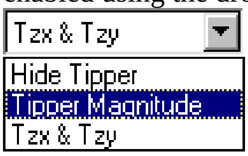
 Save button

Saves the editing changes. These changes must be confirmed when closing the station window in the main menu in order to store them in the database.

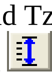
Editing Data Points

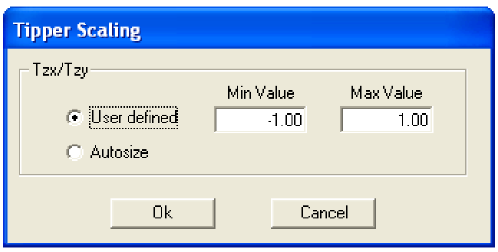
Masking data: Using the appropriate buttons, it is possible to mask individual modes at each frequency, either across all modes or for individual modes. Changes made here are reflected throughout the database, i.e. in the 2D Inversion and Pseudo-Section modules.

Note that the Mask Tipper button is enabled only when tipper display is enabled using the drop-down list at the top of the screen:



Select the Tzx & Tzy display option to display the two tipper components in separate windows.

The vertical scaling of the tipper magnitude and Tzx/Tzy components can be set using the Tipper Vertical Scaling button , which appears towards the left end of the tool bar when either Tipper Magnitude or Tzx/Tzy are visible. Clicking this button opens the Tipper Scaling dialog box:



Settings made here are retained in subsequent WinGLink sessions.

The MT masking buttons are used in the same way as the TEM masking buttons, see description above.

Shift

Shift data points

1. Click and select the data points to be shifted by single or multiple selection.
2. Move the data points to the desired positions and click the left mouse button.
3. Click the **Shift** button again to confirm the action.

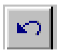
Smooth

Calculate and display a smoothed curve

Click the button to display the smoothing options

Cancel

Cancel the current action

 Undo button

Undo all the editing changes made since the last time the curve was saved. All editing steps can be undone provided the station has not yet been saved since editing.

Swapping

Swapping allows you to assign a data point of the XY curve to the YX curve (at the same frequency) and vice versa. This is useful where analytic rotation results in inconsistent mode assignment.

To Swap the Data Points of Apparent Resistivity or Phase Curves:

1. Open the station or select its window if already open.
2. Select the **Edit** command from the main menu.
3. Click the Swap button.
4. Click the data point(s) you wish to swap with the left mouse button.
5. Click the Swap button again.

To swap the entire curve, use the right mouse button at Step 4.

Static Stripping

Static stripping is an analytic technique for eliminating the frequency-independent offset of one apparent resistivity curve from the other.

To Perform Static Stripping on an Apparent Resistivity Curve:

1. Select the **Edit** command on the main menu.
2. Click the Strip button.

3. Click a data point on the curve and release the mouse button.
4. Move the curve to the desired location and click the left mouse button.
5. Click the Strip button again.

Normally, you would select a point at the high-frequency end of the curve. Practical experience suggests that most static shift cases result in one curve being moved down relative to the other. You therefore move the lower curve up to the higher one.

When the original apparent resistivity curve is dragged to its new position, a scalar multiplier of the e-field value is derived. This is applied to a recalculation of all the impedances of that sounding. Thus, when you click **Strip** for the second time, you should see the hour-glass symbol, indicating that this calculation is taking place. The curves are then redrawn. Only the apparent resistivity curves are affected by this procedure.

If the data show a very large static shift, or are strongly 3D at high frequencies, the results can sometimes be unstable. You may need to repeat the operation several times.

Static Shifting

Static shifting allows you to vertically shift all data points of a curve.

To Statically Shift a Curve:

1. Select the window with the sounding to edit
2. Select the **Edit** command on the main menu.
3. Click the S-Shift button.
4. Click one of the curve's data points and release the button.
5. Using the mouse, move the curve to the desired location. The amount of static shifting is displayed in the bottom of the window:
   Shift xy=1.002 Shift yx=0.319
   This number is a scalar factor that multiplies the original resistivity values of the curve. If equal to 1, the curve is not shifted.
6. Click and release the left mouse button.

The apparent resistivity range changes so that the curves appear centered in the display box.

A useful guide for determining the resistivity level to which a curve should be shifted is obtained by displaying the pseudo-MT curve for a co-located TDEM station.

To Reset the Static Shift to Zero:

1. Click the SS Reset button.

2. 

to run Smoothing routines

to add/subtract 360° to phase data points

to show pseudo-MT curve from TDEM model

Calculating Smoothed Curves

Resistivity and phase smoothed curves can be calculated from the edited curves using three different techniques. They are automatically updated whenever further editing is made on the curves.

To (Re)calculate a Station's Smoothed Curves:

1. Open the station or select its window if already open.
2. Select the **Edit** command on the main menu.
3. Click the 

button.
4. Select **Smooth Options** and click the **OK** button.

The available smoothing options are described below

Sutarno Phase Consistent Smoothing

This is an application of the Hilbert transform to give an apparent resistivity curve from the phase curve. (Sutarno, D. and Vozoff, K., 1991, Phase-smoothed robust M-estimation of magnetotelluric impedance functions: Geophysics, 56, 1999-2007). It is used primarily to confirm that apparent resistivity and phase are consistent.

D+ Smoothing

D+ relates apparent resistivity and phase of the same component (xy or yx) through a D+ function. In essence, this finds the one-dimensional earth which best fits both parameters. The procedure has been shown to be valid for most 2D data and for some 3D cases.

The errors attributed to the data can be those estimated by the original data processing or can be imposed by the user. In the first case, the two parameters (apparent resistivity and phase errors) are left at the default value of -1. In the second case, the user can estimate the errors (e.g. 10%). By appropriately selecting errors, one parameter can be down-weighted at the expense of the other. *Reference*: Beamish, D., and Travassos, J.M., 1992, The use of the D+ solution in magnetotelluric interpretation. Jo. Appl. Geophys., 29, 1-19.

Numerical Smoothing

Numerical smoothing is an FFT-based low-pass filter which calculates an independent smooth curve for each of the four components. It does not have an underlying geophysical process to support it, but may nevertheless be useful in certain noisy situations. The user selects a smoothing factor which gives an appropriate result. 0 gives no smoothing at all, and useful values are typically << (half the number of data points).

Reference: Press et al., 1989. Numerical Recipes, *pub.* Cambridge University Press, Section 13.9.

As is described in the “MT Soundings” and “MT 2D Inversion” chapters of this manual, smooth curves created while in Database Mode can be used as the data source for 2D MT modeling.

Adding ± 360° to Phase

This feature is useful in the case of phase curves which wrap around the normal -180 to +180 range:

To Add 360° to Data Points of a Phase Curve:

1. Select the **Edit** command from the main menu.
2. Click the [+ 360°] button.
3. Select the data points of the phase curve you want to shift.
4. Click the [+ 360°] button again.

To Subtract 360° from Data Points of a Phase Curve:

1. Select the **Edit** command from the main menu.
2. Click the [- 360°] button.
3. Select the data points of the phase curve you want to shift.
4. Click the [- 360°] button again.

Batch Commands

Most of the edit commands available in the standard operating mode of the Data Analysis program can also be performed as a batch operation on a set of EDI files. While it is not possible to achieve the same level of accuracy for some commands as is possible in actual Edit mode (e.g. masking), batch operation considerably accelerates initial editing.

The following functions are available in batch mode:

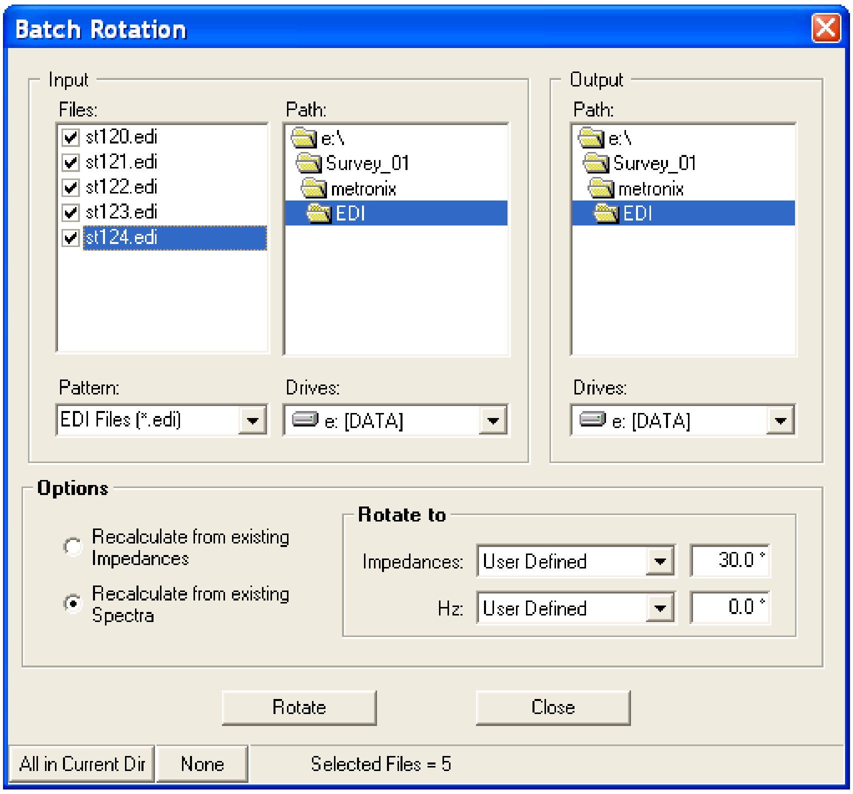
- Printing
- Rotating
- Smoothing
- Masking
- Static stripping
- Static shifting

To Access the Batch Commands:

1. Close all open stations.
2. The **Batch Tools** command appears on the main menu.
3. Select the desired operation.

4. Enter the parameters as requested.

The parameters requested in the dialog windows for the individual batch functions are largely self-explanatory. Generally, these require the user to specify the files to which the batch function is to be applied and any function-specific parameters. Shown below as an example is the window for Batch Rotation:



Note: By default, the output path is identical to the input path. If you do not wish to overwrite the original data, you must define a different output path.

8: EDI Utilities MT

Overview

WinGLink provides a number of tools for working with EDI files:

- Convert to EDI
- Split multi-site EDI files
- Combine EDI
- Dump EDI to text files

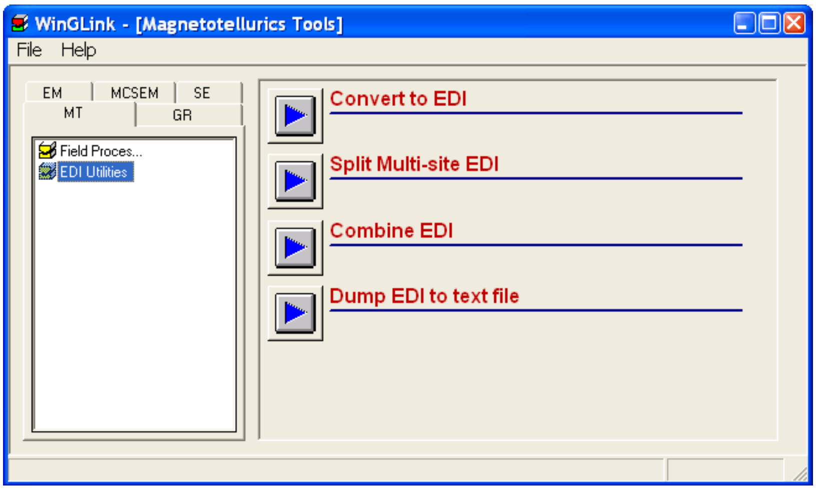
These tools are frequently needed when working with data which have been output by system or processing software to a non-EDI format.

This chapter will cover the use of each of these tools.

Getting Started With the EDI Tools

Opening the EDI Tools

After opening WinGLink in Tool Mode, select **MT** in the left panel of the tool form and select **EDI Utilities** in the left frame. The individual EDI tools can then be selected by clicking the button to the left of each:



Note: For information on starting WinGLink in Tool Mode, refer to the "Getting Started With WinGLink" chapter of this manual.

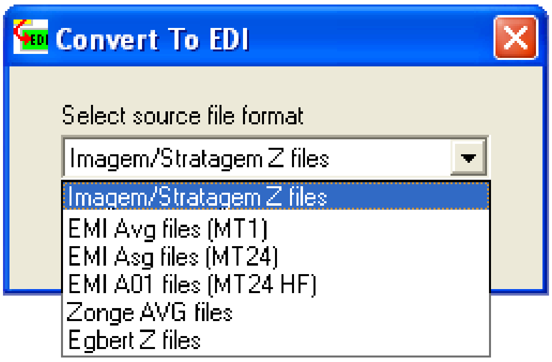
Convert to EDI

The Convert to EDI program provides functionality for converting the following file types to EDI format:

- Imagem/Stratagem Z files
- EMI AVG files (both MT1 and MT24)
- EMI A01 files (MT 24 HF)
- Zonge AVG files
- Egbert Z files

Converting Files to EDI Format

After clicking the button to the left of Convert to EDI in the Tools window, the program opens with the following window:

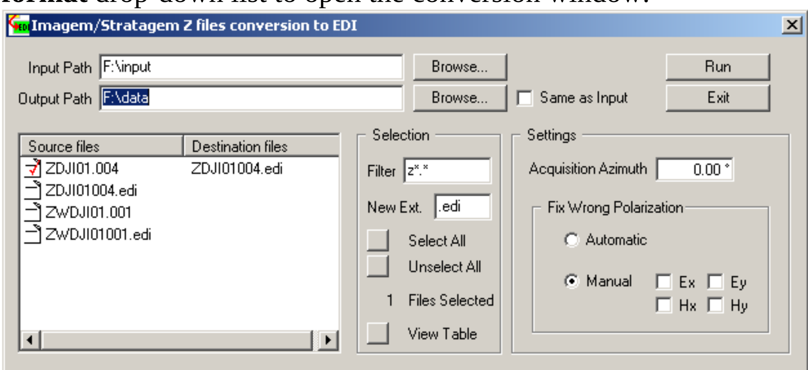


Associated with each of the file formats is a dialog window which takes into account the specific characteristics of the selected file format, e.g. channel mapping. Common to all of the dialog boxes are fields for specifying input path and the output path.

The general procedure for converting to EDI is outlined below using the Imagem/Stratagem interface as an example. While there are some differences between this dialog and those provided for the other data types, they are largely self-explanatory.

Converting Imagem/Stratagem Files to EDI:

1. In the Convert to EDI selection window, select the **Imagem/Stratagem Z files** option from the **Select source file format** drop-down list to open the conversion window:



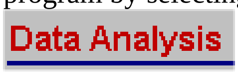
2. Use the **Browse** buttons in the **Input Path** and **Output Path** fields to set the respective paths for data input and output. As

an alternative to specifying the output path, the **Same as Input path** checkbox can be selected. In this case, the output path is automatically set to the same path as that used for the input.

3. After selecting the input path, all files matching the filter string specified in the **Selection area** of the dialog window are listed for selection in the left box under the **Source files** column.
4. Use the mouse to select the z*.* files you wish to convert. For multiple selections, hold down the <SHIFT> or the <CTRL> keys on the keyboard while clicking the file names with the mouse.
5. A red symbol appears to the left of the selected file names.

   The equivalent output files, i.e. with the EDI extension, are shown to the right under the **Destination files** column. The names of the destination files can be edited by clicking. After editing, confirm the new name with **<Enter>**. For a global view of selected input and output files, click **View Table**
6. In the **Settings** section, enter the acquisition azimuth.
7. In the **Fix Wrong Polarization** section of the window**,** select any channel that might have incorrect acquisition polarization (inverted connectors). Alternatively, you may select the **Automatic** option, and the conversion program will automatically correct the polarization if it is incorrect.

   Note: The manual option should only be used to fix field settings known to be bad.

   Note also that options in the **Settings** area are specific to Imagem/Stratagem files. Different settings are provided for other file types.
8. Click the **Run** button to start the conversion.
9. When finished, exit the program to return to the main **Tools** menu.
10. The output EDI files can now be opened in the Data Analysis program by selecting the **Field Processing |**
    

    option in MT Tools for a quick check of the converted files or imported into a WinGLink database.

Note: Source files may or may not contain station coordinate information, e.g. due to acquisition system, processing software. In this case, the output EDI files will not contain any information on the site location. Site coordinate information must then be imported or added after importing the EDI files by either importing a coordinate file or manually editing the coordinate information in the Maps program.

Split Multi-Site/EDI Files

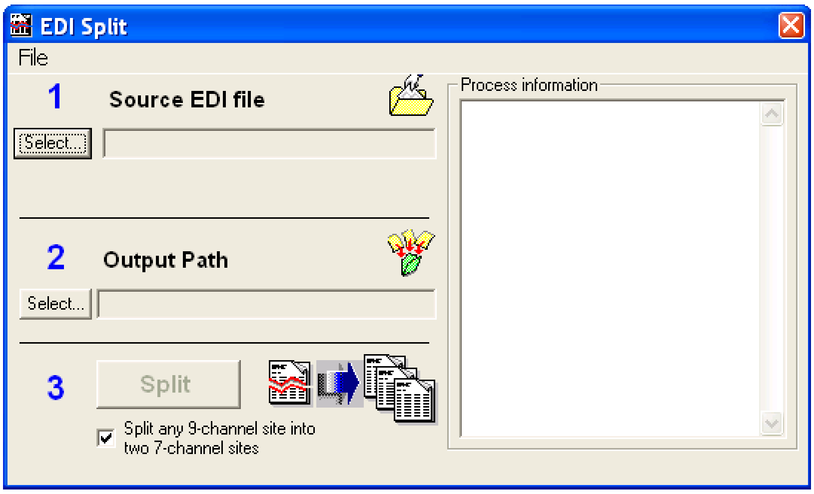
This tool is used to split:

- 9-channel EDI files into two 7-channel EDI files

- Multi-site EDI files into single-site EDI files
- Multi-site, 9-channel EDI files into single, 7-channel EDI files

Splitting Multi-Site/EDI Files

After clicking the button to the left of Split Multi-Site/EDI Files in the Tools window, the program opens with the following window:



1. Select the source file and output path in the provided fields
2. Click the Split button split the files.

The output files are assigned the name of the source multi-site file plus a numeric suffix in ascending order.

Example:

File ASD.EDI, containing 4 sites, will be split into the following 4 EDI files:

ASD_1.EDI, ASD_2.EDI, ASD_3.EDI, ASD_4.EDI

Combine EDI

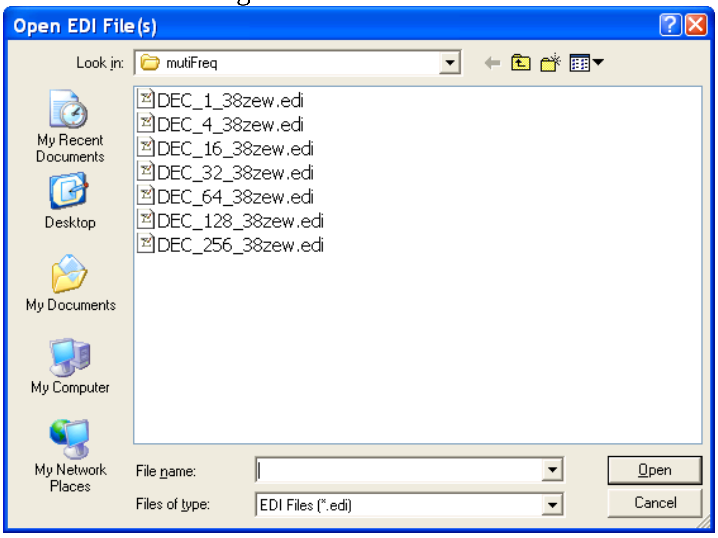
The Combine EDI tool merges EDI files containing data at different frequencies for the same MT site. Such data may result from data acquisition, data decimation or data processing. In this tool, each component EDI file can be viewed, edited with frequency masking, and eventually merged with other EDI files into output a single, consolidated EDI file. The program interface includes functionality for displaying coherencies or tipper magnitude.

Interactive, self-explanatory commands control the inclusion/exclusion of each component EDI file from the set of files to be merged.

Combining EDI Files

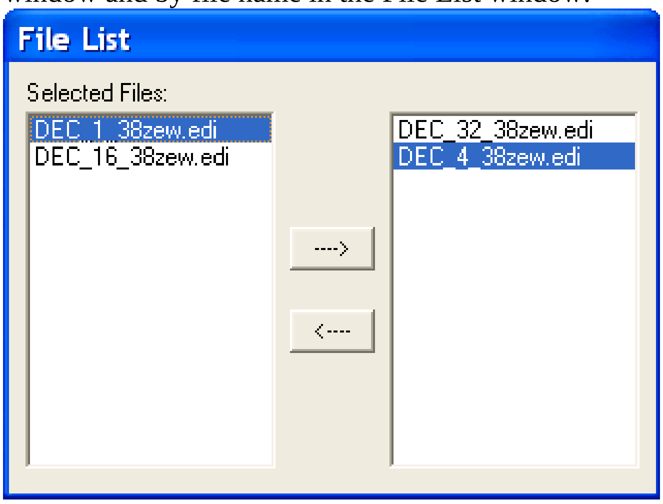
1. Click the button to the left of Combine EDI in the Tools window to open the Combine EDI interface.

2. To begin work with the tool, select the **File | Open** menu command and browse to the directory which contains the files which are to be merged:



Use the <SHIFT> and <CONTROL> keys to select the desired files. Note that you must select at least two files. All of the files to be merged must be selected in a single operation. Each time files are selected from the Open EDI Files window, any open files are closed. Click **Open** to open the selected files in the Combine EDI interface.

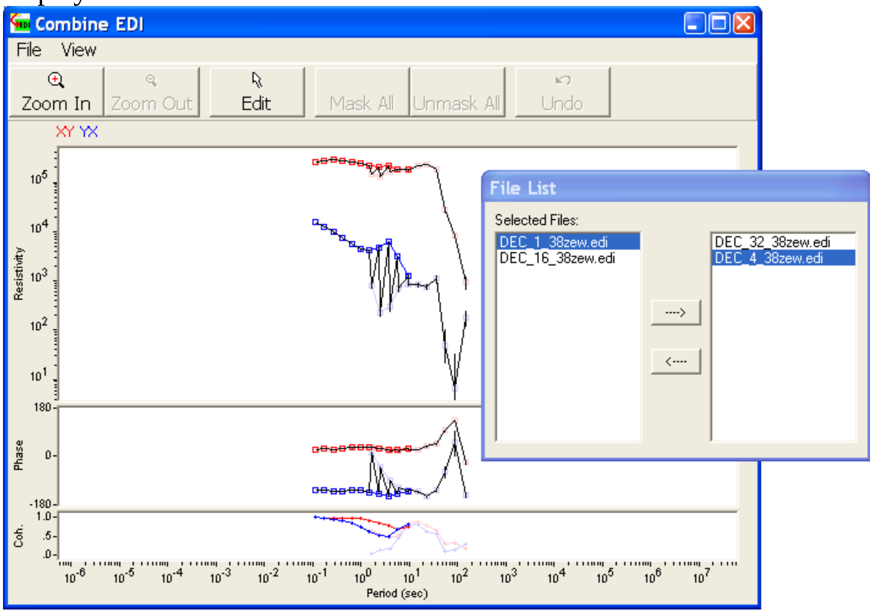
3. The selected files are now displayed, both as curves in the main window and by file name in the File List window:



4. Curves associated with the files on the left side of the screen are plotted in the main part of the window and can be edited (data points masked) using the edit tools provided at the top of the main window. Furthermore, files present on the left side of the window are included in the merged file. Files on the right are neither displayed nor included in the merged file. While editing files, it is often useful to move some of the files to the right-hand column with the arrow buttons to provide a better view of the files being edited.

5. The file selected in the left-hand window is the file to which edit operations are applied. Data points for this file are

displayed in color:



6. To edit, i.e. mask data points, click the Edit button to enter edit mode and click any of the data points associated with the frequency to be masked. Masked points can be reactivated by again clicking a data point at the given frequency. The **Mask All** and **Unmask All** buttons can be used to mask or unmask all data points associated with the current file.

   To assist in the editing process, the merged XY and YX curves resulting from the files present on the left side of the File List window are, by default merged in real time. This function can be disabled with the **View | Show Stacked Curves** menu command.

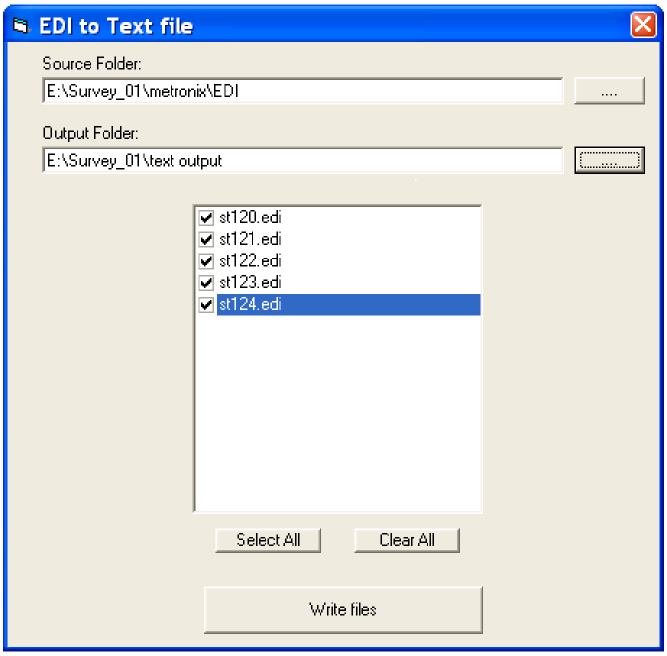
   Files, once edited, can be moved to the right side of the File List window and unedited files moved to the left.

7. After making the desired edits, move all files to be included in the merged EDI to the left side of the File List window. Execute the **File | Save** as menu command to save the merged file to a new file.

Dump EDI to Text File

Largely self-explanatory, this tool is used to output impedance values to columnar text files. Output includes frequency and real and imaginary components of impedance tensors and tipper as well as the respective variances.

Select arrow to the left of the **Dump EDI to Text File** entry in the Tools window to open the program interface:



Enter the source and output folders, select the EDI files to be output as text files and then click the **Write files** button to perform the conversion.

9: TemMerge ^{TDEM}

Overview: The TemMerge Program

The TemMerge program is used to convert raw, time-domain electromagnetic data for most standard acquisition systems into a common data exchange format. The program also includes basic editing functionality and can be used to merge multiple soundings onto a single curve.

TemMerge supports the following input formats:

- Geonics: PROTEM (TEM58, TEM57, TEM47 & TEM37 receivers)

- Sirotem: SIROTEM3 (SIROTEM2 ***not*** supported)

- Zonge: GDP-16/32

- Amira format

- Free format (space-, comma-, semicolon- or tab-delimited columnar data)

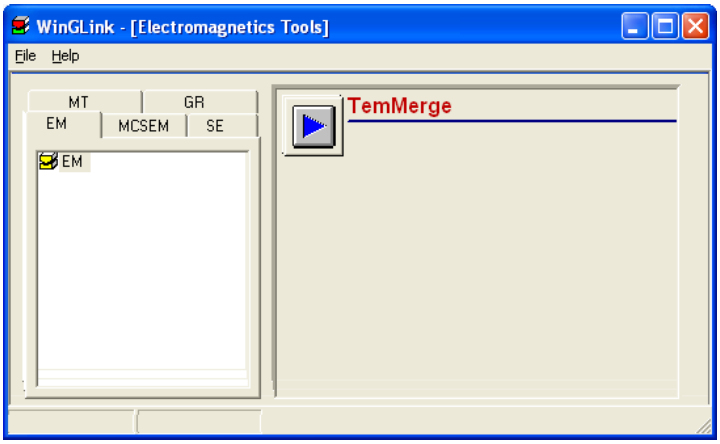
The imported and merged data can be output to ASCII files in the TemMerge format (*.TEM). The output files contain station and system information, window times (corrected for system delays), voltage values and apparent resistivity values.

Getting Started

This section provides an overview of the steps necessary to import a standard system dump file into TemMerge and merge multiple soundings onto a single curve.

Starting the Program

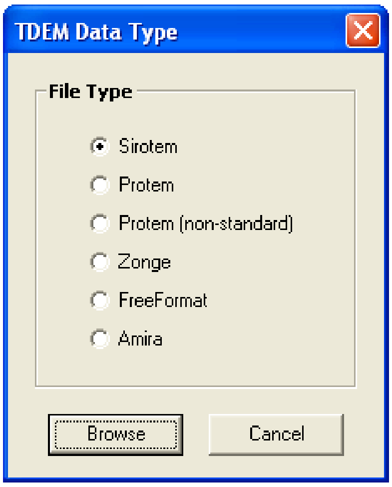
After opening WinGLink in tool mode, select **EM** in the left panel of the tool form, select **EM** in the left frame, and click the arrow to the left of TemMerge.



Note: For information on starting WinGLink in Tool Mode, refer to the "Getting Started with WinGLink" chapter of this manual.

Importing System Dump Files

1. After starting TemMerge, the TDEM Data Type dialog box opens. Select the appropriate file type and click the **Browse** button:



2. TemMerge now prompts you to select the powerline frequency at the acquisition site. The value selected here (50 or 60 Hz) affects the repetition rates for Sirotem and Protem data. The repetition rate for Zonge data is included in the file header, i.e. the value selected here does not affect the resistivity calculation for Zonge data.
3. Browse the directories in the Open File(s) dialog box and select the file to be opened.

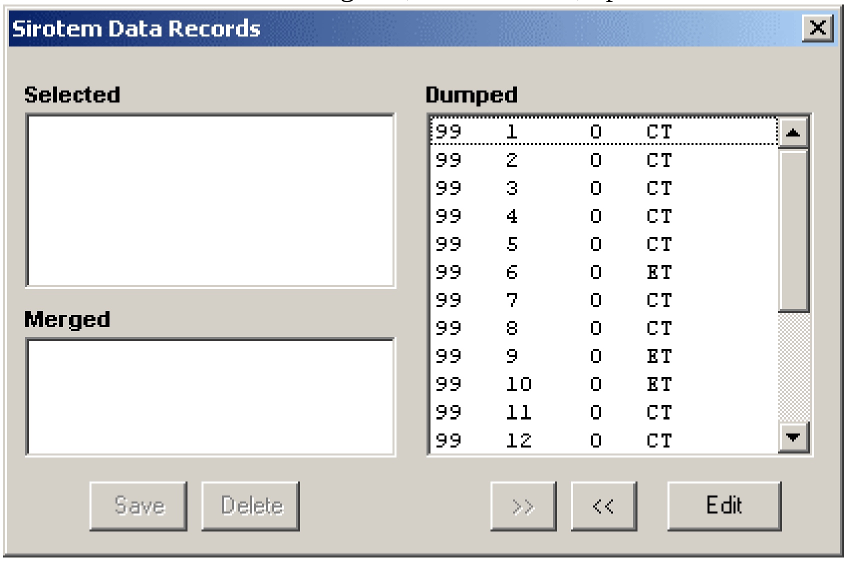
Note: The non-standard Protem format was implemented at the request of a small group of users and does not reflect the format output by the Protem system. For details, please contact Geosystem.

Importing Free-Format Text Files

Text files containing time, voltage and resistivity values in columnar form can be imported into TemMerge. The data values on each line of data must be delimited by a space, comma, semicolon or tab character. While this import option is a convenient way to import the occasional text file into WinGLink (first by converting to a TEM file in TemMerge, and then subsequently importing the TEM file into WinGLink), processing a large number of text files is time-intensive due to the fact that the text files do not contain any system information, e.g. loop configuration and sizes, delays etc. This information must be entered manually in the Edit window (see below).

The Data Records Selection Dialog Box

After selecting a file of any type (with the exception of FreeFormat), the Data Records selection dialog box, shown below, opens.



This dialog box is the heart of the TemMerge program. Here, you can edit system data for individual soundings, select the files to be merged, and save and delete merged files. All soundings associated with a file remain available for selection as long as the Data Records dialog box is open. If the dialog box is closed, the file is removed from memory. Files can be selected for import by selecting the **File | Import** menu command.

The Data Records dialog box contains three panels and five buttons:

All soundings contained in the dump file are listed in the **Dumped** panel.

The Data Records dialog box shown above contains a set of Sirotem data. The records in the dump file are listed in the Dumped panel of the dialog box. Although in principle the same, the individual data records for the various system types are labeled slightly different as outlined below:

Sirotem:		**Group#**	**Run #**	**Gain**	**Time series**
Geonics:		**Run#**	**Line***	**Station**	**Time series**
Zonge:	**Run#**	**Time****	**Group#**	**Rep. Rate**	
Amira:	**Run#**	**Line***	**Station**	**Time series**	

*Line = logger record # at line start
**Time = measurement time

Soundings which have been selected for merging are listed in the **Selected** panel. All soundings listed in the **Selected** panel are plotted as voltage or resistivity vs. time in the main TemMerge window as blue curves. The window supports multiple sounding curves.

Highlight one of the soundings listed in the **Dumped** panel and click the Edit button to open the **Edit** dialog box. Here, receiver, transmitter and timing information can be set and individual measurement values edited. The Edit dialog box is described in the **Editing soundings** section of this chapter.

The merged soundings are listed in the **Merged** panel. Merged soundings are plotted in the main TemMerge window as red curves. The window supports multiple merged sounding curves.

The left << and right >> arrow buttons are used to move soundings between the **Dumped** and **Selected** panels. Note that the right button is enabled only when soundings are listed in the **Selected** panel.

Use the Save button to save a merged curve to a *.TEM file (TemMerge exchange format) and the Delete button to remove a merged curve from the **Merged** panel. These two buttons are enabled only when merged curves are present in the **Merged** panel.

Selecting and Merging Soundings

Use the left and right arrow buttons as described above to move soundings between the **Dumped** and **Selected** panels of the Data Records dialog box. Soundings listed in the **Selected** panel are plotted in the main TemMerge window.

Select individual soundings for merging in the **Selected** panel by clicking soundings with the mouse. Multiple soundings can be selected by clicking the soundings while holding down the **Ctrl** key on they keyboard.

Provided the soundings were recorded with the *same repetition rate* and *delay*, the soundings can be merged into a single curve using either the **Merge** button or the commands available on the **Merge** menu:

Only if the repetition rates and delays are the same for all selected soundings is the **Merge** button enabled.

TemMerge does provide special functionality for Geonics/Protem soundings, which may be recorded at up to three different sampling rates. Soundings sampled at each of the sampling rates can be merged into single curves. The delays for each of the soundings must, however, be the same.

Note: Delay times and repetition rates can be edited in the Data Record dialog box, described in the “Editing Sounding Records” section of this chapter.

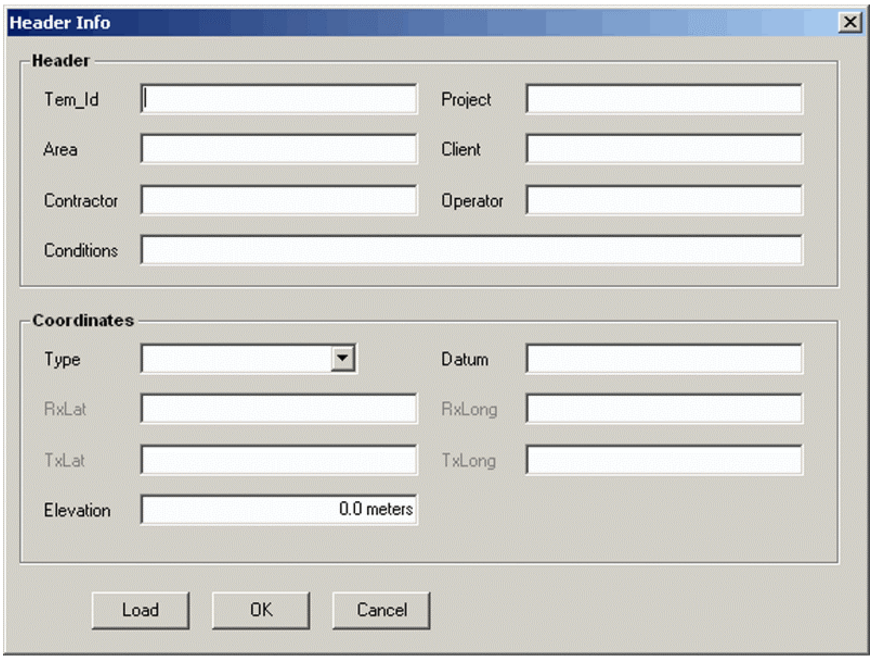
As seen on the **Merge** menu above, two merging algorithms are available: **Average** and **Polynomial**. The selection made on the **Merge** menu applies to merge operations performed with the **Merge** button. Note that when using the polynomial merge function, if a negative voltage is present at a given gate time in any of the soundings being merged, that point is omitted.

Different combinations of soundings can be selected and merged in a single session. The merged soundings can then each be saved to individual TEM files using the **Save** button, as described below in the "Saving Merged Soundings" section of this chapter.

Saving Merged Soundings

Merged soundings are saved to TemMerge (TEM) files in a two-step process. First, header information is entered in a dialog box, then a path and file name are specified.

After merging a sounding (or multiple soundings for the case of Geonics/Protem data), click the **Save** button in the Data Records dialog box to open the Header Info dialog box:



The information entered in the **Header** area of the window is stored in the header block of the TEM file. These fields are intended to provide an area for saving station information and, with the exception of the Tem_Id field, are not used within WinGLink. The Tem_Id is used as the station identifier in WinGLink, and is thus a mandatory field.

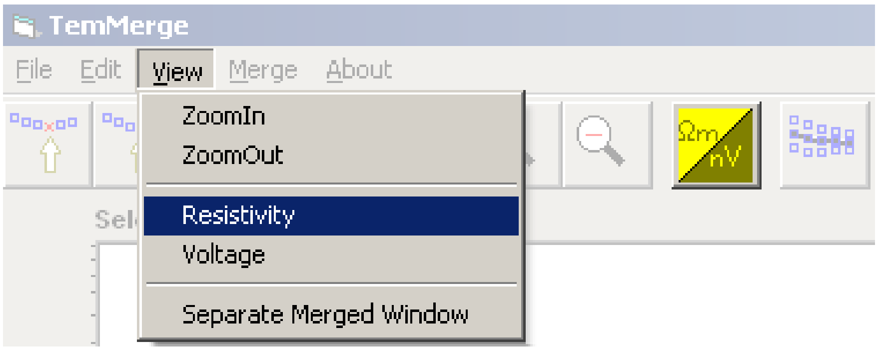
The fields in the **Coordinates** area of the window are used to specify station coordinates after the file has been imported into WinGLink. Note that stations coordinates stored in the metric format are imported into WinGLink using the central meridian specified for the database. Thus, when using metric coordinates, make certain that the central meridian to which the station coordinates are mapped is the same as that used in the WinGLink database.

Header information stored in existing TEM files can be automatically inserted into the fields in the Header Info window by selecting a file with the **Load** button.

After completing the fields, click the **OK** button to open the standard Windows dialog box for saving files.

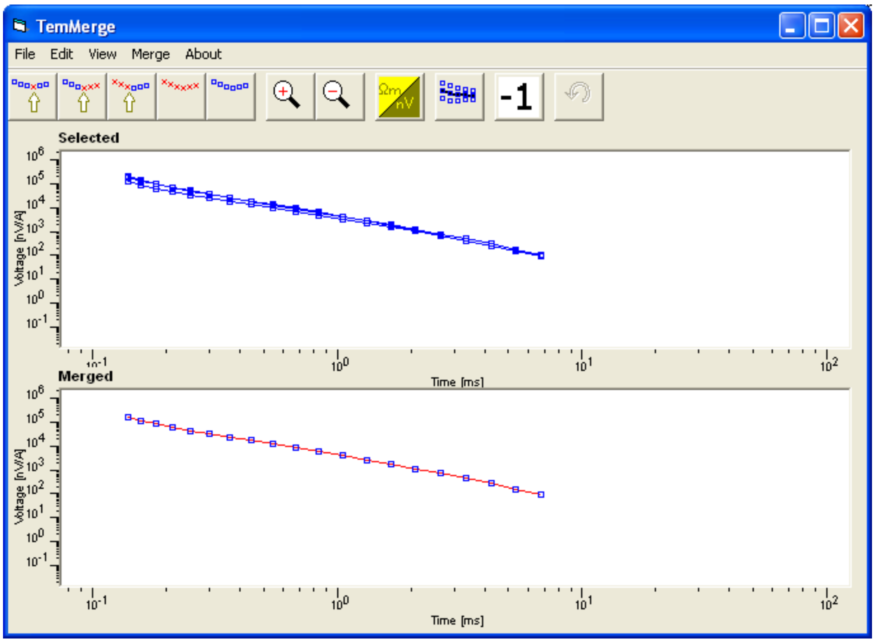
Display Options

By default, all soundings, both individual and merged, are displayed together in the workspace as Voltage [nV/A] vs. Time [ms]. Using the menu items and button shown below, you can toggle the display between voltage and resistivity:



Voltage values displayed as a red square represent negative voltage values. The resistivity at negative voltage points is zero.

Select the **Separate Merged Window** menu command to display the individual curves and merged curve in separate panels:

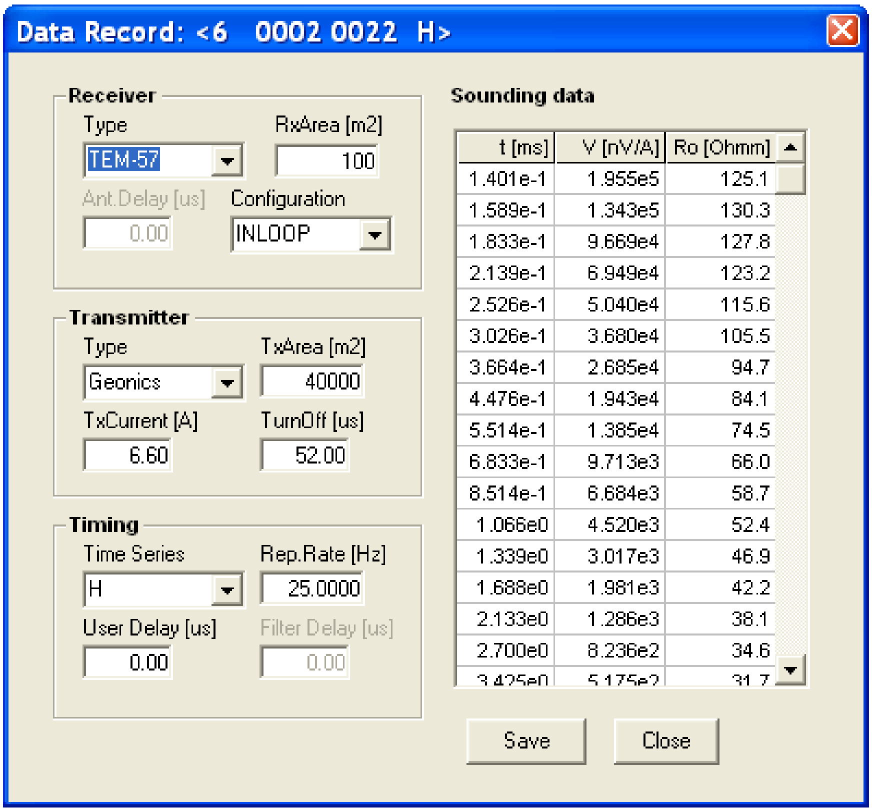


Editing

TemMerge offers two different functions for editing the raw sounding data: acquisition parameters can be edited via a dialog box and data points can be masked in the sounding display window.

Editing Sounding Records

As already mentioned, click the **Edit** button located at the bottom of the Data Records selection dialog box to open an edit dialog in which all sounding parameters can be viewed and edited:



On the left half of the dialog box, acquisition system parameters can be edited. Changes to parameters which are used in the calculation of the resistivity are immediately reflected in the resistivity values displayed in the right half of the dialog box. Note that resistivities are calculated using the Sirotem late time formula shown below:

$$r = 6.32 \times 10^{-12} \times A^{2/3} \times b^{4/3} \times (V/I)^{-2/3} \times t^{-5/3}$$

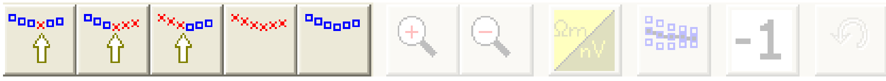
where:

r	=	apparent resistivity
A	=	effective loop area (turns area product) of the receiver loop or coil
b	=	transmitter loop side length (meters)
V/I	=	transient response (volt/amp)
t	=	delay time from start of transient decay (secs)

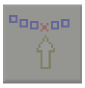
Note: The time, voltage and resistivity values displayed on the right are not editable.

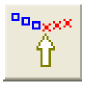
Editing Sounding Curves

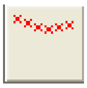
Once a sounding has been selected and moved to the **Selected** panel of the Data Records dialog box, several masking functions can be used to mask/unmask individual data points from the sounding. Masked points are not included in either the merged sounding or in the .TEM file output by TemMerge. The masking functions can be accessed either via the five buttons at the left end of the toolbar or on the **Edit** menu:

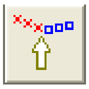


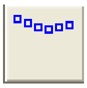
From left to right, the masking buttons serve the following functions:

 Mask individual points

 Mask all points to the end of the curve

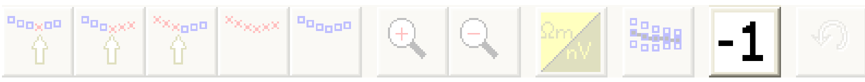
 Mask all points

 Mask all points to the start of the curve

 Unmask all points

Polarity Inversion

Use the **–1** button (or the **Edit | Invert** menu command) to invert the polarity of all points in the active sounding:



Timing

TemMerge corrects the raw times contained in the system dump files to take into account any system, filter, antenna or user delays. As a result, the first time in the .TEM files output by TemMerge is that of the first window. All other times are adjusted relative to this. TemMerge performs this calculation as follows for the respective system types:

Sirotem:

t(i) = t0(i) + [USER_DELAY]

Geonics/Protem:

t(i) = [TX turnoff time setting on receiver] + [USER_DELAY] + t0(i)

Zonge:

t(i) = t0(i) + [T/O time setting on receiver]

Output Format

TemMerge outputs merged files in a data exchange format defined by Geosystem: TEM. This format contains all information in the original dump files which is necessary for modeling in the WinGLink Soundings module. A sample .TEM file is shown below:

```
<HEADER INFO>
TEM_ID        CD_0023
PROJECT       Demo
AREA          B21
CLIENT        Geosystem
CONTRACTOR    Geosystem
OPERATOR      Geosystem
CONDITIONS    Sunny

<COORDINATES>
TYPE          LATLONG
DATUM
RXLAT           47°09'11.0000"
RXLONG          47°09'11.0000"
TXLAT          -88°45'12.0000"
TXLONG         -88°45'12.0000"
ELEVATION     290

<BLOCK>
<ACQUISITION PARAMETERS>

ACQUISITION_DATE            05-05-99
ACQUISITION_TIME            09:32
RECEIVER      Sirotem Mk3
TRANSMITTER   Sirotem
CONFIGURATION COINCIDENT
TXAREA        40000.0       # [m2]
RXAREA        40000.0       # [m2]
CURRENT       1.0           # [A]
REPTRATE      2.3           # [Hz]
TIMESERIES    CT
TURNOFF       0.0           # [us]
ANTENNA_DELAY 0.0           # [us]
FILTER_DELAY  0.0           # [us]
USER_DELAY    0.0           # [us]


<PROCESSING PARAMETERS>
FILEDATE      5/5/2005
PROGRAM       TemMerge Version 1.2.3
FILEVERSION   TEM 1.2


<INPUT FILE INFO>
INFILE        E:\_VBProjects\Progs\TemMerge\Data\EGIPT.OUT

RUNS           2
# RunNum, GroupNum, Gain
RUN_1          4 99 0
RUN_2          5 99 0


<DATA>

# Reported time is calculated by the following formula:

# t(i) = t0(i) + [USER_DELAY]

 *** TIMES RELATIVE TO T(0) ***

# Time [ms], Voltage [nV/A], Resistivity [Ohmm], Std.Dev. [nV/A]

5.0000e-2     -1.7963e7     0.00          2.9208e7

1.0000e-1     6.5872e8      0.21          7.8078e8
```

1.5000e-1	2.6703e8	0.19	3.2793e8
2.0000e-1	1.5821e8	0.17	1.9690e8
2.7500e-1	8.8571e7	0.15	1.1277e8
3.7500e-1	4.7174e7	0.13	6.2310e7
4.7500e-1	2.8394e7	0.13	3.8721e7
5.7500e-1	1.8317e7	0.12	2.6025e7
7.2500e-1	1.0997e7	0.12	1.6273e7
9.2500e-1	6.1862e6	0.11	9.5540e6
1.1250e0	3.9933e6	0.11	6.2137e6
1.3250e0	2.6839e6	0.11	4.3393e6
1.6250e0	1.7604e6	0.10	2.6907e6
2.0250e0	1.0022e6	0.10	1.6024e6
2.4250e0	5.6799e5	0.11	9.6644e5
2.8250e0	3.7545e5	0.11	6.8625e5
3.4250e0	2.6578e5	0.10	3.9127e5
4.2250e0	1.7728e5	0.10	2.1098e5
5.0250e0	1.2051e5	0.09	1.4074e5
5.8250e0	7.3940e4	0.10	9.0270e4
7.0250e0	4.0139e4	0.11	4.9498e4
8.6250e0	2.3182e4	0.11	1.7494e4
1.0225e1	2.4420e4	0.08	3.1367e4
1.1825e1	2.2872e4	0.07	4.7107e4
1.4225e1	7.4475e2	0.49	2.0181e4
1.7425e1	5.2660e3	0.10	1.2591e4
2.0625e1	5.3985e3	0.07	2.2848e4
2.3825e1	7.8975e3	0.04	9.8473e3
2.8625e1	3.9615e3	0.05	1.4142e4
3.5025e1	2.2153e3	0.05	1.8663e4
4.1425e1	-3.6525e3	0.00	1.0762e4
4.7825e1	6.9018e3	0.01	8.8008e3
5.7425e1	-1.9174e3	0.00	1.6514e4
7.0225e1	6.2065e3	0.01	5.9221e3
8.3025e1	1.8623e3	0.01	6.2920e3
9.5825e1	4.8125e2	0.03	1.1549e4

<END FILE>

As with an MT EDI file, this format organizes information according to blocks. For the most part, the format is self-explanatory. Worthy of mention, however, is the <INPUT FILE INFO> block. Here, the system dump file is listed as are the number of runs and identification codes for the runs which were merged.

TEM files created using Zonge and Sirotem data may contain only one set of sounding data. TEM files created using Geonics data may, however, contain up to three sets: one for each of three different sounding bands. In this case, all blocks with the exception of <HEADER INFO> and <COORDINATES> are included for each of the bands, as different system parameters may be used at the various sampling rates.

Using TEM Files

TEM files exported from TemMerge can be imported into WinGLink projects of type EM. To facilitate the exchange of TEM files, they can be exported from WinGLink's EM Soundings module. Planned additions to the format include adding model information as well as edits made to soundings in WinGLink.

10: Gravity Tools

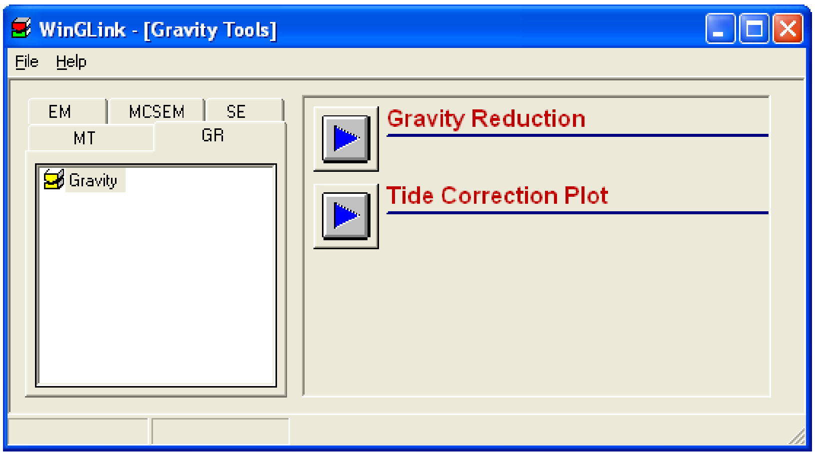
Overview

WinGLink provides two tools for use with gravity data: the **Gravity Reduction program**, which is used to compute observed gravity value at each station from instrument readings, and the **Tide Correction Plot program**, used to plot tidal correction at a given location over a user-defined period of time.

Getting Started With Gravity Tools

Opening Gravity Tools

After opening WinGLink in Tool Mode, select **GR** in the left panel of the tool form and select **Gravity** in the left frame. The two gravity tools can then be selected by clicking the button to the left of each:



Note: For information on starting WinGLink in Tool Mode, refer to the "Getting Started with WinGLink" chapter of this manual.

The Gravity Reduction Program

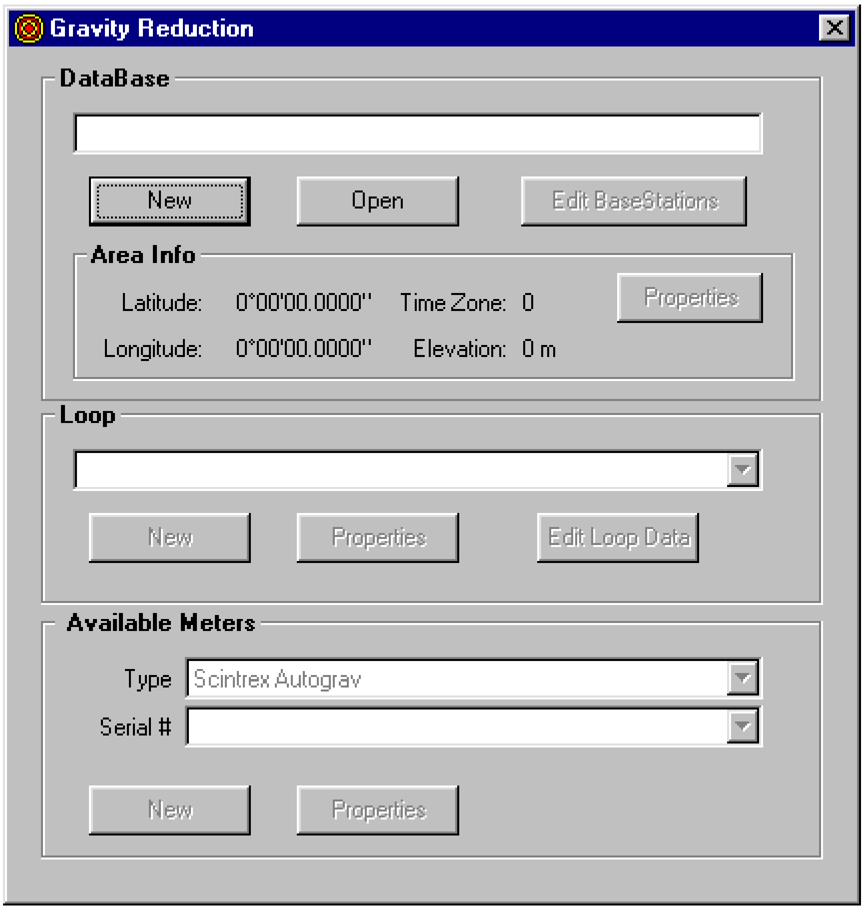
As already mentioned, the Gravity Reduction program computes the value of the observed gravity at each station from the instrument readings.

Use of this program involves the following steps:

1. Creation of a survey database (note: not a WinGLink database); entry of station names and gravity values in database.
2. Selection of a meter type
3. Creation of loop
4. Computation of gravity values
5. Export of gravity values for import into WinGLink

Creating a Survey Database

The main window of the Gravity Reduction program opens after selecting the program from the WinGLink Tools window:



In the **Database** frame, click **New** to create a database for your gravity survey. Databases are in Microsoft Access format and have the default extension *.mdb.

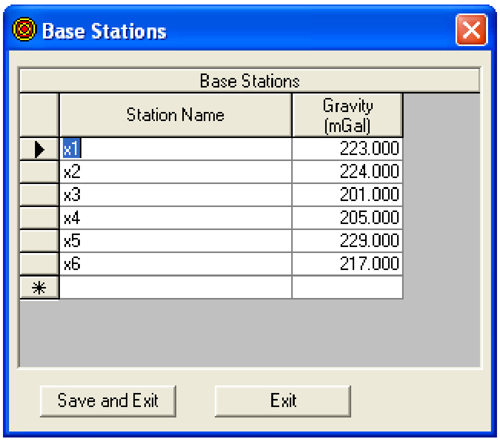
Note: The filename for the gravity reduction database **must not** contain more than 8 characters.

Entering Observed Gravity Values for Base Stations

1. **Create** or **Open** the Gravity Reduction database
2. In the **Area Info** section of the **Database** frame, click **Properties** and complete the field information as appropriate.

When finished, choose **Save and Exit.** The entered values are displayed in the frame for review.

3. Click the **"Edit Base Stations"** button:



4. Enter the name and the corresponding observed gravity value for each base station. When finished, click the **Save and Exit** button. The program will now recognize each base station by name.

Defining a Set of Available Meters

1. **Create** or **Open** the Gravity Reduction database
2. In the **Available Meters** frame, select the **Type** of the gravity meter to be added. Choose between Scintrex Autograv and Lacoste G-meter. Click **New.**
3. For **Scintrex Autograv** meters, enter the serial number of the meter.
4. For **LaCoste** meters, enter the serial number of the meter, the multiplying factor, and the calibration data. Follow the screen instructions to complete this input sequence. When finished, click **Save and Exit** to confirm and save the data.

Creating and Opening Loops

1. **Create** or **Open** the Gravity Reduction database
2. In the **Loop** frame, click **New** to add a new loop, then enter the loop name and select the gravity meter used for the loop from the list of available meters. Alternatively, you can select an existing loop.
3. Click **Properties** to edit the loop name and associated meter.
4. Click **Edit Loop Data** to enter or edit the **date, time and reading** for each station. Base stations are automatically detected by the program by name, and an **X** is displayed in the **Base** column for each base station.

Note: When working with the Scintrex Autograv meter, dump files (*.dmp) with raw data can be imported in the **Edit Loop Data spreadsheet** using the **Import** command on the **File** menu.

Computing Observed Gravity Values

1. **Create** or **Open** the gravity reduction database.
2. In the **Loop** frame, click the **Edit Loop Data** button and complete the input as appropriate.
3. On the main menu, click the **Compute** command.
   This command updates the contents of the spreadsheet columns with the computed data.

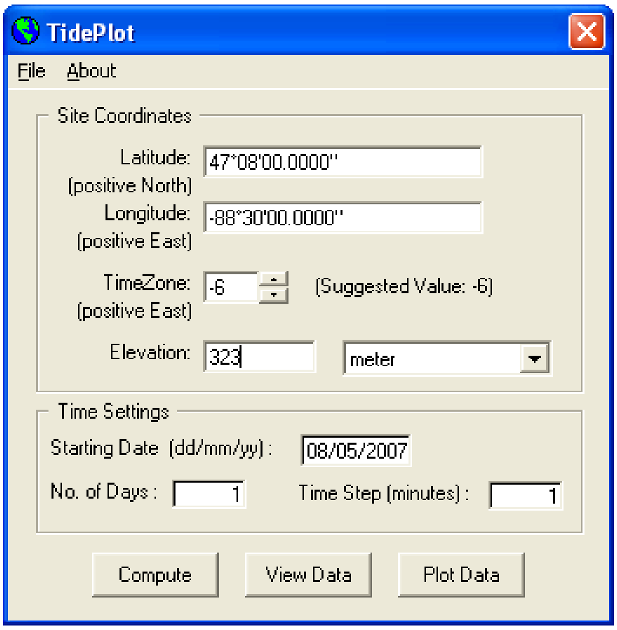
Exporting Observed Gravity Values

1. On the **File** menu, select **Export** to generate a text file with the following data for each station: **Name, date, time, drift correction, observed gravity.**
2. The first record of each exported file contains the type and serial number of the used meter. These files can then be imported into WinGLink. For further information, see the "Data Import" chapter of this manual.

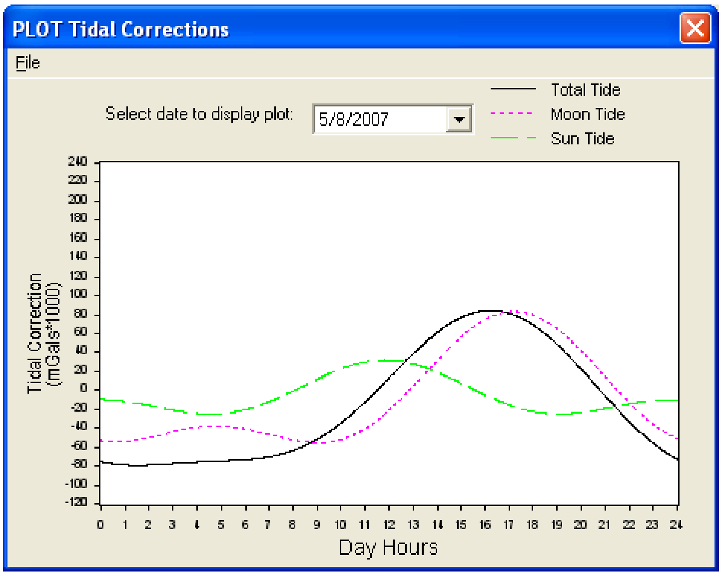
Plotting Tide Corrections

This simple utility calculates and plots tidal correction at a given location over a user-defined period of time.

Select the **Tide Correction Plot** item from the WinGLink Tools window to open the program:



After entering the requested data, use the buttons at the bottom of the window to compute the tide value and view the results:


PLOT Tidal Corrections
File
Select date to display plot:
5/8/2007
Total Tide
Moon Tide
Sun Tide
Tidal Correction
(mGals*1000)
Day Hours

11: The WinGLink Shell

Overview

This chapter is intended to familiarize WinGLink users with the WinGLink shell, the central interface used to create and configure WinGLink databases, create and manage projects, import data and to access all licensed program modules. From the shell, you can also switch from Database Mode, which contains the actual program modules, to Tool Mode, from which a suite of tools for processing, editing and analyzing data can be accessed.

Topics covered in this chapter include:

- An overview of databases and projects
- Starting the program
- Creating a database
- Setting database properties
- Sharing databases
- Creating projects

WinGLink Databases and Projects

WinGLink Databases

A WinGLink database is typically created for a given area of interest. All supported geophysical data, i.e. MT, TDEM, gravity etc., can be imported into a single database. Each type of data must be imported into a project of the appropriate type. The types of data which can be imported into a WinGLink database on a given computer are dependent on the license configuration.

Each database contains all data and information relative to the area of interest, including data and settings for the metric and geographic coordinate systems and the preferred linear units for distances and depths.

WinGLink Projects

A WinGLink project is a collection of stations. Stations may contain either observed or synthetic data.

The type of data acquired at the stations is the data type of the project.

Examples of projects:

- a magnetotelluric survey
- a gravity survey
- a collection of wells
- a group of towns

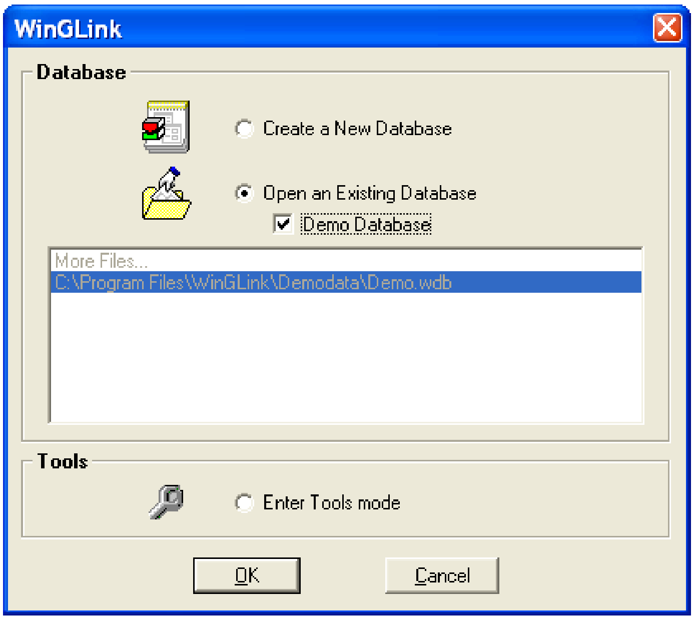
Projects are used to store data associated with surveys carried out in the database area. In cases where no observed data yet exists, i.e. in the case of a feasibility study, synthetic data can be generated in certain program modules and then used as observed data.

Stations are added to projects by entering the station name, coordinates and elevation. This may be performed either manually or by importing station data from external files.

As will be described later, WinGLink offers two types of project: single and integrated. As the name implies, single projects are handled as discrete data sets. Integrated projects, on the other hand, may consist of any number of single projects.

Using WinGLink in Database Mode

To use WinGLink in Database Mode, check one of the options in the **Database** frame of the initial dialog box, i.e. select either an existing databases, the demo database or create a new database**.** To get started with the program, open the demo database:



As soon as a database is loaded, the main Database Shell opens. This is divided into three panels:

Database Properties panel (left):

This provides information on the database coordinate system. This applies to all the stations of the projects contained in the database.

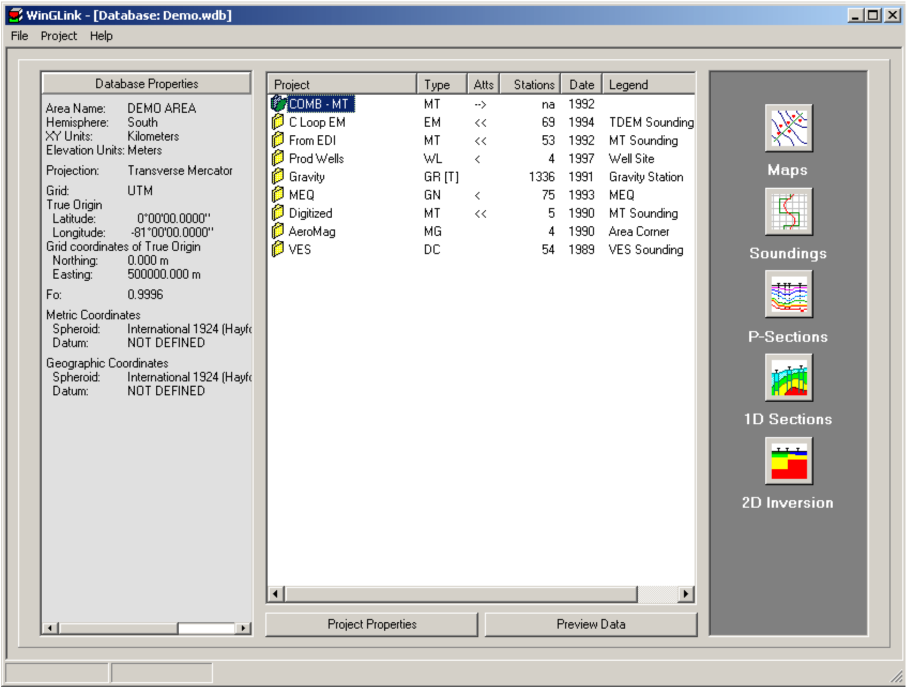
Project panel (center)

Projects available in the database are listed here. The first column lists the name given by the user to the project, the other columns provide online information for each project.

Program panel (right)

Contains icons of the application programs available for the selected project.

The Database Shell, as it appears for an MT project, is shown below:



To ***select*** a project, click the project name with the left mouse button. Once selected, (see the **COMB-MT** project in the example above) the project becomes the active project. This means that any application program launched in the right-hand panel will load the data of that project. In addition, only those program icons for the programs which are compatible with the respective project type are displayed in the right-hand panel.

WinGLink Databases

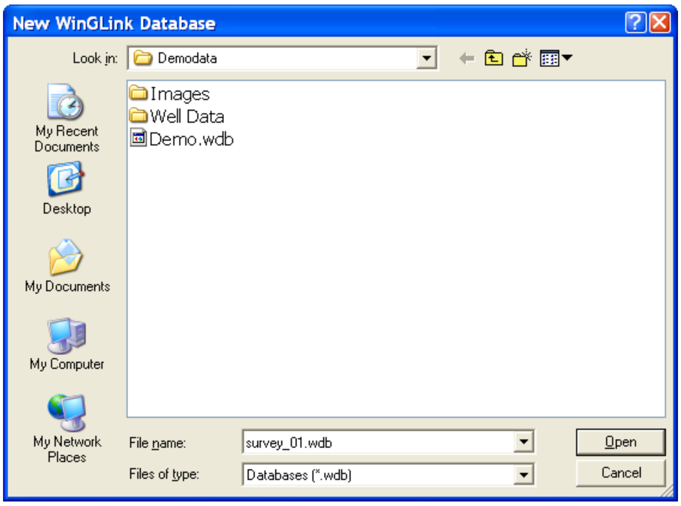
Creating Databases

Databases can be created either by selecting the **Create a New Database** option in the database selection dialog box, which opens when the program is first started, or with the **File | New Database…** menu command in the WinGLink shell.

Regardless of which method for database creation is selected, WinGLink opens a the first of a series of windows which guide you through a series of steps.

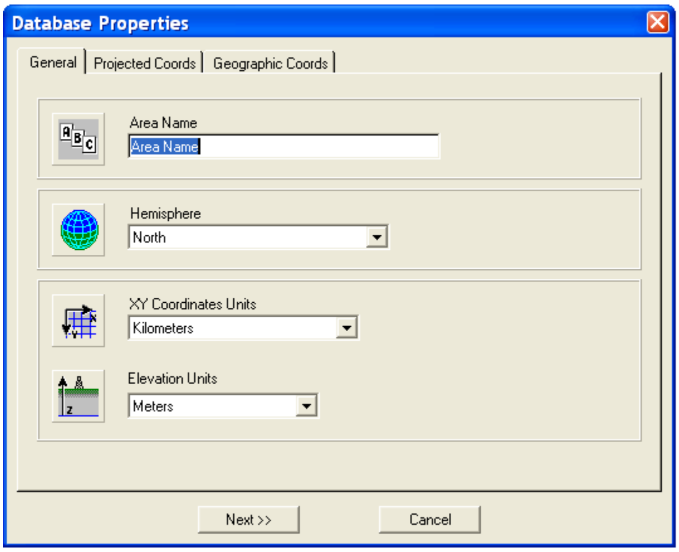
The Database Creation Wizard

1. After executing either of the two options for creating a new database, the New WinGLink Database window opens.

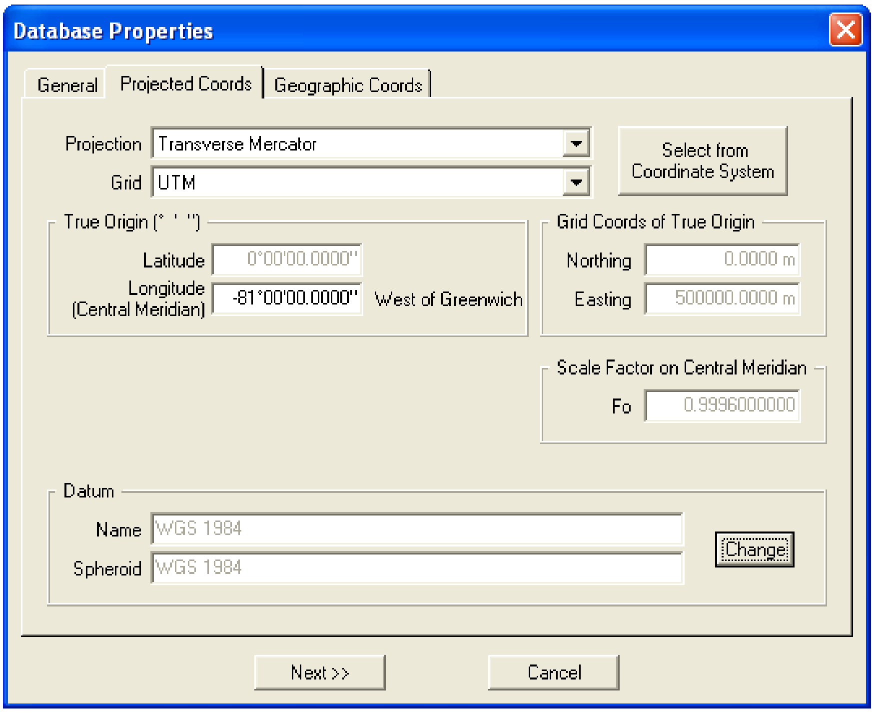


Specify the file name and path, and click the **Open** button:

2. The program now opens a window with three tabs in which all information specific to the database location is entered. On the first tab, enter the area name, and coordinate and elevation units:



3. Click the **Next** button to advance to the **Projected Coords.** tab of the window:

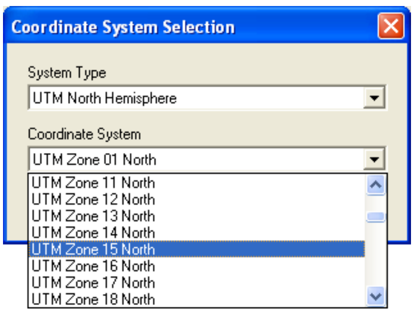


On this tab, specify the projection, grid, central meridian and datum used to project stations for Cartesian coordinates.

Note: The coordinate systems in a database are the same for all of the projects in the database. Changing the settings for a system will automatically update the coordinates for all of the projects in the database.

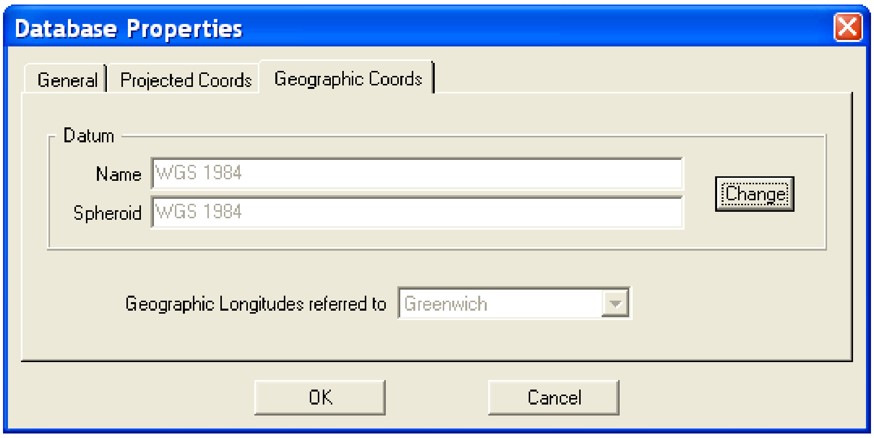
It is extremely important that the coordinate information, including central meridian, be set correctly for the location of the stations used within the database. Conversions between metric and geographic coordinates may otherwise not be accurate.

The UTM zone (and thus central meridian) can be easily set by clicking the **Select from Coordinate System** button to open the Coordinate System Selection dialog:

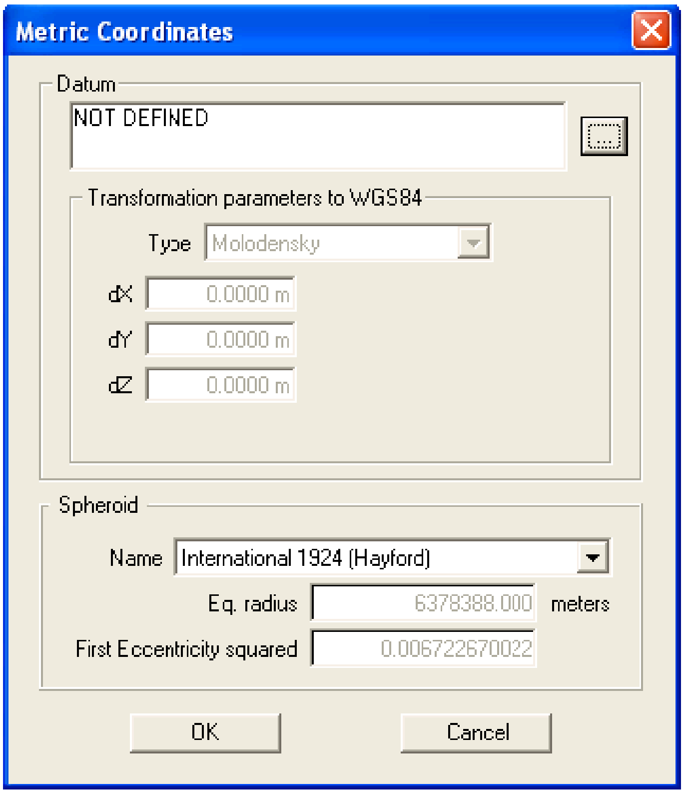


After making a selection, the central meridian for the given zone will automatically be entered in the respective field in the **Projected Coords** tab.

4. After entering the information for the projected coordinates, click **Next** to advance to the **Geographic Coords.** tab:



To make adjustments to the geographic coordinates, click the **Change** button to open the Geographic Coordinates window:



A list of all supported datums can be opened by clicking the **…** button at the right side of the **Datum** field.

5. Click the **OK** button in the Database Properties window to complete the creation process. WinGLink displays an info box alerting you that a project must first be created before you can begin work with the program:

Database-Specific Menu Commands

Databases are created, edited and saved using several commands available in the WinGLink Shell. Specifically:

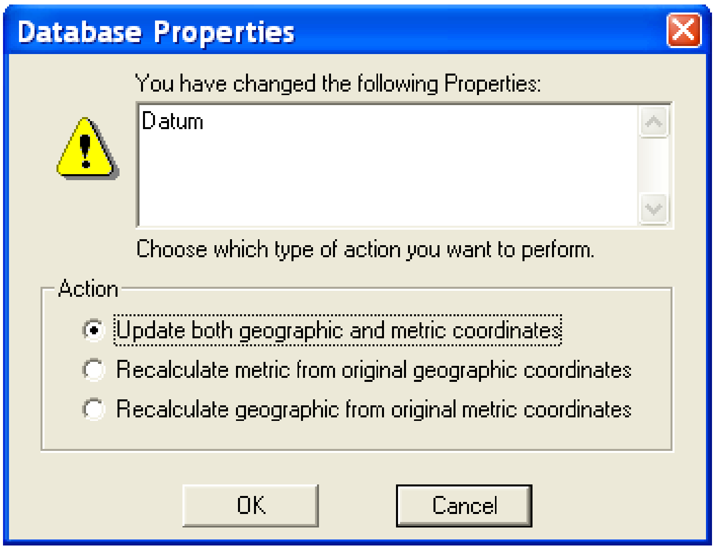
Command to use	for
File | New Database	Creating a new database in a user-selected disk directory. The properties of the new database are entered at this stage
File | Open Database	Opening an existing database
File | Close	Closing the open database with its original name and directory location
File | Save As	Closing the open database and saving it with a different name or directory location
File | Compact Database	Compacting the database; should be performed occasionally to optimize database structure and minimize size
File | Database Properties	Editing the properties of the active database

Fixing Wrong Settings in the Coordinate Systems

The geographic and metric coordinates for each station of a database are linked to each other based on the respective coordinate system settings. Each time a metric (or geographic) coordinate is changed, WinGLink appropriately updates the corresponding geographic (or metric) coordinates, and vice versa.

If the settings for one or both coordinate systems are not entered correctly, either the metric or the geographic coordinates will not be displayed with their correct values. For certain projections, e.g. Transverse Mercator, it is particularly important that the Central Meridian be set close to the actual station locations.

To correct errors in the database made in the Coordinate Settings, enter the new, correct settings (Central Meridian, Projection, etc.). When saving the new settings, WinGLink will offer you the option to use the new settings to recalculate the metric (or geographic) coordinates of all the stations contained in the database starting from the geographic (or metric) coordinates of each station:



Database Sharing

A WinGLink database may be opened and used concurrently by more than one user. To prevent inadvertent data loss, however, restrictions on how the database may be shared between users are dynamically enforced, depending on which WinGLink applications are active. These restrictions vary from simple warnings to limiting use of the database to a single user.

The most restrictive state imposed by WinGLink is the "Locked" state. This state is enabled in the event that changes to the database will affect all projects or could change the structure of any of the projects in certain ways. This state is set only in the main WinGLink shell in the following cases:

- when a user is editing the database area properties
- when a user is importing external data into the database

In these cases the database is locked and may not be used by any other user. It follows that database area properties may be changed and external data imported only when no other users are using any WinGLink application.

During database sharing, any operation which would result in a loss of data integrity is not permitted.

In general, a given application will try to lock the active project and all associated members, i.e. stations and profiles. While in the locked state, that project and all members associated with it cannot be used by any other WinGLink application. Should another user attempt to open a locked project, or a project which makes use of member marked as locked, a warning message appears and the WinGLink application is aborted. For example, while one user is using the Maps program to edit the stations used on a profile in an MT project, the MT Soundings program cannot be used to view any of the stations contained in that project.

The WinGLink applications used for 2D modeling (magnetotelluric (MT & CS), gravity (GR) and magnetics (MG)) are slightly less restrictive. When a 2D modeling application is opened, the stations used by a model are marked as being “in use”. While in this state, they can be viewed and used in other WinGLink applications. For example, while one user is running a MT 2D inversion, another user may create a 1D model for one of the stations used in the 2D model.

If an application may be used by a second user, but with restrictions, a warning message appears indicating what functions are disabled.

Note: Every attempt has been made to prevent loss of data integrity and loss of data. WinGLink generally prevents any operation which would damage the database or result in lost data. Database sharing is, however, not without risk. Certain operations which would not result in data loss, yet may still result in lost work, are possible. For example, it is possible to modify a profile in the Maps program while a user is using the MT2D program. When sharing a database, make certain that any actions you take do not inadvertently affect the work of other users.

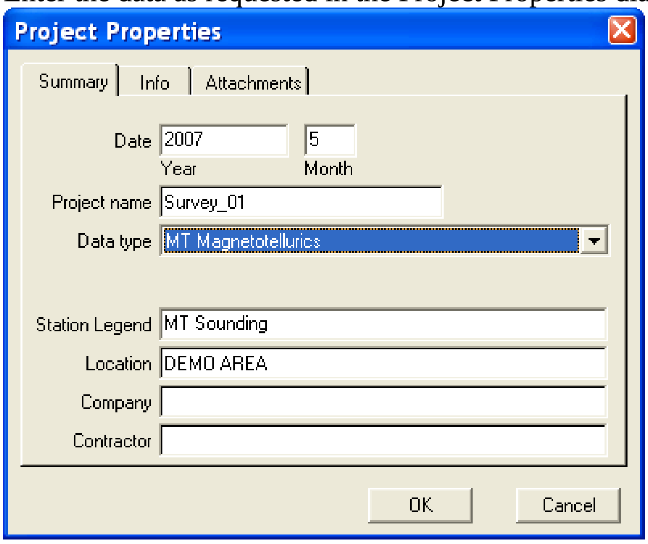
WinGLink Projects

Adding Projects to a Database

Projects are added to the database in the WinGLink Shell using the **Project | New…** menu command**.**

In the sub-menu:

1. Choose **Single…** if you want to add a project with new data to the database.
2. Choose **Integrated…** if you want to add a project which uses data from other projects already contained in the database.
3. Enter the data as requested in the Project Properties dialog box:



4. When finished, click the **OK** button**.**

Deleting Projects from a Database

To Delete a Project from a Database:

1. Open the WinGLink shell.

2. In the **Project** panel of the window, use the mouse to select the **name** of the project to be deleted.
3. Click either the **Delete** command from the **Project** menu or press the **Del** key on your keyboard.
4. In the dialog box which appears, confirm that you would really like to delete the project.

Warning: Deleting a project is a **permanent** action and cannot be undone.

Editing Project Properties

Select the **Project | Properties** menu command in the WinGLink Shell or click the **Project Properties** button located at the bottom of the WinGLink Shell. to open the Project Properties dialog box. The **Summary** tab of this window is, by in large, identical to the window used when creating a new project. Use of the **Attachment** tab is described below in the “Combining Projects” section.

Field	Meaning
Date	For observed data, the year and month the survey was carried out. For magnetic projects, this date is used to calculate IGRF values
Project name	This is the code name of the project as it will be displayed in the Projects list of the Database window
Data type	From this dropdown list, select the type of data for your project (example: gravity, MT, generic, etc.). The data type cannot be changed if the project has one or more stations.
Station legend	The legend you wish to automatically associate with the stations of the project when printing maps and sections.
Location	Survey location
Company	Client name
Contractor	Contractor name

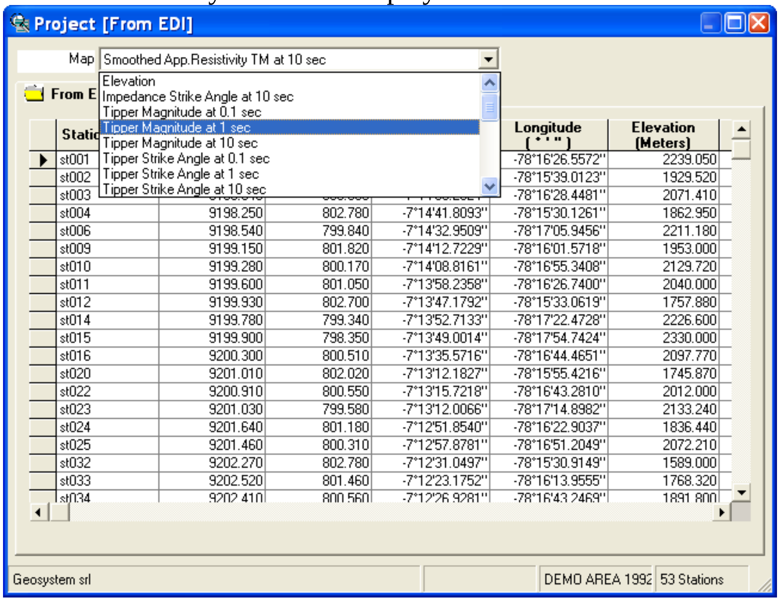
Previewing the Contents of a Project

Regardless of project type, a project contains stations which are identified by name, coordinates, elevations and, usually, values.

To Review the Station Coordinates, Elevations and Values from the Database Window:

1. On the **Projects** menu, click **QuickView** or click the **Preview Data** button located at the bottom of the WinGLink Shell.

2. Select the values you wish to display:



Note: This function displays station data and values in ***read-only*** mode. To edit the stations, you must start the **MAPS** program.

Setting a Project as the Source for the Topography

The topography of a project is obtained by interpolating the elevations of its member stations. When a project has only a few stations, this topography may be severely inaccurate.

To avoid this problem, any project in a database can be set as the source of topography for all other projects in that database. When a project is set as default topography, the topographic profiles for the sections of all other projects in the database are extracted from the gridded elevations of this project.

For maximum accuracy, a DEM (digital elevation) map may be imported into a project and this project then used as the default topography for the database. For information on importing DEMs, please refer to the “Importing External Grids” section of the “Maps” chapter of this manual.

This way, all sections along the same profile have the same topography, regardless of the project to which they belong.

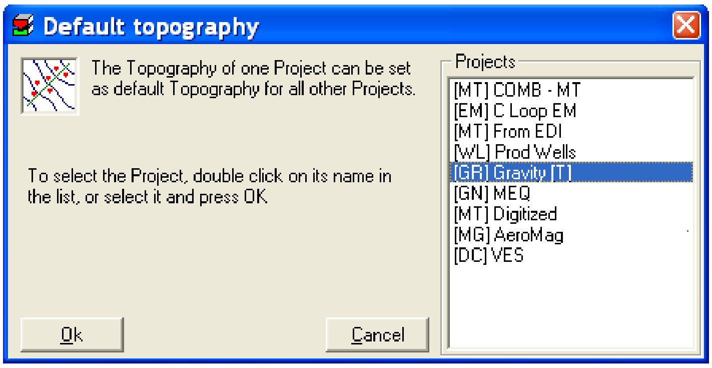
It is a general practice to use an Integrated project which includes a gravity project as source for the area topography.

Note: The MT 2D Inversion program uses actual station elevations to generate the profile, not the values in the project used as default topography.

To Set a Project as the Source for Topography:

1. Open the WinGLink Shell.
2. On the **Project** menu, select the **Default Topography | Set** menu command.

3. Click the name of the project you wish to use as the topography source:



4. Click the **OK** button.
5. A suffix **[T]** appears next to the project type in the Type column.

Combining Projects

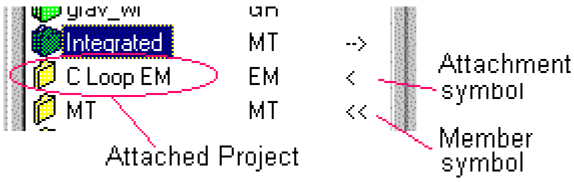
The projects of a database can be combined in order to produce maps and sections using the stations of more than one project.

There are two ways to combine projects:

Attaching projects to the current project:
The resulting maps and sections are computed using **only** the data of the **stations contained in the active project**. The stations of the **attached projects** are posted on maps and sections, but their datasets are not used in the processing.

Merging projects into an integrated project:
The resulting maps and sections are computed using **all the equivalent data** of the stations of the combined projects (**member projects**). The stations of the member projects are posted on maps and sections, **and** their datasets are used in the processing.

For a quick display of **attached** and **member** projects, select a project in the Project column of the Database Project list. In the example shown below, the "Integrated (MT)" project has been selected. The arrow on the right-hand side points to the attached and member projects:



 **Single projects** can only have attached projects.

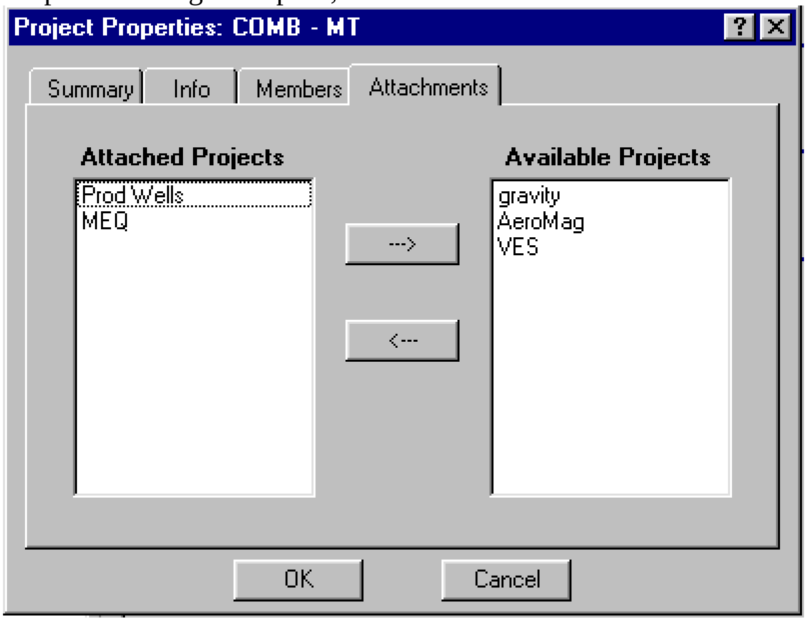
 **Integrated projects** can have both attached and member projects

Attaching a Project to Another Project

Attached projects are single projects whose stations are "attached" to the current project for simultaneous display on maps or sections.

To attach a project to another project:

1. Open the WinGLink Shell.
2. In the **Project** list panel, click the name of the ***project to which attachments are to be added***.
3. Select the **Project | Properties** menu command**.** The Project Properties dialog box opens; click the **Attachments** tab:



   The left box lists the projects which have already been attached; the right box lists the projects available for attachment.

4. Use the arrow buttons to move projects in/out the attachments list.
5. When finished, click the **OK** button.

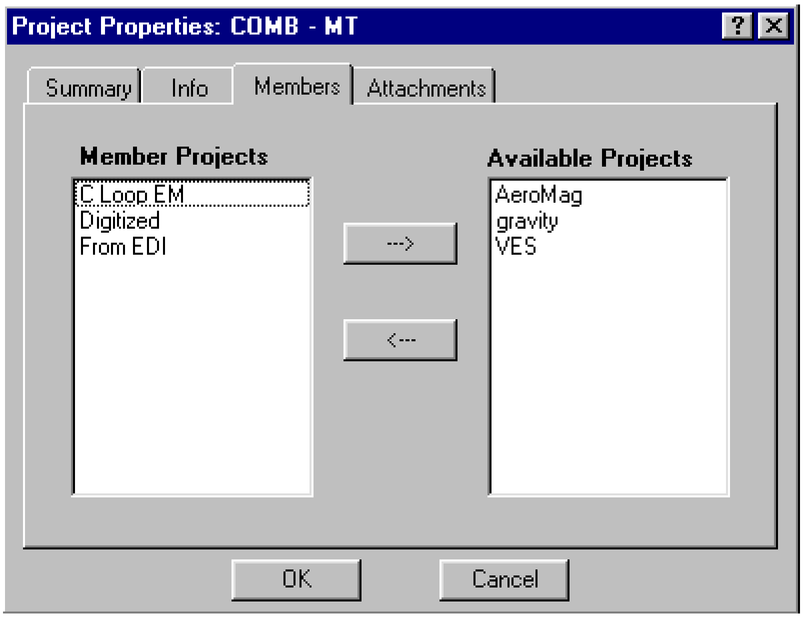
Note: When attaching projects to integrated projects, member projects are not listed among the projects available for attachment.

Combining Projects into an Integrated Project

An integrated project is a project which has no stations of its own, but instead uses the stations of other single projects **(members)** to produce integrated maps and sections.

1. Select the **Project | New** menu command in the WinGLink Shell, then select the **Integrated** option.

2. Click the **Members** tab to display the following dialog box:



3. The left box lists the current members of the project; the right box lists the other available projects.
4. Use the arrows to add/remove projects from the member list.
5. When finished, click the **OK** button**.**

12: Data Import

Overview

WinGLink can import survey data (station names, coordinates, values and/or datasets) from a variety of source files. The imported stations and associated datasets are organized and stored in a database project.

This can be either a new project created with the purpose of containing the imported data, or an existing project that already has stations. For example, it is possible to first import station IDs and coordinate information, and then import the actual station data. In the this case, the imported data are added or merged with the existing information.

Depending on the data type, data can be imported from the following sources:

- From external files
- From a WinGLink database
- From another project in a database

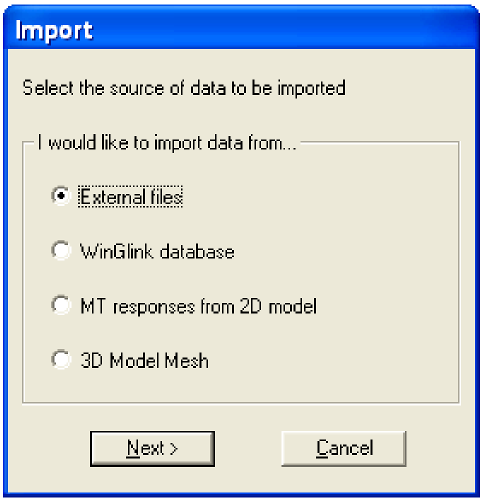
This chapter provides instructions on importing all data types which can be imported into a WinGLink database from the WinGLink Shell.

Getting Started with Data Import

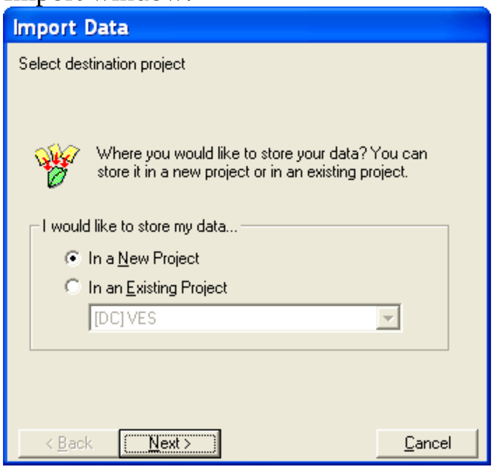
Regardless of data type, the initial steps in the data import process are identical for all data types. The description provided below covers the first few steps common to nearly all types of data import.

Import Steps Common to Most Data Types

1. On the **File** menu in the WinGLink Shell, click the **Import…**, menu command. In the Import window which opens, select the source from which you wish to import your data:



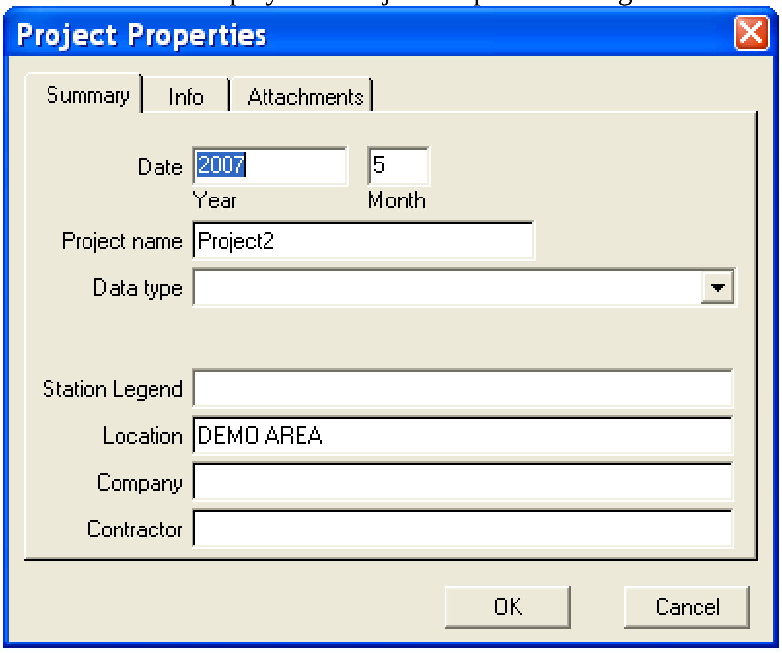
2. For importing most types of files, select **External files** option and continue by clicking the **Next** button to advance to the next Import window:



Here, the program gives you the option of importing data into either a new or an existing project.

3. If importing into an existing project, select the appropriate option and, on the next screen, select the desired project from the list presented. Otherwise, select the **In a New Project** option. WinGLink now displays a message alerting you to the fact that a new project will now be created. Acknowledge the window and click the **Next** button to continue.

4. WinGLink now displays the Project Properties dialog box:



Mandatory fields in this window are the **Project name** and **Data type**. When selecting the data type, make certain to select the type which matches the data to be imported.

The subsequent steps in the import process are, to a certain extent, data-type dependent. The remaining import process for each of the different data types is described in detail in the following sections.

Importing Station Coordinates

Station coordinates may be imported either alone or together with the actual station data. This may be the case with EDI files, which can contain both coordinate information as well as magenetotelluric data. Frequently, station coordinates are not included with the actual data files. This program option allows you to import the data in two steps, first by importing the coordinate information and then importing the actual data (or vice versa).

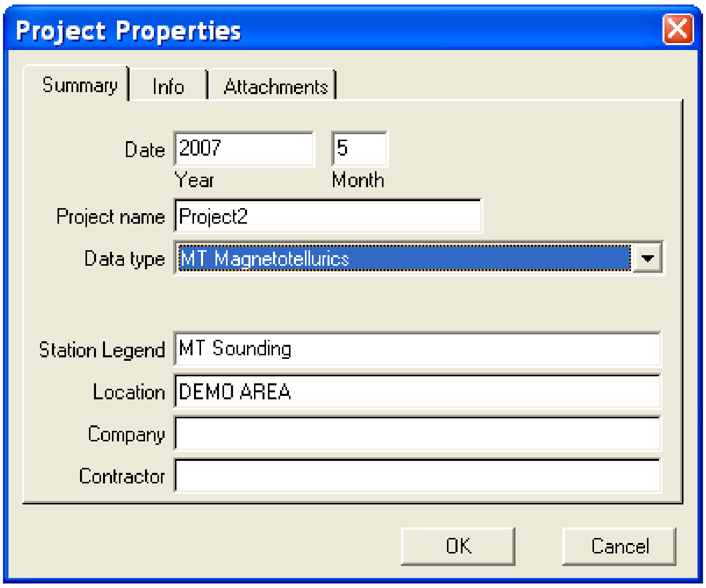
This section provides a description of the process for importing coordinate information which is contained in a separate text file. These text files must be organized in columns, whereby the order of the columns is immaterial

To Import Station Coordinates Contained in Text Files:

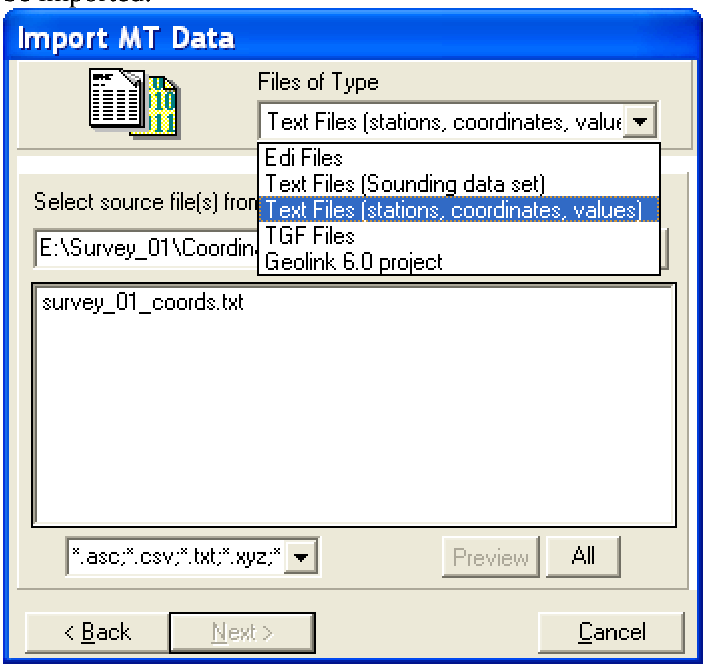
WinGLink can import station data (i.e. coordinates and values, sounding datasets, well courses, or layered/numerical data) from text files with data organized in columns. The process for importing coordinates is illustrated below for the case of magnetotelluric data. With the exception of project type, the process is very similar for all data types.

1. To import station coordinates into a new project, follow steps 1-3 of section “Getting Started with Data Import”. Otherwise, if importing coordinates into an existing project, follow the first two steps, selecting the project into which the coordinates are to

be imported instead of creating a new project.



2. Click the **OK** button to advance to the Import MT Data window. In the **<Files of Type>** box, select the type of data to be imported:



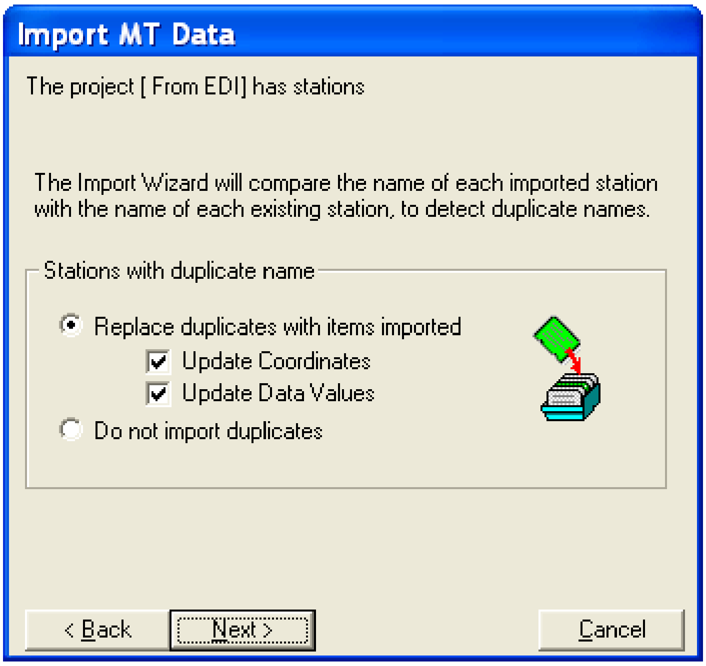
Note that the file types available for selection are dependent on the current project type.

3. Browse the directories until you find the text file to be imported.

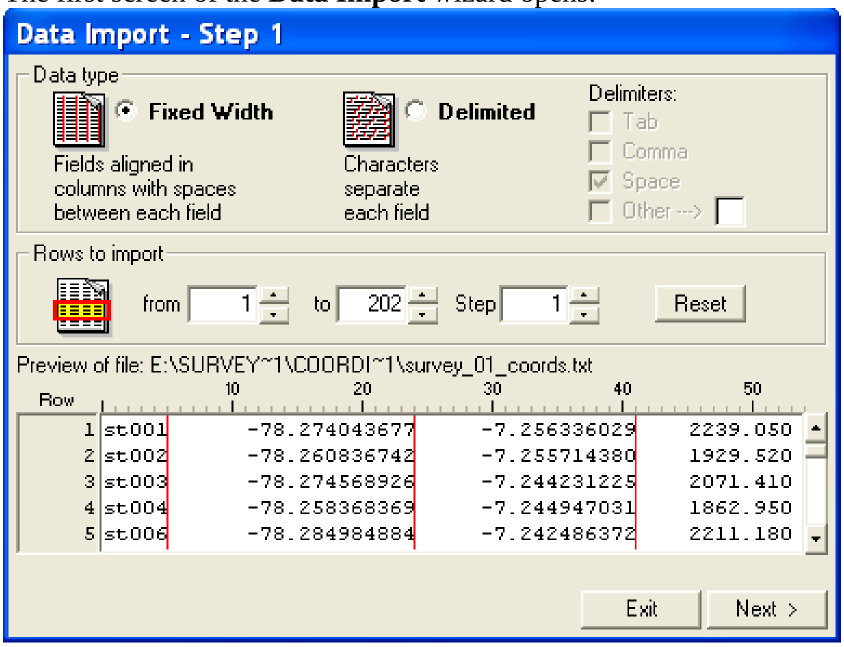
4. Click the filename of the file to be imported to select it (use the ***Preview*** command if you would like to quickly check the file contents). Click the **Next** button.

5. If the destination project already has stations, choose the desired options for updating duplicate stations. Click the **Next**

button.



6. The first screen of the **Data Import** wizard opens:



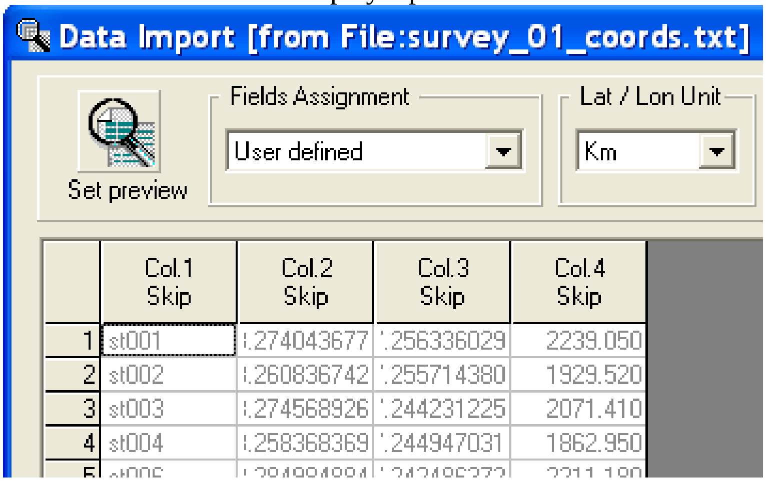
Check the appropriate boxes in the **Data type** frame at the top of the Data Import dialog box . These are used by the import wizard to guess the field separation in the text file being imported. You can check the data type in the preview window at the bottom of the dialog box.

Specify which rows are to be imported in the **Rows to import** section of the dialog box. Use this option to skip header and/or footer lines.

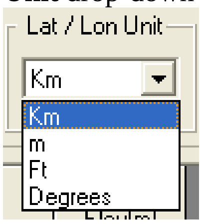
In the **Preview of file** area of the dialog box, check that the input file is read correctly. If necessary, use the mouse to correct the field separation. **Red** lines represent field breaks. To

CREATE a field break, click the mouse at the desired position; to DELETE a field break, double-click the line.

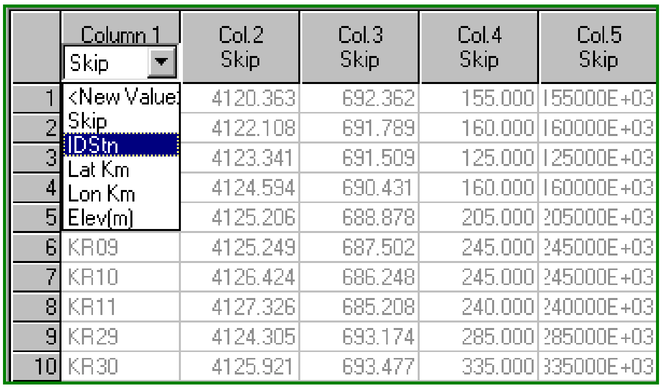
7. Click the **Next** button to display a preview of the file columns:



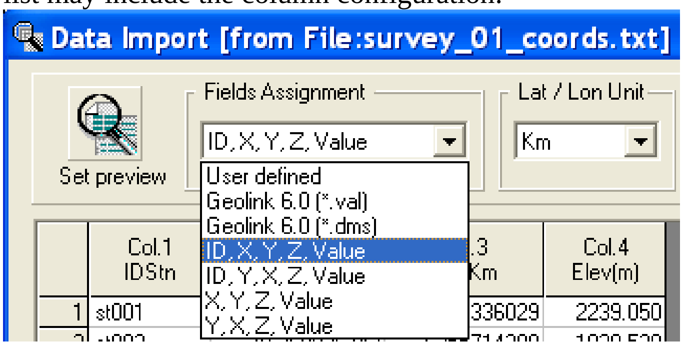
First, select the units for latitude and longitude in the **Lat / Lon Unit** drop-down list:



Then click the header of each column to select the field for each column:

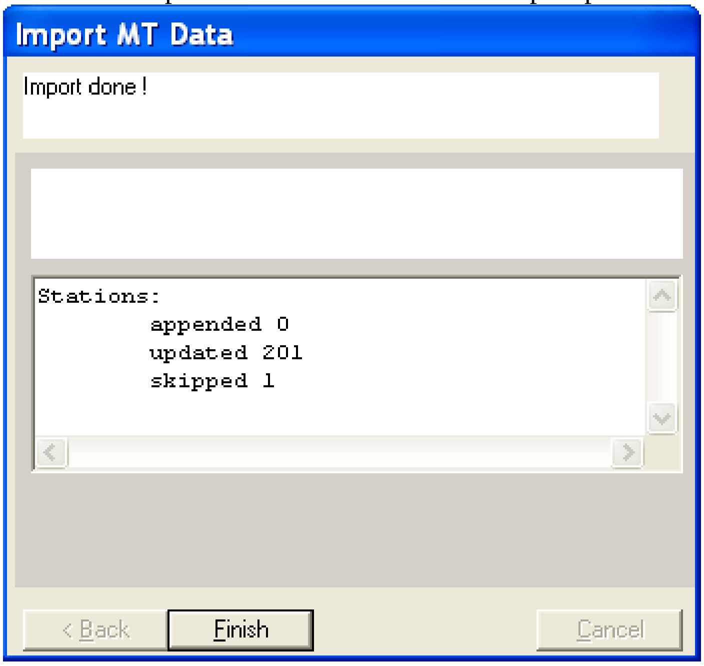


8. Alternatively, the option in the **Fields Assignment** drop-down list may include the column configuration:



9. When finished, click the **Save** button to confirm the import.

10. The program displays the final screen of the Import Wizard, a screen which provides information on the import process:



Note: Columns marked with <**Skip**> are ignored during the import.

Importing MT Data

MT data can be imported into WinGLink in a variety of formats:

- Single-site EDI files

- Text files

- WinGLink projects

- WinGLink databases

- Geolink projects

- TGF files

Furthermore, multi-site EDI files and system files (e.g. Stratagem / Imagem) can be converted to standard, single-site EDI files using the EDI Tools available in Tool Mode. See Chapter 10, MT Tools for details.

This section provides instructions on importing

- Single-site EDI files

- Text files

- 2D MT forward responses

-3D meshes

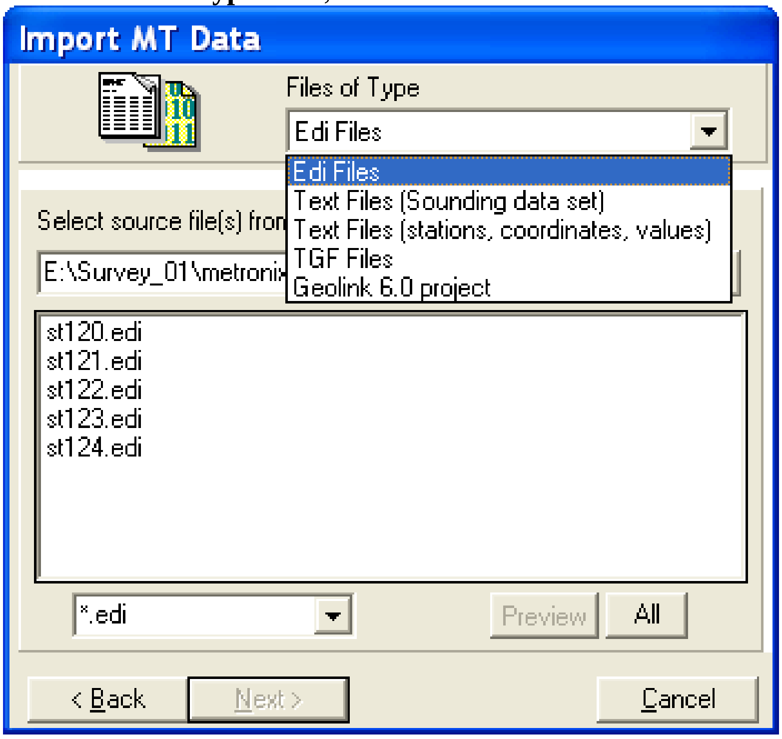
Note: Information on importing data from within a WinGLink database, TGF files and Geolink projects is provided at the end of this chapter as the procedure is nearly identical for all data types

Importing Single-Site EDI Files

The EDI format is the de facto standard for magnetotelluric station data. This standard includes data blocks for all typical data associated with a sounding. WinGLink can import multiple EDI files at the same time, one for each sounding.

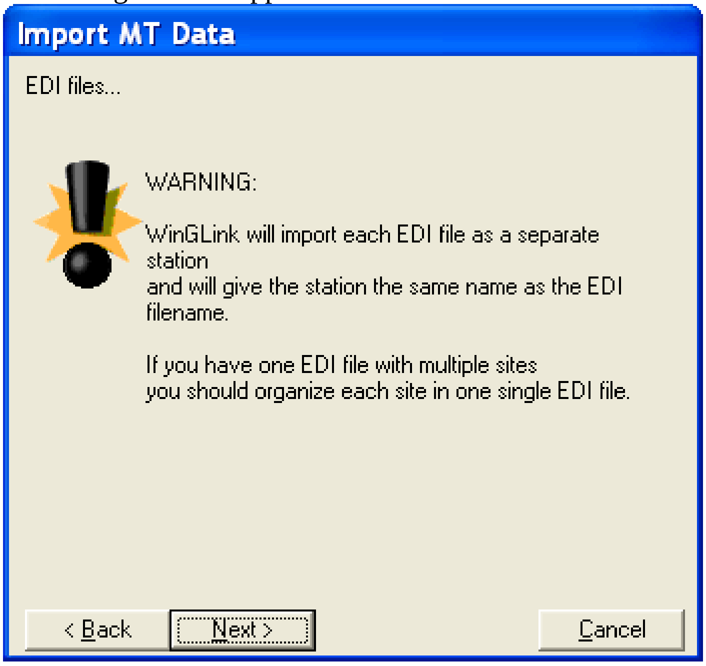
To Import a Single-Site EDI File:

1. To import a single-site EDI file into a new project, follow steps 1-3 of section "Getting Started with Data Import". Otherwise, if importing an EDI file into an existing project, follow the first two steps, selecting the MT project into which the EDI file is to be imported instead of creating a new project.
2. In the <**Files of Type**> box, select **EDI files**.



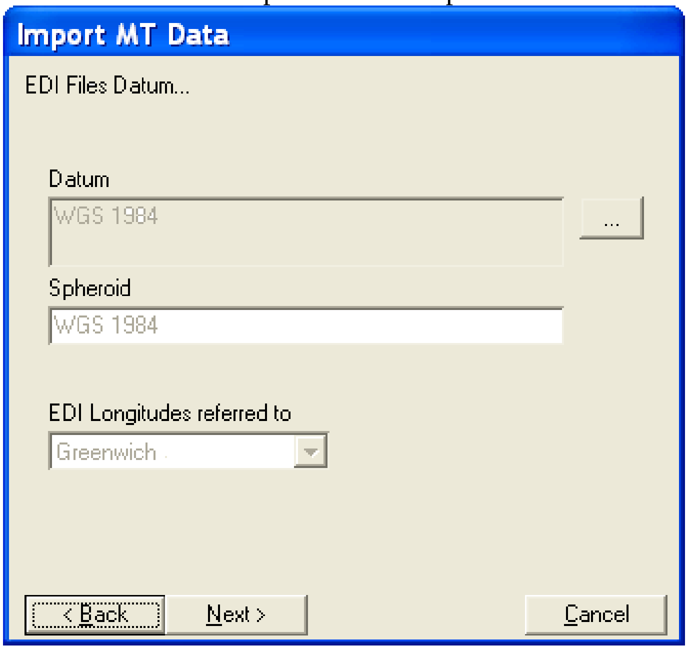
3. Browse the directories until the EDI files to be imported are listed.
4. Click the names of the file(s) to be imported; click the **All** button to select all EDI files in the current directory. Click the **Next button**.

5. A warning window appears:



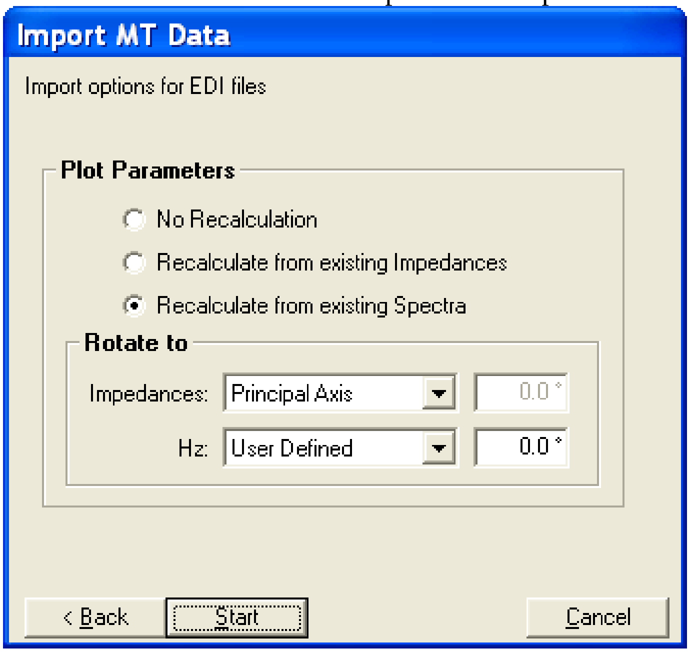
If attempting to import multi-site EDI files, they should first be broken down into single-site files using the tools provided in the MT Tools .

After acknowledging a warning window with the **Next** button, the first of the EDI import windows opens:



Here, you can specify the datum and spheroid of the station coordinates in the EDI files. The coordinates in the EDI files are then recalculated to the database coordinates. If there are either no coordinates in the file or the coordinates in the file and the database are the same, continue by clicking the **Next** button. Otherwise, make the appropriate settings in the provides fields and then click the **Next** button to continue.

6. The next window of the EDI import wizard opens:

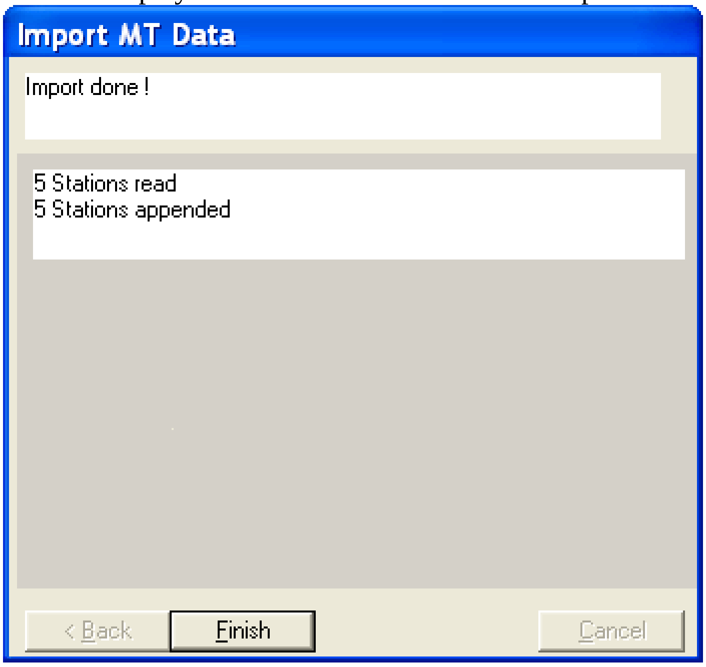


Here, you can specify whether recalculation of impedances or spectra should be performed and whether impedance and/or tipper rotation should be performed.

Use the impedance rotation options as follows:

Import option	When to use
No Recalculation	To import the plot parameters saved in the EDI file without recalculating them from existing impedances or spectra If plot parameters are missing, the program will calculate them from existing impedances. If impedances also are missing, plot parameters and impedances will be calculated from existing spectra.
Recalculate from existing Impedances	To compute plot parameters from existing impedances, without recalculating impedances from spectra.
Recalculate from existing Spectra	To recalculate plot parameters and impedances from spectra.

7. Click the **Start** button to execute the import. Import statistics are then displayed in the final window of EDI Import Wizard:



Note 1 - Spectra are not saved in the database: For each site, only plot parameters and impedances are imported and saved in the database. Spectra are read and used to recalculate impedances, but they are **not** saved in the database.

Note 2 - Site coordinates: The geographic coordinates of each site are read in from the REFLAT, REFLON fields of the EDI file, converted into metric coordinates according to the area settings of the database, and then corrected for the offset of the corresponding E dipoles. To correct for missing or incorrect site coordinates, launch the **MAPS** program and edit the site coordinates using the **Stations | View Data…** command. Site coordinates, if using a datum or spheroid different than that used in the database, can be converted during import by specifying the datum/spheroid used in the EDI files in the first of the EDI import windows as shown above.

Note 3 - Rotate to: If there is a ZROT block in the EDI file, the data are first "back rotated" to 0 degrees. Thus, to match the rotation in the EDI file, you must specify the angle to which the impedances are to be rotated in the User Defined field. The angle specified in the Import Wizard is used for all frequencies.

When rotating to the principle axis, the program rotates the impedance matrix to the maximum and minimum values of Zxy and Zyx, respectively.

If there are tipper data present in the EDI file, these data are likewise "back rotated" to 0 degrees from the angle specified in the TROT block. If there is no TROT block, it is assumed that the data are already at 0 degrees and no back rotation is performed. **The ZROT block is NOT used to specify the tipper rotation in the case that no TROT block is present.**

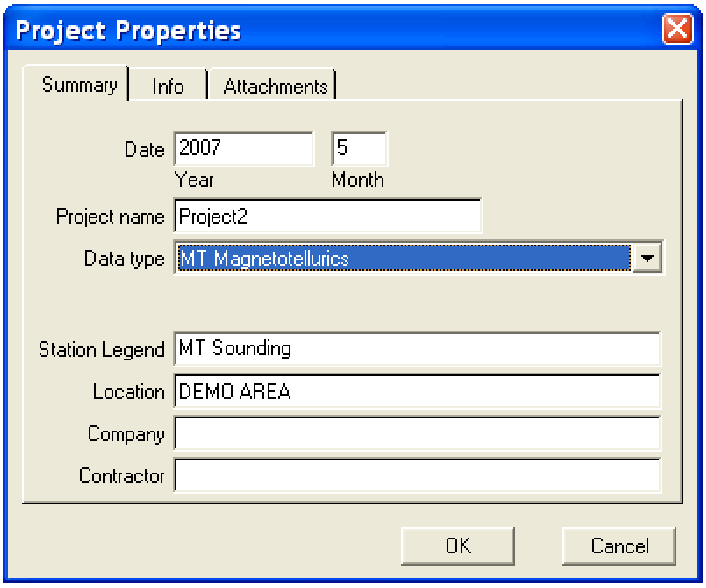
When using the 2D MT inversion module, tipper should be rotated to strike.

Importing Text Files

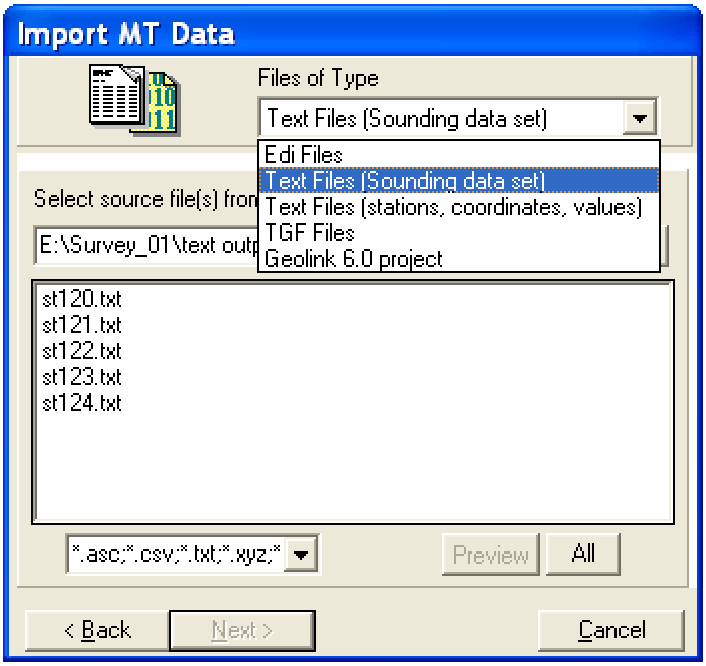
The Text File Import Wizard, which is also used to import station coordinate files, can be used to import MT data contained in column-based text files. Files of this type may contain either resistivities and phases or impedance values.

To Import MT Data Contained in Text Files:

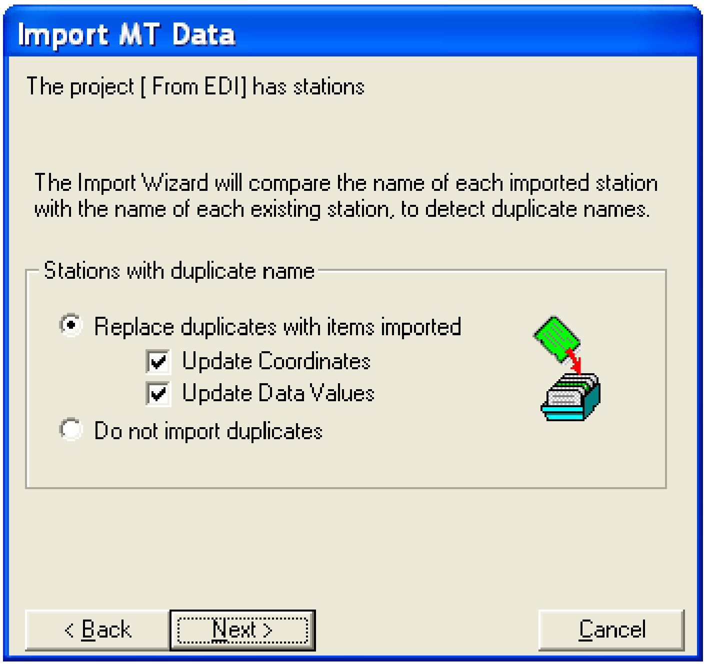
1. To import MT data into a new project, follow steps 1-3 of section "Getting Started with Data Import". Otherwise, if importing MT data into an existing project, follow the first two steps, selecting the project into which the data are to be imported instead of creating a new project.



2. Click the **OK** button to advance to the Import MT Data window. In the **<Files of Type>** box, select the type of data to be imported:

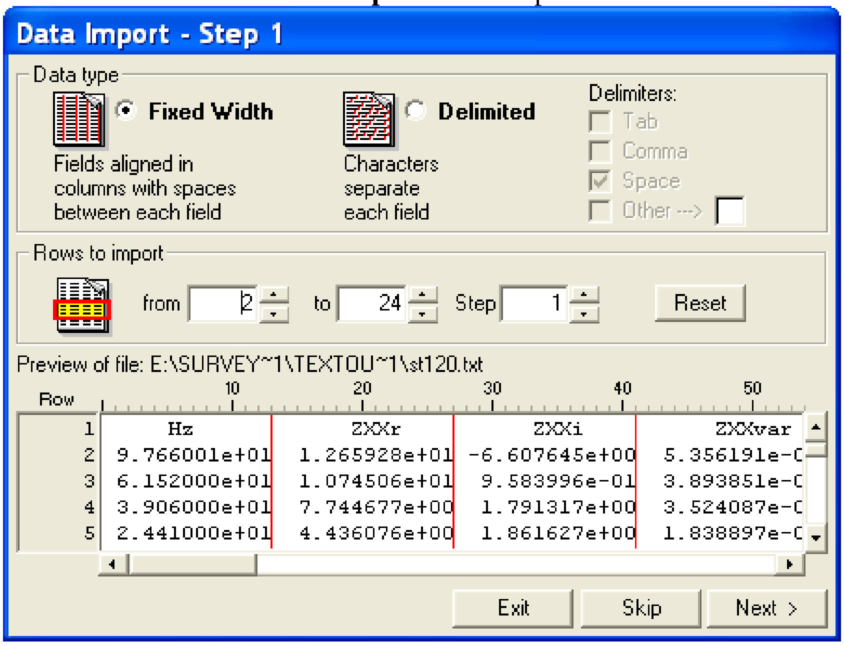


3. Browse the directories until you find the text file to be imported.
4. Click the filename(s) of the file(s) to be imported to select (use the ***Preview*** command if you would like to quickly check the file contents). Click the **Next** button.
5. If the destination project already has stations, choose the desired options for updating duplicate stations. Click the **Next** button.



6. An info box appears informing you that a station will be created for each input file and given the same name as the input file. Acknowledge the info box by clicking the **Start** button.

7. The first screen of the **Data Import** wizard opens:

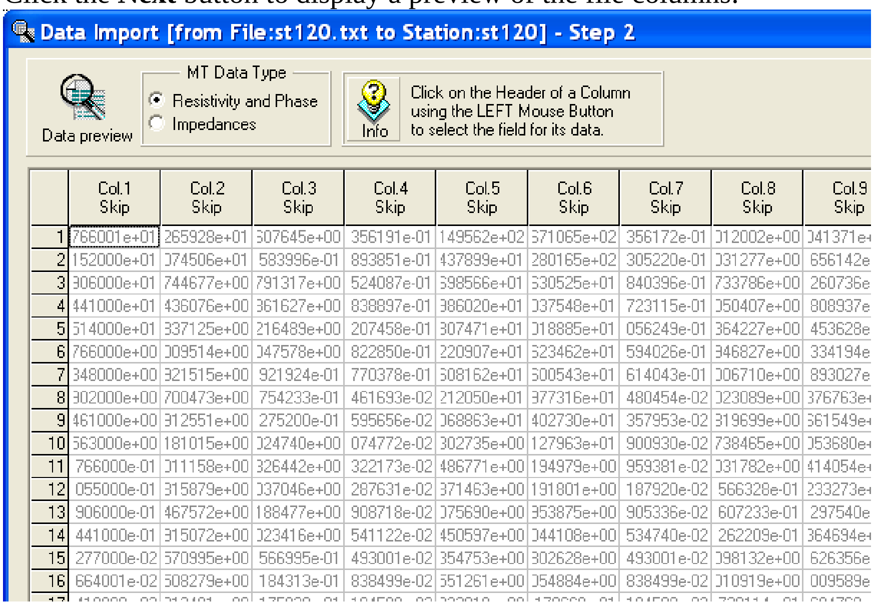


Check the appropriate boxes in the **Data type** frame at the top of the Data Import dialog box . These are used by the import wizard to guess the field separation in the text file being imported. You can check the data type in the preview window at the bottom of the dialog box.

Specify which rows are to be imported in the **Rows to import** section of the dialog box. Use this option to skip header and/or footer lines.

In the **Preview of file** area of the dialog box, check that the input file is read correctly. If necessary, use the mouse to correct the field separation. **Red** lines represent field breaks. To CREATE a field break, click the mouse at the desired position; to DELETE a field break, double-click the line.

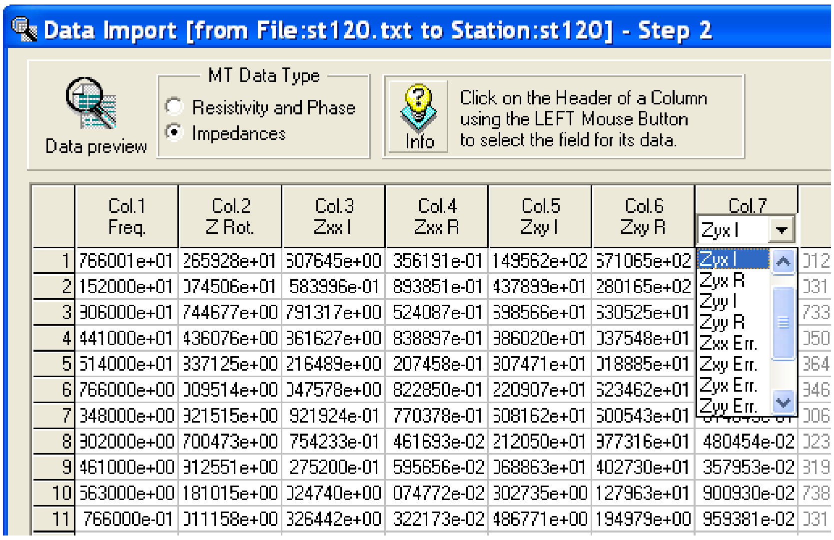
8. Click the **Next** button to display a preview of the file columns:



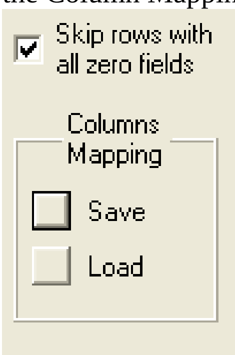
First, specify whether the file contains resistivity and phase data or impedance data in the **MT Data Type** area at the top of the window.

Note that columns marked with **<SKIP>** are ignored during the import process.

Then click the header of each column to select the field for each column:



9. Column mappings can be saved by clicking the Save button in the Column Mapping area at the right edge of the window:



The .MAP files created here can be reloaded the next time a file with the same column arrangement is to be imported by clicking the **Load** button.

10. When finished, click the **Save** button at the bottom of the window to confirm the import.

11. If importing multiple text files, the Import Wizard returns to step 7. After the steps outlined above have been completed for all of the text files, the program displays the final screen of the Import Wizard. Here, the program displays statistics on the import process.

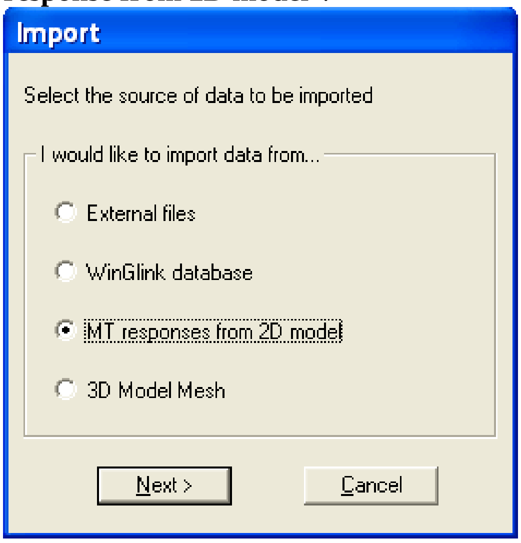
Importing MT 2D Forward Responses

The stations of an MT profile for which 2D modeling results have been calculated and saved in the current database can be imported into a new or an existing MT project. The computed 2D modeling results (i.e. computed TE and TM resistivity and phase) are imported as observed data for the stations of the profile.

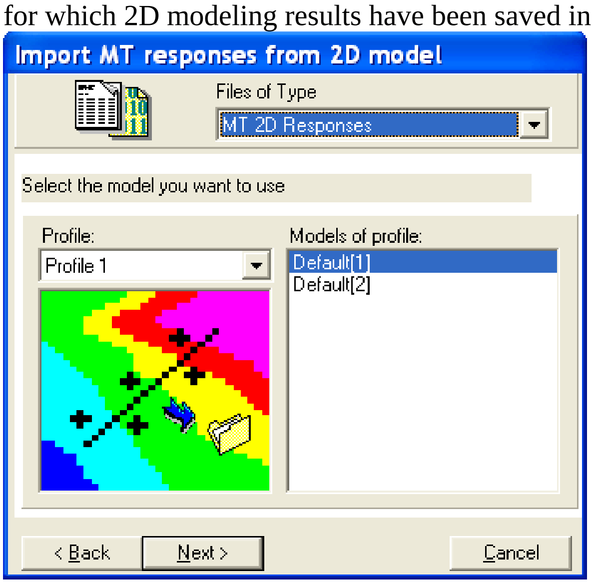
In this way, an inversion can be run using synthetic 2D MT model results as observed data.

To Import MT 2D Forward Responses

1. Select the **File | Import…** menu command in the WinGLink Shell.
2. In the Import dialog box which then appears, select: "**MT response from 2D model**":

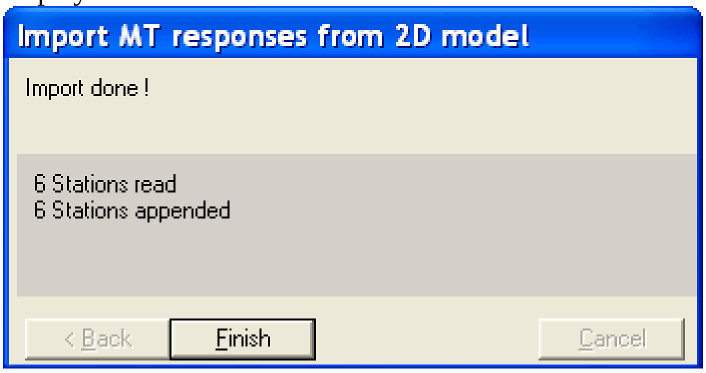


3. Select the destination project. This can be either a new or an existing MT project.
4. After selecting/creating the project into which the responses are to be imported, the program presents a list of available profiles for which 2D modeling results have been saved in the database:



5. Select the desired 2D model and click the **Next** button. The program imports the responses into the selected project and

displays the results:



Note: Because the 2D inversion routine returns only the Ty vertical component, certain interpretational qualities which require both Tx and Ty, i.e. induction arrows, cannot be displayed.

Importing 3D Meshes

WinGLink configurations which include the MT-4 module provide support for importing 3D meshes in the Randy Mackie format. This format is defined at the end of this section. These meshes are MT - specific and can only be imported into MT projects.

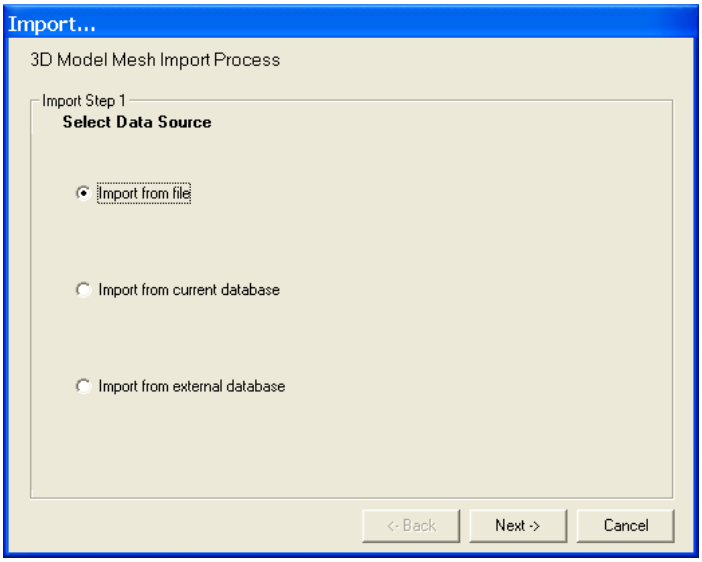
Once imported into a database, a 3D mesh remains associated with its project and can only be accessed from that project. The 3D Mesh Importer, which can be accessed from the WinGLink shell as well as from the 3D Modeling program, can be used to import 3D meshes from external files, the active database and other WinGLink databases.

Profiles placed on 3D meshes in the Maps program can be used to extract mesh sections in both the X-Sections program as well as the MT 2D Inversion program. WinGLink generates these sections by interpolating between mesh elements. Topographic information is retained during this interpolation. Sections extracted from 3D meshes in the MT 2D Inversion program can also be used as starting meshes for 2D models. Please refer to the “MT 2D Inversion” chapter of this manual for details.

To Import a 3D Mesh into WinGLink:

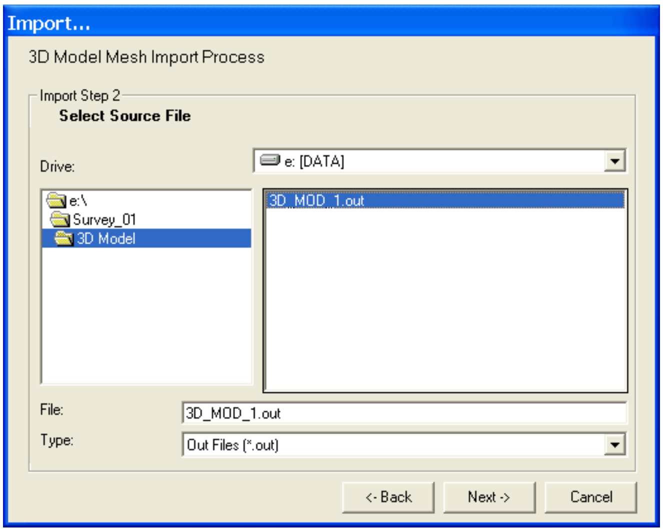
1. Select the **File | Import…** menu command in the WinGLink Shell.
2. In the Import window which opens, select the 3D Model Mesh option.
3. Select either an existing MT or project or opt to import the 3D Mesh into a new project.
4. The first of five 3D Mesh Import Wizard windows opens. Specify the data source: from file, the current database, or an

external database:

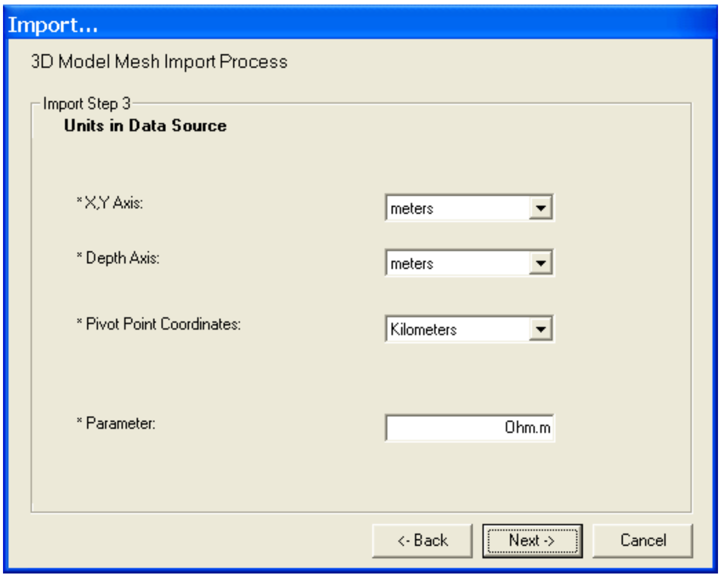


If you choose to import from either the current database or a different WinGLink database, additional dialog windows open prompting you to select e.g. the database name and the mesh to be imported.

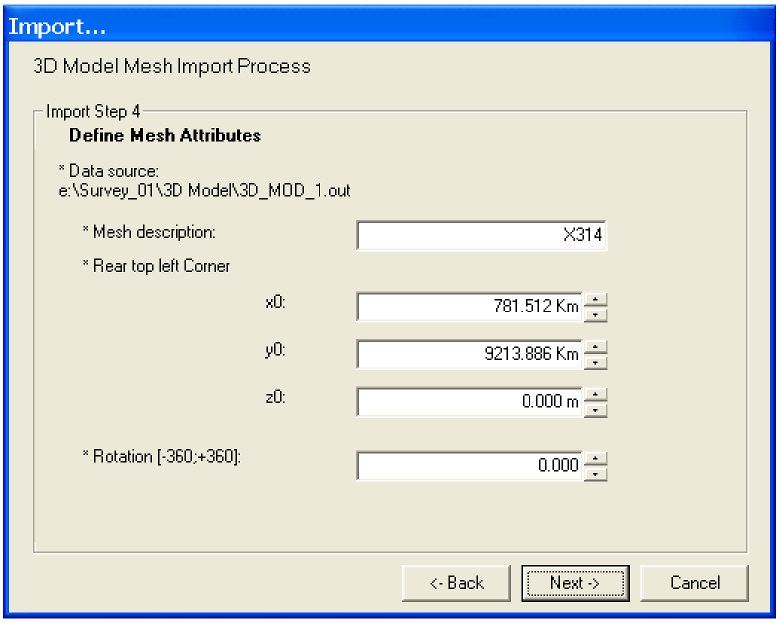
5. To import a 3D mesh form a file, now select the source file. The file must be in the Randy Mackie format specified at the end of this section.



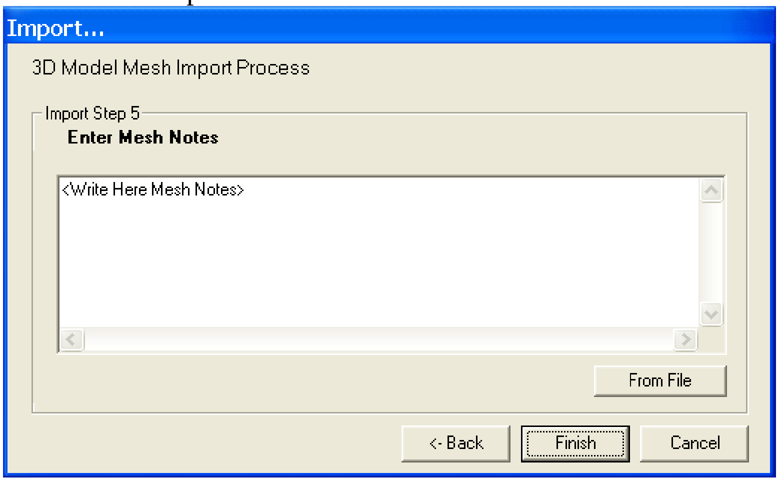
6. In step three of the import process, specify the units in the data source:



7. In step four, specify the mesh attributes, including mesh name, location of the rear top left corner as well as the angle to which the mesh is to be rotated:



8. In step 5, you can enter notes regarding the mesh. Alternatively, notes can be imported into the field from an external file:



9. In the final step, WinGLink performs the import operation. Import progress and any encountered errors are displayed on screen.

For details on working with 3D meshes, refer to the following sections in the WinGLink manual:

Chapter Maps, “Displaying 3D MT meshes”
Chapter Sections, “X-Sections”
Chapter “MT 3D Modeling”
Chapter “MT 2D Inversion”

Note: In addition to the method described above, 3D meshes can also be imported into projects from within the 3D Modeling program. For details, please refer to the “3D Modeling” chapter of this manual.

Randy Mackie 3D mesh specification

The mesh format is that specified by Randy Mackie plus an additional data block, located at the end of the file, which contains georeferencing information.

```
NX, NY, NZ (#’s of blocks) [Nair (layers)], [MAP.
VAL[UES]]
X block sizes (NX values in meters)
Y block sizes (NY values in meters)
Z block sizes (NZ values in meters)
1 (layer #)
NX*NY codes or values in free format: x varies
fastest
-----
-----
NZ (layer #)
NX*NY codes or values in free format: x varies
fastest
0. resistivity(for code 1) resistivity(for code
2) ....
resistivity(for largest code) [resistivity for
sea water]
end (optional termination text)
```

```
-----
-----
WINGLINK
ABC (site name)
I J (block numbers)
0000.000 0000.000 (real world coordinates)
0 (rotation)
```

Importing TDEM Data

Like MT data, TDEM data can also be imported into WinGLink in a variety of formats:

- TemMerge files

- Temix datasets

- Zonge AVG files

- Geolink projects

- TGF files

- USF files (a data exchange format defined by Phoenix Geophysics for the Phoenix V8 system)

This section provides instructions on importing the file types listed above with the exception of TGF files and Geolink projects, which are described at the end of this chapter as the import procedure is nearly identical for all data types..

TemMerge Data Files

Raw time-domain electromagnetic (TEM) data can be imported into WinGLink in a two-step process. The first step involves using WinGLink's TemMerge program to convert system dump files from the respective acquisition systems into a standardized format which can then be imported into WinGLink. Systems currently supported include:

- Sirotem

- Protem

- Zonge

- as well as Amira and free-format data)

The conversion is performed using the TemMerge program, which can be accessed in the EM area of the Tools section of WinGLink. For instructions on how to use TemMerge, please refer to the TDEM Tools chapter of this manual. For instructions on how to use TemMerge, please refer to the TDEM Tools section of the WinGLink help system.

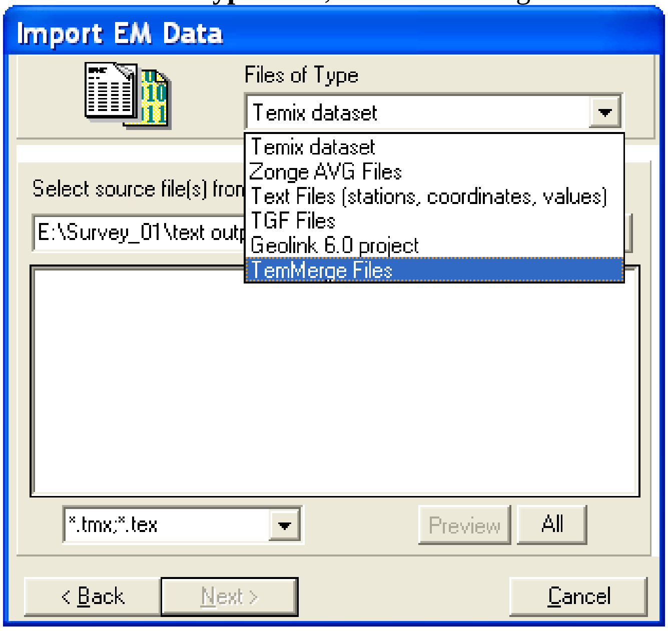
Once the TDEM data have been converted to TemMerge-format data, they can be imported into WinGLink using the import wizard.

To Import TemMerge Data Files into WinGLink:

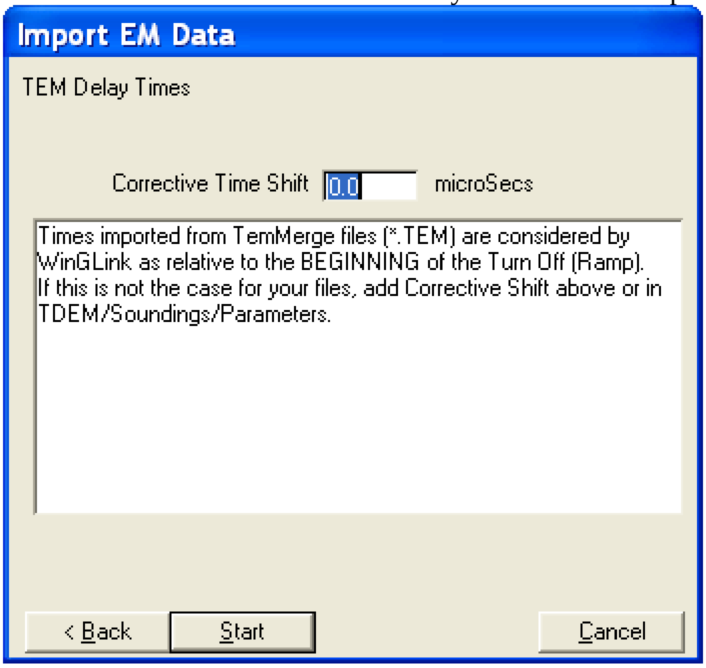
1. To import TemMerge data files into a new project, follow steps 1-3 of section "Getting Started with Data Import". Otherwise, if importing TemMerge files into an existing project, follow the first two steps, selecting the EM project into which the

TemMerge files are to be imported instead of creating a new project.

2. In the <**Files of Type**> box, select TemMerge files:



3. Browse the directories until you find the files to be imported.
4. Select the name of each file to be imported.
5. Click the **Next** button. The TEM Delay Times window opens:



In the **Corrective Time Shift** field, enter any time shift which was not included in the TemMerge file. This may be the case, for example, if the system operator did not enter the turn-off time during acquisition. The value entered here can be edited later in TDEM Soundings by executing the **File | Sounding Properties** menu command in the Soundings module.

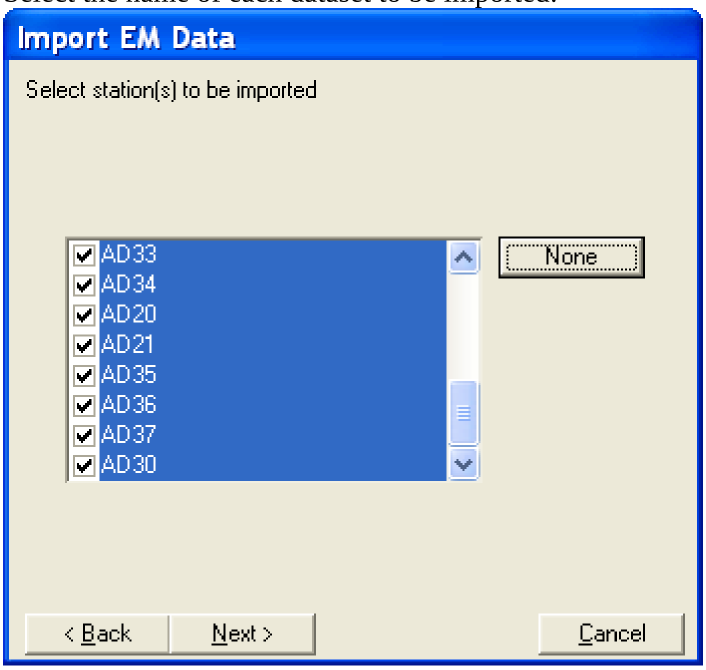
6. Click the **Start** button to initiate the import process. Once completed, WinGLink displays a window with information on the number of files which were successfully imported.

Importing Temix Datasets

TDEM datasets created with the Interpex Temix software can be imported into WinGLink.

To Import Temix Datasets into WinGLink

1. To import Temix datasets into a new project, follow steps 1-3 of section “Getting Started with Data Import”. Otherwise, if importing Temix datasets into an existing project, follow the first two steps, selecting the EM project into which the Temix datasets are to be imported instead of creating a new project.
2. In the <**Files of Type**> box select **Temix**.
3. Browse the directories until you find the dataset(s) to be imported.
4. Select the name of each dataset to be imported:



5. Click the **Next** button. WinGLink displays a window with information on the number of files which were successfully imported.

Importing Zonge AVG Files

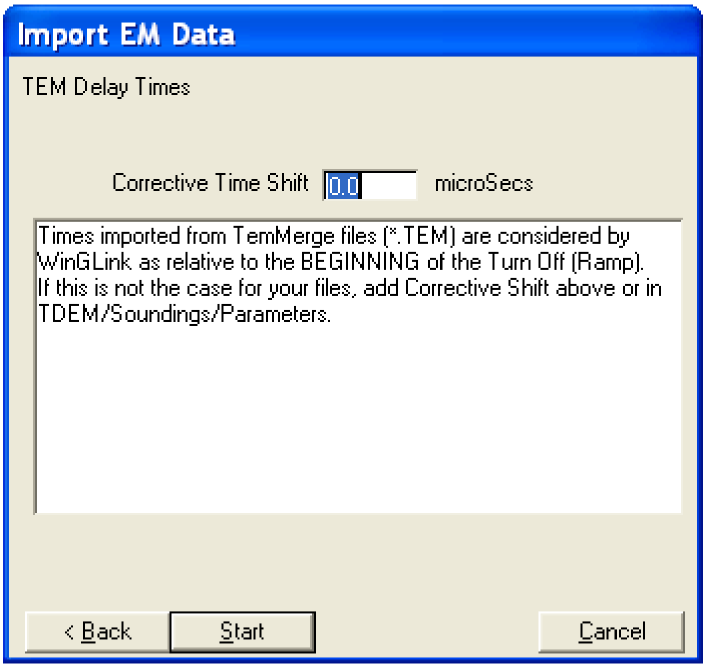
Zonge AVG files, unlike raw Zonge system dump files – which must first be converted to TemMerge files, can be imported directly into WinGLink.

To Import Zonge Files into WinGLink:

1. To import a Zonge AVG file into a new project, follow steps 1-3 of section “Getting Started with Data Import”. Otherwise, if

importing a Zonge file into an existing project, follow the first two steps, selecting the EM project into which the stations contained in the AVG files are to be imported instead of creating a new project.

2. In the <**Files of Type**> box select **Zonge AVG**.
3. Browse the directories until you find the dataset(s) to be imported.
4. Select the name of each station to be imported.
5. Click the **Next** button to advance to the next screen of the import wizard. Here, select either **m** or **km** as the XY units for the Zonge data.
6. In the next screen, WinGLink prompts you to enter any time shift which was not included in the Zonge AVG file. This may be the case, for example, if the system operator did not enter the turn-off time during acquisition. The value entered here can be edited later in TDEM Soundings by executing the **File | Sounding Properties** menu command in the Soundings module.



7. Click the **Start** button to initiate the import process. Once completed, WinGLink then displays a window with information on the number of stations which were successfully imported.

Importing USF Files

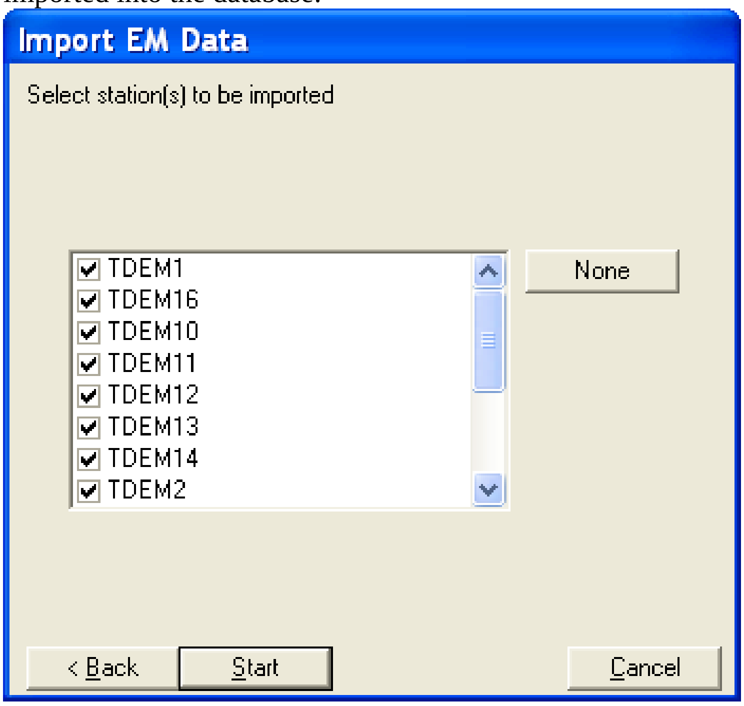
TDEM data contained in USF files, a data format defined by Phoenix Geophysics for their V8 system, can be directly imported into WinGLink.

To Import USF Files into WinGLink:

1. To import a USF file into a new project, follow steps 1-3 of section “Getting Started with Data Import”. Otherwise, if importing a USF file into an existing project, follow the first

two steps, selecting the EM project into which the stations contained in the USF file is to be imported instead of creating a new project.

2. In the <**Files of Type**> box select **USF TEM Files**.
3. Browse the directories until you find the file to be imported.
4. Select the name of the desired file (only one file can be imported at time).
5. Click the **Next** button to advance to the next screen of the import wizard. Here, select the stations() which are to be imported into the database:



6. Click the **Start** button to initiate the import process. Once completed, WinGLink then displays a window with information on the number of stations which were successfully imported.

Importing Gravity / Magnetic Data

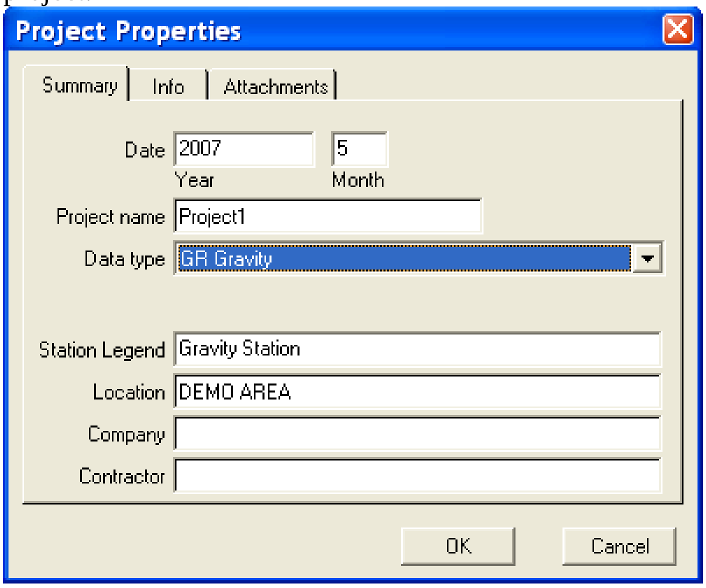
Gravity and magnetic data are typically imported into WinGLink as column-based text files. Files typically contain a station ID, coordinates, elevation and a measurement value, e.g. observed gravity or terrain correction.

In addition to the procedure outlined below for text files, gravity and magnetic data can also be imported using TGF files or from Geolink projects. For information on importing data contained in these last two types of files is provided at the end of this chapter as the procedure is nearly identical for all data types.

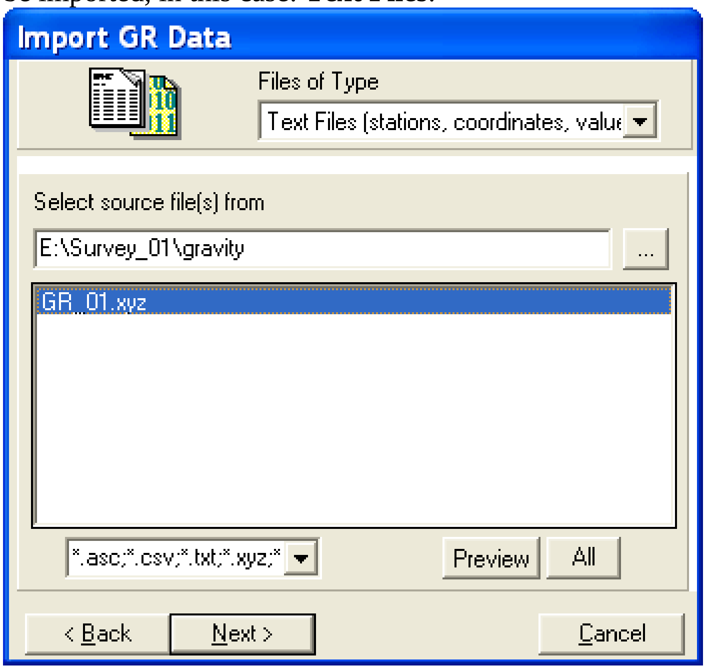
The description provided below is applicable to both gravity and magnetic data, whereby for magnetic data a project of type MG must be selected/created, and for gravity data a GR project must be used.

To Import Gravity / Magnetic Data into WinGLink

1. To import gravity or magnetic data into a new project, follow steps 1-3 of section “Getting Started with Data Import”. Otherwise, if importing gravity/magnetic data into an existing project, follow the first two steps, selecting the project into which the data are to be imported instead of creating a new project.

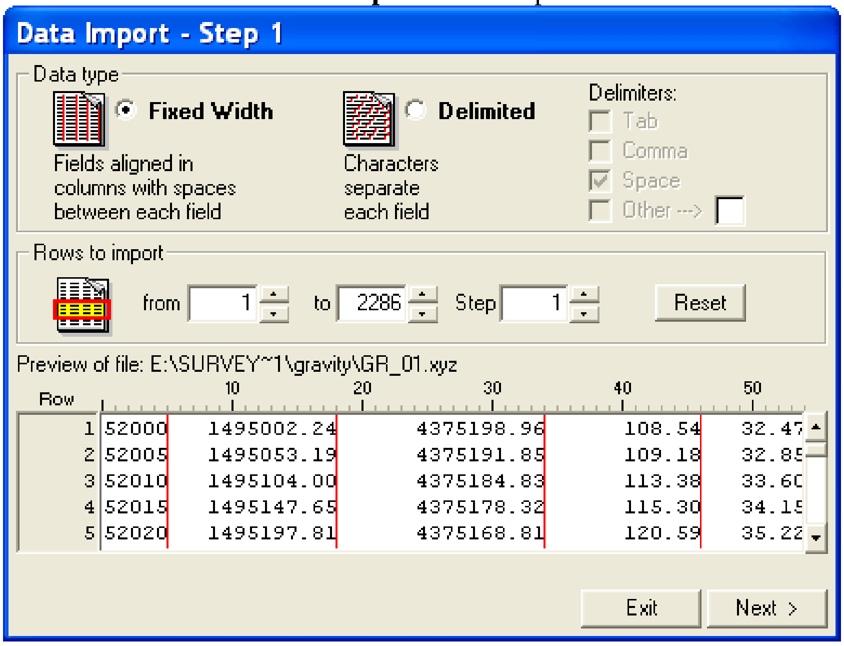


2. Click the **OK** button to advance to the Import GR/MG Data window. In the **<Files of Type>** box, select the type of data to be imported, in this case: **Text Files**:



3. Browse the directories until you find the text file to be imported.
4. Click the filename(s) of the file(s) to be imported to select (use the ***Preview*** command if you would like to quickly check the file contents). Click the **Next** button.

5. If the destination project already has stations, choose the desired options for updating duplicate stations. Click the **Next** button.
6. The first screen of the **Data Import** wizard opens:

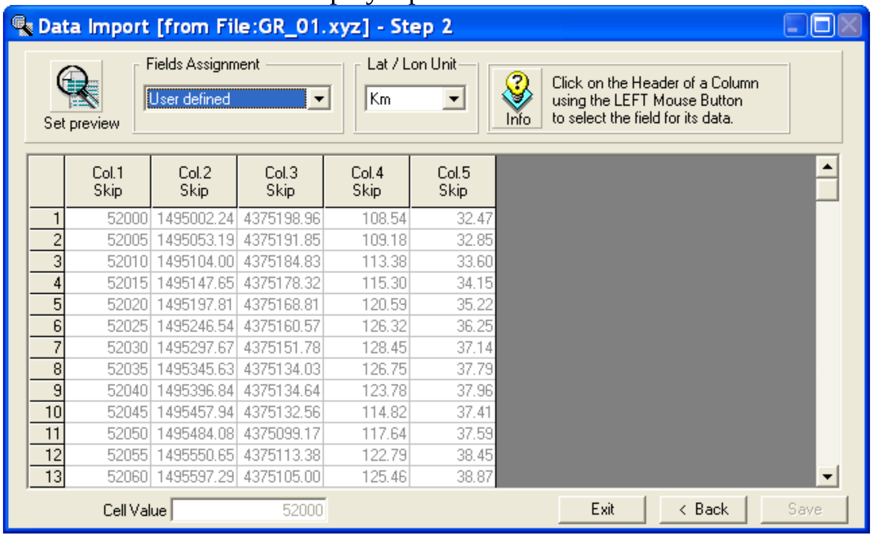


Check the appropriate boxes in the **Data type** frame at the top of the Data Import dialog box . These are used by the import wizard to guess the field separation in the text file being imported. You can check the data type in the preview window at the bottom of the dialog box.

Specify which rows are to be imported in the **Rows to import** section of the dialog box. Use this option to skip header and/or footer lines.

In the **Preview of file** area of the dialog box, check that the input file is read correctly. If necessary, use the mouse to correct the field separation. **Red** lines represent field breaks. To CREATE a field break, click the mouse at the desired position; to DELETE a field break, double-click the line.

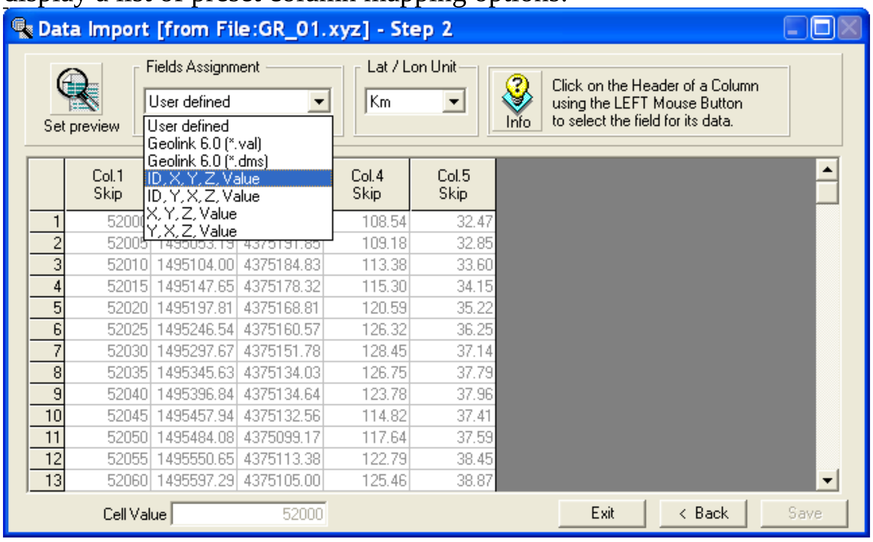
7. Click the **Next** button to display a preview of the file columns:



First, specify the units for latitude and longitude in the drop-down list at the top of the window.

Note that columns marked with **<SKIP>** are ignored during the import process.

Then click the header of each column to select the field for each column or open the **Field Assignments** drop-down list to display a list of preset column mapping options:



8. When finished, click the **Save** button at the bottom of the window to confirm the import. WinGLink then displays a window with information on the number of stations which were successfully imported.

Importing Well Data

All data associated with a given well is not typically contained in a single file. Instead, one file might contain coordinate information for one or more wells. Another series of files might contain the actual well traces; one for each well. Layer data, e.g. lithography or pressure, might be contained in other files. Thus, for optimal workflow with well log data, we recommend importing well data in the following order:

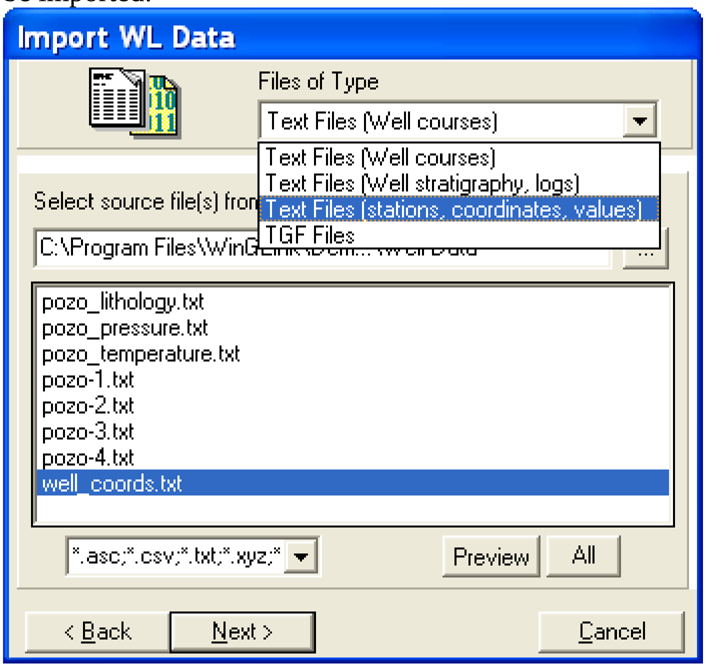
1. Import coordinate information
2. Import well courses
3. Import layer data

What all of these files have in common is that they are column-based text files. Thus, they can all be imported using the hopefully now familiar Text File import wizard.

The description shown here utilizes the demo well data provided in the WinGLink installation directory, typically:
`C:\Program Files\WinGLink\Demodata\Well Data`

To Import Coordinate Information

1. To import well coordinates into a new project, follow steps 1-3 of section "Getting Started with Data Import". Otherwise, if importing the coordinates into an existing project, follow the first two steps, selecting the project into which the coordinates are to be imported instead of creating a new project.
2. Click the **OK** button to advance to the Import WL Data window. In the **<Files of Type>** box, select the type of data to be imported:



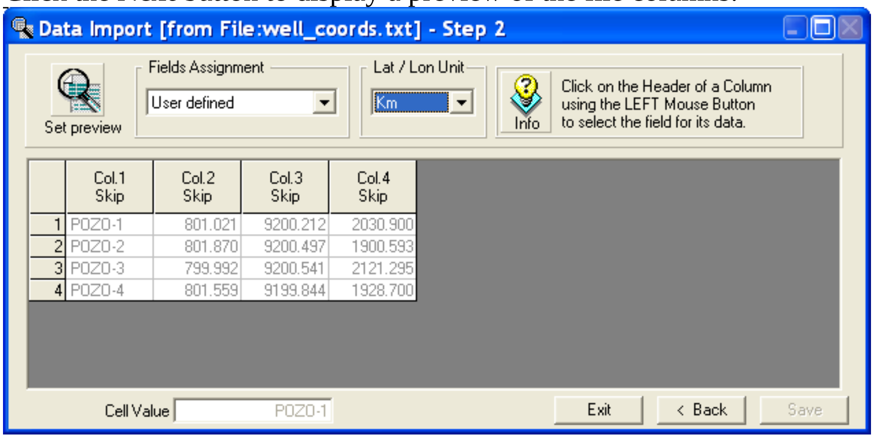
3. The first screen of the **Data Import** wizard opens.

   Check the appropriate boxes in the **Data type** frame at the top of the Data Import dialog box . These are used by the import wizard to guess the field separation in the text file being imported. You can check the data type in the preview window at the bottom of the dialog box.

   Specify which rows are to be imported in the **Rows to import** section of the dialog box. Use this option to skip header and/or footer lines.

In the **Preview of file** area of the dialog box, check that the input file is read correctly. If necessary, use the mouse to correct the field separation. **Red** lines represent field breaks. To CREATE a field break, click the mouse at the desired position; to DELETE a field break, double-click the line.

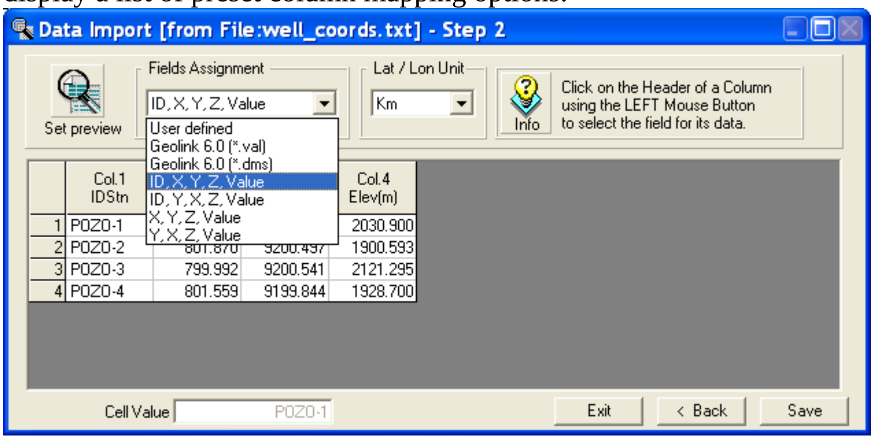
4. Click the **Next** button to display a preview of the file columns:



First, specify the units for latitude and longitude in the drop-down list at the top of the window.

Note that columns marked with **<SKIP>** are ignored during the import process.

Then click the header of each column to select the field for each column or open the **Field Assignments** drop-down list to display a list of preset column mapping options:

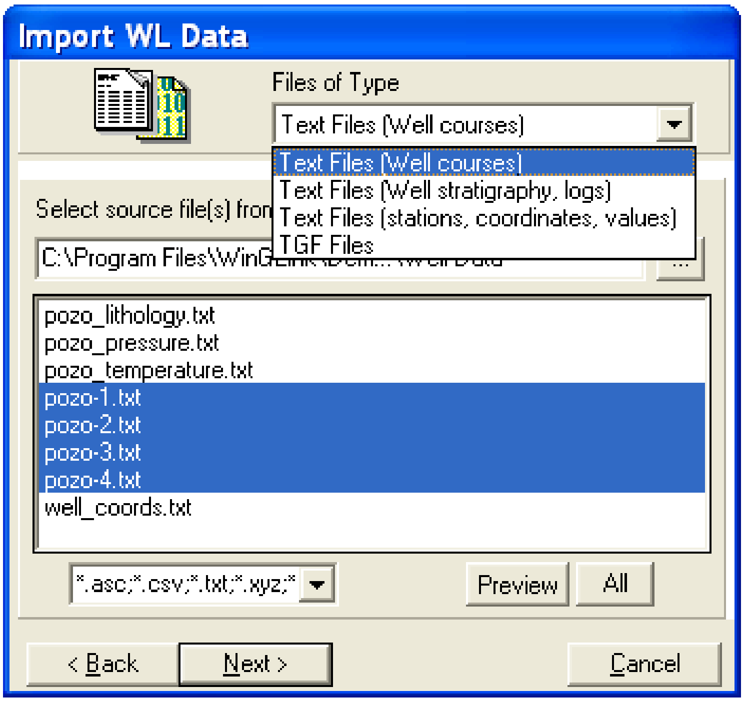


5. When finished, click the **Save** button at the bottom of the window to confirm the import. WinGLink then displays a window with information on the number of stations which were successfully imported.

To Import Well Courses

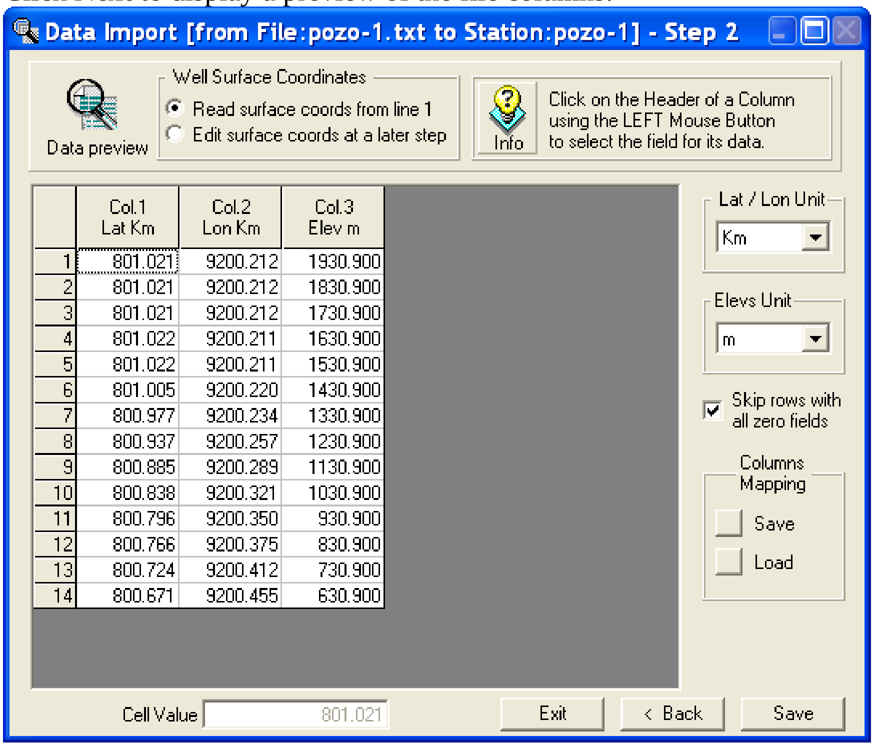
1. Proceed as outlined above for station coordinates, making certain to select the same project as was used to import the well coordinates. Instead of selecting **Station Coordinates**, select **Well Courses** in the Import WL Data window and select the

desired well course files:



2. The first screen of the **Data Import** wizard opens. Make the appropriate settings and click the **Next** button to continue.

3. Click **Next** to display a preview of the file columns:



On the right-hand part of the window, ***first*** select the units for latitude, longitude and elevation. ***Then*** click the heading of each column to select the appropriate field designations. **Latitude**, **Longitude** and **Elevation** must each be assigned to a column. Other columns are to be set to **Skip** (default):

After you have assigned the column headings, click the **Save** button to import the well log file into the database.

Tip:
If, in Step 3, you selected multiple files, the same column headings will automatically be set as the default headings for the remaining files in the import session. If, however, you will be importing further files with the same mappings at a later time, you can save the mappings by clicking the **Save** button in the **Column Mappings** panel located on the right side of the window. The mappings can be restored at a later time by clicking the **Load** button.

About Layer Data

Various types of well log data can be imported into WinGLink:

Layer data: stratigraphic data between two different well depths:

Well	From(M.D.)	To(M.D.)	Descr
AA-1	8.23	250.00	Sand
AA-1	250	1000.00	Clay
AA-1	1000	1300.00	Shale
AA-4	8.23	200.00	Sand
AA-4	200.00	1200.00	Clay
AA-4	1200.00	1500.00	Shale
AA-3	0	350.00	Sand
AA-3	350.00	400.00	Clay
AA-3	400.00	1100.00	Sand
AA-3	1100.00	1400.	Shale

Numerical data: value data at depth, e.g. temperature or pressure at various well depths. The files must contain 3 columns, either fixed width or delimited using any standard or user-defined delimiter.

```
Name   Depth           Value
AA-1   800.00  20
AA-1   1000.0010
AA-1   1545.0050
AA-2   200.00  14
```

Depth tags: labels to be assigned along the well trace at specified depths.

Well	Depth	Descr
AA-1	750.00	Low
AA-3	1300.00	High
AA-4	150.00	Med
AA-4	750.00	High

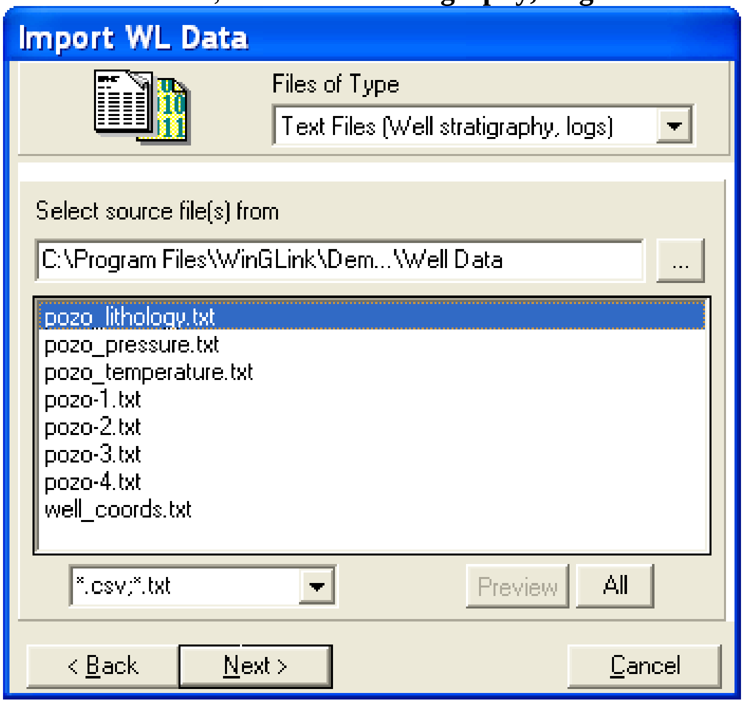
Note: Depth tags can be displayed only in Interpreted Views and Montage. They must, however, be imported into WinGLink using the import functions provided in the WinGLink database window.

To Import Layer Data

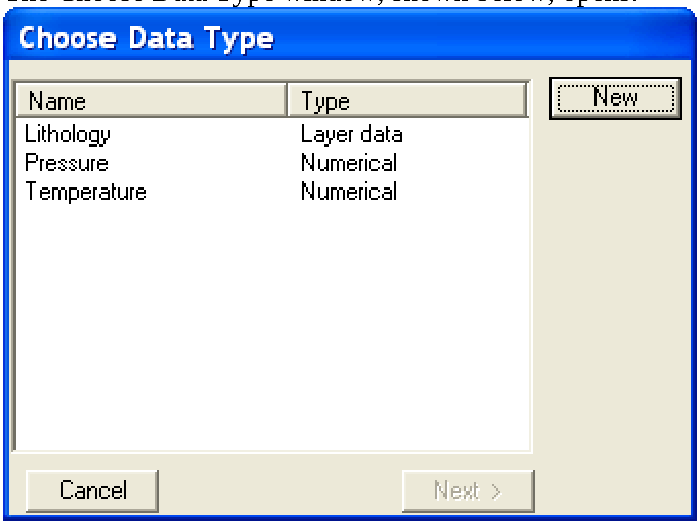
As with well coordinates and well courses, layer data are imported into WinGLink using the Import Wizard:

1. Proceed as outlined above for station coordinates and well courses, making certain to select the same project. In the Import

WL Data window, select **Well Stratigraphy, Logs**:



2. The Choose Data Type window, shown below, opens:

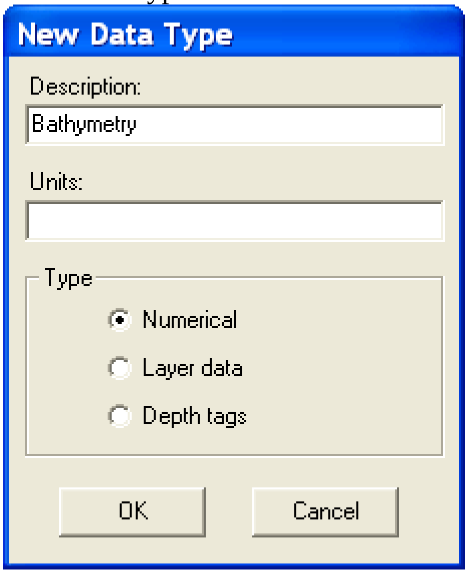


This window lists all previously imported data types, e.g. lithology, hydrology etc. Every time a file of a specific type is imported, each of the descriptive elements in a given file is appended to a list associated with the type.

For example, if a file XX contains layer data for lithology containing descriptive elements A,B and C, and it is associated with a data type called Lithology_1, the elements in the file will be linked to this data type. If a file ZZ containing elements D and E is imported and also associated with the Lithology_1 data type, these elements will also be linked to the Lithology_1 data type.

As is discussed in the “Common Functions” chapter of this manual, each descriptive element can have associated with it a fill pattern. By using data types, these fill patterns can be easily reused throughout a database.

To create a new data type, click the **New** button to open the New Data Type window:

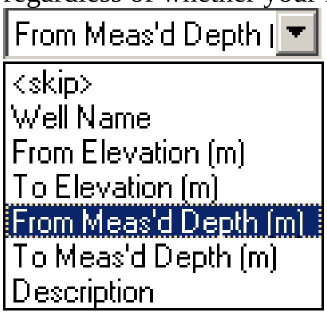


When importing numerical data, specify the units to be used when displaying the values. Leave this field blank for layer data and depth tags.

Confirm your entries with **OK**. The data type should now be listed in the Data Type window.

3. Proceed with the **Next** button to the next Import Wizard screen. Use the provided fields and checkboxes to specify the file format as you did when importing well courses.

4. In the following Import Wizard screen, specify the field headings. Note here that options are provided for both elevation and measured depth. Select only one of the two options, regardless of whether your files contain values for both:



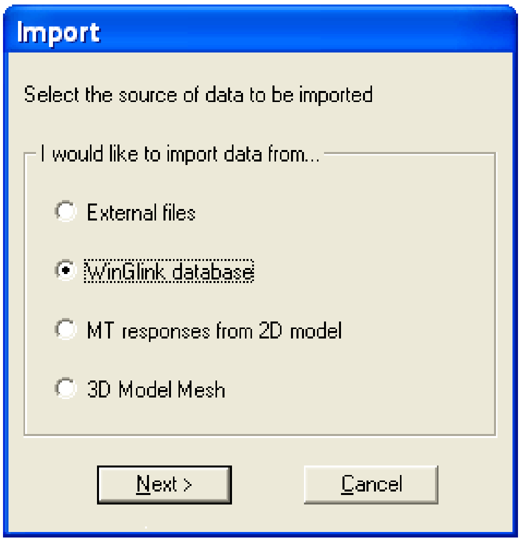
Proceed as described above for each of the layer data files to be imported.

Generally Applicable Import Methods

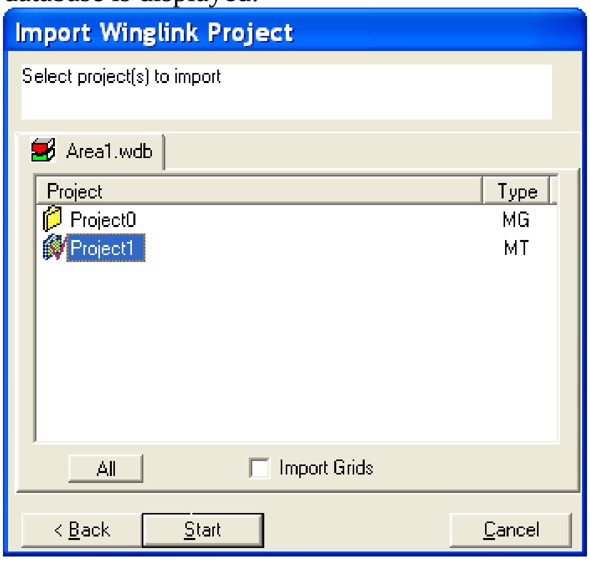
Importing a Project from a WinGLink Database

It is possible to import a project from another WinGLink database: the imported project will be appended to the projects of the destination database.

1. Create or open the database into which your data are to be imported.
2. On the **File** menu in the WinGLink Shell, select the **Import…**, menu command. In the Import window which opens, select the source from which you wish to import your data, here WinGLink database:



3. Browse the directories until the desired database is displayed.
4. Select the database file (*.wdb).
5. Click the **Next** button. The list of projects found in the selected database is displayed:



6. Select the project(s) to be imported (**SHIFT** and **CTRL** keys can be used for multiple selection).

7. When finished, click the **Start** button at the bottom of the window to confirm the import. WinGLink then displays a window with information on the number of stations which were successfully imported.

Importing TGF Files

The TGF format is an expandable data format specified by Geosystem for the purpose exchanging geophysical data across applications. It is used, among other applications, to import vertically distributed data into WinGLink. A given TGF file may contain station data for one or more stations, e.g. name and coordinates, as well as data values for one or more values, e.g. temperature and density. Details on the TGF specification can be found in Appendix A, The TGF file format.

TGF data are imported into WinGLink using the **Importing station data** command in the main Database window.

To import TGF files into WinGLink:

1. On the **File** menu in the WinGLink Shell, select the **Import…**, menu command. In the Import window which opens, select **External Files** as the data source and click **Next** to continue
2. In the Import dialog box, select TGF files in the **<Files of Type**> drop-down list.
3. Browse the directories until you find the file(s) to be imported.
4. Select the name of each file to be imported.
5. Click the **Next** button.
6. Click each of the stations which are to be imported.
7. When finished, click the **Start** button at the bottom of the window to confirm the import. WinGLink then displays a window with information on the number of stations which were successfully imported.

Note: If no stations are listed, either no stations are contained in your TGF file or your file is not conformant with the TGF specifications

---

Importing Data from a Geolink 6.0 Project

1. To import Geolink data into a new project, follow steps 1-3 of section “Getting Started with Data Import”. Otherwise, if importing Geolink data into an existing project, follow the first two steps, selecting the project into which the data are to be imported instead of creating a new project.
2. In the <**Files of Type**> box, select Geolink 6.0 project.
3. Browse the directories until the header of the desired project is displayed.
4. Select the name of the file to be imported.
5. When finished, click the **Start** button at the bottom of the window to confirm the import. WinGLink then displays a window with information on the import operation.

Importing Stations Referenced to a Different Coordinate System

To import stations with longitude and latitude referenced to a different coordinate system:

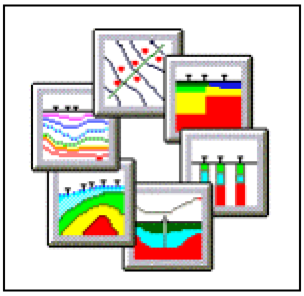
1. Create a temporary database called <Temporary> or similar.
2. When prompted for the database properties, enter the parameters that match the coordinate system of the stations to be imported.
3. Import the stations into a new project of the <Temporary> database.
4. Close the <Temporary> database.
5. Open the final destination database. This is the database which contains the coordinate system to be used.
6. On the **File** menu, click **Importing station data**.
7. In the Importing station data dialog box, select **A Project of another WinGLink database**.
8. Browse the directories until you find the database <Temporary.wdb>.
9. Select the database <Temporary.wdb> and click **Next**.
10. Select the project in which the stations were previously saved, then click **Next**.
11. WinGLink will show a warning to inform you that the source and target databases have different coordinate systems, which is correct.
12. Confirm the import with coordinate conversion.
13. The stations will be imported and their coordinates will be converted during the import.

13: Common Functions

Overview

This chapter contains:

- An introduction to WinGLink windows and the utilities available to the user to organize and use their contents.
- A description of common features and functions used for editing, formatting and setting the display options for maps and sections, including colors and line contours.
- A general overview of printing options for both maps and sections

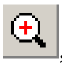


The Display Area

Zoom and Pan Functions

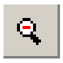
Described in this section are two functions common to all WinGLink modules which display gridded sections or maps, e.g. the Pseudo-Sections and Maps modules: zooming and panning. Both functions can be accessed from the toolbar located below the menu bar of the program modules:

Zooming

To **Zoom In** click , then:

1. Move the mouse pointer to a spot in the map near the zone into which you would like to zoom.

2. Left-click and release the mouse button.
3. Move the mouse pointer to drag a rectangle over the area which is to be enlarged.

After zooming in on an area, the zoom out button  is enabled. This button can be clicked as many times as the **Zoom In** button was clicked, thereby allowing you to move backwards through the previously executed zoom operations.

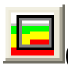
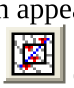
Panning the Display

This function is used to move or "pan" the current display area. Used in combination with the Zoom function, the Pan function allows you to quickly display specific areas of the active window in detail.

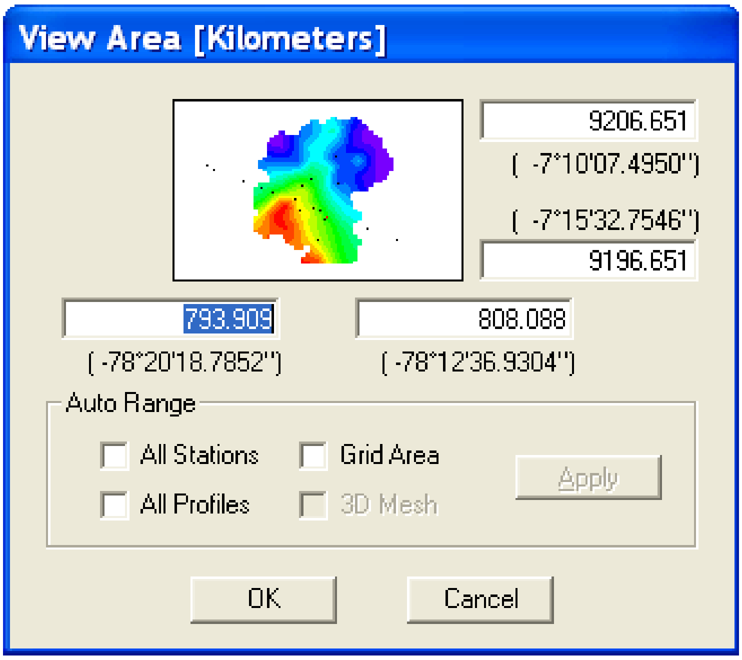
To **Pan** a window**,** click ; then:

1. Move the mouse pointer to a spot in the map near the zone into which you would like to zoom.
2. Move the selection area.
3. Left-click in the section or map to set the new window position.

Setting the View Area

To set the display area, i.e. the area actually displayed in a given map or section window, select the **Window | Set View Area….** menu command to open the View Area window. This window can also be opened by clicking the **Set View Area** button located in the toolbar. Depending on module, this button appears as either:
(sections) or  (maps).

Shown below is a typical View Area window.



Note that the fields for defining the view area dimensions vary from module to module depending on the data being plotted.

Managing Color Fill

This section provides an overview of WinGLink's color management functions. These include functions for enabling/disabling color fill, defining color ranges and working with color palettes.

Enabling Color Fill

To toggle color filling on and off, click the button on the toolbar

- or -

1. Select the **View | Display Options** menu command.
2. In the Display Options window, select the **Color Fill** check box; then click the **OK**.

Note: The color fill range definition is based on the grid of each map. Therefore, color ranges can only be plotted if the map values have been interpolated.

Color Scale

The range distribution is shown in a color scale which is generally displayed at the right edge of each map or section by clicking the button. In the 3D Modeling module, the color scale is displayed in its own window and includes additional functionality (see the "3D Modeling" chapter for details).

The Range Editor

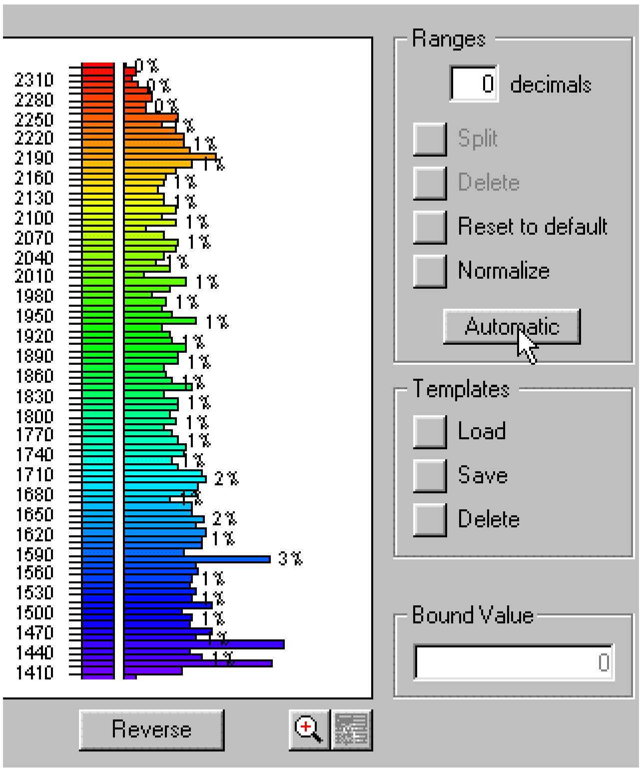
The interpolated values of a map or a section are organized in adjacent ranges. Each range is defined by two numerical bounds (upper and lower bound) and is associated with a color. This color is applied to all values within the range.

On the **Contours** menu, select **Color Ranges** to open the Range Editor.

Defining the Number of Ranges

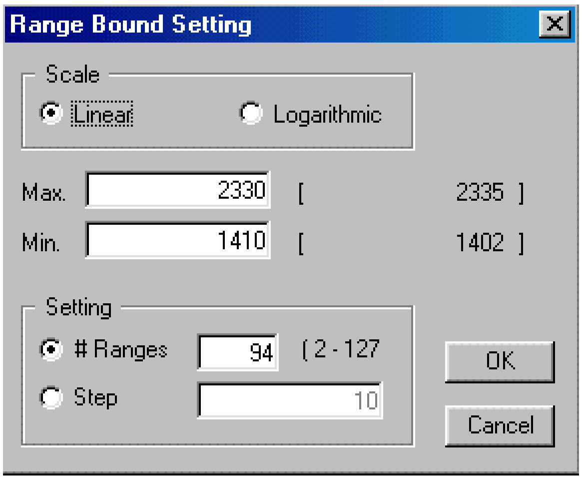
By default, the Range Editor creates 32 equally spaced intervals based on the difference between the maximum and minimum values. The maximum allowed number of ranges is 128.

The number of ranges can be changed at any time by using the **Automatic** button in the **Range Editor**:



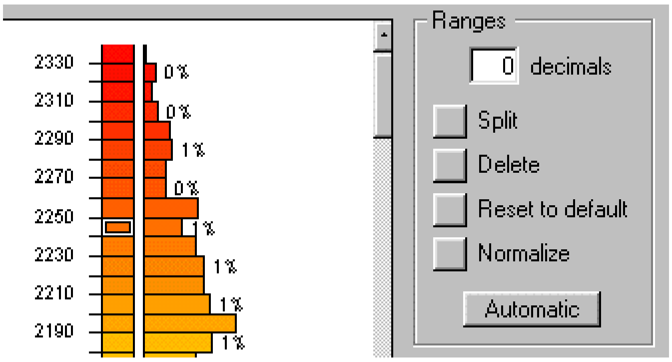
Another dialog box opens in which the user can enter the desired number of ranges and the criteria to be used for automatically generating range bounds:

When using a logarithmic scale, make certain that the number of decimals in the main form is high enough to resolve the definition of the bounds. If necessary, increase the number of decimal in the Range Editor dialog



Adding or Deleting a Range

In the Range Editor, click the mouse on the color box of the range you wish to edit. The box is highlighted:

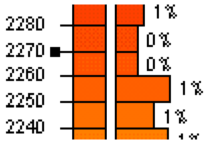


Choose **Split** or **Delete** from the Ranges frame.

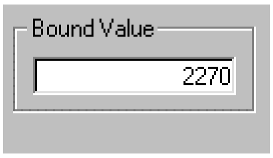
Editing Range Bounds

Range bounds can be edited by:

- Changing the number of ranges
- Selecting the **Normalize** button in the **Ranges** frame. This option keeps the number of ranges unchanged, but resets all range bounds to achieve an equal distribution of the histogram.
- **Manually** editing the range bounds. This is done by clicking the mouse on the line which separates two ranges:



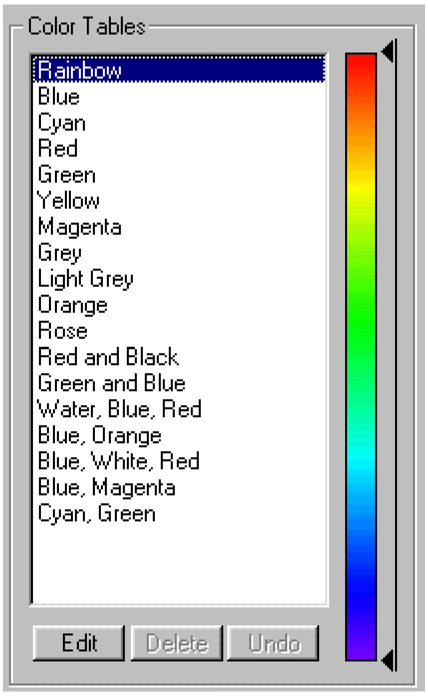
The bound value is shown in an edit box.



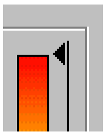
Enter the new value, making sure it is compatible with the adjacent bounds.

Selecting and Calibrating Color Palettes

Color palettes can be selected in the right-hand frame of the main Range Editor:



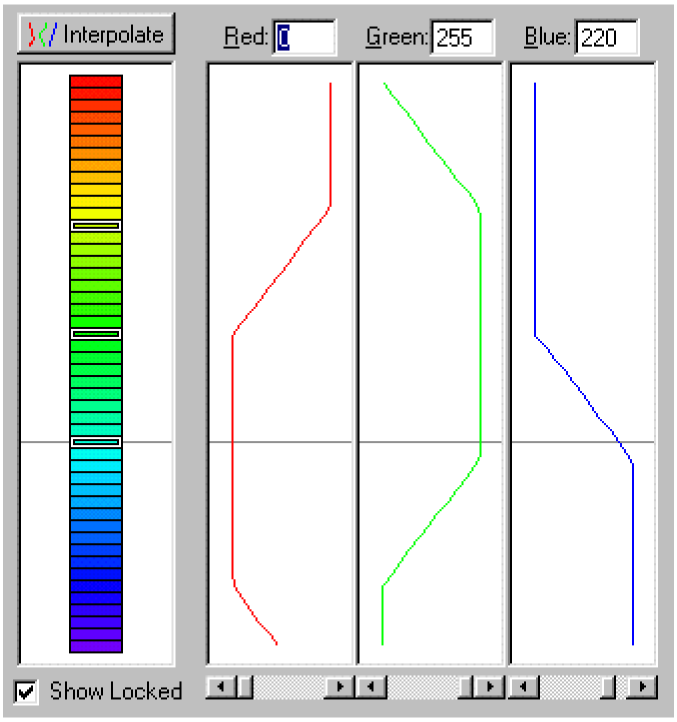
The range of the color table can be modified by clicking the upper or lower markers:



and then dragging them to a new position. The color table is then restricted to the area between the two markers.

Editing Color Palettes

1. Use the **Contour | Color Ranges** menu command to open the Range Editor.
2. In the Color Table frame, click the **Edit** button to open the following window:



Each color can be modified by double clicking and choosing the new color from a standard palette or by editing the RGB components using the graphic tool on the right-hand side of the color table.

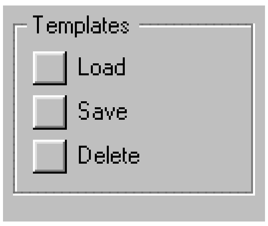
The colors with a border have been locked by clicking the defining box while holding down the **Shift** key. By clicking the **Interpolate** button, a new color table is generated by interpolating the colors between the locked colors.

3. To save the new color table, click **Save as New**, and enter a name for the new palette. **Note:** Default palettes cannot be modified and saved with the same name.

Managing Templates

Templates are fixed combinations of color palettes and range bounds which can be saved in the database for use in other map or section presentations.

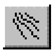
The **Template** frame:



is always available in the Range Editor. Its use is self-explanatory.

Contour Lines

Displaying Contour Lines

To toggle the display of contour lines on and off, click the  button on the tool bar

or:

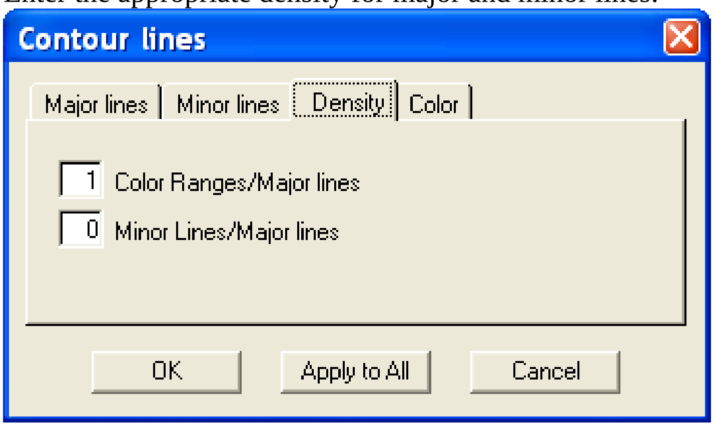
1. Select the **View | Display Options** menu command
2. In the Display Options dialog box, select the **Contour Lines** check box.
3. Click the **OK** button.

Setting Contour Lines

WinGLink has two levels of contour lines which can be formatted independently of one another when displaying a map: major contour lines and minor contour lines

To set the number of major and minor contour lines:

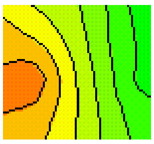
1. Select the **Contours | Contour Lines…** menu command.
2. Click the **Density** tab.
3. Enter the appropriate density for major and minor lines:



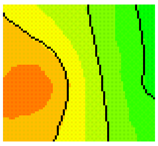
Major Contour Lines

These are contour lines which are defined with respect to the color ranges set for the map. The user chooses how many color ranges should be contained between two major contour lines.

Shown below is a sample display with the value 1 entered for **Color Ranges/Major Lines:**



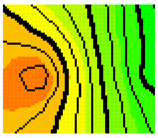
And this is what is displayed with the value 2 entered for **Color Ranges/Major Lines**:



Minor Contour Lines

These are contour lines which are defined with respect to major lines. Use this option to select how many minor contour lines should be present between two major contour lines.

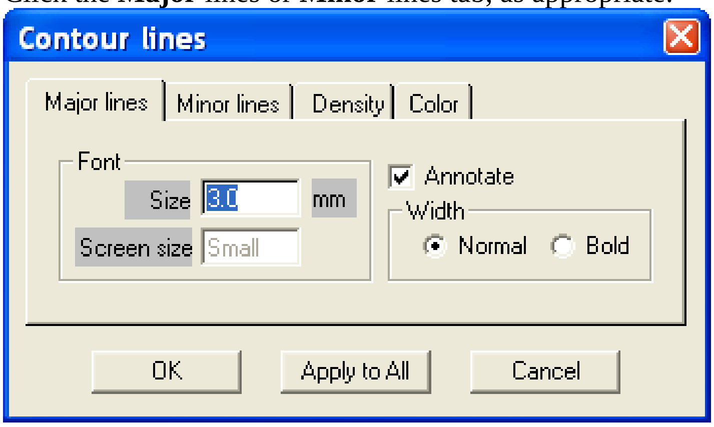
2 minor lines/major line (bold):



Formatting Contour Lines

To change the format of major or minor contour lines:

1. Select the **Contours | Contour Lines…** menu command.
2. Click the **Major** lines or **Minor** lines tab, as appropriate:



3. Select the **Annotate** check box to display the contour line value.
4. Choose the font size for the annotation.
5. Choose the contour line **Width**.
6. Click either the **OK** or the **Apply to All** button. **Note: Apply to All** is available as an option only if multiple windows containing contours are displayed on the screen.

Setting Decimals in Contour Annotation

The number of decimals shown in the contour lines annotation depends on the ranges defined for the map.

To Change the Number of Decimals:

1. Select the **Contours | Color Ranges…** menu command to open the Range Editor.

2. In the **Ranges** frame, enter in the decimals field the numbers of decimals you would like to display (set this number to 0 if you do not want to display any decimals).
3. Click the **OK** button.

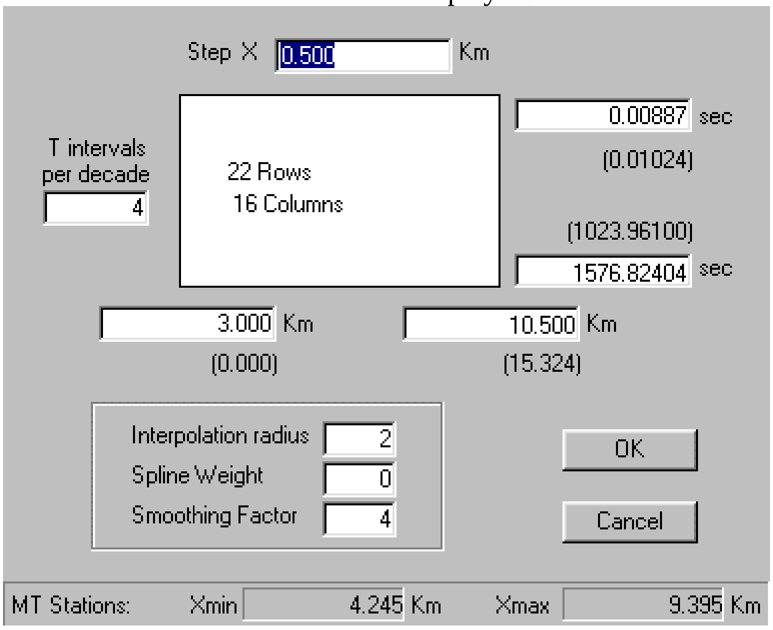
Gridding

In order display contours and gridded colors in maps and sections, station values must be interpolated on a regular grid. This operation creates a two-dimensional array of values, also referred to as a **grid**. Each of the WinGLink module with gridding functionality includes a Gridding window which is displayed the first time a new dataset is to be displayed. This window can also be opened by selecting the **Gridding | New Grid** menu command.

Although the gridding parameters vary according to data type, the principle remains the same regardless of in which program the Gridding window is opened.

To Grid Station Data

1. Select the section or map to be gridded.
2. Select the **Gridding | New Grid…** menu command. A form similar to that shown below this is displayed:



3. The numbers in parenthesis on the gray background show the maximum and minimum values of the data. The current value to be used to define the grid extent must be entered in the entry field adjacent to each number.
4. In the **Step X** field, enter the step size in the specified units. This number determines the number of columns in the grid, which will be displayed in the center white box. This white area represents the section.

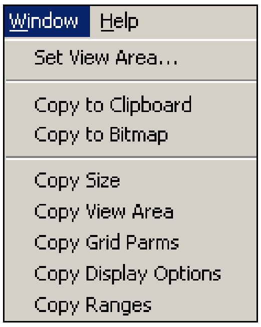
5. In the **T intervals per decade** field, for the case of a pseudo-section grid, enter the number of grid rows per decade.
6. Fill the fields with the values for the interpolation radius, spline weight and smoothing factor.
7. Click the **OK** button to generate and draw the grid.

Note: As the interpolation radius increases (values >5), the gridding algorithm attempts to fill any holes present in the data by interpolating between relatively distant data points. In some cases, such as when creating difference pseudo-sections, we recommend using a low interpolation radius (2) to prevent the creation of misleading pseudo-sections.

Copying Display Parameters

The individual application programs which comprise **WinGLink** function by opening multiple windows, each one containing a different map or section. For example, in the MT Pseudo-Sections program, a number of different sections may be displayed simultaneously, each one representing different components of one or more datasets.

Nearly all WinGLink modules include a number of functions which can be used to apply the display settings of one window (e.g. one x-section display) to one or more other currently open windows. For example, if the view area of one x-section is set to cover a selected length of a profile, say from 2 – 10km, and all other windows currently display the entire profile length, the display area of the other windows can be adjusted by first selecting the "master" window, then using the **Window | Copy View Area** menu command to activate the function, and then clicking each window which is to be adjusted to the view area of the "master" window:



WinGLink provides a number of commands which function in an identical manner. The available commands vary from module to module, but generally include functions such as:

- Copy Size
- Copy View Area
- Copy Grid Parameters
- Copy Display Options
- Copy Ranges

Copying Screen Contents

For display and presentation purposes, it is often necessary to capture the screen contents. All WinGLink modules provide two functions for this purpose:

- **Copy to Clipboard**

and

- **Copy to Bitmap**

The **Copy to Clipboard** utility, accessed with the **Window | Copy to Clipboard** menu command, copies the current map or section to the clipboard for pasting into other Windows-based applications.

The **Copy to Bitmap** utility copies the current map or section to an external file. After selecting the **Window | Copy to Bitmap** command, you are prompted to enter a file name under which the image is to be stored.

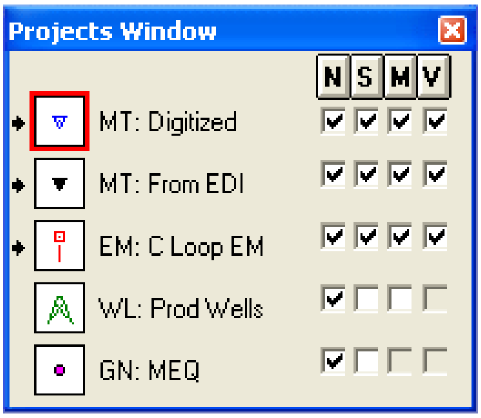
Editing and Formatting Stations

Setting the Display Options for the Stations of a Project

An ancillary window, called the Projects Window, can be opened to control the display and the editing of the stations of each project.

On the **View** menu, select **Projects Window.**

A typical *Projects Window* looks like this:



For each project:

Check box:	To apply the corresponding option to all project stations:
N	Display names
S	Display symbols
M	Display 1D model (X-Sections only)
V	Post vertical dataset values

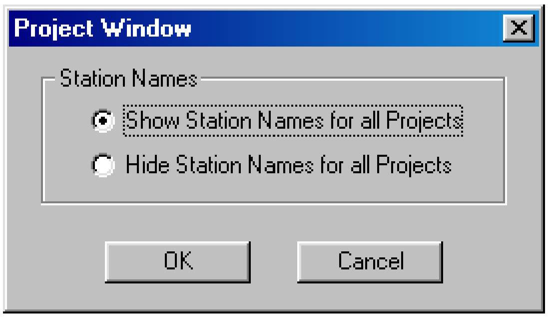
Note: The new settings are applied to all windows but ***do not become active until a given window is clicked***

Applying the Same Display Options to the Stations of All Projects in Maps and X-sections

Global commands are available to set the same display options for the stations of all projects displayed in a map. For example, to show the station names for ***all*** projects in a map or X-section

Open the Project Window using the **View | Project Window** menu command, then click the heading **N** :

The following dialog box opens:

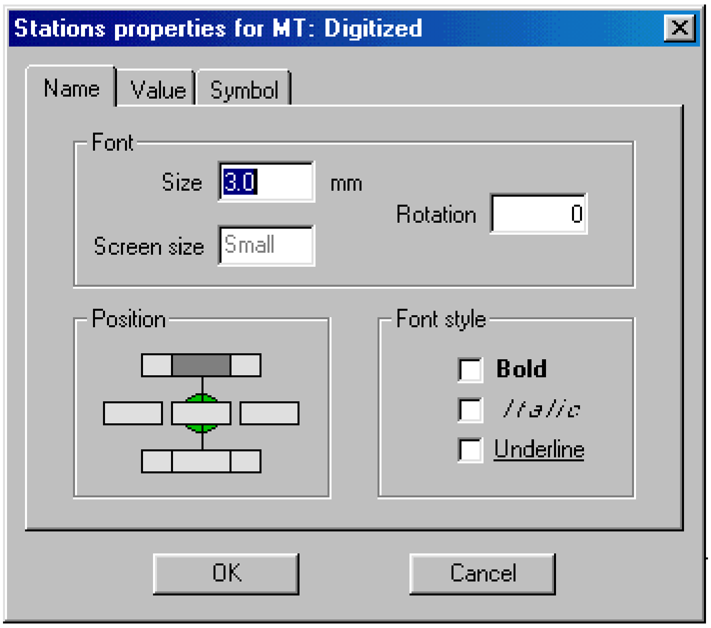


Formatting the Appearance of the Station Names, Symbols and Posted Values for Each Project

Stations can be displayed with different font size, color and symbol by saving the settings in the Project Station Properties.

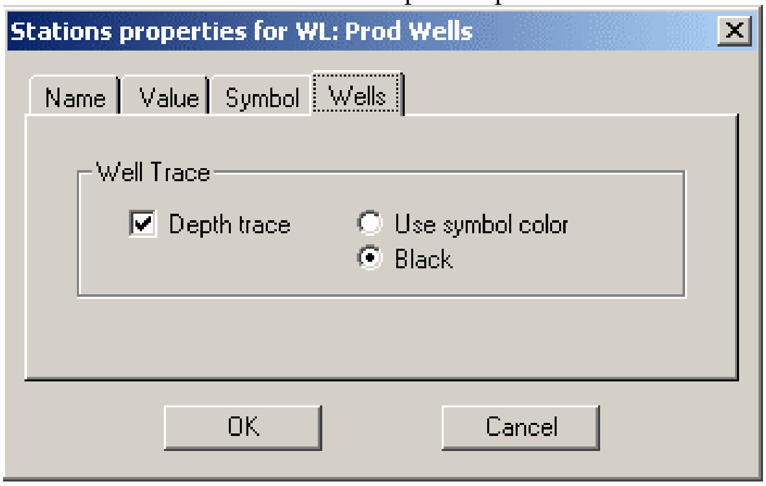
To Edit the Stations Properties for a Project:

1. Select the **Stations** | **Properties** command or:
2. Open the Project Window using the **View | Project Window** menu command.
3. In the Project Window, **click** the symbol of the project whose stations you want to format. The symbol of the active project will be displayed with a bold red border
4. **Double-click** the project symbol **or** select the **Stations | Properties** menu command.
5. The Station Properties dialog box opens, similar to the following:



6. Select the appropriate tab and set fonts and symbols for the project stations as desired.

The Station Properties window for projects of type Wells (WL) contains an additional tab which includes options specific to well courses:



The station format used in the Maps program can be different from the station format used in the sections programs.

The format settings saved for the stations of a project in a map are used for every map containing the stations of that project. Different settings can be saved for displaying sections, but the settings saved for one section will be used on all sections.

The Category and Pattern Editors

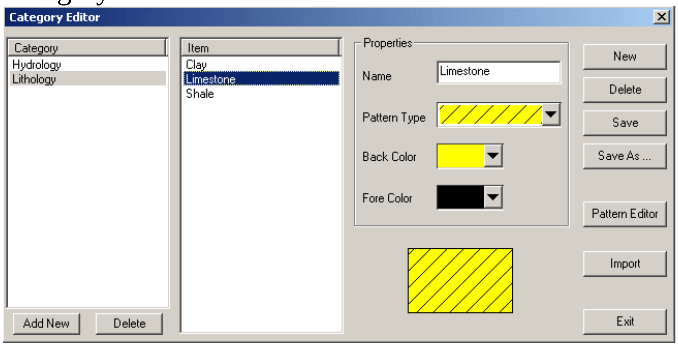
WinGLink's Category and Pattern editors are used to create patterns and assign them to specific categories. These features are used, for example, to associate a specific pattern to a given layer of strata in a well or to assign a pattern to a specific region in a map or section in the Interpreted Views and Montage (IVM) program.

In addition to IVM and the Well Editor, the Category and Pattern editors is available in the X-Sections and 2D Inversion modules. The categories are centrally stored in the database. Thus, regardless of the program in which categories are edited or new categories created, the modifications are reflected throughout the database.

Once categories, or types of layers, have been created, any time another layer of that type is encountered in a section within the database, WinGLink automatically performs filling.

To Assign Fill Patterns to Layer Data:

1. Select the **Tools | Category Editor** menu command to open the Category Editor:



   Displayed in the Category frame are all existing categories. Categories are created automatically when importing layer data into WinGLink and can also be created using the **Add New** button located at the bottom of the left column.

   The elements in the **Item** frame are the description elements contained in layer-data files which were linked to the given category type during import. New items can be created using the **New** button located in the upper right corner of the Category Editor.

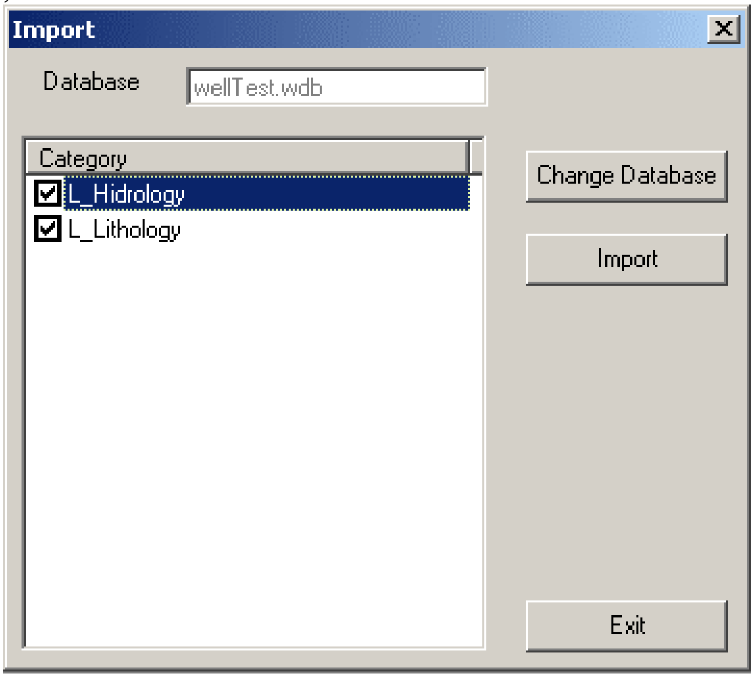
2. Select a category and item with the mouse. The current pattern type is displayed in the **Pattern Type** dropdown box.

3. Click the arrow at the right end of the **Pattern Type** dropdown box to open a box containing all available fill patterns. Choose back color and fore color for the filling using the provided drop-down boxes.

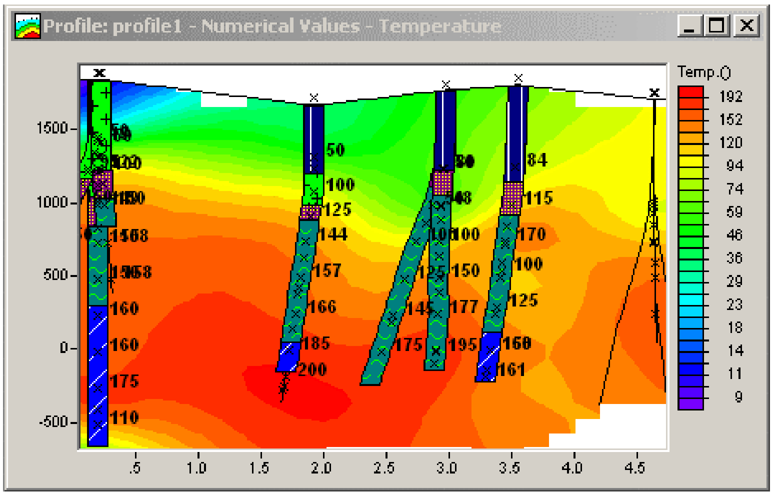
4. The Pattern Editor, described below, can be used to create new patterns. This editor can be opened using either the button provided in the Category Editor or by selecting the **Tools | Pattern Editor** command on the main menu.

The **Import** function, which is called up using the **Import** button in the Category Editor, provides a function for importing entire categories from other databases. If you import a duplicate category, the Category Editor can be used to merge the two categories together.

;



Upon closing the Category Editor, well courses are filled with the appropriate fill patterns:

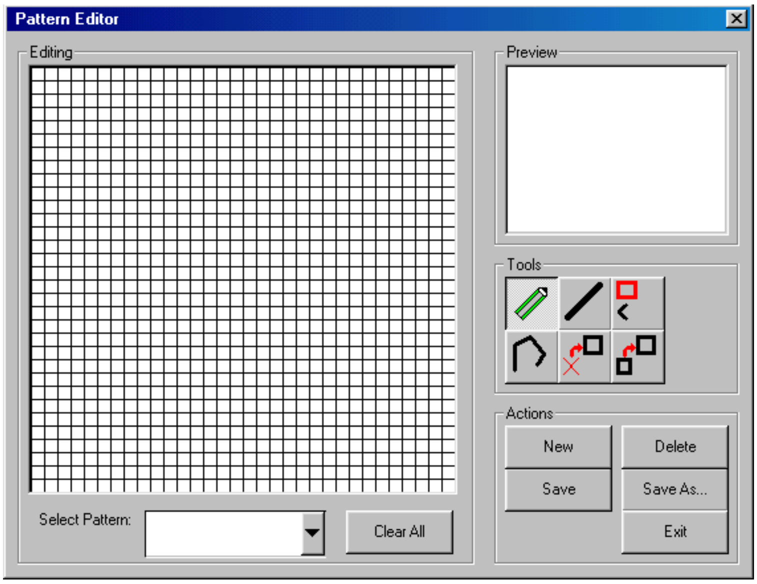


The Pattern Editor

The Pattern editor is used to create and edit fill patterns. Theses patterns can be used, for example, to fill shapes used in cultural data layers or as fill patterns to describe layers of vertically distributed data, e.g. wells. Patterns created in the Pattern Editor are available in the property windows of all elements which support fill patterns, both in IVM and in other WinGLink modules. In the main WinGLink modules, this feature is available only for sections which contain well courses, e.g. X-Sections.

To open and use the Pattern editor:

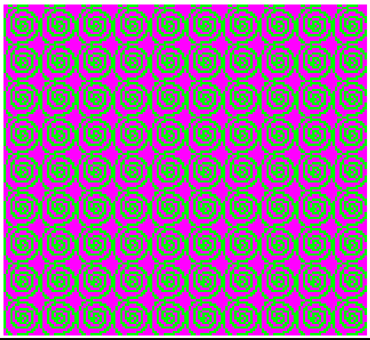
1. Select the **Tools | Pattern Editor** menu command to open the Pattern Editor:



2. To create a new pattern, position the mouse over the Editing window, then click and hold down the left mouse button. Slide the mouse across the edit window to create a pattern. You may alternatively click individual squares to create a pattern. When finished, click the **Save** button.

3. To edit an existing pattern, open the **Select Pattern** dropdown list. Make any changes to the pattern, then click the **Save** button to overwrite the pattern with your changes, or **Save As…** to save your changes to a new pattern.

Note: The only color available in the pattern editor is red. When using patterns as a fill, you may, however, select both foreground and background colors when assigning the pattern in the Category Editor.

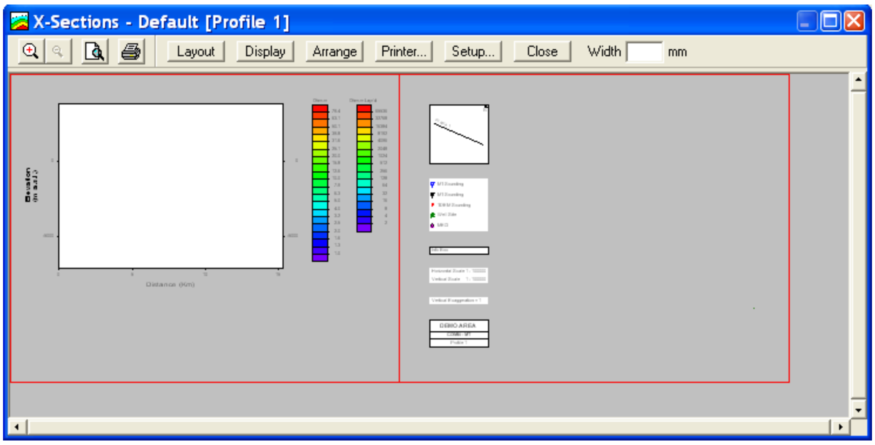


Printing

Overview

To print the section or map displayed in the active window, select the **File | Print** menu command**.** This opens another window, the Layout

Definition window. Here, the map or section is represented by a white rectangular box, surrounded by axis tic marks.



Other boxes are shown representing the:

- **color scale**: use the mouse to move or resize this item.
- **legend, info** and **title box:** these items can be moved, resized and edited.
- **scale** and **scale bar:** this item can be moved, resized and edited. To edit, enter the desired scale and choose from different scale bar formats.

Information on changing display parameters of these elements is provided further below.

A **red border**  indicates the ***paper size*** with respect to the size of the items to be printed. To change the paper size, use the **Printer** button. This button is also used to set the printer

Margins and page orientation are set using the **Setup** button.

Each box can be sized and edited interactively until a satisfactory output layout has been obtained. The appearance of the final presentation can be checked by pressing the **Preview** button.

When ready, click the  button to send the map or section to the printer.

Setting the Printer

On the Layout Definition form, click the 

button.

Setting up the Page

To set the page margins:

1. Click the **Setup** button.
2. Set the margins and the orientation and click the **OK** button.

To set the page size:

1. Click the 

button.

2. From the dropdown list, choose the printer and click the **Properties** button.
3. The printer driver dialog box opens; select the paper size.
4. Exit the printer driver dialog box.

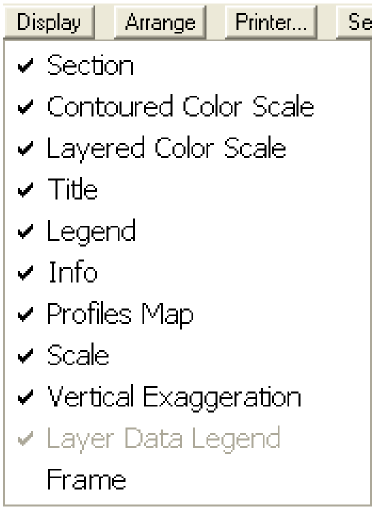
Modifying the Print Layout

For a given map or section, a print layout consists of the position of the **items to be printed** (and whether or not they will be printed) as well as their positions on the printout and their sizes. These items are represented by either white rectangular boxes in the layout page or, for the smaller items, by scaled-down versions the items. These include:

Map, Title, Legend, Info, Color scale, Map Scale, North Arrow (maps only)

After you have defined a layout, you can save it and apply it to the same or other maps at a later time.

Items included in the layout can be selected by clicking the **Display** button at the top of the window:



Use the mouse to select/deselect the items to be displayed.

Moving and Resizing Items on the Print Layout

Any of the print items can be moved with respect to the page margins and resized.

To Move and Resize Objects Using the Mouse:

1. Click the item; a white dot appears next to each of the vertices. The mouse pointer changes to a four headed arrow.
2. To move the item, left-click the item and move the mouse while holding down the button; release the button to drop the item.
3. To resize the item, move to one of its vertices until the mouse pointer turns to a double-headed arrow; left click and hold down the button; move the mouse to resize the item and release the button when the desired size has been reached.

To Move and Resize Objects Using the Keyboard:

1. Click the item; a white dot appears next to each of the vertices. The mouse pointer changes to a four headed arrow.
2. To move the item, hold down the **CTRL** key and press one of the cursor keys.
3. To resize the item, hold down the **SHIFT** key and press one of the cursor keys.
4. To move or resize multiple items, click the items while holding down the **SHIFT** key and follow steps 2 - 3.

To Change Display Parameters

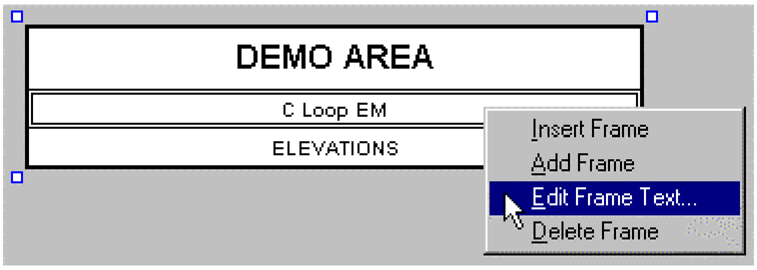
Double-click any of the screen items to open a parameter window in which the display parameters for the given element can be edited.

Editing the Textual Print Items

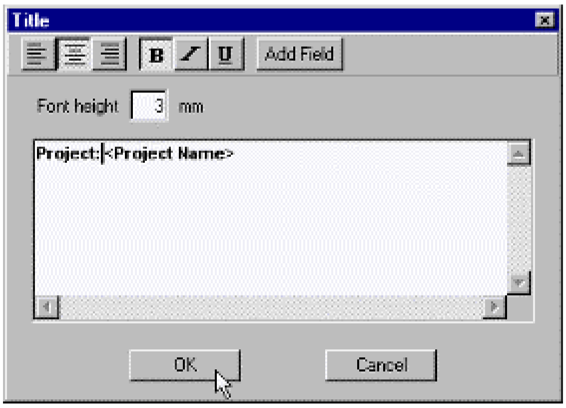
Each of the **Title**, **Legend** and **Info** items can be subdivided into one or more frames in which text can be edited and formatted. Frame text can include fields linked to the active database, e.g. area name, equipment used, etc.

To Edit the Contents of the Title, Legend or Info Print Items:

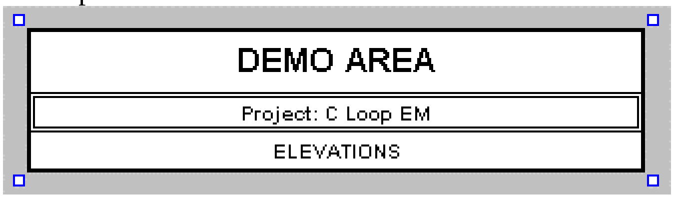
1. Right click the item frame to be edited. On the shortcut menu, select **Edit Frame Text…**



2. Edit the frame text. Words enclosed in <> signs are fields taken from the database. Additional fields may be added by using the **Add Field** button.



3. You can format the text using the buttons located in the toolbar at the top of the window. Click **OK**.



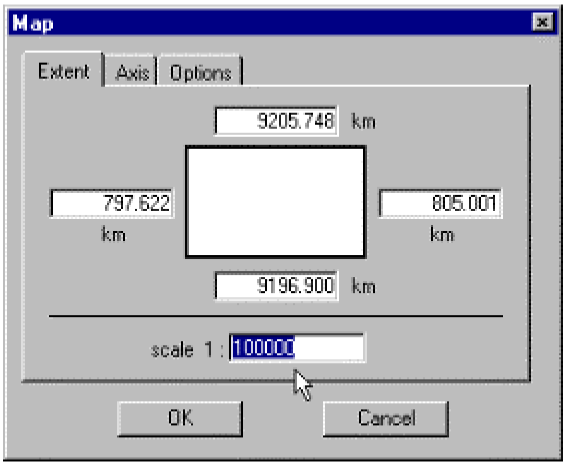
Frames can be added and removed. **Insert frame** adds a frame just before the frame which was right-clicked; **Add frame** adds a new frame in the last position.

Changing the Scale

The map scale can be redefined. If the scale is changed, the size of the map on the print pages will be modified accordingly and vice versa.

To Redefine the Scale:

1. Double-click the map scale.
2. Enter the desired ratio in the scale field.

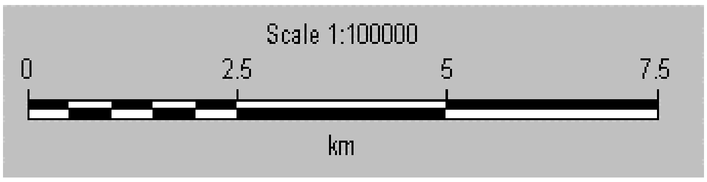


3. Click the **OK** button.

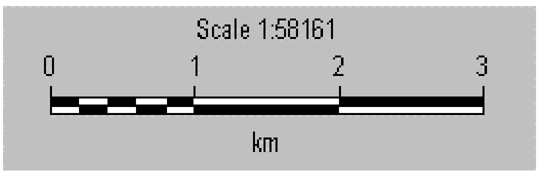
The scale also changes if the map item is resized.

Example

Before map resizing



After map resizing



Splitting the Printout Over Several Pages

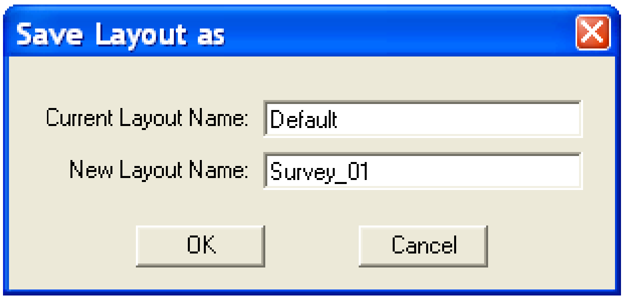
A map or section may be enlarged in such a way that, when sent to the printer, it is split and printed on more than one page.

Whenever you drag an item outside the boundaries of a page or resize an item so that it falls outside the page, new pages are added to the layout in order to print all the items.

Saving a Layout

To Save a Print Layout:

1. Click Layout and select **Save as…**

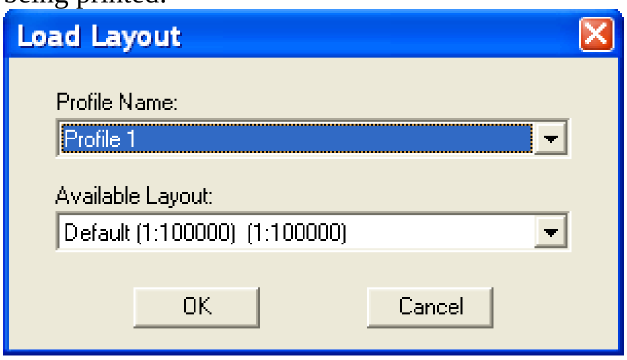


2. Enter the new layout name and click the **OK** button.

Loading a Layout

To Load a Previously Saved Print Layout:

1. Click Layout and select **Load…** .There can be more than one layout for each value type. Note that you may select a layout which is associated with a value type different from the one being printed:



2. Select the source and one of the available layouts; click the **OK** button.

Printing to File

In addition to the direct output to a selected printer, the Print functionality found throughout WinGLink can be used to send printer output to a number of file formats:

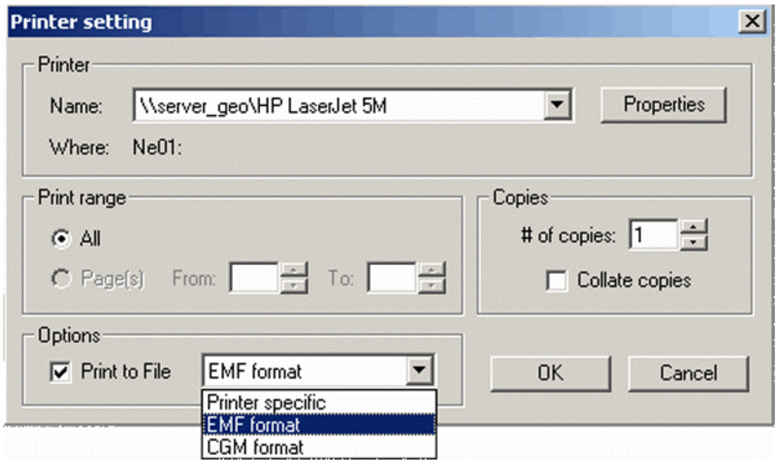
- Printer specific (e.g. *prn, *ps). The printer-specific format is dependent on the output types supported by the printer driver. A PostScript (ps) file can be output by printing to a

PostScript printer driver, regardless of whether a PostScript printer is actually connected to the computer.

- EMF
- CGM
- PDF

To Print to Files of Type Other Than PDF:

1. Select the **File | Print** menu command as you would to print to a printer.
2. Select the **Print to File** checkbox in the **Options** section of the Printer Setting dialog box.
3. Select the desired file format from the dropdown list:



Important:
When printing to EMF of CGM files, note the following: Although output is not sent to a physical printer, the paper properties of the printer selected in the Printer section of the Printer Settings dialog are used. Each output file can be considered to be a page of printer output. Because the entire output is sent to a single file, however, you must ensure that the paper size selected for the printer is large enough for the entire print area, i.e. fits within the red border which defines the first page of output in the Print Preview screen. Any information outside of the first page is clipped in the output file.

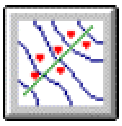
To Print to PDF Files:

The driver used to create PDF files, the Acrobat Distiller available from Adobe, functions in the same way as a regular printer driver. PDF files can be created only if this software has been installed on your system.

1. Select the **File | Print** menu command as you would to print to a printer.
2. Select the **Acrobat Distiller** printer from the **Printer** section of the Printer Settings dialog box and set the properties as required by clicking the **Properties** button.
3. **Do not** select the **Print to File** checkbox in the **Options** section.

14: Maps

Overview



The primary uses of the **Maps** program are to:
- manage stations
- create profiles
- create and display contoured and color-filled maps of the different values associated with the stations of a project

In this chapter we will first present an overview of how the program creates maps and will then provide general instructions on how to create maps, including information on creating and working with profiles as well as managing stations. The chapter will conclude with detailed information on working with specific data types in the Maps program.

Getting Started with the Maps Program

This section provides an overview of basic functions of the Maps program, including:

- Creating a new map
- Creating a profile
- Assigning stations to a profile

This is the bare minimum for preparing a WinGLink project for use with section-based modules, such as 2DMT, X-Sections and Pseudo-Sections.

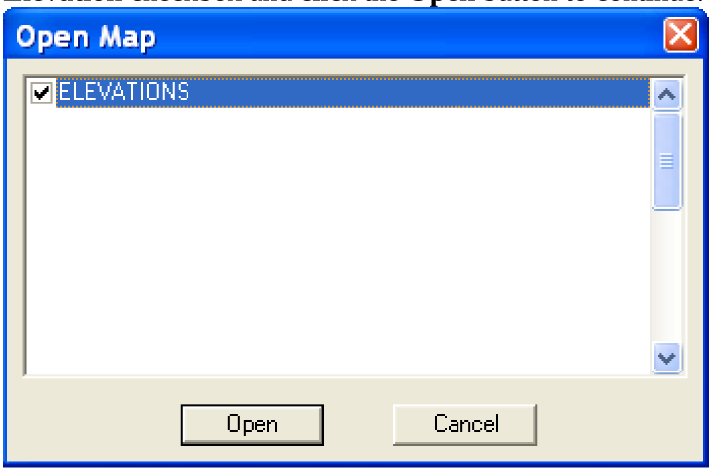
Detailed information on creating specific map types, managing stations and working with specific data types is provided in the sections which follow.

Creating a New Map

The first time the Maps program is started for a given project which already contains stations, the program opens with a dialog box which contains options for creating standard map types for the given data type. At a minimum, this dialog box contains an option for creating an elevation map. When beginning work with a new project, the elevation map is a often good place to perform the initial “administrative” work which, depending on data type, needs to be taken care of with a new project, i.e. profile creation, assignment of stations.

Starting the Maps Program and Creating an Elevation Map:

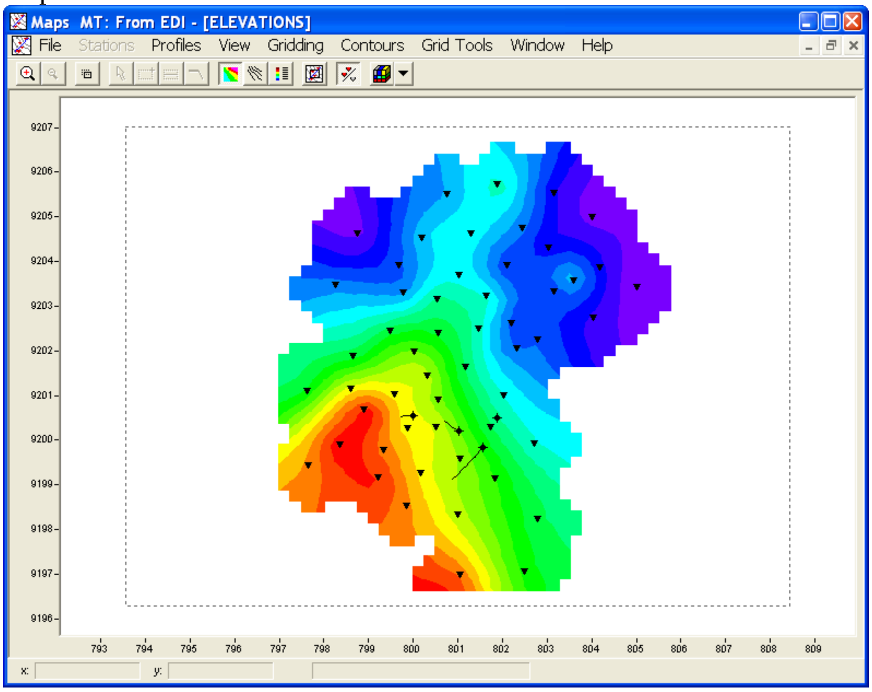
1. In the WinGLink Shell, use the mouse to select the project for which a map is to be created.
2. Click the Maps icon.
3. The program starts with the Open Map dialog box. Select the **Elevation** checkbox and click the **Open** button to continue:



4. If creating the elevation map for the first time, the program now prompts you to select the gridding options:

The **Grid using default parameters** option generally provides suitable gridding results. Select the second option to set the individual gridding parameters. Information on setting these parameters can be found further below in the "Map Grids" section of this chapter and in the "Common Functions" chapter of this manual.

Provided the project contains at least two stations with elevation values, the Maps program opens with the elevation map for the data:



Creating and Using Profile Traces

A profile trace is a polygonal line added to the database area and to which individual stations can be associated. A profile trace added to a map is available to all projects in the database and can be used for:

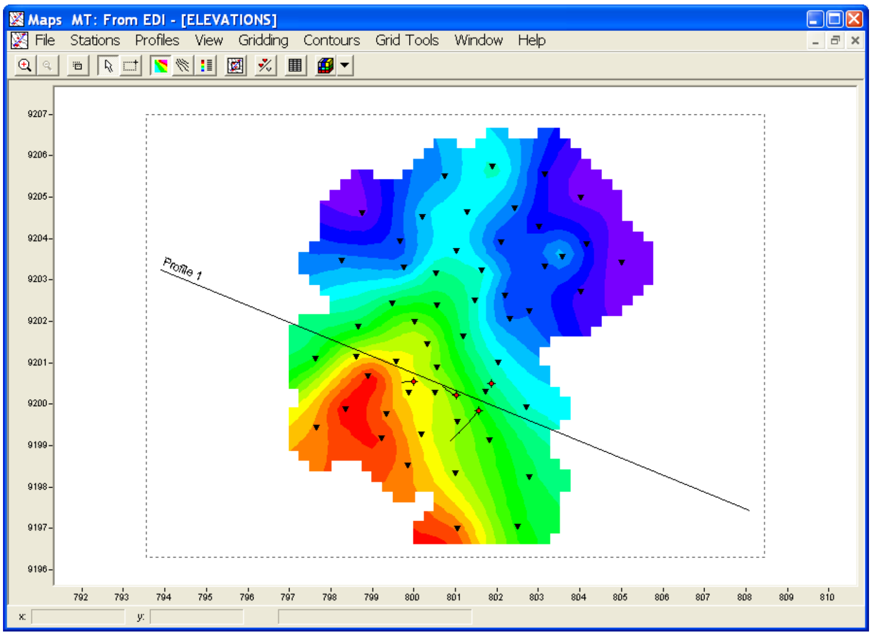
- **Constructing sections along the profile** by adding stations to the trace and using the station datasets to build different types of sections (examples: pseudo-sections, imaged sections, etc.).
- **Extracting an interpolated value profile** from the grid of a map (example: interpolated Bouguer anomaly profile).
- **Constructing a value profile by adding stations** to the trace, and plotting the station values vs. the distance along the profile (example: Bouguer anomaly profile using station values).

The use of the ***same profile trace*** for all type of sections is the basis for the integrated display of interpretive sections. For example, a 2D gravity model can be easily superimposed on a resistivity cross section because they are referenced to the same profile trace.

Adding a Profile Trace

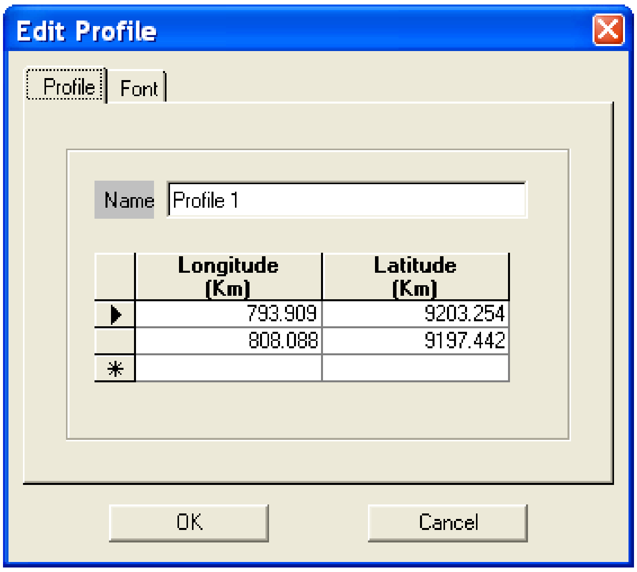
1. Select the **Profiles | Add profile trace** menu command.
2. Move the mouse pointer to the location at which you would like to insert the first node of the trace.
3. Click the left mouse button, release it and move the cursor to the next node.
4. To draw more nodes, repeat the above step; click the right mouse button to add the last profile node.

5. Enter a name for the profile trace.
6. If you would like to add another profile trace, press the **Next** button and repeat steps 2-5.
7. Click the **Close** button.



Editing the Nodes of a Profile Trace

1. Select the **Profiles | Edit profile trace** menu command.
2. Click the profile trace you would like to edit.
3. Edit the name and/or the longitude and latitude values of the nodes which define the trace:



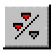
4. Click the **OK** button.

Adding Stations to a Profile Trace

This section explains how to associate ***existing*** stations to profile traces in order to construct value profiles or sections. For help on how to insert

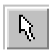
new stations at a regular spacing along a profile trace, see “Inserting New Stations Along a Profile Trace” in this chapter.

The basic commands used to construct a section along a profile trace are activated by pressing:



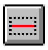
To Switch to Profile Mode

1. Click the profile trace which to you would like to add stations; the profile trace turns **red** to indicate that is **selected**.
2. If the selected profile trace consists of more than one segment, the **unselected** segments are displayed in light **green.**



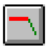
To Add One Station at a Time

1. Click the arrow button and then click the symbol of each station to be added to the profile.
2. Added stations turn **red**.



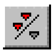
To Add All Stations Located Within a Given Distance from the Profile:

1. Click the button shown above, then left-click the profile trace and hold down the mouse button.
2. Resize the band by moving the mouse pointer away from the profile trace
3. Release the mouse button when band has reached the desired size.
4. Selected stations turn **red**



To Select Another Profile or Profile Segment

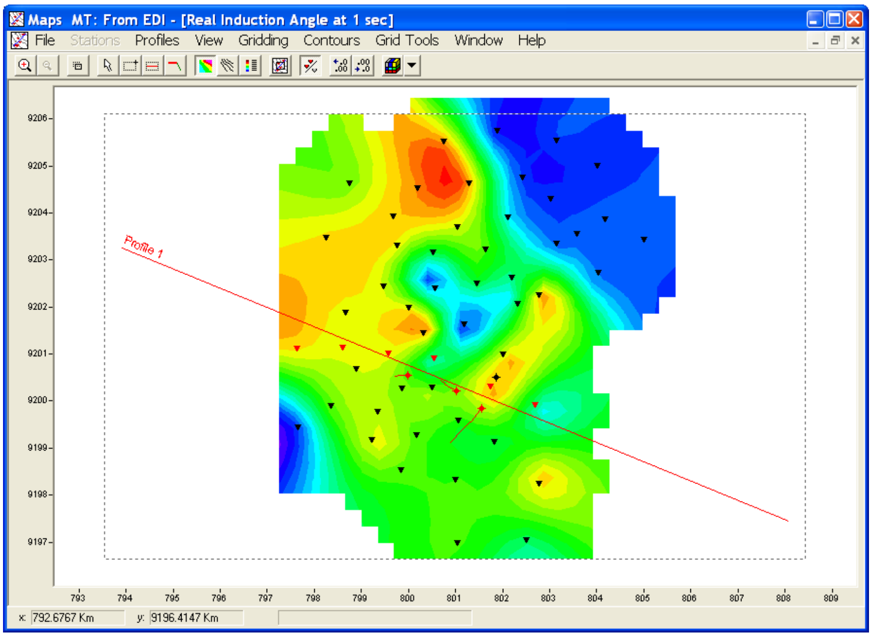
1. To activate another profile, click the button shown above, then click the new profile trace. For multi-segment profiles, each segment is edited separately and should be edited as a separate profile



To Return to Station Mode (or select Profiles | goto Station Mode)

1. Click the button shown above to return to station mode. Alternatively, you may use the **Profiles | goto Station Mode** menu command to return to station mode.

Shown below is a selected profile in profile mode with attached stations:



Note: The program can handle over 32,000 stations on a single profile.

Creating Maps

Each time the Maps program is launched, the stations of the project which is currently highlighted in the WinGLink Shell are loaded. For multiple projects, i.e. integrated projects, the stations of all member projects are loaded.

When running the Maps program, you may open many maps at the same time, one for each type of value. For example, for gravity data, maps may be open for elevation, observed gravity, terrain correction – among others. New maps can also be created by importing, computing or extracting new values. The resulting maps can then be saved and reopened during subsequent sessions.

The user is presented with a list of all maps which have been saved during previous sessions when the Maps program is launched as well as given the option of creating standard maps available for the given data type.

Items Displayed in a Map

Maps are displayed by combining the display of different items, which the user can interactively modify by editing the respective data and properties. The items represented in a WinGLink map are:

Item	Meaning
Stations	The stations contained in the current project and those of any attached project. Each project can be assigned a different symbol to identify its stations. Stations are plotted with their names, values and symbols.
Color Ranges	For each map, the value distribution is organized in ranges of different colors. Default color palettes are available. The user may also create new or modify existing palettes.
Contour Lines	Annotated contour lines are drawn based on the defined value ranges. The user can define the contour interval and the line thickness.
Profiles Traces	Polygonal lines can be added to the area to serve as a base for profile and section construction. Profiles are elements of a map which are shared by any other maps contained in the database.
3D MT Meshes	3D MT meshes, in the Randy Mackie format, which have been imported into a WinGLink database, can be superimposed onto MT project maps.

Parameter Values Used to Generate Maps

Each map contained in a WinGLink database is defined by the name of the displayed parameter, e.g. elevation or observed gravity. The values used to build each map are:

1. The values of the parameter at each station of the project
2. The interpolated values of the same parameter

whereby interpolated values are added to the maps by one of the following methods:

- interpolating the parameter values at each station
- importing an external grid
- transforming an external grid

Creating Standard Maps

As will be described in more detail, each project type has associated with it certain standard maps. These maps are available for all projects of a given type. For example, the standard maps available to gravity projects include elevation, observed gravity and terrain correction. They represent fixed, measured values. The standard maps available with a given project type are listed in the Open Map dialog box which appears when the Maps program is started. This dialog box can also be opened with the **File | Open Map** menu command.

Creating New Map Definitions

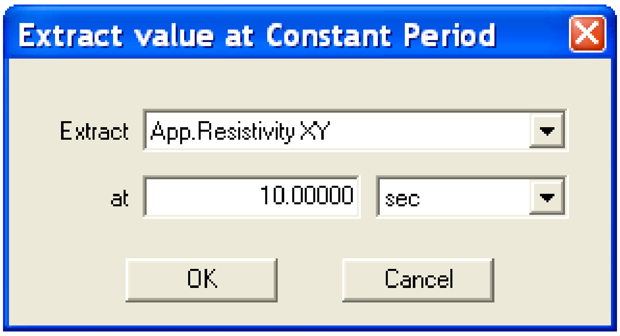
About Map Definitions

In addition to the standard maps mentioned above, it is possible to generate maps using values extracted from a given data set. With MT data, for example, it is possible to extract a resistivity map for a specific tensor component at a given frequency. The number of possible maps is, therefore, limitless. Each time a new map is created in a project, the definition of the parameter values used to generate the map is saved in the database and made available to all other projects. The same map can then be created using the data contained in any other project, if available.

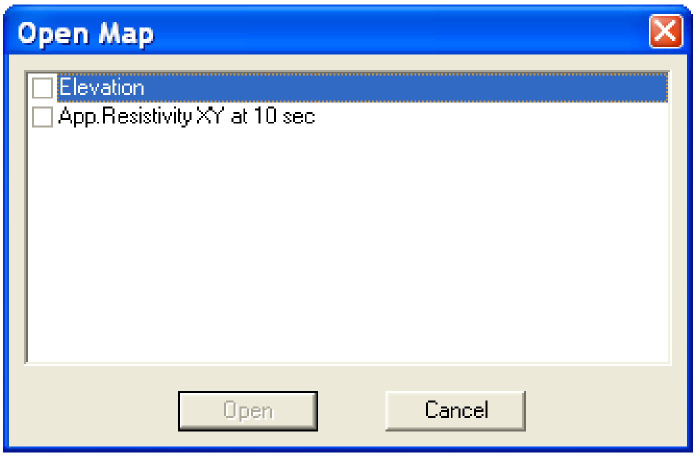
Assume, for example, that we have a database which contains two different MT projects:

Prj_1 and Prj_2

We create the map of "Apparent resistivity at T=100 sec" by extracting it from the station datasets of Prj_1:



This map definition will then be available in the list of existing map definitions when the Maps program is started for the Prj_2 project, or any other MT project:

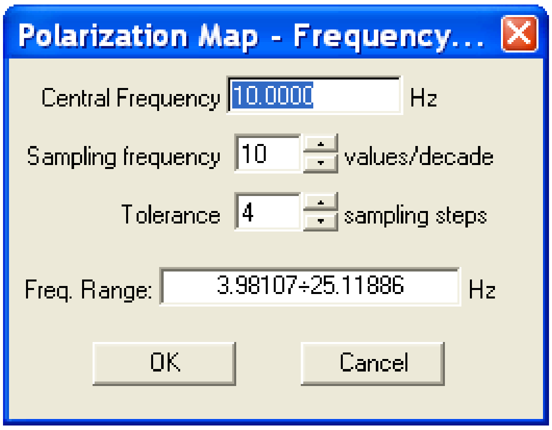


Thus, the same map definition ("Apparent resistivity at T=10 sec") can be used to extract equivalent values from any other MT project without needing to recreate the map definition..

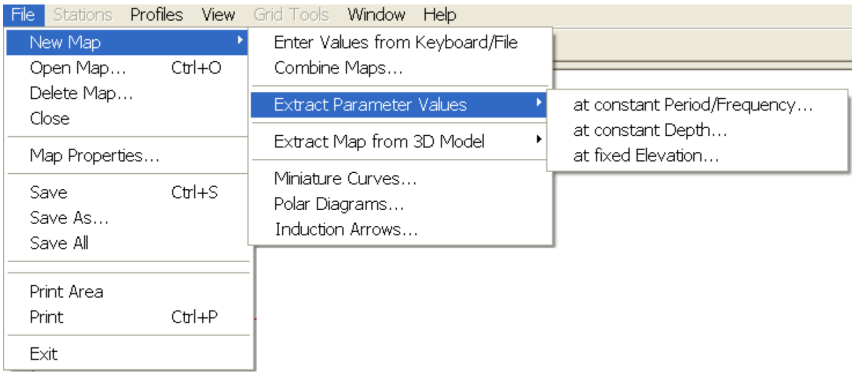
Types of Map Definitions

Different project types have associated with them different data types from which data can be extracted to generate new maps. In the example used above, we created an apparent resistivity map for the XY component at a period of 10 seconds.

Associated with the various project types are also map types which can be generated on the basis of user-defined parameters, e.g. induction arrows at a specific frequency for MT data:



Options for generating new map definitions can be accessed with the **File | New Map** menu command. The figure below shows the options available for an MT project:



Creating a New Map

In addition to the standard map types associated with a given project type, the Maps program provides a number of different methods for creating new maps. New maps can be created using station values, either imported or entered in the Maps program via the keyboard or by combining two existing maps.

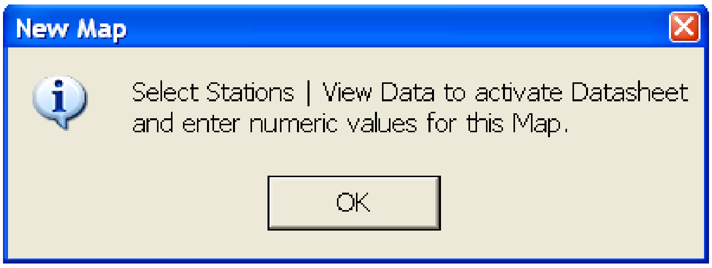
The options for creating new maps can be accessed with two options available with the **File | New Map** menu command:

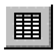
- **Enter Values from Keyboard/File**
- **Combine Maps…**

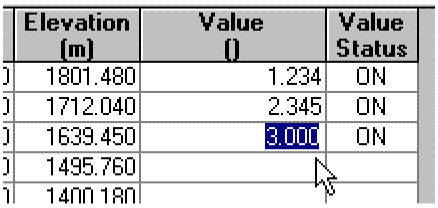
Entering Values from the Keyboard/File

1. Select the **File | New Map** menu command and click the **Enter Values from Keyboard/File** option. The Maps program displays the following window, instructing you to open the

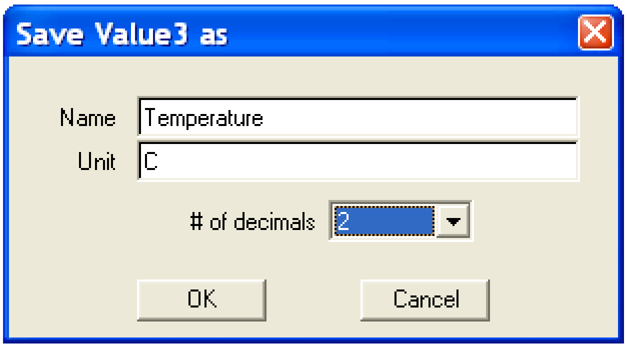
View Data window:



2. On the Stations menu, click **View Data…** or click  in the toolbar.
3. The data sheet form for the new value type appears. Listed in the first five columns (from left to right) are the station names, states, latitudes, longitudes and elevations. The sixth column is empty and will be used to enter the values.
4. Enter the station values in the data sheet. You may leave some values blank.



5. Alternatively, you may import the data from an external file by executing the **File | Import** menu command. The file must contain at least the station ID and the value to be imported, though may also contain coordinate and elevation information. The program guides you through the import process using the Import Wizard described in the “Data Import” chapter of this manual.
6. After entering/importing the data, select the **File | Exit** menu command; confirm changes by clicking the **Yes** button.
7. Enter the Name, Unit and # of decimals for the new value type. Click the **OK** button:



When selecting this option, a new map definition is created with a default name Value ( ). The map properties can be accessed and modified at a later time by selecting **Map Properties** from the **File** menu.

Adding or Subtracting Map Values

New maps can also be created by combining (adding or subtracting) the values of two existing maps.

1. Select the **File | New Map** menu command and click the **Combine Maps** option.
2. Select the source map and click the **Next** button.
3. Select the operator (Add Map or Subtract Map) and click the **Next** button.
4. Select the second map and click the **Next** button.
5. Enter the name for the computed map and click the **Finish** button.

Creating Maps by Merging Data from Different Projects

The data source for a given map is dependent on the project type. As described in the “WinGLink Shell” chapter of this manual, WinGLink gives the user the option of creating two different types of projects: single and integrated.

If an integrated project, containing for example two projects named Prj_1 and Prj_2, is created grouping the two projects, any created map will include station data from both projects.

Separate maps can be generated for each project by opening the single projects, i.e. the projects which are members of the integrated projects, in the Maps program. Gridding is performed on basis of the stations contained in each project.

Loading an Existing Map

To Open a Map:

1. Select the **File** | **Open Map** menu command**.**
2. Select the map by clicking the check box located to the left of each map name.
3. Click the **Open** button.

Each map opens in a separate window. Use the options provided on the **Window** menu to switch between or tile the windows

Map Grids

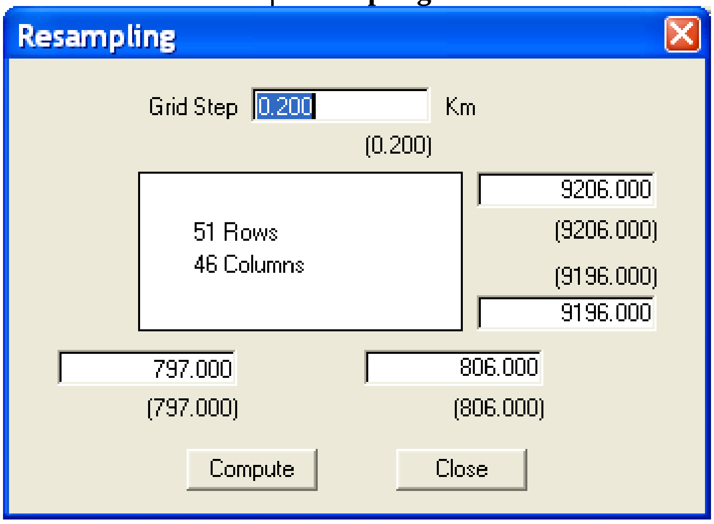
Gridding a Map

To display map contours and color fill ranges, the station values must be interpolated on a regular grid. This operation creates a two-dimensional array of values, also referred to as the map **grid**.

Regridding a Map Using Different Parameters

This option lets you create a new grid by resampling the current map grid:

1. Select the window containing the map you would like to regrid.
2. Select the **Grid Tools | Resampling** menu command:

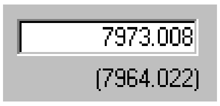


3. Enter the new grid step and the grid endpoints
4. Click the **Compute** button.

A window is opened for the new map with the new grid. Use the **File | Save** menu command to save this map. Station values can be extracted from this new map using the **Extract Value** command described below.

To Interpolate the Parameter Values:

1. Open the map which contains the values to be interpreted.
2. Select the **Gridding menu** | **New Grid…** menu command
3. The numbers in parenthesis show the coordinates of the stations having maximum and minimum Northing and Easting. The new grid boundaries can be entered in the fields located above each of these numbers.



4. Choose the preferred Gridding Option (Normal / Logarithmic);
5. Complete the fields with the values for the **Interpolation Radius**, **Spline Weight** and **Smoothing Factor**.
6. Click the **Compute** button.

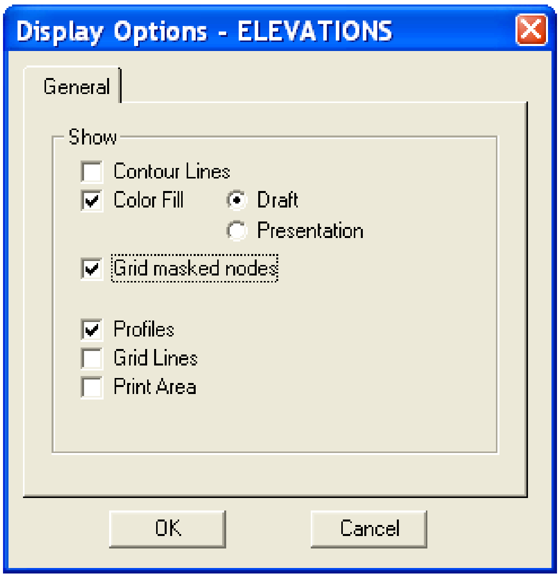
When the station data are changed as the result of subsequent editing, a new map grid must be created in order to show an updated map.

To Regrid the Map Using the Previous Grid Settings:

1. Select the window containing the value you would like to regrid.
2. Select the **Gridding** | **Regrid** menu command.

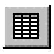
To Enable the Gridding of Masked Nodes:

1. Select the **View | Display Options** menu command. In the Display Options dialog, select the **Grid masked nodes** checkbox on the **General** tab:



Extracting Station Values From Grids

This option assigns to each station the value calculated by interpolating from the grid nodes the value at each station's location, as defined by the station coordinates.

1. Select the window which contains the map.
2. Select the **Stations | View Data** menu command or click the  button.
3. Click the **Value** column header.
4. Choose **Operators** | **Extract Value** from the Grid menu.
5. Select the **File | Exit** menu command and confirm the changes.

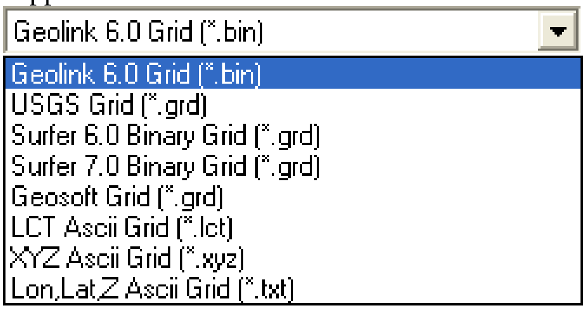
Importing External Grids

External grids can be imported into WinGLink and added to the current project. All standard grid formats are supported, e.g. USGS, Surfer, Geopack, LCT.

To Import an External Grid into the Current Project:

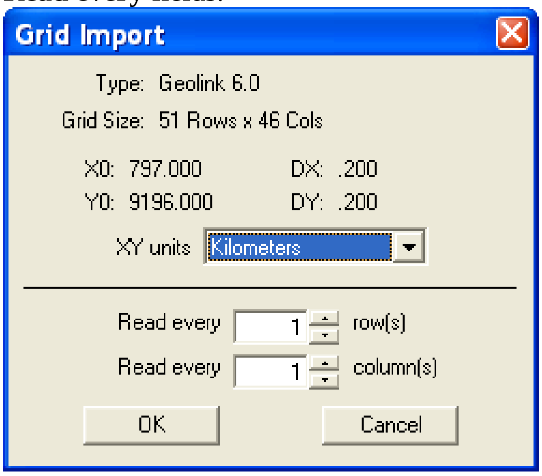
1. Load the map into which you wish to import the grid, or
2. Create a new map which will correspond to the imported grid.

3. Select the **File | Import Grid** menu command. WinGLink supports the standard formats shown below:



4. Select the appropriate format for the grid which is to be imported from the "**Files of type**" list.

5. In the Grid Import window, select the XY units of the grid. If the grid is very large, you can reduce the number of data points which are imported by making the appropriate selections in the **Read every** fields.



6. If necessary, use the **Grid Tools | Resampling** menu command to resample the grid using user-defined values.

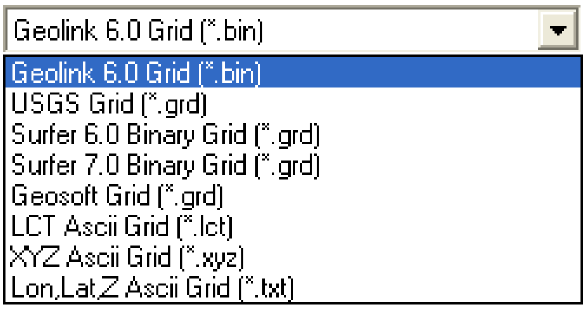
Exporting grids

Map grids created in WinGLink can be exported to external files in a number of different formats for use in other applications or for importing into another WinGLink project.

To Export a Grid to an External File:

1. Create or open the map for which the grid is to be exported

2. Select the **File | Export Grid** menu command and select the desired file format:

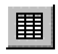


Adding and Editing stations

Importing Stations From a Text File

Station coordinates, elevations and values can be imported from external text files (columnar data). Station data can only be imported into single projects. When the current project is an **integrated** project, the import function is disabled.

To Import Stations From a Text File:

1. Load the map into which you would like to import the value ***or***
2. Select **File | New Map | Enter Values from Keyboard/File** to create a new map using the imported data.
3. On the **Stations** menu, click **View Data** or click the  button on the toolbar.
4. The Maps datasheet form appears. Listed in the first five columns (from left to right) are the station name, states, latitudes, longitudes and elevations. If you selected a map other than elevations, two additional columns appear on the right side: a column for the value and a column for the value status.
5. Select the **File | Import** menu command on the datasheet menu.
6. Browse the directories to select the file to be imported and click **Open**.
7. The **WinGLink Text File Import Wizard** starts. Refer to the instructions provided in the “Data Import” chapter of this manual for more details.
8. When finished, use the **File | Exit** menu command on the datasheet menu to exit the datasheet. Click **Yes** to confirm the import.
9. If the data have been imported into a new map, the Map Properties form will open. Assign the map a name, a measurement unit (used only in legends) and the number of decimals to be saved for each imported value. Click **OK**.

The file containing the data which are to be imported must be a text file organized in columns where one of the columns contains station names and the other three columns contain the station longitude, latitude and elevation. Another optional column may contain a value to be imported.

Warning: The WinGLink import function identifies stations by name. This may cause the program to overwrite existing data if the import options are not set correctly.

Importing New Values for Existing Stations

The **import stations** function can also be used to import new values for existing project stations or to update them by importing revised data from a text file.

Note: This procedure is only enabled for **single** projects.

To import new values for existing stations:

Make sure that the file to be imported has <u>at least</u> two columns:

- one column containing the values to be imported
- one column containing the corresponding station name

Load the map into which the file containing the new or updated data are to be imported.

Refer to **"Importing Stations From a Text File"** above for detailed instructions.

When selecting the fields in the file to be imported, ***skip*** any field that should not be imported or updated.

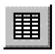
At the end of the import process, a window opens which contains information regarding the number of stations contained in the imported file which were ***merged, skipped*** *or* ***appended.***.

A station is ***merged*** if a station with the same name is found in the destination project. In this case the existing station takes the value imported from the file as well as the values, if any, of other fields not tagged with the **Skip** keyword.

A station is ***skipped*** if no station with the same name is found in the destination project *and* no field of the imported file is assigned to be latitude or longitude for the station.

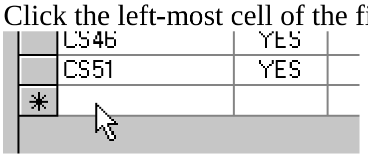
A station is ***appended*** if no station with the same name is found in the destination project: the station is added to the current project and is assigned the values of any field not tagged with the **Skip** keyword.

Using the Datasheet to Add, Delete or Edit Stations

The Maps program has a built-in datasheet which can be used to **add, delete and edit** the station information used when building a map. To open the datasheet, select the **Stations | View Data** menu command or click the  button on the toolbar.

Adding Stations

1. Click the left-most cell of the first empty row:

2. Enter the fields described in the column headers. If unedited, the **Active** and **Value Status** fields are automatically filled with the YES and NO values, respectively.

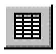
Deleting Stations

1. In the datasheet, highlight the rows of the stations which are to be deleted (use the **Shift** and **Ctrl** keys for multiple selection.)
2. Press the **Del**ete key, and confirm the delete command by clicking the **OK** button.

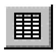
Editing Stations

1. Click the cell which contains the field to be edited.
2. Edit the data.

Applying Operators to Station Values

1. Click the  button to open the datasheet.
2. Select **Operators** from the menu in the datasheet window, then choose the desired operator and enter the requested parameters.
3. **Warning:** When saving the map, choose the **Save As** command if you do not want to overwrite the original data.

Extracting Elevations from Grids

1. Click the  button to open the datasheet.
2. Select any element in the Elevation column (i.e. the column heading or an elevation value).
3. Select **Operators** from the menu in the datasheet window, then choose the **Extract Elevation from Grid** menu command. Here, select either the **Using project topography** or **Using Area topography** option.
4. The program then automatically extracts the elevation values from the specified source and uses these values in the elevation grid.

Updating Extracted Values After Editing Stations

If editing is performed on stations after parameter values have been extracted, the corresponding map can be updated with the new values using the **File | Update Map** command.

Using the Interactive Graphic Editor to Add, Delete and Edit Stations

Station editing can be performed using an interactive graphic feature of the Maps program.

Note: When working with integrated stations, the operations listed below apply to the project currently selected in the Project window. It is only possible to delete, move or edit stations belonging to that project. Likewise, new stations are added to the current project.

Adding Stations

1. Select the **Stations | Insert Station** menu command.

2. Move the mouse pointer to the location on the map at which you would like to place the station then click.
3. Fill the Insert Station window with the appropriate field values.
4. Continue adding all desired stations.
5. Right-click the mouse or press the **Esc** key to stop adding stations.

Deleting Stations

1. Select the **Stations | Delete station** menu command.
2. Move the mouse pointer over the station you wish to delete and click.
3. In the window which then opens, place a check mark to the left of the stations to delete the station, then click **Delete**.
4. Right-click the mouse or press the **Esc** key to stop deleting stations.

Editing Stations

1. Select the **Stations | Edit station** menu command.
2. Move the mouse pointer over the station you wish to delete and click.
3. Edit the value field and select the value status (ON/OFF).
4. Click the **OK** button.
5. Right-click the mouse or press the **Esc** key to stop editing stations.

Moving Stations

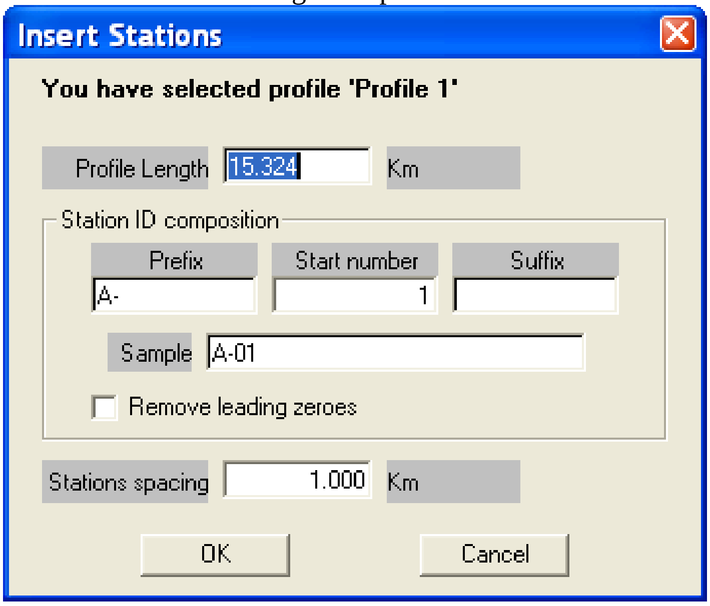
1. Select the **Stations | Move station** menu command.
2. Move the mouse pointer over the station you wish to move and left-click.
3. Move the mouse pointer to the location on the map where you would like to place the station to be moved and click.
4. The Move Station window opens. In the **ID** field, enter the name of the new station; fill the **elev** field with the elevation and edit the x and y fields if necessary. If you selected a map other than elevations, you can supply the value for the new station and its ON/OFF status as well.
5. Click the **OK** button.

Inserting New Stations Along a Profile Trace

This interactive operation is used to automatically insert stations along a profile trace at regular distances:

1. Make sure the desired profile trace is on your map. If not, add a new profile trace.
2. Select the **Stations | Insert stations along profile…** menu command.

3. Move the cross-shaped mouse pointer to a location on the desired profile trace and click it to select the profile.
4. The Insert Stations dialog box opens:



5. Fill the fields as appropriate.
6. Click the **OK** button to insert the stations.

Note: The program can handle over 32,000 stations on a single profile.

Profile Traces and Value Profiles

The general procedure for creating profiles and assigning stations to them is described in the "Getting Started with the Maps Program" section of this chapter. The description here is limited to:

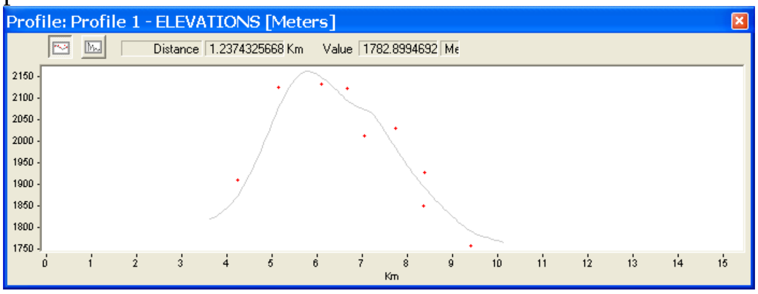
- Viewing profile values
- Working with sections
- Exporting profiles

Viewing Profile Values and Displaying the Frequency Spectrum

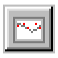
To display a sectional view of the profile in a separate window:

1. Select the **Profiles | goto Profile Mode** menu command or click the  button. This enables you to select the profile trace.
2. Select the **Profiles | Select Profile** menu command.
3. Click the profile to be displayed as a section.

4. Select the **View | Profile Graph** menu command to display the profile window:



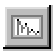
This window displays one of two different graphs, depending which button is enabled.



XY Graph

The **observed values** of the stations attached to the profile are plotted as **red dots.**

The **interpolated values** extracted from the map grid along the profile trace are displayed as a **continuous gray line.** The y-axis of the graph represents the values along the profile, while the x-axis represents the distances along the profile trace.



Spectra

Spectra calculated along the profile are plotted vs. frequency.

Constructing Sections Along Profile Traces

As described in brief in the “Getting Started with the Maps Program” section of this chapter, before 2D sections can be created in other WinGLink modules, the following steps must be performed:

1. **Add a profile to the database area**

   When a profile trace is added to a map, the trace is saved in the database and can be used in any other map of any project contained in the database. Profile traces are elements of the area to which the database refers.

5. **Add stations to the profile**

   These stations provide, in the case of vertical soundings or wells, the datasets needed to build the section. In the case of gravity and magnetic projects, the observed anomaly values are extracted along the profile trace from the gridded map, and the stations associated to the profile provide the information on the measured values .

When **Maps** is launched on a single project, the stations contained in the project can be added to a profile trace to construct the project sections.

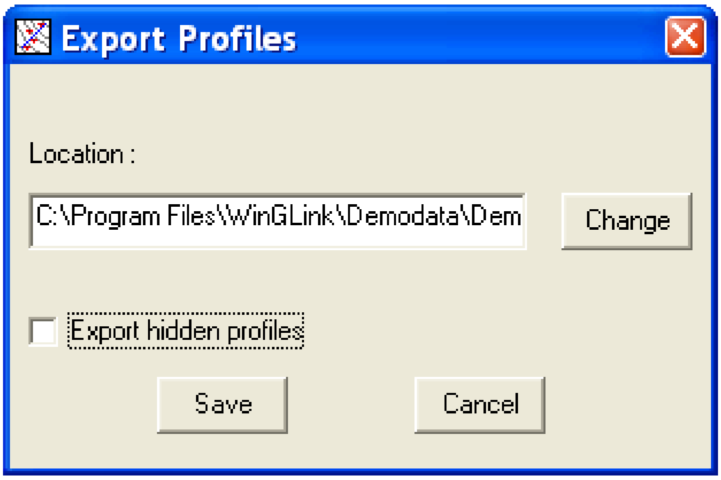
When Maps is launched on an integrated project, the stations of each member project can be added to a profile trace, if not yet associated.

Note: A station added to a profile trace in an integrated project is also added in its original project.

Stations from many different projects can be added to a given profile trace. As a result, integrated sections can be obtained by using, for example, equivalent datasets from MT, EM and DC projects (i.e. imaged sections, 1D models, cross sections).

Exporting Profiles

Use the **Profiles | Export Profiles** menu command to export all profiles visible from within either the current project or all profiles in the database to an external text file. By default, only the visible profiles are exported; to export all profiles in the database, select the **Export hidden profiles** checkbox.

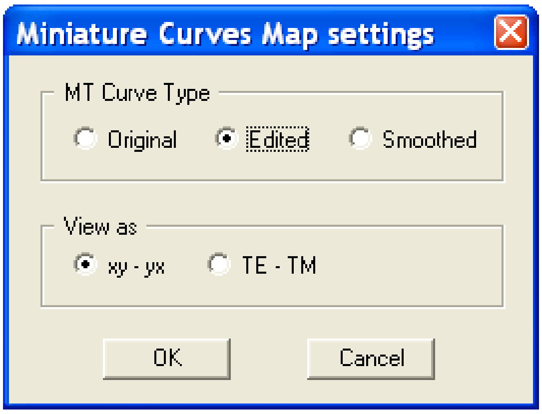


MT Maps

Mapping MT Parameters

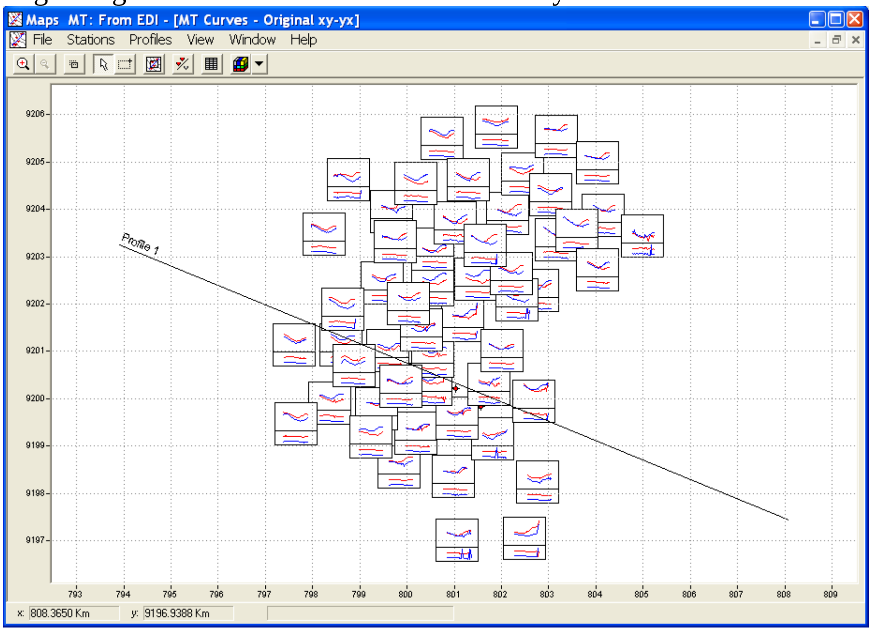
To Plot Resistivity and Phase Curves

1. On the **File** menu, select the **New Map** | **Miniature Curves** menu command:



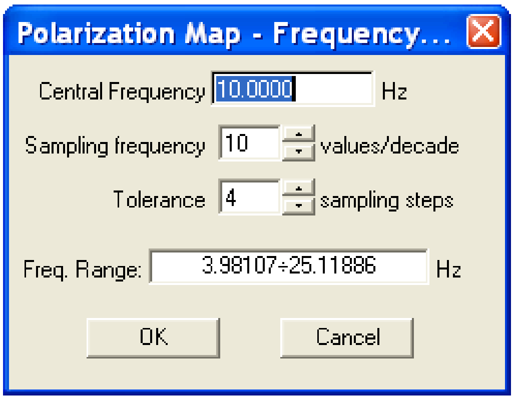
2. Make the desired settings and click the **OK** button. The resistivity and phase curves for each sounding of the project are calculated and plotted in a box centered on the station locations

to give a global view of the curves in the survey area:



To Plot Polar Diagrams

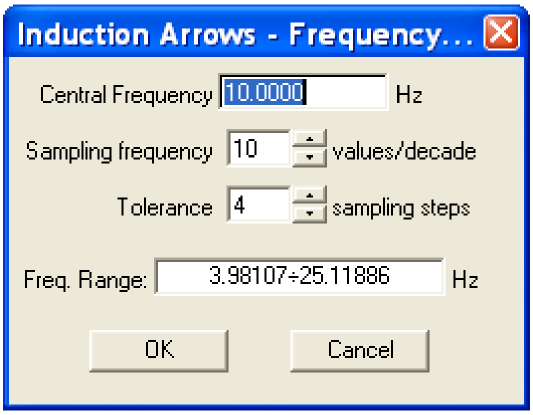
1. On the **File** menu, select the **New Map** | **Polar Diagrams** menu command:



2. Enter the central frequency for which the polar diagrams should be calculated, as well as the other parameters requested in the dialog window.
3. Click the **OK** button to display the polar diagrams for that frequency.
4. Repeat the steps described above to display maps of polar diagrams for other frequencies.
5. On the **View** menu, select the **Display Option** menu command to edit the size of the polar diagram on both monitor and printer.

To Plot Induction Arrows

1. On the **File** menu, select the **New Map | Induction Arrows** menu command:



2. Enter the central frequency for which the induction arrows should be calculated, as well as the other parameters requested in the dialog window.
3. Click the **OK** button to display the induction arrows for that frequency.
4. Repeat the steps described above to display maps of induction arrows for other frequencies.
5. On the **View** menu, select **Display Option** to edit the size of the real and imaginary arrows on both monitor and printer.
6. On the **View** menu, check **Legend** to display the unit-length reference segments.

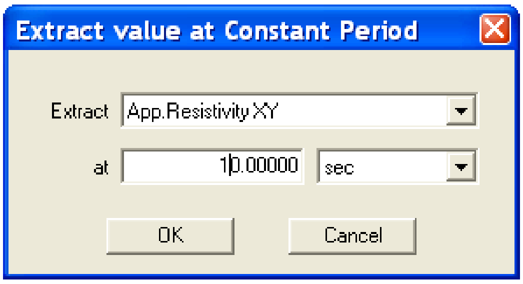
Note: In WinGLink, induction arrows (real and imaginary) always refer to geographical North.

Extracting Parameter Values from Station Datasets or Models

In addition to creating maps for the MT parameters as described above, maps can also be created for a number of different MT parameters at a given period/frequency or depth/elevation. Using available options, it is possible to generate maps for any standard MT component, measured or derived, at a specific period or frequency. Maps can be extracted at a given depth or elevation using values resulting from 1D, 2D or even 3D modeling.

Extracting Parameter Values

1. On the **File** menu, select the **New Map** | **Extract Parameter Values** menu command.
2. Select the desired extraction key from the list of available options: **At constant period/frequency…**, **At constant depth…**, or **At fixed elevation...** Shown below is the dialog which for extracting at a constant period or frequency:



3. Select the parameter value to be extracted from the drop-down list.
4. Specify whether to extract by period or frequency.
5. Click the **OK** button; a window for the new map opens.
6. On the **Gridding** menu, select **New Grid…**; set the gridding parameters as necessary.

Updating Extracted Values after Editing Stations

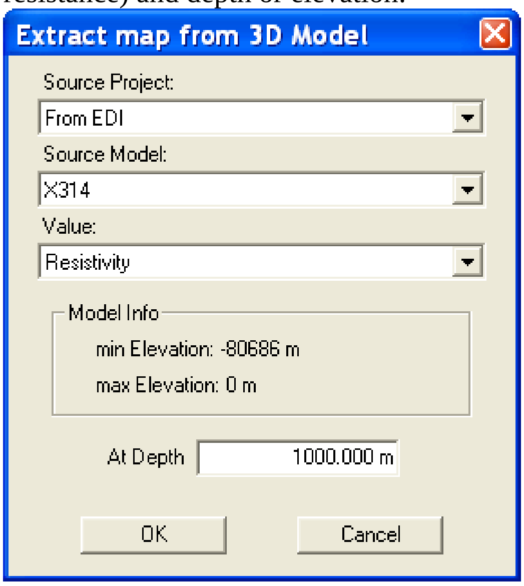
If editing is performed on stations after parameter values have been extracted, the corresponding map can be updated with the new values using the **File** | **Update Map** command.

Extracting a Map from 3D Meshes

If one or more 3D MT meshes have been imported into a WinGLink database, it is possible to extract maps at any specified depth or elevation. Note that this option is available only in MT projects.

To Extract a Map from a 3D Mesh:

1. On the File menu, select the **New Map | Extract Map from 3D Model** menu command; select from the available options: **At constant depth** or **At fixed elevation**.
2. In the Extract map dialog box which opens, select the source 3D mesh, value (resistivity, conductance or transverse resistance) and depth or elevation:

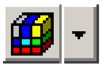
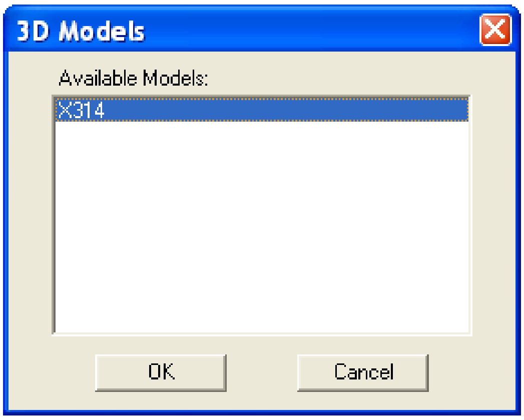


3. After clicking the **OK** button to confirm, a gridding window appears. Accept the default values or make any desired changes and click the **Compute** button to extract the map.

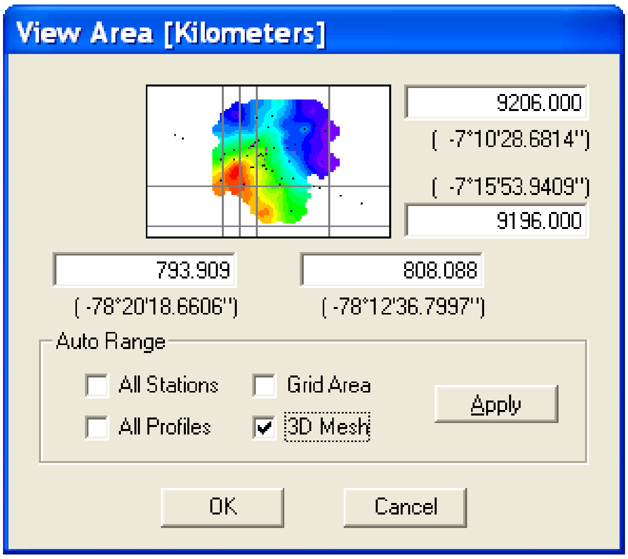
Displaying 3D Meshes

3D meshes which have been imported into WinGLink can be superimposed onto MT projects maps. Profiles projected onto a 3D mesh can be used in the 2D MT and X-Sections programs to extract sections from a mesh.

To Display a 3D mesh:

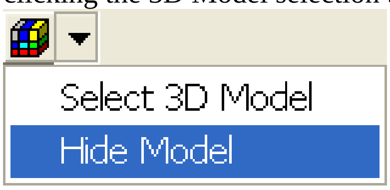
1. Open an MT project
2. On the Maps toolbar, click the 3D Model selection button:

3. The 3D Model selection window opens, listing all models contained in all MT projects in the database. Select the desired model with the mouse and click the **OK** button:


4. Depending on the current view area settings and the geographic coordinates for the 3D mesh, the mesh may or may not be displayed on the map after selecting the mesh. You can force the mesh to be displayed by altering the view settings. To do this, open the View Area window with the **Window | Set View Area…** menu command:



The two mesh-specific options in the **Auto Range** area of the window, **Grid Area** and **3D Mesh** will force the mesh to be displayed on screen. The **Grid Area** option attempts to fit the entire grid area onto the map, whereas the **3D mesh** option zooms in on the map far enough to display the entire mesh on the map. By selecting all of the options in the **Auto Range** area of the window, you can ensure that all map elements are visible on the map.

5. The 3D mesh can be toggled on and off by selecting the **Hide Model / Show Model** commands, which can be accessed by clicking the 3D Model selection button:



Note: For details on importing 3D meshes, refer to section “Importing 3D Meshes” in the “Data Import” chapter of this manual.

TDEM Maps

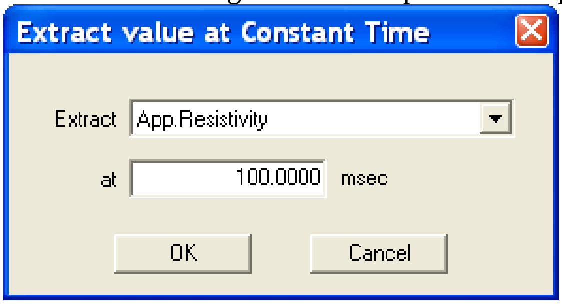
Extracting Parameter Values From Station Datasets or Models

As with MT data, maps can also created for a number of different TDEM parameters at a given period/frequency or depth/elevation. Using available options, it is possible to generate maps from observed or modeled resistivity or voltage at a given time, resistivity or conductance at a given depth/elevation from a smooth or layered model.

Extracting Parameter Values

1. On the **File** menu, select the **New Map** | **Extract Parameter Values** menu command.

2. Select the desired extraction key from the list of available options: **At constant period/frequency…**, **At constant depth…**, or **At fixed elevation...** Shown below is the dialog which for extracting at a constant period or frequency:



3. Select the parameter value to be extracted from the drop-down list.
4. Depending on the section made above, specify the time/frequency or depth/elevation for the extraction operation.
5. Click the **OK** button; a window for the new map opens.
6. On the Gridding menu, click **New Grid…**; set the gridding parameters as necessary.

Updating Extracted Values After Editing Stations

If editing is performed on stations after parameter values have been extracted, the corresponding map can be updated with the new values using the **File** | **Update Map** menu command.

Gravity and Magnetic Maps

Gravity and Magnetic Station Anomalies

Computing the Free Air Anomaly

A map of the free air anomaly can be calculated by selecting the **File | New Map | Free Air Anomaly (FAA)** menu command.

Computing the Bouguer Anomaly

The Maps program can calculate the simplified (no terrain correction) or complete Bouguer anomaly for the stations of a gravity project, assuming that the observed gravity and terrain correction values have been entered for each station. For details on how to calculate the terrain correction, please refer to the next section, “Terrain Correction Maps”.

For stations with missing observed gravity values, the Bouguer anomaly computation will return an empty value.

Observed gravity values can be imported from text files using the WinGLink text files import wizard, as detailed in section “Importing Stations From a Text File” in this chapter.

Computing IGRF (Magnetic Projects)

This command calculates the international geomagnetic reference field (IGRF) at each magnetic project station.

1. With the Maps program launched from a magnetic project, select the **File | Compute IGRF** menu command.
2. A window appears informing you which reference field will be used to calculate IGRF. To continue, click the **OK** button.

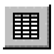
A new map named **IGRF** is created.

Note: The IGRF is computed using the project date entered in the project properties when the project was created. To correct this date, exit **Maps** and modify the project date using the menu command **Project | Properties** or pressing the **Project Properties** button in the main Database window:

The IGRF-10 model data are used for the IGRF calculation.

Removing IGRF (Magnetic Projects)

With this option you can subtract the IGRF from the magnetic field map.

1. Load the magnetic field map.
2. Select the **View | Stations Data** menu command or click the  button.
3. Click the **Value** column header.
4. From the datasheet menu, select **Operators | Remove IGRF**.
5. Select the **File | Exit** menu command and confirm the changes.
6. Save the new map with an appropriate name.

Aeromag (Magnetic Projects)

The magnetic anomaly values for aeromag maps are calculated at the observation point. The observation point is one of the following:

1) Ground survey: Data are collected at ground level

2) Drape survey: Data are collected at a constant positive distance from the ground level

3) Constant altitude survey: Data are collected at a constant elevation above the ground level

To select the observation point property:

1. Open the magnetic (MG) project in Maps.
2. Load the value "AeroMag".
3. Select the **File | Map Properties** menu command. For cases 2 and 3 a flight altitude must be specified, where 0 is the ground. A drape survey with a fight altitude=0 is equivalent to a ground survey.

Gravity and Magnetic Field Transformation

Filtering (Low-, High- or Band-Pass Filters)

Use this option to low/high/band-pass filter 2D data grids.

Note that, following conventional terminology, the terms "low" and "high" pass refer to spatial *frequency*, i.e. 1/wavelength. To eliminate short-wavelength noise, therefore, a low pass filter is used, and vice-versa.

LOW PASS the minimum wavelength to be passed is requested.

HIGH PASS the maximum wavelength to be passed is requested.

BAND PASS wavelengths in the entered range are passed.

Several filters are available for minimizing the effects of ringing (i.e. leakage of unwanted frequencies into the desired output): square, Bartlett and Hamming-Tukey. The anti-ringing factor is the narrowness of this filter: a value of 0 minimizes its width in the frequency domain, while a value of 10 is so broad that the effects of filtering are minimal.

To filter a grid:

1. Select the **Grid Tools | Filtering** menu command.
2. Enter the filtering parameters and click the **OK** button.
3. If you wish to have station values extracted from the new grid, select **Yes** when prompted.

The filtered grid is assigned to a new map.

Polynomial Fitting

With this option, you can obtain the regional and residual field of a selected map.

The regional field is approximated with an n^{th} order polynomial surface.

The residual field is the difference between the polynomial surface and the selected map's grid.

Both the regional and the residual fields are obtained as maps.

To obtain the regional and residual field of a map:

1. Select the **Grid Tools | Polynomial fitting** menu command.
2. Enter the name for the map that is to contain the regional field.
3. Enter the name for the map that will contain the residual field
4. Enter the degree of the polynomial surface and click the **OK** button.
5. If you wish to have station values extracted from the new grid, select **Yes** when prompted.

Derivatives

This option can be used to calculate the 1^{st} horizontal and vertical, and the 2^{nd} vertical derivatives. This transformation makes use of the fact that nearby disturbing sources have a greater effect on gradient maps than on anomaly maps.

The 1st horizontal derivative is calculated directly in the space domain (see Blakeley and Simpson, 1986). The remaining two are calculated by conventional Fourier transformation. As usual, the use of derivatives is limited by their tendency to emphasize noise.

To calculate the derivative of a grid:

1. Select the **Grid Tools | Derivatives** menu command and the type of derivative you wish to apply.
2. Select the map containing the input grid (the current map is the default).
3. Enter the name for the new map that will contain the output grid.
4. Click the **OK** button.
5. If you wish to have station values extracted from the new grid, select **Yes** when prompted.

Up/Down Continuation

This option allows one to compute anomalies as they would be observed on a surface parallel to but below the observation surface (DOWNWARD) or above the observation surface (UPWARD). The user must enter the level of continuation in kilometers.

DOWNWARD CONTINUATION emphasizes the effect of local shallow anomalies. This may lead to very noisy maps if the level of continuation is too large (a maximum continuation level of 2*grid step is recommended).

UPWARD CONTINUATION will produce smoothing of the anomalies.

To carry out an upward/downward continuation:

1. Select the **Grid Tools | Up/Down Continuation…** menu command
2. Select the map containing the input grid (the current map is the default).
3. Enter the name for the new map that will contain the output grid.
4. Enter the continuation parameters and click the **OK** button.

Computing Pseudo-Gravity Anomalies

This operator applies only to magnetic project data. It functions by reducing the magnetic field to the pole (assuming that remanence effects are negligible) and filtering with the appropriate frequency-domain operator. The output requires scaling by a factor equal to

$$|\mathbf{m_a}| \ / \ \mathbf{G}\rho$$

where:

$\mathbf{m_a}$ apparent magnetization (susceptibility x field strength)

ρ the corresponding assumed density contrast

G is the gravitational constant and does not need to be entered

You will also have to enter the values for the inclination and declination of geomagnetic field at the survey location.

This is a useful method for viewing complex magnetic data as it tends to eliminate much of the “clutter” caused by dipolar fields.

To compute the pseudo-gravity anomaly:

1. Select the **Grid Tools | Pseudo Gravity…**menu command.
2. Select the map containing the input grid.
3. Enter the name of the output grid (or accept the default name).
4. Fill the remaining fields and click the **OK** button.
5. A new map is created. If you wish to have station values extracted from the new grid, select Yes when prompted.

Computing Pseudo-Magnetic Anomalies

This operator applies only to gravity project data, and is used to compute the pseudo-magnetic anomaly from the gravity anomaly. The output requires scaling by a factor equal to

$$G\rho \; / \; |\mathbf{m_a}|$$

where

$\mathbf{m_a}$ apparent magnetization (susceptibility x field strength)

ρ the corresponding assumed density contrast

G is the gravitational constant and does not need to be entered

You will also have to enter the values for the inclination and declination of geomagnetic field at the survey location, assuming that the desired result is to be comparable with observed magnetic data. When the field is calculated at the magnetic pole, it is (except for the scaling factor) equivalent to the first vertical derivative of the gravity field.

For example, suppose that the intensity of the geomagnetic field is 47000nT; assume that the source magnetic susceptibility is 1000mcgs and we wish to replace this with rock of “density” contrast 1.0gm.cm-3. The magnetization to be entered is then 47000 x 1000 x 10-6 = 47 nT

To compute the pseudo-magnetic anomaly from a gravity anomaly:

1. Select the **Grid Tools | Pseudo Magnetics…**menu command.
2. Select the map containing the input grid (which should be the gravity anomaly).
3. Enter the name of the output grid (or accept the default name).
4. Fill the remaining fields and click the **OK** button.
5. A new map is created. If you wish to have station values extracted from the new grid, select **Yes** when prompted.

Reduction to the Pole

This function converts magnetic data which have been recorded in the inclined earth’s magnetic field to data which would have been produced by a vertical geomagnetic field (see Baranov, 1957; Gunn, 1975, p.306). The primary use of this function is to simplify magnetic maps obtained in areas with low magnetic latitudes. Symmetric anomalies then result from simple vertical bodies, with the maximum located above the source. The assumption is made that all magnetic anomalies in the survey area are caused by magnetization in the direction of the current geomagnetic field.

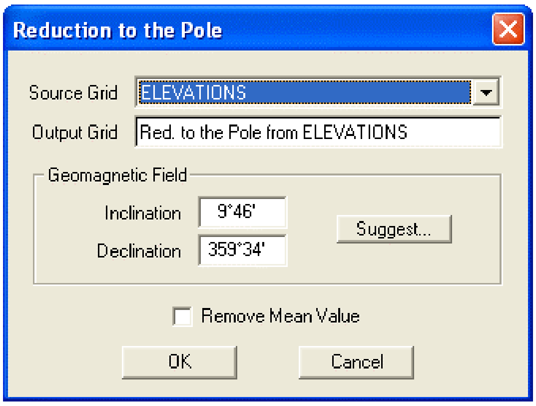
The following parameters must be entered:

- Inclination of geomagnetic field

- Declination of geomagnetic field

To perform a reduction:

1. Select the **Grid Tools | Reduction to the Pole…** menu command:



2. Select the map containing the grid to be reduced.
3. Enter the name of the output grid (or accept the default name).
4. Enter the inclination and declination of the geomagnetic field. Click the **Suggest** button, and the program will calculated recommended values.
5. If you would like to have the mean value of the input grid be removed from the output grid, select **Remove Mean Value**.
6. A new map is created. If you wish to have station values extracted from the new grid, select **Yes** when prompted.

Vertical to Total Field Transformation

If data obtained with a vertical-field fluxgate magnetometer need to be compared with more modern total-field measurements, they can be transformed using this module.

The theoretical background for this is given by Gunn (1975, pp 306-307).

The user is required to enter the following parameters:

- Inclination of geomagnetic field

- Declination of geomagnetic field

To transform the vertical field to total field:

1. Select the **Grid Tools | Vertical to Total field…** menu command
2. Select the map containing the vertical field data;.
3. Enter the name of the output grid, which will contain the total field data (or accept the default name).

4. Enter the inclination and declination of the geomagnetic field.
5. If you would like to have the mean value of the input grid be removed from the output grid, select **Remove Mean Value**.
6. A new map is created. If you wish to have station values extracted from the new grid, select **Yes** when prompted.

Terrain Correction Maps (Gravity)

The WinGLink Maps program provides functionality for calculating terrain corrections to take into account local and distant topographic features. These corrections can then be used in the calculation of complete Bouguer anomalies for stations contained in WinGLink gravity projects.

The approach used involves combining the inner terrain correction (ITC) and outer terrain correction (OTC) to calculate the total terrain correction. The ITC, which corrects for local topographic features, is performed using the Hammer methods. The OTC, which corrects for more distant topographic features, is performed using one or more external grids containing digital elevation models (DEM) for the area of interest.

Terrain correction data can also be imported from external text files directly into the Maps program for both new as well as existing stations.

As with other map types, calculated terrain correction values can be exported to external text files. These external files contain station ID, coordinates, elevation and data value, in this case terrain correction. External files can then used to import station and terrain correction data into other projects, providing a convenient tool for combining project data.

The total terrain correction is the sum of the outer and inner terrain corrections. The order in which the individual correction values are calculated is irrelevant. The following discussion describes first how to create a terrain correction map, followed by how to calculate the inner, and then the outer terrain correction.

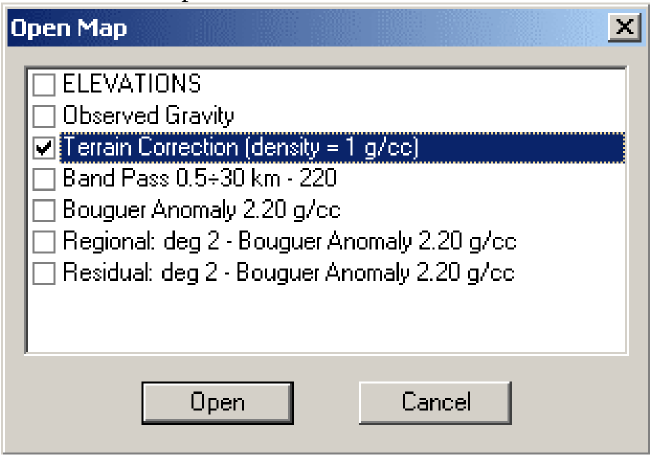
Creating a Terrain Correction Map

Among the default map types available for Gravity projects within the WinGLink Map program is the Terrain Correction map. The values contained in this map are used in the calculation of the complete Bouguer anomaly map.

To Create a Terrain Correction Map:

1. Open WinGLink, and in the main database window, select a project of type Gravity [**GR**]; in the right-hand panel, click the Maps application icon.

2. The **Open Map** dialog box opens. Select the **Terrain Correction** map:



3. After clicking the **Open** button, the Maps program prompts you to select the type of gridding to be used for the map. If opening the map for the first time, opt to continue without gridding, as two or more stations with values are required in order to calculate a grid.

Calculating the Inner Terrain Correction

The calculation of the inner terrain correction calculation is based on the Hammer method (Geophysics, v. 4, pp. 184-194, 1939.). This method involves creating a set of concentric rings, or zones as they are referred to here, each of which is divided into a specified number of equal-sized segments. The average elevation for each of these segments is entered relative to the elevation specified for a given station. In this way, the topography immediately surrounding each station can be defined and the inner terrain correction calculated.

To Calculate the Inner Terrain Correction for a Station:

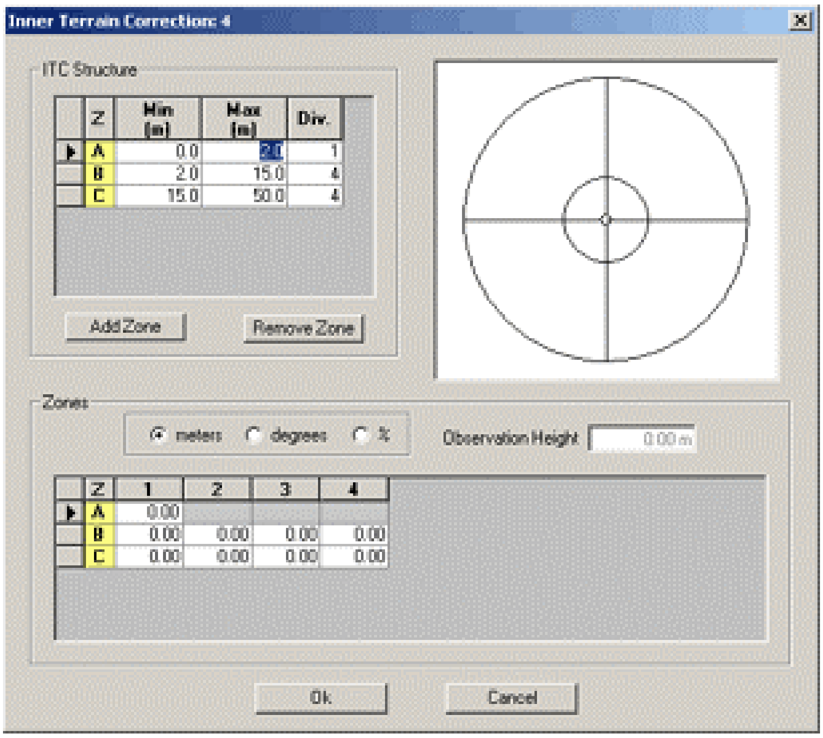
1. Continuing with the terrain correction map created above, open the Data Sheet window by either selecting the **Stations | View Data…** menu command or by clicking the  button on the toolbar.

2. The data sheet window for terrain correction maps contains, in addition to geographic coordinates, columns for ITC, OTC and Value. The value column in this case represents the total terrain correction and, unless the value has been edited manually updated during a file import, is equal to the sum of ITC and OTC.
   To set the ITC value for a station, position the mouse over the cell in the ITC column for the station in question and click.

Now click the arrow which appears at the right edge of the cell:

	Elevation (m)	ITC (mGal)	OTC (mGal)
15	1339.000	0.000000	0.000000
90	1314.300	0.000000	0.000000
28	1298.400	0.000000	0.000000
48	1329.600	0.000000	0.000000
65	1355.900	0.000000	0.000000

3. Click the arrow to open the Inner Terrain Correction window in which the set of zones is defined and relative elevation values are entered.:



The Inner Terrain Correction window, shown above, is divided into three areas. The set of zones is defined in the ITC Structure frame, located in the upper-left corner of the window. This frame contains a spreadsheet with four columns. The first column, **Z**, designates the zone name, **Min** and **Max** specify the respective minimum and maximum radii of the circles which define each zone, and **Div**. specifies the number of segments into which the zone is to be divided.

Zones can be added and removed using the buttons provided at bottom of the ITC Structure frame. The program allows the creation of up to 5 zones, each of which may be divided into up to 12 sections. In addition, the program forces zone continuity, i.e. the Max and Min values of two adjacent zones must be identical.

The current zone configuration is shown in the display panel to the right of the ITC Structure frame. Changes made to the number of zones and the zone definitions are reflected immediately in this display panel.

The elevation values for each of the segments are entered in the Zones frame of the Inner Terrain Correction window. The upper area of this frame contains a frame with option buttons for **meters**, **degrees**, and **%**. When using the meters option (default), the values are entered as elevation values relative to the actual station elevation, i.e. if the average elevation of Section 2 of Zone C is determined to be 1000 m and the actual station elevation is 900m, a value of 100 m would be entered for this section.

Depending on how the average elevation values are determined for the individual sections, it may be more convenient to use the degrees or % options. When using degrees, specify the number of degrees relative to the horizontal plane projected from the station location to a representative point in the given zone. When using the percent option, specify elevation difference between the station location and the zone section as a percentage. As the observation may not necessarily be made at the station elevation, specify in the **Observation Height** field the height above the station elevation from which the measurement of the zone elevation was made, i.e. if a transit of height 1.7 m was used to make the measurements, enter 1.7 in this field.

After specifying how the zone values are to be entered, enter the relative elevation values for each segment in the spreadsheet-like table.

4. Click **OK** after entering values for all of the sections. The values can be edited at a later time by reopening the Inner Terrain Correction window for the station.

The inner terrain correction value is calculated immediately after closing the Inner Terrain Correction window:

	Elevation (m)	ITC (mGal)	OTC (mGal)	Value (mGal)
15	1339.000	0.000000	0.000000	
30	1314.300	0.000000	0.000000	
28	1298.400	0.000000	0.000000	
48	1329.600	0.295070	0.000000	0.30
55	1355.900	0.000000	0.000000	0.00

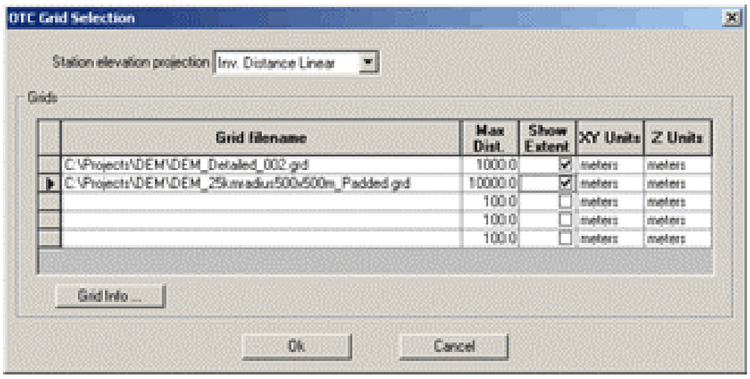
The inner terrain correction value for each station is to be calculated using the procedure outlined above. ITC values cannot be manually entered in the ITC column of the Data Sheet window.

Calculating the Outer Terrain Correction

Unlike the inner terrain correction, which must be performed separately for each station, the outer terrain correction makes use of one or more digital elevation models to calculate the effect which non-local topographic features have on each station contained in the map.

To Calculate the Outer Terrain Correction:

1. Continuing with the terrain correction map used above in the description of the inner terrain correction, open the OTC Grid Selection window by selecting the **File | OTC Grid Settings…** menu command:



2. To add a grid, position the mouse over a line in the **Grid filename** column and click. Now click the arrow which appears at the right edge of the field to open the Import Grid window. Browse the directories until the desired grid has been found; click the **Open** button. All standard grids are supported: e.g. Geolink, USGS, Surfer, Geopack, Geosoft, LCT).

3. Set the parameters for each grid as necessary. The **Max Dist.** value specifies the correction distance from each station. Select the **Show Extent** checkbox to display the area covered by he given DEM. Select the units used in the DEM in the two columns to the right.

   The **Station elevation projection** drop-down list contains three options for the station elevation: **None**, **Inv. distance linear**, and **Inv. distance square**. Select **None** to use the actual station elevation in the calculation of the OTC. The **Inv. distance linear** and **Inv. distance square** options extrapolate station elevations from the given DEM using either a linear or quadratic algorithm, respectively, based on the mesh values and the geographic coordinates of the stations.

   To view the physical parameters which define a grid, click the grid name in the **Grid Filename** column to select the grid, and then click the **Grid Info…** button located in the bottom part of the screen. The Grid Info window opens, displaying the grid name and type, min. and max. X and Y values, X and Y step sizes and numbers of rows and columns.

4. After setting the digital elevation models, select the **Compute TC** command on the **File** menu to calculate the terrain correction. Computation progress is indicated by a progress bar in the status bar. Upon conclusion of the calculation, a log window is displayed which lists any encountered problems.

Total Terrain Correction

The total terrain correction is the sum of the outer and inner terrain corrections. Thus, a value for the total terrain correction exists for a given station provided one of the two terrain correction values has been determined.

To view the values in tabular form, open the Data Sheet window by either selecting the **Stations | View Data…** menu command or by clicking the button on the toolbar:

Elevation [m]	ITC [mGal]	OTC [mGal]	Value [mGal]
1339.000	0.062403	0.012793	0.08
1314.300	0.107559	0.176990	0.28
1298.400	0.143419	0.516518	0.66
1329.600	0.143419	0.033332	0.18
1355.900	0.081024	0.392252	0.47

Neither the ITC nor OTC values can be edited manually. It is however, possible to change the total terrain correction value by selecting the field with the mouse and overwriting the value. Any changes are retained until the terrain correction is recalculated using the **Compute TC** command on the **File** menu, in which case the total terrain correction is again set equal to the sum of the ITC and OTC, or until terrain correction data are imported from an external file.

To update the map grid, or if no grid yet exists for the terrain correction, click the **Regrid** command on the **Gridding** menu.

Importing and Exporting Terrain Correction Data

As with other types of value data, it is possible to import and export terrain correction data from/into the Maps program. In this way, terrain correction data, including station coordinates, can be passed from one project to another as well as to and from other applications.

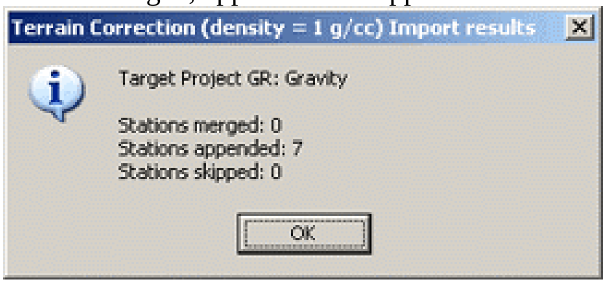
Terrain correction data are imported into the Maps program in the same way as other types of value data. The data must be in text format and organized by column. The files should include station ID, geographic coordinates, elevation and the values for total terrain correction.

To Import Terrain Correction Data:

1. Open a terrain correction map, then open the Data Sheet window by either selecting the **Stations | View Data…** menu command or by clicking the button on the toolbar.
2. On the **File** menu of the Data Sheet window, select the **Import** command.
3. In the File Import dialog box which opens, browse the directories until you have found the file to be imported; select and click the **Open** button.
4. The File Import wizard opens. This Wizard guides you through the import process, step by step, prompting you for details regarding the file structure and the assignment of the column headings and units. For further details on how to use this

Wizard, please refer to the “Data Import” chapter of this manual. For further details on how to use this Wizard, please refer to the “Data Import” topic in the of the WinGLink help system.

5. After specifying file format in the Import Wizard and clicking the **Finish** button, the Maps program imports the file. A status report is then displayed in an info box, indicating the number of stations merged, appended or skipped:



Click the **OK** button to conclude the import process.

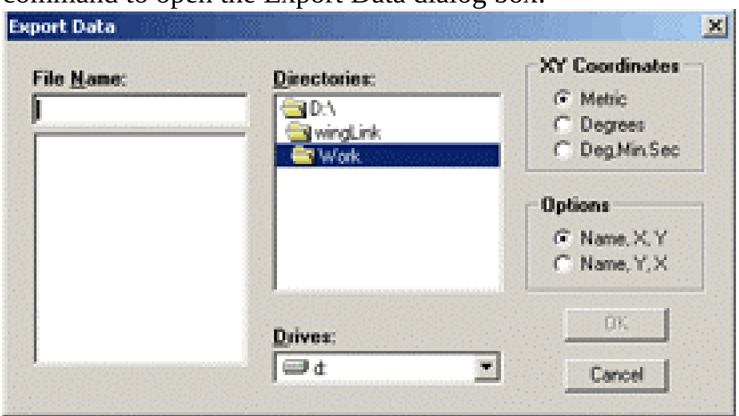
6. To update the map grid, or if no grid yet exists for the terrain correction, click the **Regrid** command on the **Gridding** menu.

Terrain correction data can be exported from the Maps program as text files with fixed-width columns. These files can be imported into maps in other WinGLink projects with no difficulty.

To Export Terrain Correction Data

1. Open a terrain correction map, then open the Data Sheet window by either clicking the **Stations | View Data…** menu command or by clicking the  button on the toolbar.

2. On the **File** menu of the Data Sheet window, click the **Export** command to open the Export Data dialog box:



3. In the **XY Coordinates** area of the dialog box, select the format in which the data are to be exported. The format specified here does not have to be identical to the current format, i.e. any required conversions are performed by the Maps program during file export. Likewise, specify the order of the X and Y coordinate values in the Options area of the dialog box. After specifying a file name, click the **OK** button.

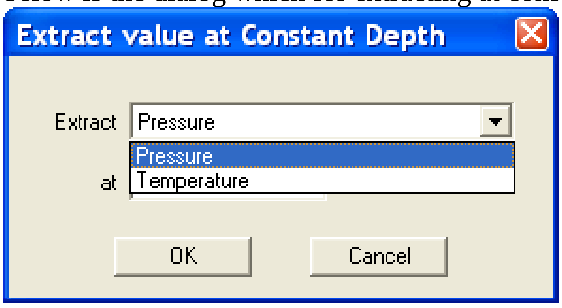
Well Maps

Creating a New Map from Numeric Well Data

As with other value types, it is possible to create maps at specific depths using numeric values imported for well stations. (For details on importing well courses and well layer data, please refer to the "Data Import" chapter of this manual.)

To Create a New Map from Numeric Well Data:

1. On the **File** menu, select the **New Map** | **Extract Parameter Values** menu command.
2. Select the desired extraction key from the list of available options: **At constant depth…**,or **At fixed elevation...** Shown below is the dialog which for extracting at constant depth:



3. All of the numeric data imported to the active wells project should be listed in the **Extract** drop-down list. Select an item and enter a depth.
4. The program displays the number of stations extracted and number skipped during extraction.
5. If data were extracted for at least two stations, you will be able to grid the map (specify how you would like the map gridded in the Gridding Window which opens following extraction).

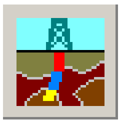
Note: When extracting parameter values from well maps, note that the position of the point at the specified depth or elevation may not necessarily be coincident with the station coordinates. The position is instead that at the specified depth along the well trace. This location is indicated by a dot on the new map.

15: Wells

Overview

This chapter focuses on the use of the Well Data Editor. Following a detailed description of how to use the program, an overview is provided on how to work effectively with well data within WinGLink. There, references to the appropriate locations of the manual can be found for instructions on how to use wells in the respective WinGLink modules.

Using the Well Data Editor



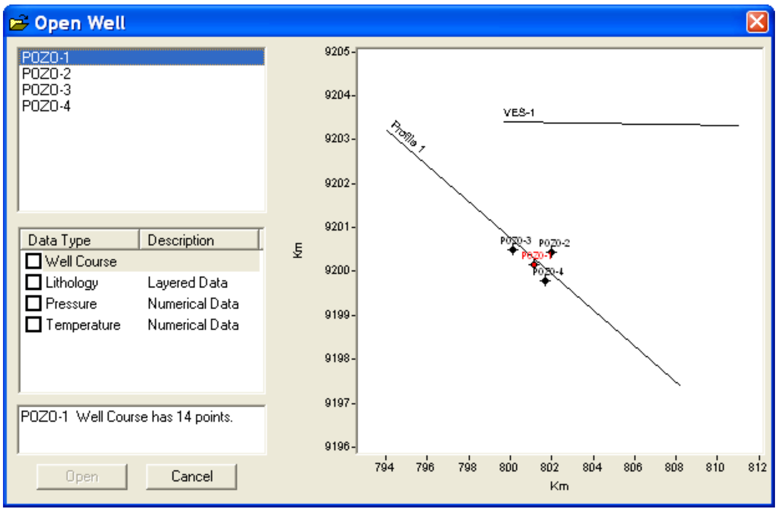
The Well Data Editor provides a powerful interface for working with well courses, including tools for editing and displaying well traces as well as for editing and assigning display properties to layered and numerical data. Note that layered and numerical data can be edited and saved only if the WinGLink license includes the WL_2 module.

Note: While well traces can be imported directly into the Well Data Editor (as well as into the WinGLink Shell), layered/numerical data, such as lithologic data or measured values, cannot. These data types must instead be imported into the database in the main WinGLink shell. For details, please refer to the “Data Import” chapter of this manual.

To Start the Well Data Editor:

1. In the main Database window, use the mouse to select the Wells project (**WL**) which contains the well trace(s) and layered/numerical data to be edited.

2. Click the Well Data Editor icon  in the applications panel of the WinGLink Shell to open the Well Data Editor. In addition to all wells contained in the project, the Open Well dialog box lists the available data types, which include the well course and any imported layered or numerical data types, e.g. lithology, pressure, temperature:



The upper left panel of the dialog box lists all well traces contained in the current project. To the right are the well course and data types available for the well selected in the upper left panel. The lower left panel displays information about the selected well. Shown in the right is an overview map; the currently selected station is displayed in red.

Note that after making a selection and opening a well window, the Open Well dialog box closes. This can be reopened by selecting the **File | Open** menu command.

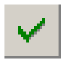
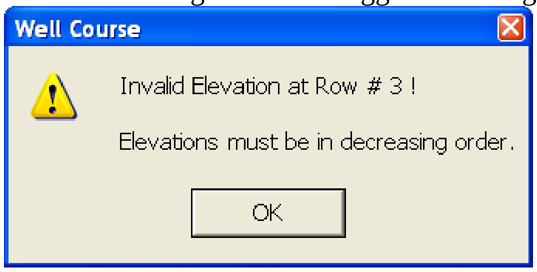
To View and Edit Well Courses

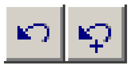
1. To open the well course for a specific well, select the well in the upper left panel, then select the **Well Course** checkbox. Click the **Open** button to display the Well Course spreadsheet:

POZO-1 - Well Course

Well Course			
Longitude (Km)	Latitude (Km)	Elevation (m)	Vertical Depth (m)
801.021	9200.212	2030.900	0.000
801.021	9200.212	1930.900	100.000
801.021	9200.212	1830.900	200.000
801.021	9200.212	1730.900	300.000
801.022	9200.211	1630.900	400.000
801.021	9200.211	1530.900	500.000
801.005	9200.220	1430.900	600.000
800.977	9200.234	1330.900	700.000
800.937	9200.257	1230.900	800.000
800.885	9200.289	1130.900	900.000
800.837	9200.321	1030.900	1000.000
800.796	9200.350	930.900	1100.000
800.766	9200.375	830.900	1200.000
800.724	9200.412	730.900	1300.000
800.671	9200.455	630.900	1400.000

2. Each field can be edited manually. Note that when editing the elevation, the program automatically calculates the vertical depth and vice versa. Note, as well, that that elevations must be in decreasing order, though several identical elevation values in succession are possible (horizontal traces).

3. After either editing a well course point or entering a new point, click the green check button on the toolbar:

Only after this button has been clicked are the changes acknowledged by the program. Any elevation values which are not in decreasing order will trigger a warning message:


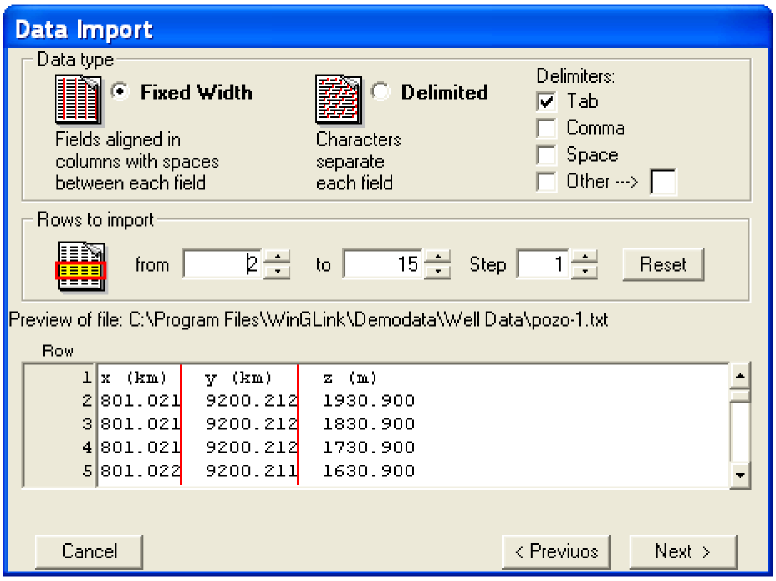
4. To delete a row, press the “Delete” key on the keyboard.
5. The delete operation as well as any other action performed in the datasheet windows can be undone using the **Undo / Redo** buttons on the toolbar:

6. To save changes made to a well course, either click the x box in the upper right corner of the spreadsheet, which will open a dialog box prompting you to save the changes, or use the **File | Save** menu command.

To Import Well Courses

Well courses can be imported into an existing well using the **File | Import** menu command. It is not possible to import new wells into a database from the Well Data Editor. This can only be performed in the WinGLink Shell. For details, please refer to the “Data Import” chapter of this manual.

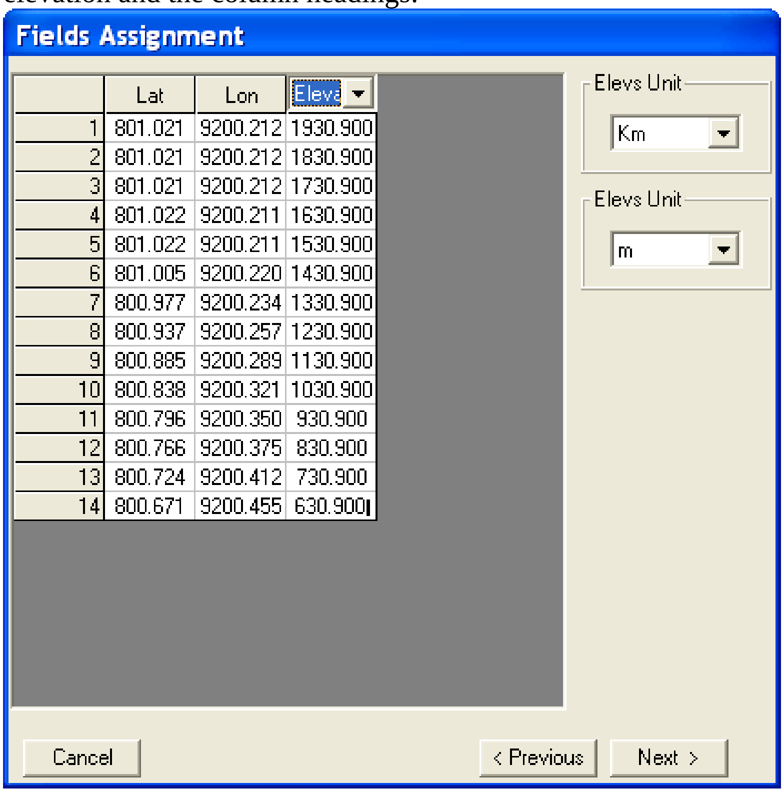
To Import Well Course Data Into an Existing Well:

1. In the Well Data Editor, open the well course into which well course data are to be imported.
2. Use the **File | Import** menu command to open a standard Windows dialog box for selecting the file which contains the well course data.
3. After selecting the file, the Data Import Wizard for text files opens:



Make the appropriate settings for the given data file and continue with **Next**.

4. In the next screen, specify the units for coordinates and elevation and the column headings:

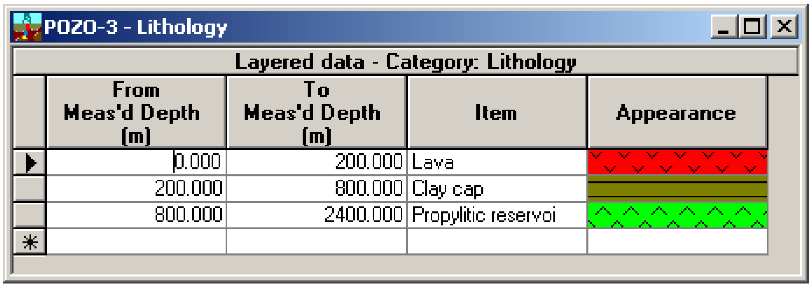


5. Click the **Next** button to execute the import.

To View and Edit Layered Data

1. As with well courses, first select a well in the left frame of the **Open Well** dialog box, then select a checkbox next to a layered data type. Note that only those data types which have been imported for the given well and are not currently open are

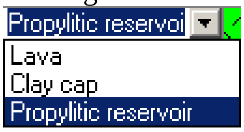
available for selection:



Layered data - Category: Lithology			
From Meas'd Depth (m)	To Meas'd Depth (m)	Item	Appearance
0.000	200.000	Lava	
200.000	800.000	Clay cap	
800.000	2400.000	Propylitic reservoi	

2. The **From** and **To measured depths**, which define the layers, can be edited. New layers can be appended in the bottom field. While layers may overlap, the bottom layer takes priority. Layers need not be in order of depth.

3. Each layer can be assigned an item type by placing the mouse pointer in the respective field in the **Item** column and then clicking the arrow button to display the drop-down list:

   All layer types associated with the current category are listed and are available for selection. The appearance associated with the selected item is automatically displayed after selecting an item.

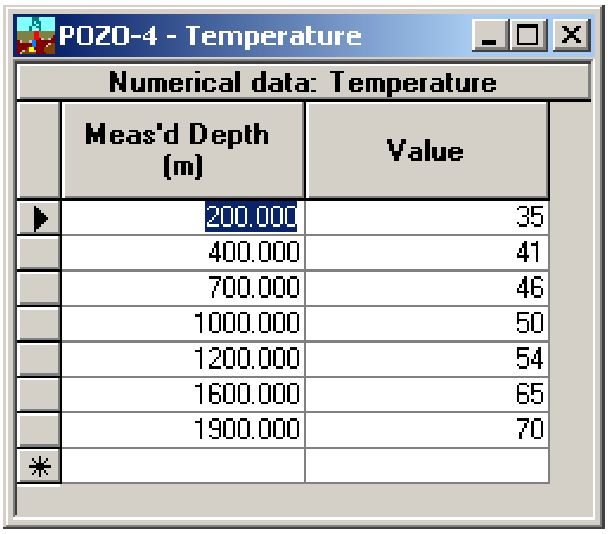
   Categories, the associated layer types, and the appearances of each layer type can be defined with the Category and Pattern Editors, which can be opened with the **Tools | Category Editor** menu command. This command is available in the Well Data Editor only when a layered data window is open.

   For information on how to use the Category and Pattern Editors, please refer to the description in the “Common Functions” chapter of this manual.

4. To save changes you have made, either click the x box in the upper right corner of the spreadsheet, which will open a dialog box prompting you to save the changes, or use the **File | Save** menu command.

To View and Edit Numerical Data

1. Use the Open Well dialog box to open the edit window for the desired numerical data type:



POZO-4 - Temperature

Numerical data: Temperature

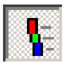
Meas'd Depth (m)	Value
200.000	35
400.000	41
700.000	46
1000.000	50
1200.000	54
1600.000	65
1900.000	70

Make changes or add new layers as with layered data. Layers do not need to be in order of depth.

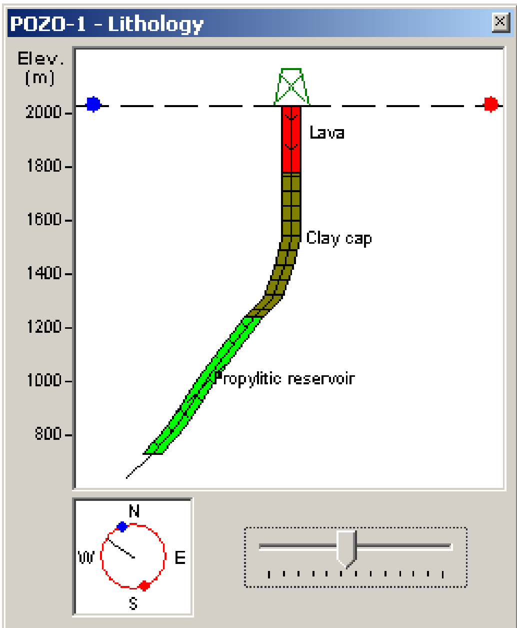
2. To save changes you have made, either click the x box in the upper right corner of the spreadsheet, which will open a dialog box prompting you to save the changes, or use the **File | Save** menu command.

To Display Sectional Views of Well Data

A sectional view can be displayed for the active spreadsheet window by clicking the Sectional View button on the toolbar:



Depending on which spreadsheet type is active, the sectional view window displays the well course with the respective layer types, e.g. lithology, layered data or numerical data. Shown below is the sectional view of well POZO 1:



The slide bar in the bottom right part of the screen can be used to change the user's perspective of the well course, allowing the user to view the well course from all angles. The blue and red dots at the top of the sectional view correspond to the blue and red dots on the compass view in the bottom left corner of the window to provide a frame of reference.

Working with Well Data in WinGLink: An Overview

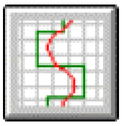
Provided below are references to the chapters of the manual which include information on working with well data, including both well courses and layer information, e.g. lithologic information.

Please refer to the chapters referenced in parenthesis for detailed instructions on working with wells in the respective WinGLink programs.

- WinGLink Shell: used to import well courses and layer data. (***Data Import***)
- Maps: used to assign wells to profiles (***Maps***)
- Well Data Editor: used to edit and display well courses (***Wells***)
- X-Sections: used to create cross sections of well log data (***X-Sections***)
- Sections: wells, both well courses and, in certain cases, layer data, can be viewed in all WinGLink modules which support sections, e.g. 2D Inversion and X-Sections. (***2D MT, X-Sections***)
- Interpreted Views and Montage: used to prepare documents for presentation. (Requires WinGLink license option WG-2, see www.geosystem.net for details.) (***Interpreted Views and Montage***)

16: MT Soundings

Overview



The Soundings icon is displayed in the program menu whenever a sounding project is selected.

When a sounding project (MT, TEM, DC, etc.) is selected and the program launched, the stations of the selected project are loaded together with their original datasets.

The discussion in this chapter is limited to the MT Soundings program.

For each station, this program can be used to:

- Compute and display the parameter curves and a number of MT parameters
- Import/export sounding data
- Edit sounding data
- Compute a 1D smooth and layered inversion model

As described at the end of this chapter, many of the functions available for individual soundings can also be performed in batch mode on any number of soundings contained in the current project.

About Editing

Edits performed on sounding data are saved in the *"edited data"* section of the database. The original data are not overwritten and can always be restored using the **File | Reload Original Data** menu command.

The edited data which are saved by the MT Soundings program are used by other WinGLink programs:

- Edited resistivity curves, as well as phase and tipper, are used to calculate resistivity **pseudosections**.
- Smooth and layered 1D models are used to calculate **imaged sections** and **cross sections,** respectively.
- Curves and 1D models are also used to **extract parameter maps** in the **Maps** program.

Sounding Data Which are Saved in the Database

For each station of the current project, the following datasets are saved in the WinGLink database:

Original Data

Apparent resistivity and phase as imported from external files or entered by keyboard. Original curves can always be recovered to restart the editing process using the Soundings menu command **File | Reload Originals**.

Impedances:

Impedances (if available) are saved in the database. If impedances are not saved in the database, editing functions such as decomposition or static stripping are not enabled.

Power spectra:

Power spectra are read during the import from EDI files and are used to compute impedances, if requested, but are ***not*** saved in the database.

Edited Data

Edited curves are initially constructed by assigning to each data point the same value of the original curves. At the end of each editing session, the edited values are saved in the database, separately from the original values.

Edited curves are used by default to construct *pseudosections* along profiles.

Smoothed Curves

These are calculated from the edited curves using the selected smoothing routine. These curves may optionally be used instead of the edited curves to construct *pseudosections* along profiles.

Smooth 1D Model

One smooth model is saved for each sounding. This provides a preliminary guess model for the layered inversion routine and is also used to produce imaged sections along profiles.

Both Bostick and Occam models are calculated and saved, one for each sounding. For each sounding, the user must specify whether the TE or TM curve is to be considered the 1D curve and used by the inversion program.

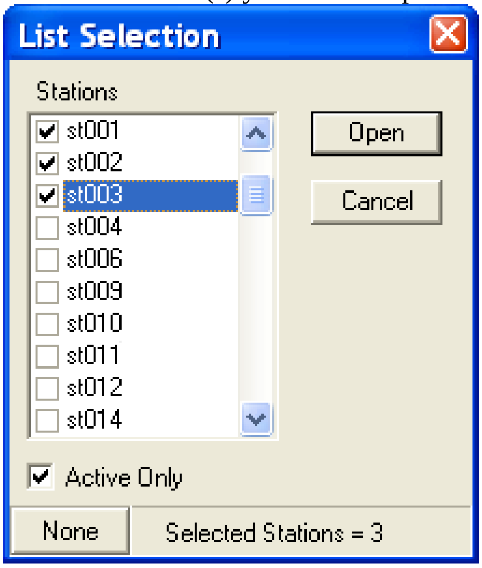
Layered 1D Model

One layered model is saved for each sounding. These models can be displayed along profiles in WinGLink's section programs and used to generate gridded sections the *X-Sections* program. The 1D curve used to calculate the layered model is the same curve used for the smooth model.

Opening, Saving and Printing Sounding Data

To Open Soundings from the Station List

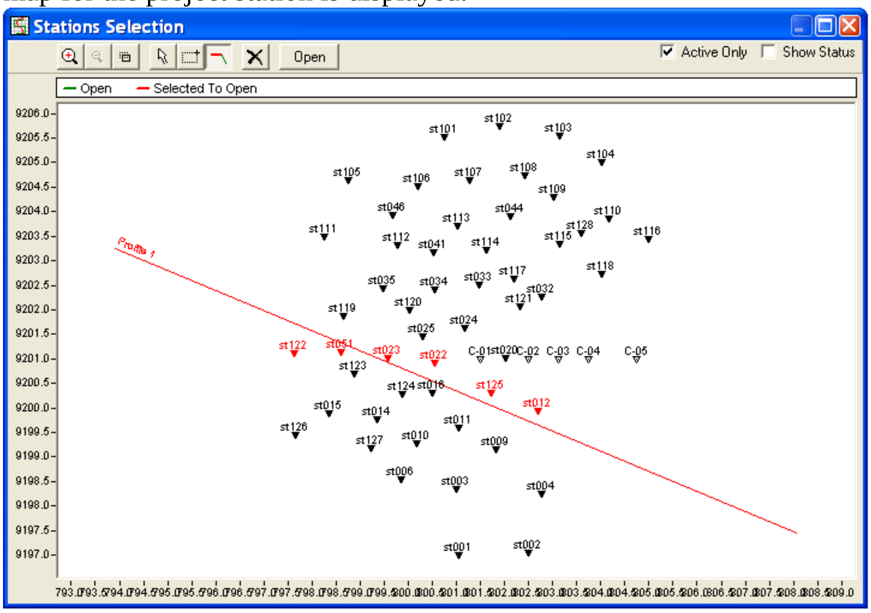
1. Select the **File | Open from List** menu command
2. Select the station(s) you wish to open:



3. If you want to display only the active stations, select the **Active Station** check box.
4. Click the **Open** button.

To Open Soundings on the Station Map

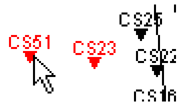
Select the **File | Open from Map** menu command; a simplified location map for the project station is displayed:



Opening Soundings by Clicking Individual Stations:

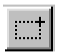
1. Click the  button.

2. Click the stations you wish to open; the selected stations turn to red:

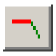


3. Click the  button.

Opening All Soundings Within a Given Area:

1. Click the  button.
2. Left-click the mouse on a corner of the area of interest and drag it to define a rectangle on the map. When releasing the mouse button, all stations in the rectangle will turn to red.
3. Click the  button.

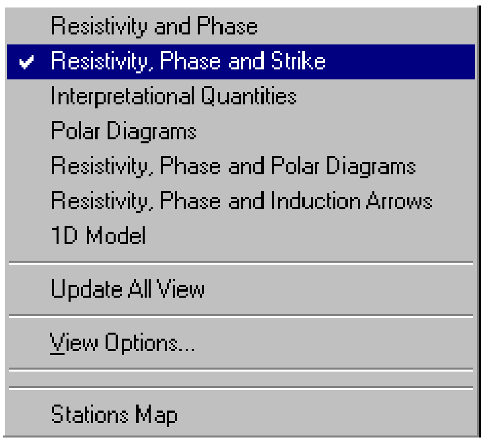
Open All Soundings Associated with a Profile Trace:

1. Click the  button.
2. Left-click the mouse on the profile or on a segment of the profile trace of interest. All stations previously added to that profile trace turn red, indicating selection.
3. Click the  button.

Viewing Parameter Curves

Each window can be set to show the parameter curves or the 1D model for each sounding.

On the **View** menu, select from the available display options:



Note on MT ellipticity:
Ellipticity is calculated using the definition given by Ranganayaki (1984, Geophysics, 49;1730-48):

$$Ellipticity = \frac{|Z'_{xx} - Z'_{yy}|}{|Z'_{xy} + Z'_{yx}|}$$

where the ‘ denotes rotated impedance tensor.

Ranganayaki points out that this quantity is limited as a dimensionality indicator because even at places where Z’xx = Z’yy, the fields could be influenced by an anomaly and require 3D resistivity structure in order to describe their variation. This has been shown to be especially true at low frequencies. This deficiency can be offset by calculating skew, which increases at low frequencies where the impedance tensor is affected by a larger volume of the earth.

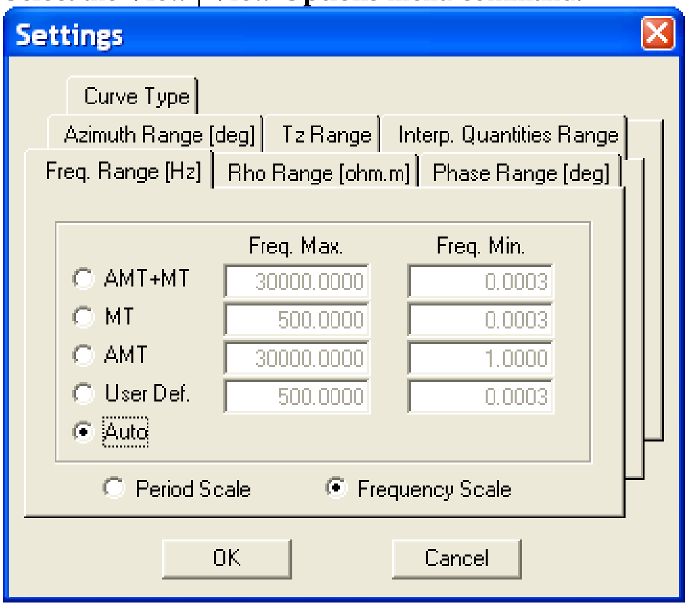
Note on induction arrows

In the MT Soundings program, induction arrows are displayed with the arrows pointing away from the conductor, i.e. the Weise or Schmucker convention. The display properties of the arrows, i.e. length and convention cannot be changed in the Soundings program. They can, however, be altered in the Maps program which also provides functionality for displaying induction arrows. Please refer to the "Maps” chapter of this manual for details.

Setting Diagram Ranges and Scale

To Set the Ranges and Scale for the Plot Diagrams:

1. Select the **View** | **View Options** menu command:



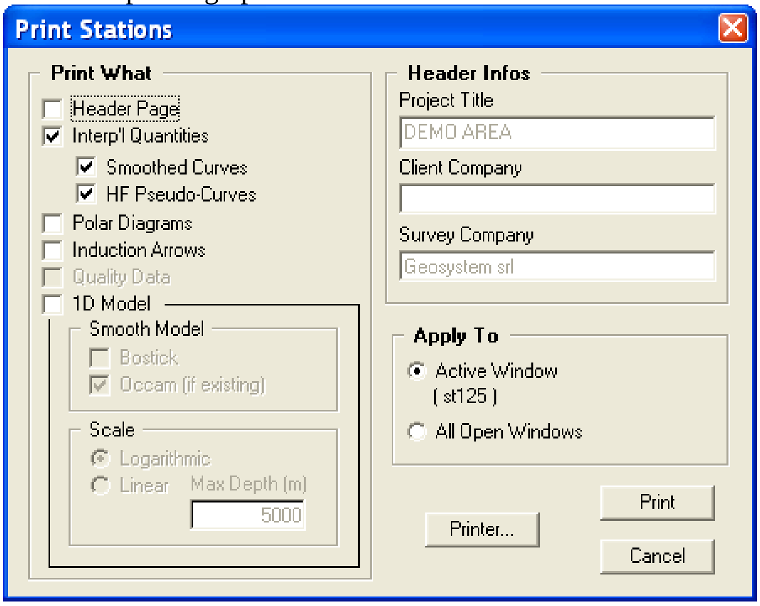
2. For each parameter, set the automatic range option or enter the max. and min. values to be plotted. The axis scale can be set to linear or logarithmic for many plots.
3. The settings made here apply to all window.

Printing Curves and 1D Models

To Print Station Curves

1. Select the station by clicking on its window;

2. Select the **File | Print** menu command.
3. Select the printing options and click the **Print** button:



Importing Sounding Data

In general, sounding data are imported into WinGLink using the import functionality provided in the WinGLink shell (see the "Data Import" chapter of this manual for details).The MT Soundings module includes an option, however, for importing new data into an existing sounding.

Import functions are available for:

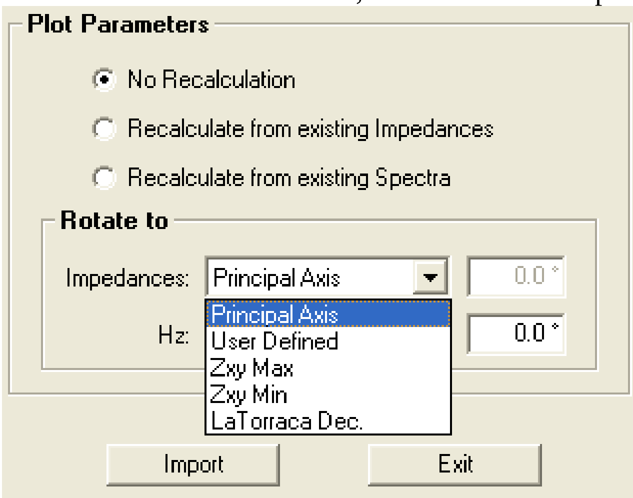
- EDI files
- Text files
- Keyboard entry

Note: This function cannot be used to import new stations into the current project.

To Import MT Data from an EDI File

1. Open the sounding into which the data are to be imported
2. Select the **File | Import new data| From EDI File…**menu command
3. Browse the directories and select the file you wish to import.

4. In the **Plot Parameters** frame, select the desired options:



If you select the **Recalculate from existing spectra** option, the program uses the spectra contained in the EDI file.

If the file does NOT contain spectra, the program uses impedances.

If the file does not contain impedances, the program uses the plot parameters contained in the file.

Impedances are rotated back to 0 degrees regardless of the values in the ZROT block in the EDI file. The impedances are then rotated to the value specified in the **Rotate to** section of the import window.

The tipper rotation is specified in the TROT block in the EDI file. If this block is not present, it is assumed that the tipper is rotated to 0 degrees and no "back rotation" is performed during import, regardless of the values present in the ZROT block. Following the "back rotation", the tipper values are rotated to the value specified in the **Rotate to** section of the import window.

Note: Any previously saved sounding data are lost when new data are imported.

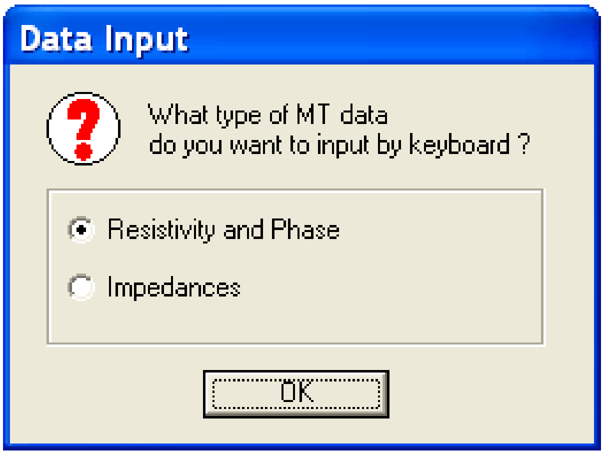
To Import Datasets from Text Files:

1. Open the sounding into which the data are to be imported
2. Select the **File | Import new data | From Text File…**menu command
3. Follow the instructions given by the Import Wizard (refer to the "Data Import" chapter of this manual for details.)

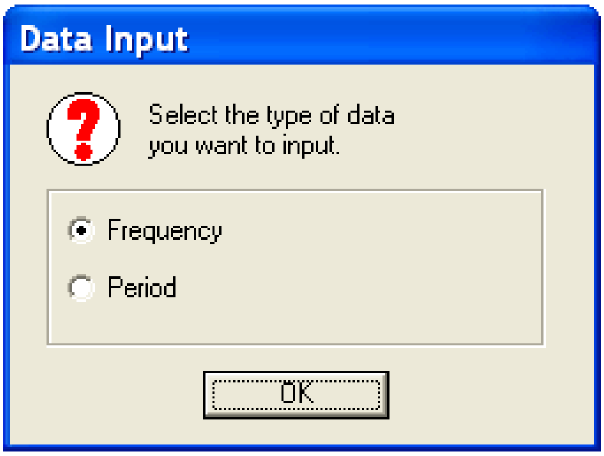
Note: Any previously saved sounding data are lost when new data are imported.

To Enter Data from the Keyboard:

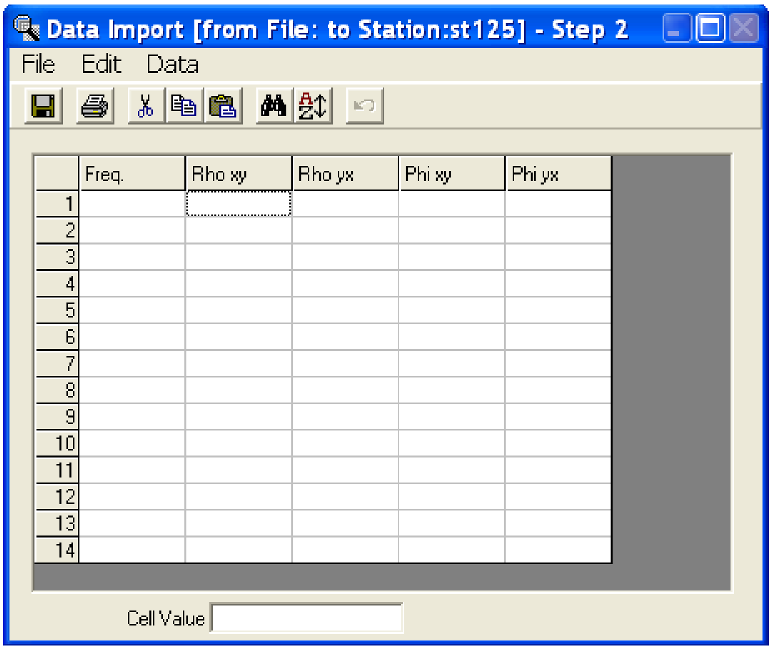
1. Open the sounding into which the data are to be imported
2. Select the **File | Import new data | From Keyboard…**menu command
3. The program alerts you that any existing data will be lost.
4. A dialog window opens prompting you to select what type of data are to be entered:



5. In the next window, specify whether the frequency or period data are to be entered:



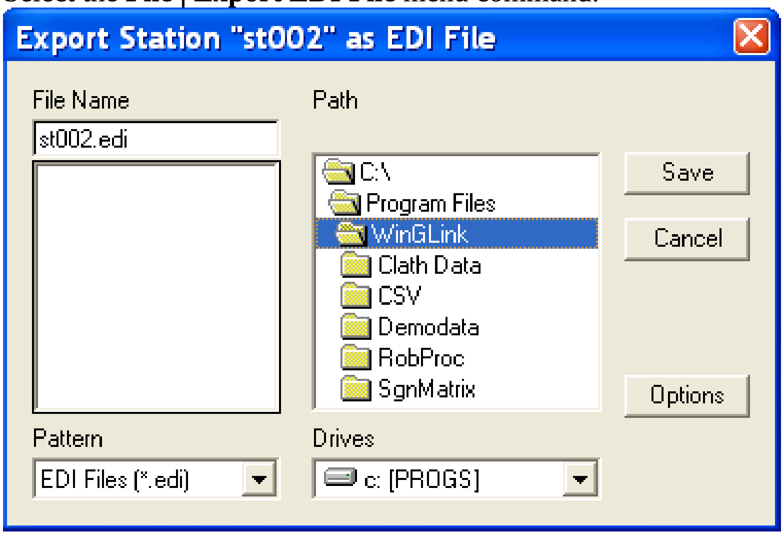
6. In the Data Import window, enter the data in the appropriate columns:



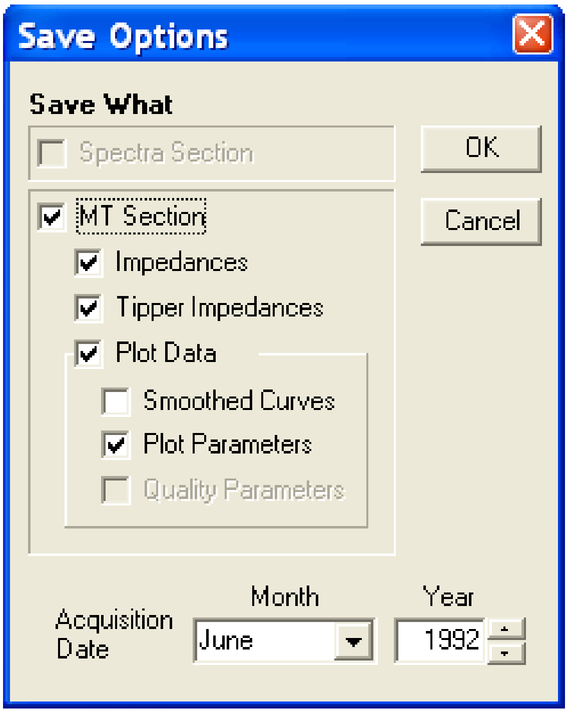
When finished, use the **File | Save** menu command to store the data in the database

To Export MT Data to an EDI File

1. Select the station by clicking on its window.
2. Select the **File | Export EDI File** menu command:



3. Click the **Options** button to open a dialog window in which the parameters to be written to the EDI file can be selected:



Make the desired selections and click the **OK** button when finished

4. Enter a name for the EDI file to be created.
5. Click the **Save** button.

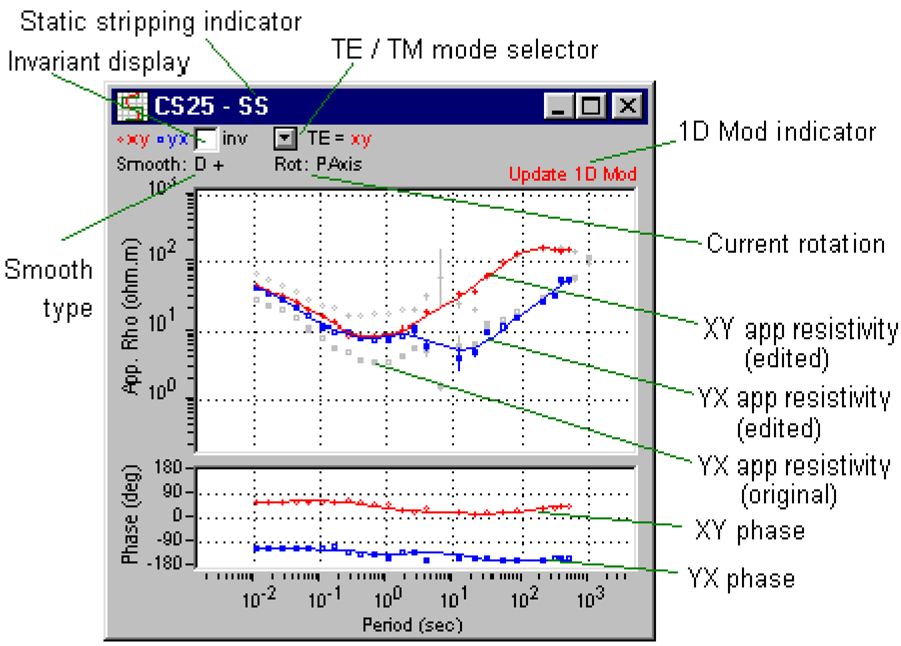
Activating / Deactivating Stations

The checkbox located in the upper area of each sounding display window can be used to activate and deactivate the current station. The active/inactive status applies to the station in all WinGLink modules, i.e. sounding data associated with deactivated stations are not included in 2DMT inversions.

Analyzing MT Data

The MT Soundings View Form

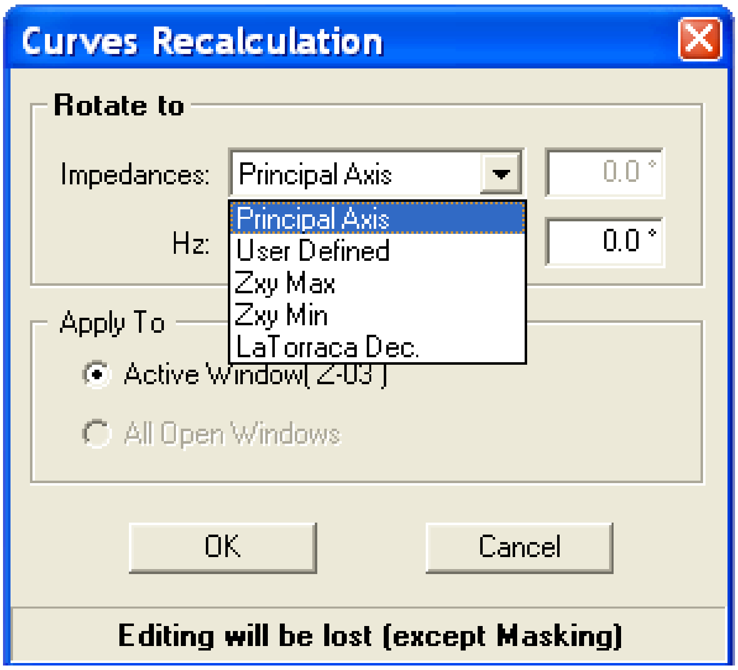
The following figure shows the MT sounding data form:



TE/TM mode selector	Click this button to change which curve corresponds to TE mode. In the above example, the xy curve is selected for TE mode (therefore, the yx is for TM mode)
1D model validation	This text is displayed whenever the 1D model of the station needs to be recalculated because a change was made on the curve.
Invariant display	If this option is checked, the invariant curves for resistivity and phase are shown. The invariant resistivity is calculated as the geometric mean of rhoXY and rhoYX; the invariant phase is calculated as the arithmetic mean of the two.
Smooth type	Indicates the algorithm used to calculate the smoothed curves.
Current rotation	Indicates the rotation (if any) applied to the impedances.
XY, YX curves	The original curves are displayed in gray, the edited curves in red (XY) or blue (YX); the continuous curves represent the smoothed data.
Static stripping	If present, this text indicates that the edited curves have been static stripped.

To Rotate Station Impedances and Tipper

1. Select the station by clicking on its window.
2. Select the **Rotate** command from the menu to open the Curves Recalculation dialog box:



3. In the **Rotate to** frame, select the rotation type and rotation angle.
4. To rotate tipper, specify the rotation type and angle in the fields to the right of the **Hz** label.
5. In the **Apply To** frame, select whether the rotation is to be applied to all stations or to the active station only.
6. Click **OK**.

To Perform a Decomposition on the Active Station

1. Select the station by clicking on its window.
2. Select **Rotate**.
3. In the **Impedances Rotation** frame, select the decomposition type.
4. In the **Apply To** frame, select whether the rotation is to be applied to all stations or to the active station only.
5. Click **OK**.

To View a Polar Diagram Map

1. Close all open stations. The **Special** command is now displayed on the main menu.
2. Select **Special | Polarization Maps**.
3. If you are already in polarization maps mode, select **File | New**.
4. Enter the values for central frequency, sampling frequency and tolerance; the frequency range is determined using these values.
5. Click **OK**.

6. To switch back to station view mode, select **File | Close All** and then **Special | Soundings**.

To Display a Miniature Map of Resistivity and Phase Curves

1. Close all open stations. The **Special** command is displayed on the main menu.
2. Select **Special | Miniature curves**.
3. To switch back to station view mode, select **File | Close All** and then **Special | Soundings**.

Editing Sounding Data

The Edited Curve

When sounding data are entered or imported by the Soundings program, they are saved as “original data” in a reserved area of the database and are never modified unless new data are imported or synthetic data are saved as original data (see the “2DMT Modeling” chapter of this manual for details). At the same time, a duplicate dataset is generated and saved in a different area of the database for use by editing functions. When an editing session is carried out on the sounding data, it is carried out on this second dataset, which is referred to as the “edited curve” of the sounding.

Saving changes following an editing session means saving an edited curve. This curve is used by default in calculating 1D inversion models and pseudosections.

Sounding data can be edited by:

1. Opening the data spreadsheet and entering new edited values.
2. Starting the graphic interactive Edit form by selecting the **Edit** command.

Editing Sounding Data in the Spreadsheet

Sounding data can be viewed and edited using a spreadsheet which shows original and edited data together in a common window.

To Open the Data Spreadsheet:

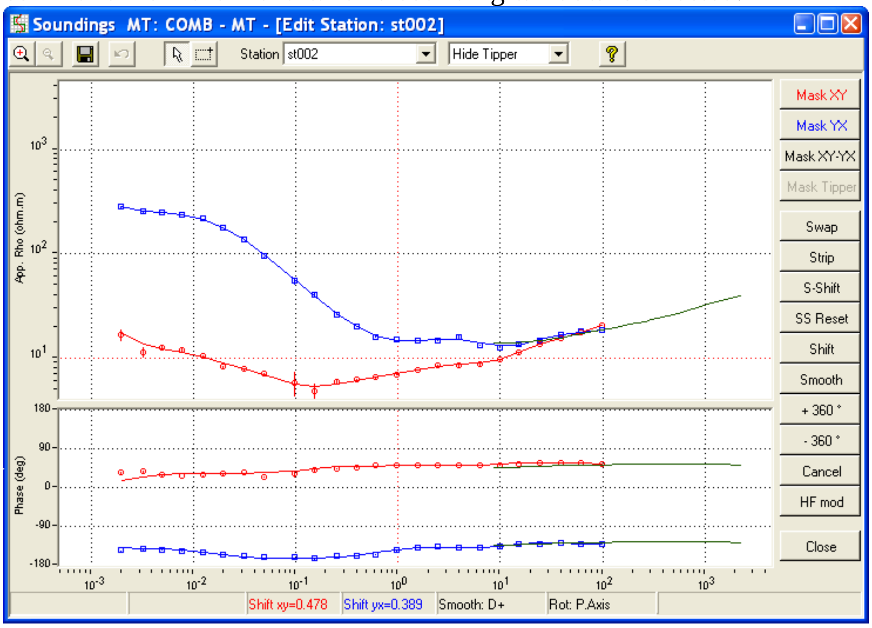
1. Select the **File | Edit Data** menu command.
2. The soundings data are displayed in columns. Columns with original data are gray and the header has a suffix (o) indicating that the data of the column are original data. Original data cannot be edited.
3. On the **Data** menu, use:
   - **Arrange columns** to select the type of sounding data to be displayed
   - **Format** to set a fixed format for each data column
   - **Operators** to apply arithmetical operators to a column

- **Sort** to sort a selected data column

4. On the **File** menu, select:
   - **Save Data** to store any editing changes
   - **Exit** to return to the sounding View form
   - **Export to file** to export the displayed data to a text file

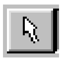
Interactive Sounding Editing Form

The interactive Sounding Editing form is opened by selecting the **Edit** command from the menu when a sounding window is selected:

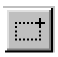


The form includes commands for performing various editing functions:

Selecting Data Points

 Single point selection

Click to select/unselect for editing single data points on the curve.

 Multiple points selection

Click to select/unselect for editing all data points included in a user-defined range: click and drag the mouse to define the area.

Zooming in on the Curve

 Zoom in

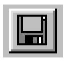
1. Click a point in the curve diagram and release the button.
2. Move the mouse to define the zoom area.
3. Click and release the left mouse button.

 Zoom out

Click to reset the previous view.

Note: this button is available only after the **Zoom in** button has been used. It remains available until the original curve size is restored.

Saving Editing Changes



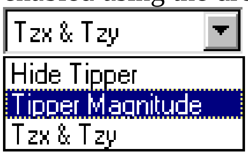
Save button

Saves the editing changes. These changes must be confirmed when closing the station window in the main menu in order to store them in the database.

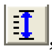
Editing Data Points

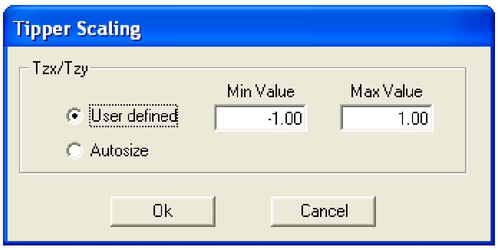
Masking data: Using the appropriate buttons, it is possible to mask individual modes at each frequency, either across all modes or for individual modes. Changes made here are reflected throughout the database, i.e. in the 2D Inversion and Pseudosection modules.

Note that the Mask Tipper button is enabled only when tipper display is enabled using the dropdown list at the top of the screen:



Select the Tzx & Tzy display option to display the two tipper components in separate windows.

The vertical scaling of the tipper magnitude and Tzx/Tzy components can be set using the Tipper Vertical Scaling button , which appears towards the left end of the tool bar when either Tipper Magnitude or Tzx/Tzy are visible. Clicking this button opens the Tipper Scaling dialog box:



Settings made here are retained in subsequent WinGLink sessions.

Shift data points

1. Click and select the data points to be shifted by single or multiple selection.
2. Move the data points to the desired positions and click the left mouse button.
3. Click the **Shift** button again to confirm the action.

Smooth Calculate and display a smoothed curve

Click the button to display the smoothing options

Cancel Cancel the current action

Undo button

Undo all the editing changes made since the last time the curve was saved. All editing steps can be undone provided the station has not yet been saved since editing.

Swapping

Swapping allows you to assign a data point of the XY curve to the YX curve (at the same frequency) and vice versa. This is useful where analytic rotation results in inconsistent mode assignment.

To swap the data points of apparent resistivity or phase curves:

1. Open the station or select its window if already open.
2. Select the **Edit** command from the main menu.
3. Click the Swap button.
4. Click the data point(s) you wish to swap with the left mouse button.
5. Click the Swap button again.

Note: To swap the entire curve, use the right mouse button at Step 4.

Static Stripping

Static stripping is an analytic technique for eliminating the frequency-independent offset of one apparent resistivity curve from the other.

To perform static stripping on an apparent resistivity curve:

1. Select the **Edit** command on the main menu.
2. Click the Strip button.
3. Click a data point on the curve and release the mouse button.
4. Move the curve to the desired location and click the left mouse button.
5. Click the Strip button again.

Normally, you would select a point at the high-frequency end of the curve. Practical experience suggests that most static shift cases result in one curve being moved down relative to the other. You therefore move the lower curve up to the higher one.

When the original apparent resistivity curve is dragged to its new position, a scalar multiplier of the e-field value is derived. This is applied to a recalculation of all the impedances of that sounding. Thus, when you click **Strip** for the second time, you should see the hour-glass symbol, indicating that this calculation is taking place. The curves are then redrawn. Only the apparent resistivity curves are affected by this procedure.

If the data show a very large static shift, or are strongly 3D at high frequencies, the results can sometimes be unstable. You may need to repeat the operation several times.

Static Shifting

Static shifting allows you to vertically shift all data points of a curve.

To statically shift a curve:

1. Select the window with the sounding to edit
2. Select the **Edit** command on the main menu.
3. Click the S-Shift button.
4. Click one of the curve's data points and release the button.
5. Using the mouse, move the curve to the desired location. The amount of static shifting is displayed in the bottom of the window:
   Shift xy=1.002 Shift yx=0.319
   This number is a scalar factor that multiplies the original resistivity values of the curve. If equal to 1, the curve is not shifted.
6. Click and release the left mouse button.

The apparent resistivity range changes so that the curves appear centered in the display box.

A useful guide for determining the resistivity level to which a curve should be shifted is obtained by displaying the pseudo-MT curve for a co-located TDEM station.

To reset the static shift to zero:

1. Click the SS Reset button.
2. Smooth to run Smoothing routines
   + 360 °
   - 360 ° to add/subtract 360° to phase data points
   EM mod to show pseudo-MT curve from TDEM model

Calculating Smoothed Curves

Resistivity and phase smoothed curves can be calculated from the edited curves using three different techniques. They are automatically updated whenever further editing is made on the curves.

To (re)calculate a station's smoothed curves:

1. Open the station or select its window if already open.
2. Select the **Edit** command on the main menu.
3. Click the Smooth button.

4. Select **Smooth Options** and click the **OK** button.

The available smoothing options are described below

Sutarno phase consistent smoothing

This is an application of the Hilbert transform to give an apparent resistivity curve from the phase curve. (Sutarno, D. and Vozoff, K., 1991, Phase-smoothed robust M-estimation of magnetotelluric impedance functions: Geophysics, 56, 1999-2007). It is used primarily to confirm that apparent resistivity and phase are consistent.

D+ smoothing

D+ relates apparent resistivity and phase of the same component (xy or yx) through a D+ function. In essence, this finds the one-dimensional earth which best fits both parameters. The procedure has been shown to be valid for most 2D data and for some 3D cases.

The errors attributed to the data can be those estimated by the original data processing or can be imposed by the user. In the first case, the two parameters (apparent resistivity and phase errors) are left at the default value of -1. In the second case, the user can estimate the errors (e.g. 10%). By appropriately selecting errors, one parameter can be down-weighted at the expense of the other. *Reference*: Beamish, D., and Travassos, J.M., 1992, The use of the D+ solution in magnetotelluric interpretation. Jo. Appl. Geophys., 29, 1-19.

Numerical smoothing

Numerical smoothing is an FFT-based low-pass filter which calculates an independent smooth curve for each of the four components. It does not have an underlying geophysical process to support it, but may nevertheless be useful in certain noisy situations. The user selects a smoothing factor which gives an appropriate result. 0 gives no smoothing at all, and useful values are typically << (half the number of data points). *Reference*: Press et al., 1989. Numerical Recipes, *pub.* Cambridge University Press, Section 13.9.

Adding ± 360° to Phase

This feature is useful in the case of phase curves which wrap around the normal -180 to +180 range:

To add 360° to data points of a phase curve:

1. Select the **Edit** command from the main menu.
2. Click the 

button.
3. Select the data points of the phase curve you want to shift.
4. Click the 

button again.

To subtract 360° from data points of a phase curve:

1. Select the **Edit** command from the main menu.
2. Click the 

button.
3. Select the data points of the phase curve you want to shift.
4. Click the 

button again.

Computing Pseudo-MT Curve from Co-Located TDEM, Schlumberger Sounding Model or DC Model

Displaying the pseudo-MT curve for a co-located TDEM or DC station

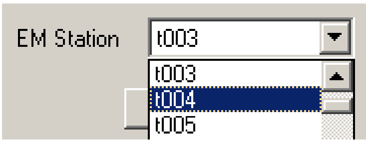
A well-established procedure in MT surveys is to use co-located time-domain EM (TDEM) or DC soundings to correct for static shift (Sternberg, B. K., Washburne, J. C. and Pellerin, L., 1988, Correction for the static shift in magnetotelluric using transient electromagnetic soundings: Geophysics, 53, 1459-1468; Pellerin, L. and Hohmann, G. W., 1990, Transient electromagnetic inversion: A remedy for magnetotelluric static shifts: Geophysics, 55, 1242-1250.). In WinGLink, we assume that central or coincident loop TEM or DC soundings have been edited and inverted, either in WinGLink or using TEMIX/RESIX (Interpex products for TEM and DC data, respectively) and then imported into WinGLink.

The appropriate sounding is selected using the procedure shown below. The imported 1D model is used to calculate a forward MT response. Using static shift, the observed apparent resistivity curves can then be shifted along the resistivity axis to coincide with the values suggested by the TDEM or DC response.

To display the MT curve calculated from a TDEM or DC model:

1. Select the **Edit** command from the main menu.
2. Click the 

button.
3. Select the EM or DC Project containing the EM/DC station. The project must contained in the current database:


select the EM or DC station located at the MT station being edited.


Note: Either **EM Station** or **DC Station** is displayed to the left of the dropdown list depending on the type of station selected.

Common Problems in MT sounding Editing

There are some problems which can occur while editing MT soundings. A list of the most common problems is given below, together with the solution:

Problem: Strip button is disabled in the Edit form

This may happen for one of the following reasons:

1. You did not rotate the impedances after importing the EDI file. *Solution*: exit Edit and rotate impedances.
2. You ran a La Torraca or Torquil Smith decomposition. *Solution*: exit Edit and reload original curves.

3. The station comes from the import of an EDI file which had neither impedances nor spectra.

Problem: Calculated smooth curves are no longer displayed

This happened because:

1. You reloaded the original curves. *Solution*: calculate the smoothed curves again.

Problem: Smoothed curves are not recalculated after shifting data points

This may happen because:

1. **The smoothed curves were calculated using the Sutarno option.** With this option, the smoothed curves are recalculated from impedances and not from the plot parameters.

Problem: When running D+ smoothing, an "invalid command" message is shown

This may happen because:

1. A La Torraca or Torquil Smith decomposition has been previously applied, and
2. You did not specify the Rho Err % and Ph Err %. Normally, these parameters are initialized with the default value of -1, which means that the variances found in the EDI file will be used. However, since the La Torraca decomposition eliminates these variances, they can no longer be used. *Solution*: try D+ smoothing again, and specify a value (in the 1-100% range) for both Rho Err and Ph Err.

Reloading Original Curves

This action overwrites the edited curves of a station with the original curves.

To reload original curves:

1. Select the station by clicking its window.
2. Select the **File |Re-Load Original** menu command.

After reloading, the smoothed curves and the 1D models need to be recalculated.

1D Inversion

1D Inversion: Overview

The 1D Model menu command starts the inversion program for the station whose window is currently active.

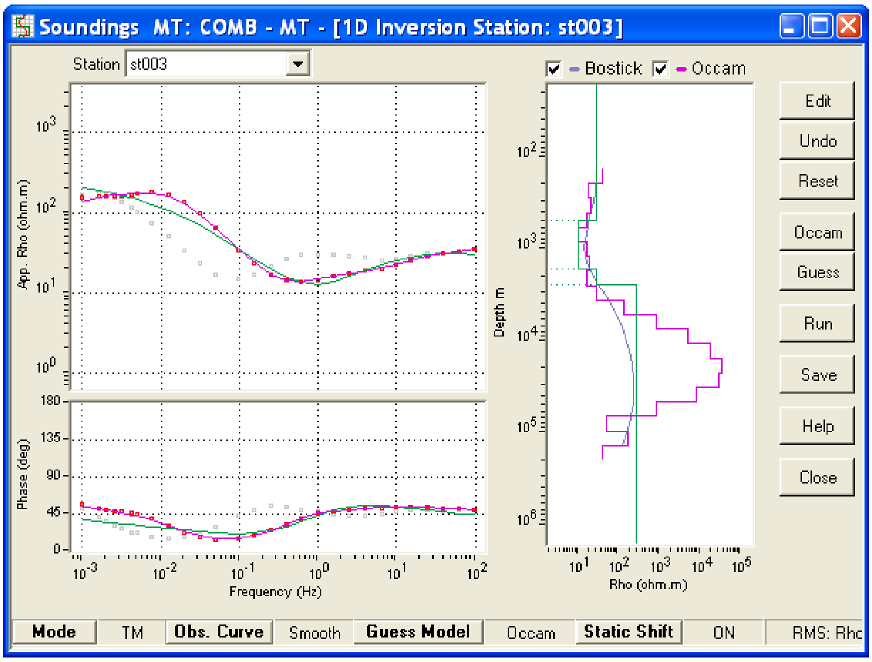
The main steps involved in the inversion process are:

Set curves to use for inversion	The inversion can be run using the edited or smoothed curve. The curve selection button,

	located on the bottom bar, allows the user to change the selected curve at any time. The models must be recalculated when the curve to be used is changed. For each sounding, the TE, TM **or** invariant curve (i.e. one of the three) can be used to calculate an inversion model. A mode selection button allows the user to change the selected curve at any time. The models must be recalculated when the mode to be used is changed.
Calculate the smooth inversion model	For each station, one smooth model is calculated and stored in the database. This model will be used to produce imaged sections for the project. Two types of smooth models can be calculated and saved separately: the Bostick model and the Occam model (the respective Bostick and Occam imaged sections will be calculated).
Guess the layered inversion model from the smooth model	A layered model can be guessed from the calculated smooth models. The number of layers is entered by the user by editing the inversion settings or by editing the model with interactive graphic commands. Only one layered model is saved for each station. A guess model selection button allows the user to change which smooth model is to be used for the guess routine.
Edit the initial model using the graphic model editor (optional)	An interactive graphic editor allows the editing of layers thicknesses and resistivities. The response is displayed in real time.
Run the inversion and save the model	The inversion process can be run with a given number of iterations and predefined RMS fitting degree. Resistivity and/or thickness values of one or more layers can be kept fixed.

The 1D Inversion Window

The following figure shows the 1D inversion window:

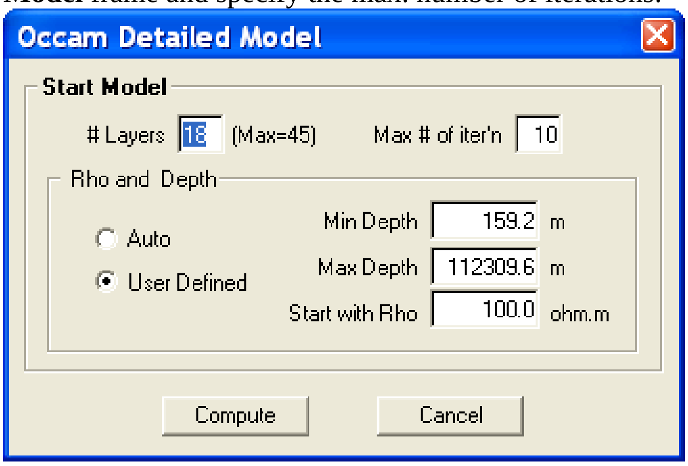


To Select the Curve to be Used for Inversion

1. Select the **1D Model** menu command to enter the inversion mode for the active sounding.
2. Click the  button, located at the bottom of the form, to select the edited or smoothed curve for inversion.
3. Click the  button, located at the bottom of the form, to switch between TE/TM and invariant curve to be used as the 1D curve.
4. Recalculate the model whenever you change the selected curve.

To Calculate the Occam Model for the Current Sounding Curve:

1. Click the  button, then fill the fields in the **Start Model** frame and specify the max. number of iterations:



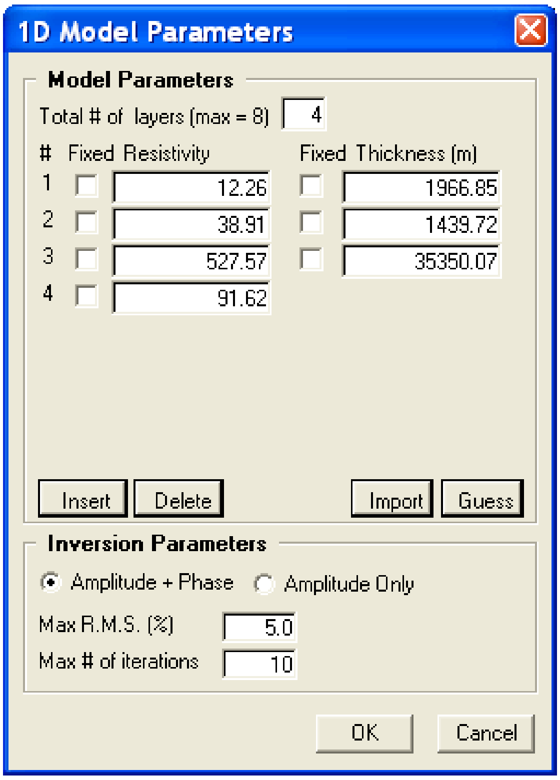
The Occam model and the corresponding curve are displayed with magenta lines.

The Bostick MT model is automatically computed for the selected 1D curve.

The Bostick model and the corresponding curve are displayed with blue lines.

To Guess the Layered Model from the Smooth Model

1. Use the **1D Model** command to enter the inversion mode for the active sounding.
2. Calculate the smooth model (magenta lines), if you have not already done so.
3. Click the  button:

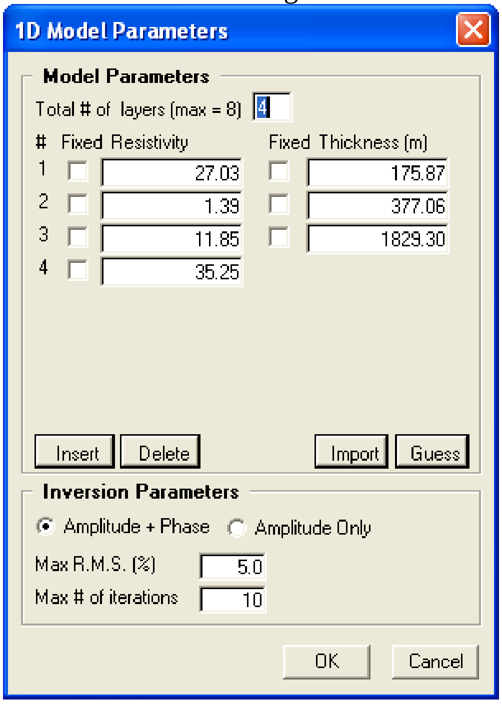


4. The layered model will be guessed (green lines).

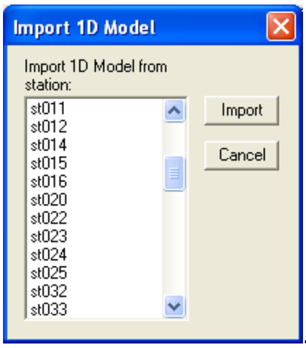
Click the  button on the bottom bar to switch between Bostick and Occam as guess models.

To Import a Model Created for a Different Station:

1. In 1D Modeling mode, click the **Edit** button to open the 1D Model Parameters dialog:



2. Here, click the **Import** button to open a dialog box which contains all available stations:



.

3. Select the desired station and click Import. Note that any changes to the imported model are *not* reflected in the original model.

To Run a Layered Model Inversion:

1. Use the **1D Model** command to enter the inversion mode for the active sounding.

2. Click the 

button.

3. The inversion is started using the model shown with green lines as the starting model and the inversion parameters set for the sounding.

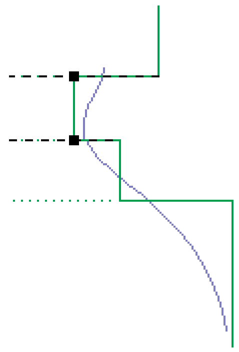
Graphic Editor for 1D Layered Models

The layered model can be edited graphically using drag-and-drop operations.

The resulting resistivity curve is updated in real-time, allowing you to compare the fit with the observed curve.

To Graphically Change the Resistivity of a Layer:

1. Click the vertical line which denotes the layer's resistivity. Two black dots appear on its edges indicating that this line can be moved:



2. Release the mouse.
3. Move the mouse pointer to the selected line; the arrow pointer changes to a double-headed arrow.
4. Left click and hold down the button; drag the line to the new layer resistivity delimiter position.
5. Release the mouse button. The resistivity curve coming from the layered model is recalculated and redrawn.

To Graphically Change a Layer's Thickness

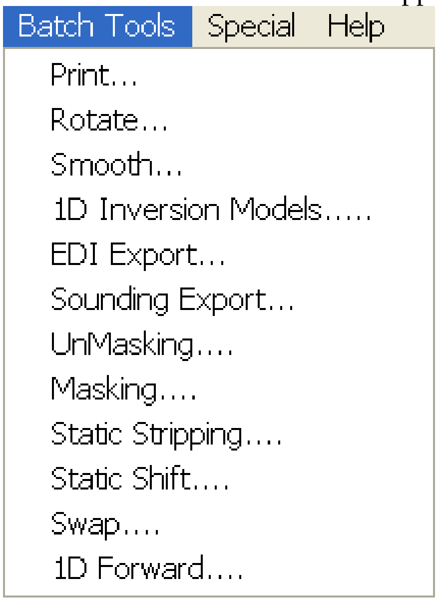
1. Repeat the procedure used to change a layer's resistivity, but instead select one of the horizontal lines which delimits the layer and move it to the desired position.

Batch Commands

A group of batch procedures are available for performing selected operations on a set of user-selected stations. To start a batch procedure:

1. Close all open stations.

2. The **Batch Tools** command will appear on the main menu:



3. Select the desired operation.
4. Enter the parameters as requested.

Operations available in batch mode include:

- Curve smoothing
- 1D inversion modeling
- Printing
- Impedance rotation
- Tipper rotation
- Exporting edited curves as EDI files
- Masking/ unmasking parameter values
- Static stripping
- Static shifting
- Swapping modes for a specific frequency range
- EDI export; this function also includes options for exporting static shift values to an external text file. File format for the static shift file is: station name, xy shift, yx shift.
- Batch 1D forward: Enables the selection of an existing 1D model and the computation of the MT forward response for all selected stations. In addition, the user can specify whether synthetic data should overwrite the observed data.

Special Menu Commands

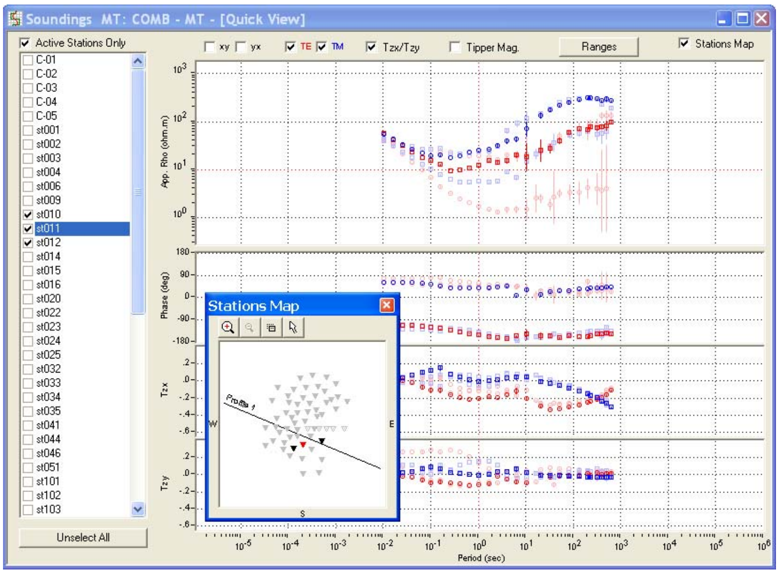
The **Special** menu, which is available on the menu bar when no soundings windows are open, contains a number of special display and edit functions:

- Quick View

- Quick Edit
- Miniature Curves
- Polarization Maps

Quick View

The **Quick View** function provides a way to display multiple soundings in the same Window. To open the Quick View window, select the **Special | Quick View** menu command:



Use the check boxes at the top of the window to select the components to be displayed.

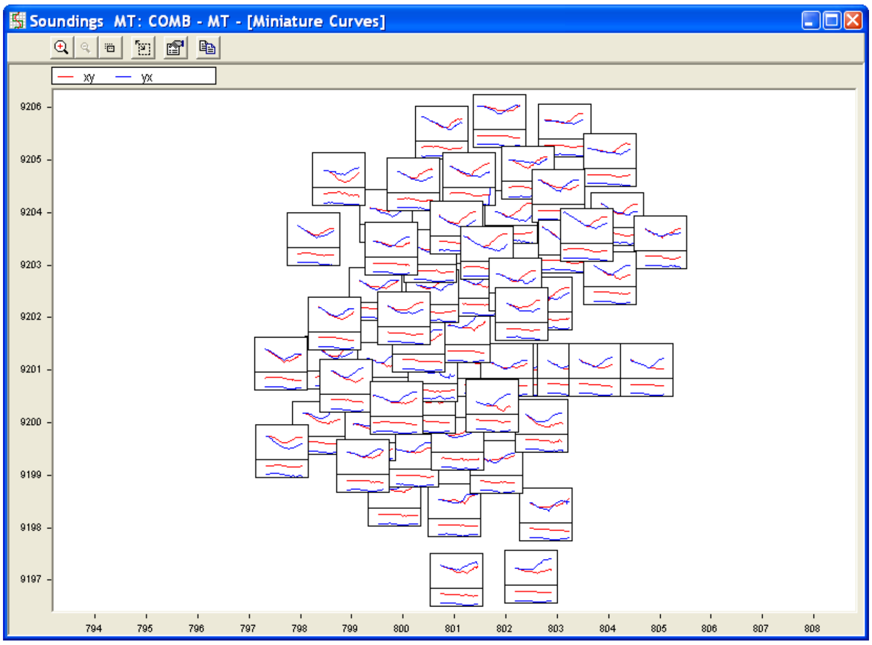
The Stations Map window, which can be toggled on and off with the corresponding check box in the upper right corner of the window, indicates the positions of the stations which are currently open (black squares) as well as the position of the active station (red square).

Quick Edit

The **Quick Edit** function, accessed with the Special | Quick Edit menu command, opens the MT Soundings edit window without requiring that individual stations be opened. To select a station for editing, using the **Station** drop-down list at the top of the screen. Limitation of this function is that only editing can be performed; in order to model and perform inversions, the stations must either be opened individually or a batch operation performed.

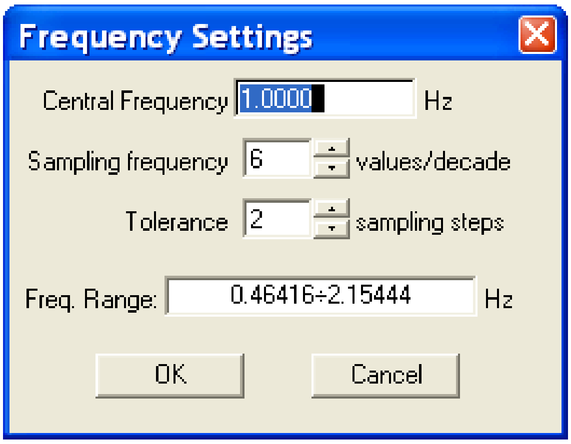
Miniature Curves

Select the **Special | Miniature Curves** menu command to display a miniature curve map with all stations in the current projects:

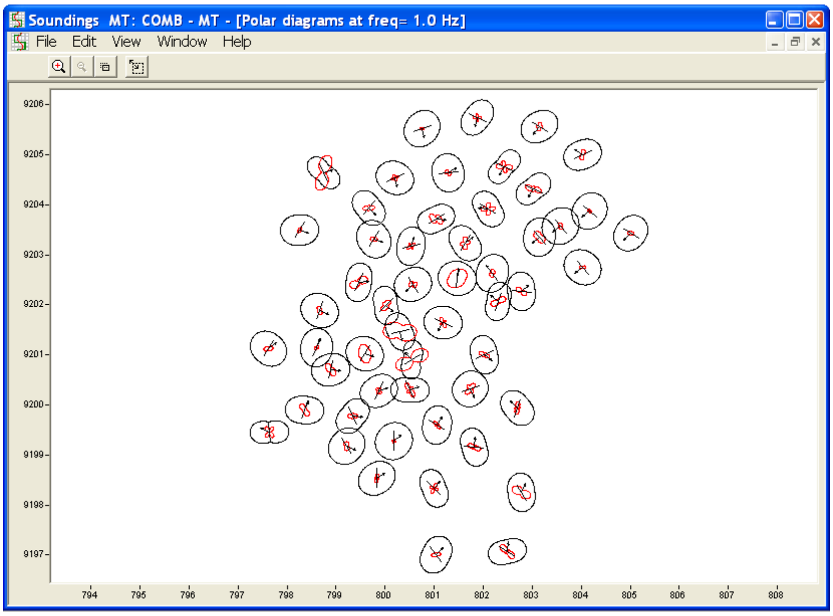


Polarization Maps

Similar to the **Miniature Curves** function, the **Polarization Maps** option on the **Special** menu can be used to display polarization maps for a specific frequency. After selecting the menu command, a dialog box appears prompting you to specify a central frequency, in addition to several other parameters, for the polarization maps:



Click the **OK** button to generate the specified polarization map:


Soundings MT: COMB - MT - [Polar diagrams at freq= 1.0 Hz]
File Edit View Window Help
9206
9205
9204
9203
9202
9201
9200
9199
9198
9197
794
795
796
797
798
799
800
801
802
803
804
805
806
807
808

17: MT Pseudo-Sections

Overview

The Pseudo-Sections program reads the apparent resistivity vs. measuring parameter values for each station associated with a profile, then interpolates them to display a section showing the lateral variation along the profile.

Pseudo-sections can be generated not only for station data, but also for 2D model responses calculated using the 2D MT module. In addition, it is also possible to calculate difference pseudo-sections for sections of the same type on the same profile. Comparisons can, thus, easily be made between 2D models created along a given profile.

Each pseudo-section is defined by a profile trace and its associated stations, which supply the datasets with the corresponding data.

With the Pseudo-Sections program, multiple sections can be open at any time. Pseudo-sections may also be opened along multiple profiles. Each section is displayed in a separate window.

Apparent resistivity and apparent phase are displayed as pseudo-sections in a single window. Tipper pseudo-sections are displayed in a separate window.

The curves used to generate the pseudo-sections can be viewed and edited, one at a time, by clicking the vertical dotted line below each station on the section.

A limited amount of station editing is possible in the Pseudo-Sections program: data points can be masked and stations can be activated/deactivated. Edit operations are stored in the database and apply throughout WinGLink, i.e. in Maps and MT 2D Inversion.

Creating a New Pseudo-Section

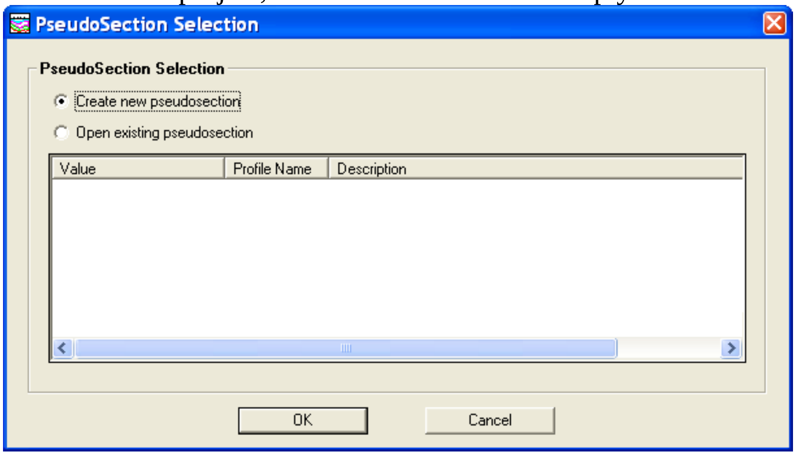
To create a **new** MT pseudo-section

1. Construct a new profile, as described in the “Maps” chapter of this manual.
2. Make sure you have imported or entered the field data for each station of the new section. You can use the **MT Soundings** program to check that the stations have data. The MT Soundings program uses the edited data, if available, or the

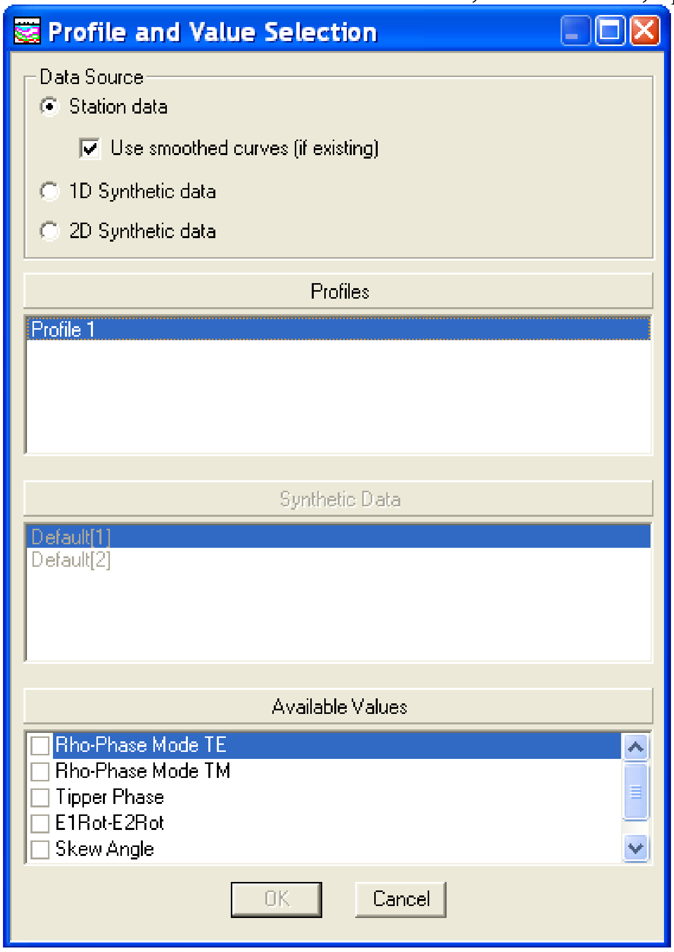
smoothed data, if requested. Alternatively, the original field data can be recalled.

3. Start the **Pseudo-Sections** program.

4. In the PseudoSection Selections window, which opens immediately after starting the program, select from the existing pseudo-sections. If no pseudo-sections have yet been created for the current project, the selection window is empty.



5 After selecting the "Create new pseudosection" option and clicking the **OK** button in the PseudoSection Selection window, the Profile and Value Selection window, shown below, opens:



In the Data Source section of the window, specify whether you would like to use station or synthetic data for the pseudo-section. If using station data, the program uses smoothed curves, if available, by default.

Synthetic data, the model response returned by the 2D Inversion program can be used as the data source by making the appropriate selection in the Data Source frame, then selecting the model to be used.

Note that when making a selection, you must be consistent with the data source, i.e. it is not possible to open station and synthetic data without first closing and reopening the Profile and Value Selection window.

Pseudo-sections which are already open are not available for selection in the **Available Values** frame.

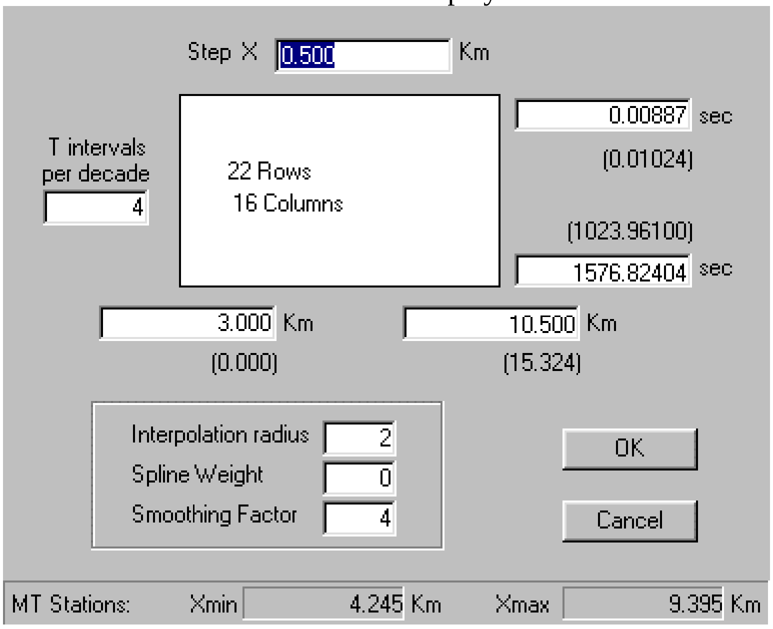
6. The section is displayed and can now be edited and saved in the database for future modifications or printing.

Gridding the Profile Data

The data of the stations used to generate each pseudo-section must be gridded in order to display contours and color ranges.

To Grid the Profile Station Data

1. Select the section you want to grid.
2. Select the **Gridding | New Grid…** menu command. A form similar to that shown below this is displayed:



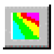
3. The numbers in parenthesis on the gray background show the maximum and minimum values of the data; the current value to be used to define the grid extent must be entered in the field-box close to each number.

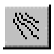
4. In the **Step X** field, enter the step in the specified units. This number determines the number of columns in the grid, which will be shown in the center white box, representing the section.

5. In the **T intervals per decade** field, enter the number of grid rows per decade.
6. Fill the fields with the values for the interpolation radius, spline weight and smoothing factor.
7. Click the **OK** button.

Note: As the interpolation radius increases (values >5), the gridding algorithm attempts to fill any holes present in the data by interpolating between relatively distant data points. When creating difference pseudo-sections (see below), we therefore recommend using a low interpolation radius (2) to prevent the creation of misleading differences.

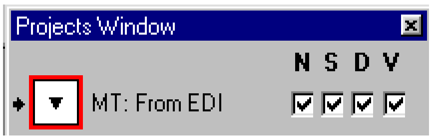
To Display Color Ranges and Contour Lines

Click the  button to display/hide color ranges.

Click the  button to display/hide contour lines.

Displaying Station Dataset Values

On the **View** menu, select the **Projects Window**, if not already displayed:

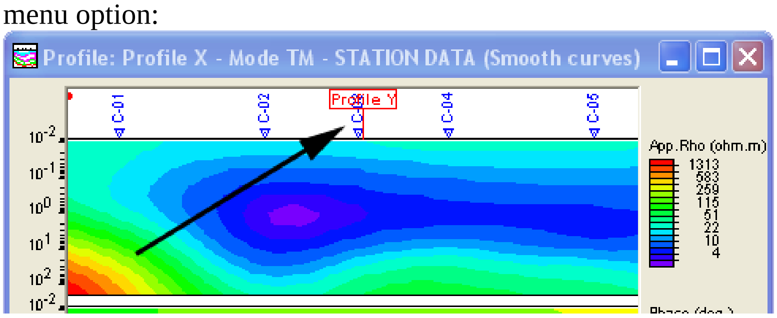


Select the **V** box to display the values of the dataset at each station.

To update the display of the section, click the section with the mouse.

Displaying Profile Intersections

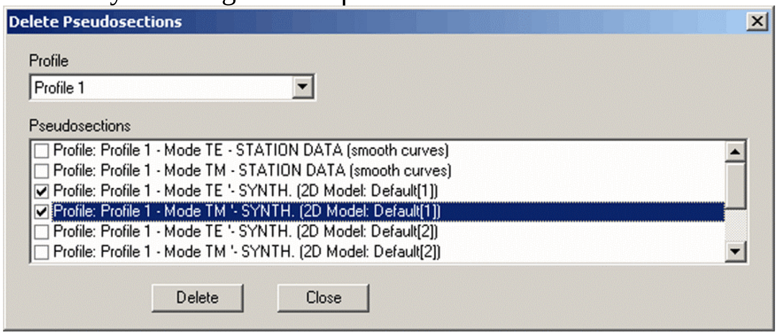
If any other profiles in the database intersect the current profile, the point of intersection can be displayed by enabling the **View | Crossing Traces** menu option:



Note: This display option applies only to the current window.

Deleting Pseudo-Sections

Any unopened pseudo-section contained in the current project can be deleted by selecting the **File | Delete Pseudo-Section** menu command:



Exporting Grids

Grids associated with the TE/TM modes of amplitude and phase pseudo-sections can be exported to an external file in a number of different formats. To export, select the **Export Grid** menu command on the **File** menu.

Editing Data Points

A number of edit operations can be performed on the data associated with each station. These include masking samples, static shift and station state, i.e. active or inactive.

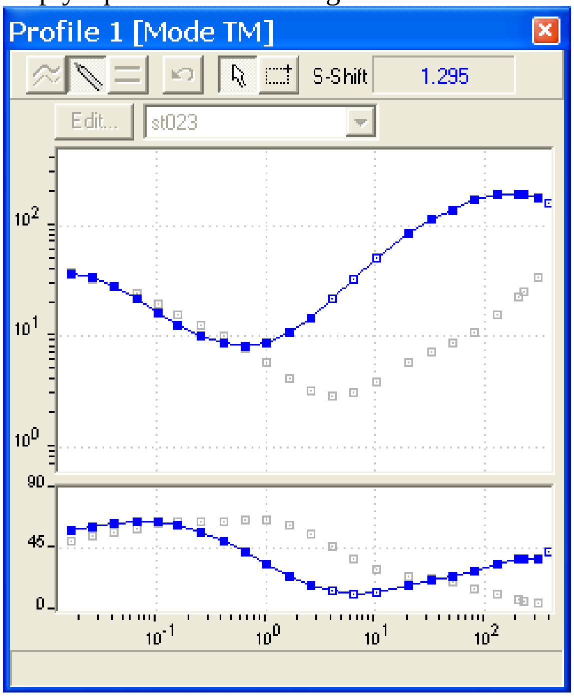
Edits made here affect the stations elsewhere in the database. Thus, a station disabled in the Pseudo-Sections program is not available in the MT 2D Inversion program.

To Mask or Unmask Station Samples:

1. Click the dataset you wish to edit in the section (the data are shown as dots below each station. This will bring up the data window for the station to be edited).
2. Click the 

button on the station data window.
3. Click the  button.

4. Click the point(s) to be masked. The solid squares become empty squares after masking:



5. Click the masked point(s) to be unmasked.

6. Click the  button again to confirm your edit changes.

7. Exit the station window, or click another station to edit its data.

8. Use the  button to switch modes.

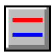
To Adjust the Static Shift of a Curve

The static shift applied to the apparent resistivity curves can be edited from within the pseudo-sections program:

1. Bring up the editing window for the desired station by double clicking on the symbol of the station along the pseudo-section.

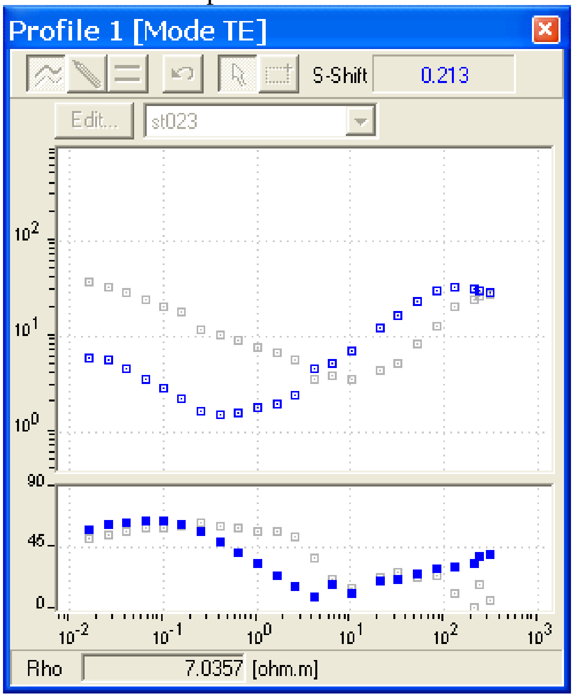
2. Click the 

button on the station data window.

3. Click the  button to select the curve you would like to shift; the unselected curve is displayed in gray.

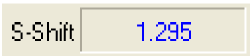
4. Click the  button.

5. Move the mouse pointer next to the curve and single-click.



6. Shift the curve by moving the pointer; when the curve is positioned, single-click.

The amount of the static shifting is displayed in the box labeled S-Shift:



This number is a factor that multiplies the original resistivity values of the curve. If equal to 1, the curve is not shifted.

To Change the State of a Station

1. Select the **Station | Edit Status** menu command.
2. Click the station symbols at the top of the profile to toggle the current state. When disabled, the stations are displayed in light gray.

Station states may only be altered if the current project is a single project, i.e. this function cannot be applied to stations contained in an integrated project.

To Change the TE/TM Sounding in a Pseudo-Section

For each profile, both the TE and the TM pseudo-sections are constructed. It is possible to specify whether the XY or YX resistivity curve of a station be used for a TE or TM pseudo-section.

1. Select the **Edit | Stations Mode** menu command; the station symbols and names in the active window turn red or blue in color. If a station is red, it means that the XY curve is used; if it is blue, the YX curve is used instead.
2. Click the symbol of the station(s) for which you would like to change the mode. If you change the mode of stations in a TE (TM) section, the mode of those stations is automatically changed in the TM (TE) section.

This function can only be used if the current project is a single project, i.e. it cannot be applied to stations contained in an integrated project.

Note: In order to see the effects of changes or edits made to the stations on the section, you must select the **Gridding | Regrid** menu command.

Note: Data points for the **tipper component** can also be masked using the same technique as is used for the TE and TM components.

Creating a New Difference Pseudo-Section

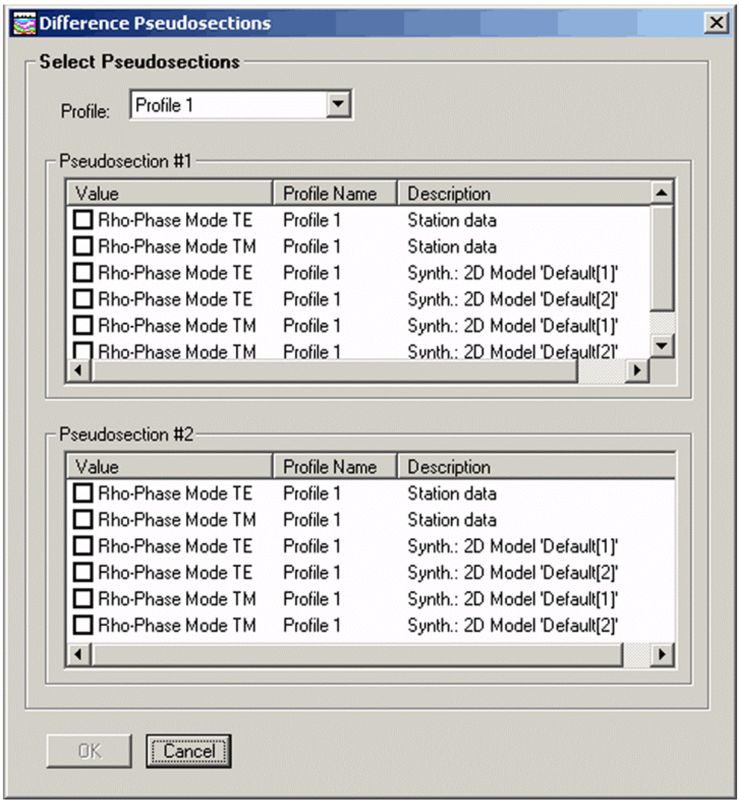
Difference pseudo-sections can be created for any two pseudo-sections of the same type, i.e. TE, TM or tipper, which are contained in the same project. Difference pseudo-sections are displayed as a percentage and provide a useful tool for determining the differences between models.

The PseudoSections program creates pseudo-sections by copying the gridding parameters of the first pseudo-section into the second pseudo-section. Both sections are internally regridded, i.e. the data are reimported from the database, whereby any editing changes remain in effect. The second pseudo-section is then subtracted from the first, i.e. the grids not the data sources. This difference is then divided by the first, and the result multiplied by 100%: ((P1 - P2) /P1) * 100%

The same gridding parameters as were used for both P1 and P2 are used to regrid the resulting difference.

To Create a Difference Pseudo-Section:

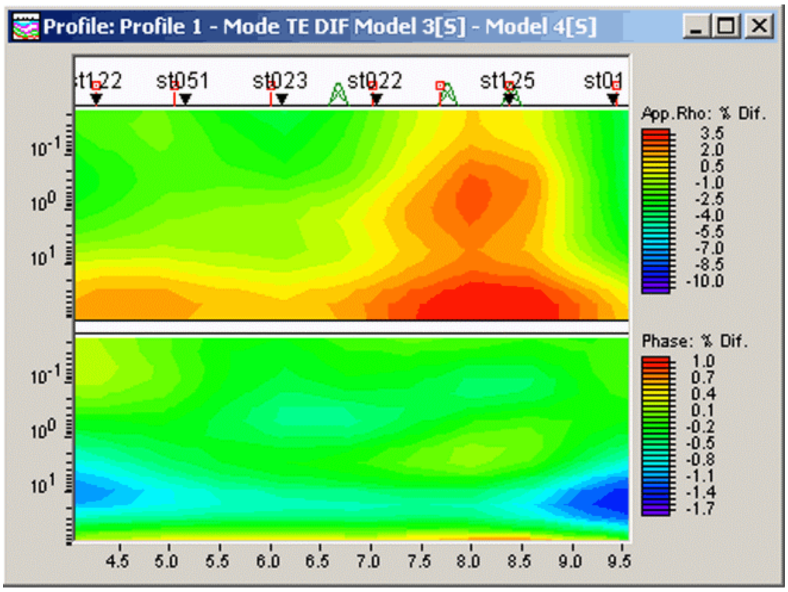
1. If not yet created, create two pseudo-sections of the same type along the same profile as described above.
2. Select the **File | Create Difference Pseudo-Section** menu command to open the Difference Pseudosections selection window:



All pseudo-sections which have been created on the selected profile and which may be used for difference pseudo-sections are listed.

Note: only TE, TM and tipper pseudo-sections created from either station or synthetic data may be used to create difference sections. Difference pseudosections themselves may not be used.

3. Select two pseudo-sections, one in each of the Pseudosection frames. Only if the pseudo-sections are of the same type, yet not identical, is the **OK** button enabled. The difference pseudo-section is calculated and displayed after clicking the **OK** button:

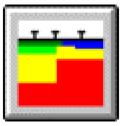


4. Make changes to the ranges as necessary using the **Contours | Ranges** commands.

The various **Copy** commands on the **Window** menu are available for difference pseudosections and can be used to copy gridding and window parameters between both difference and standard pseudosections. Please refer to the “Common Functions” chapter of this manual for further details.

18: MT 2D Inversion

Overview



The MT 2D Inversion program contains two different routines for running inversions:

1. **A smooth model inversion routine**
Developed by Randy Mackie, this routine finds regularized solutions (Tikhonov Regularization) to the two-dimensional inverse problem for magnetotelluric data using the method of nonlinear conjugate gradients. The forward model simulations are computed using finite difference equations generated by network analogs to Maxwell's equations. The program inverts for a user-defined 2D mesh of resistivity blocks, extending laterally and downwards beyond the central detailed zone, and incorporating topography.

2. **A sharp boundary model inversion routine**
Also developed by Randy Mackie, this routine is based on the inversion of 2D MT data for discrete interfaces and the resistivities of the layers between those interfaces. The interfaces are described by a series of nodes, whose horizontal positions are fixed, but whose vertical positions can vary in the inversion. The interfaces are assumed to transect the entire model, i.e., there are no closed bodies. The interface varies linearly between each interface node. The resistivity of each layer is also described by a set of nodes at fixed horizontal positions within each layer. The resistivity is assumed to vary linearly between nodes. The interface and resistivity information is projected onto a finite-difference mesh for computation, and the inversion calculates the best fitting interface node locations and resistivity nodal values in order to fit the observed data.

An assumption common to both routines is that the profile to be inverted is perpendicular to the electrical strike.

Preparing the Input for the MT 2D Inversion Program

The preparation of the input for the 2D MT inversion program requires the use of other programs to edit the data and construct each section. Refer to the “Maps” chapter of this manual or the WinGLink Help System for more details:

 MT Soundings

- Edit and save observed MT curves.
- Assign TE/TM mode for each sounding.

Maps

- Construct sections by adding MT stations to a profile trace.

Pseudo-sections

- Further editing of data and mode assignment of MT curves.
- Masking bad data in each separate TE/TM curve. (The editing carried out in the **Soundings** module will mask frequencies but not individual samples for each separate TE/TM component)

Creating and Loading Models

Elements of a Model

A typical 2D inversion model consists of:

A Mesh

This is a finite-difference mesh, which must have at least one column for each station of the profile. Each cell of the mesh is assigned a user-defined resistivity value. The model mesh is the input model for the smooth inversion routine.

The program allows two types of automatic mesh generation

1. **Coarse mesh** generation, where the thickness of the rows increases with depth according to a fixed, pre-assigned scheme (suitable for most MT data).
2. **Fine mesh** generation, where the thickness of the rows increases with depth based on the initial resistivity set for the model and the frequencies involved (more suitable for high-frequency data).

Resistivity Interfaces

Resistivity interfaces are elements of the model used **only** by the sharp boundary inversion routine. The interfaces are polygonal lines added to the model to create layers of different resistivity, hence assuming a sharp variation of the resistivity within the mesh.

Model validation criteria: To run a sharp boundary inversion, you need to define a **valid** initial model. This is a model with:

1. **at least** 1 resistivity interface in addition to the default interface corresponding to the air-earth interface (topography).
2. **at least** 1 resistivity node on *each* interface (including the air-earth interface).

An error message appears when trying to run a sharp boundary inversion on models which do not meet the above criteria.

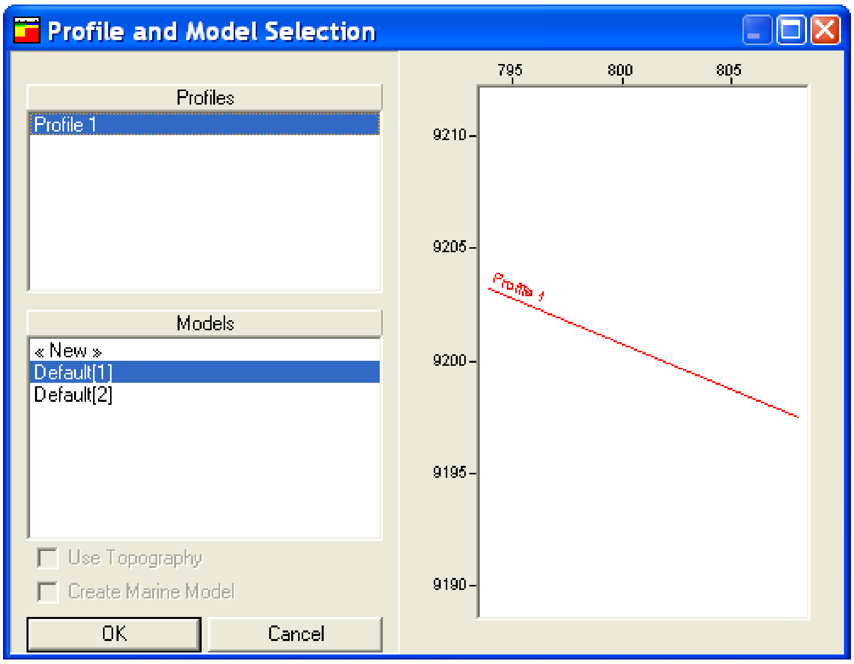
Resistivity Nodes

Resistivity nodes are elements of the model used **only** by the Sharp Boundary inversion routine.

The nodes are markers put on the interfaces to define the lateral variation of the resistivity below the interface. At least one node for each interface must be set, including the topography which is considered by the program as the top interface.

Creating and Loading Models

Each profile having stations may have many models associated with it. To display the list of models associated with a profile, select the **File | Open Model** menu command and select the desired profile from the Profiles box:



In the **Models** box, select the model name, then click the **OK** button.

To create a new model, select the **New** item, then click the **OK** button.

Each model is based on a finite difference mesh. Whenever a new model is added, a new mesh is generated with:

- at least one column for each station.
- the same resistivity for all cells.
- a predefined number of rows and columns.

If the **Use Topography** checkbox is selected, the model incorporates the topography along the profile according to the following rules:

1. From the area topography grid as defined in the WinGLink shell.
2. From the project topography grid.
3. From the elevations of the stations attached to the profile.

You can then use the **Edit Model** commands to modify the current model. The **Edit Model** commands are made available when you click one of the following two icons:

This button starts Edit Mesh mode. The mesh can be edited by:

- Adding rows or columns
- Deleting rows or columns
- Changing the size of a row or a column
- Changing the resistivity of the cells
- Locking/unlocking cell resistivity values

This button starts the Edit Resistivity Interfaces mode.

This is used to build or edit a sharp boundary inversion model. The model can be edited by:

- Adding model interfaces
- Adding resistivity nodes to the interfaces

This button starts the Edit Tears mode.

This is used to define tear areas on a model (use of this function is described further below).

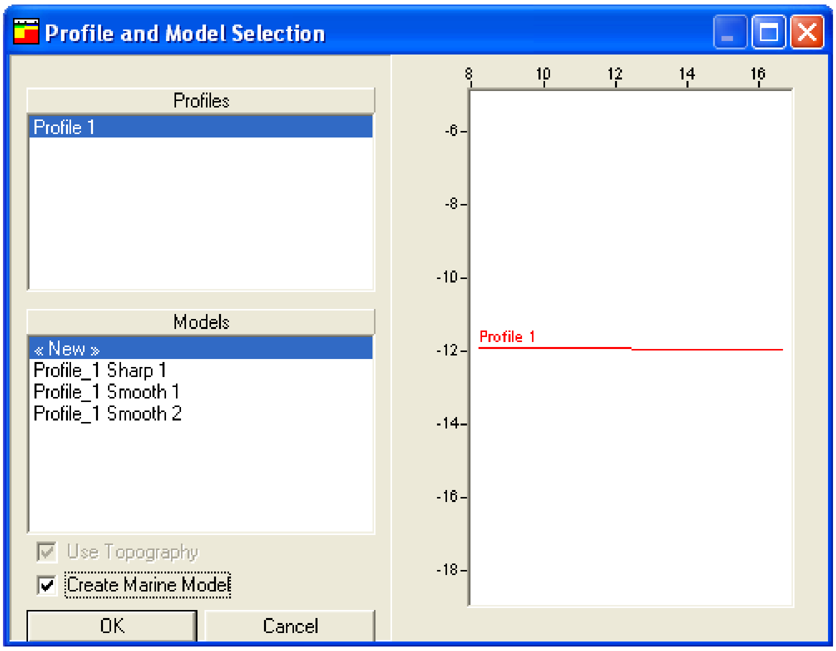
To **exit** the edit modes, reclick the same button used to enter the respective edit mode: the program will ask if you wish to confirm the changes you made. Use of the edit functions is described further below.

Creating Marine / Amphibious Models

Marine or amphibious models can be created for profiles along which at least some of the stations have elevations less than 0m. The default resistivity of cells located between sea level and the sea floor for sections of submarine profile is 0.33 Ohm.m. This resistivity value remains fixed during model inversion. Marine/amphibious models are inverted in the same way as standard terrestrial models.

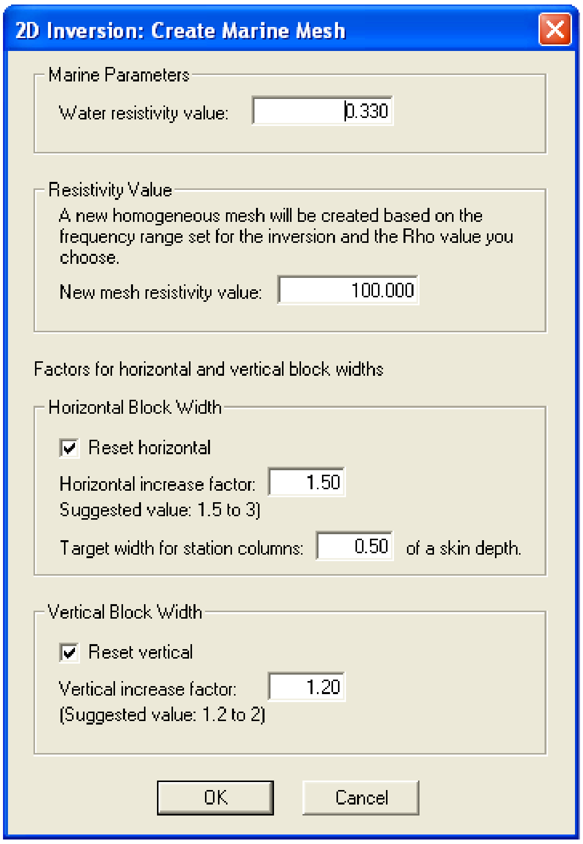
To Create a Marine (or Amphibious) Model:

1. Select the **File | Open Model** menu command. The Profile and Model Selection dialog box opens:



Select the **<New>** item, and select the **Create Marine Model** check box; the **Use Topography** check box is then automatically activated.

2. The mesh can be regridded using user-defined block-width parameters and the water resistivity and default mesh resistivity values set using the **Edit | Reset Marine Mesh** menu command:



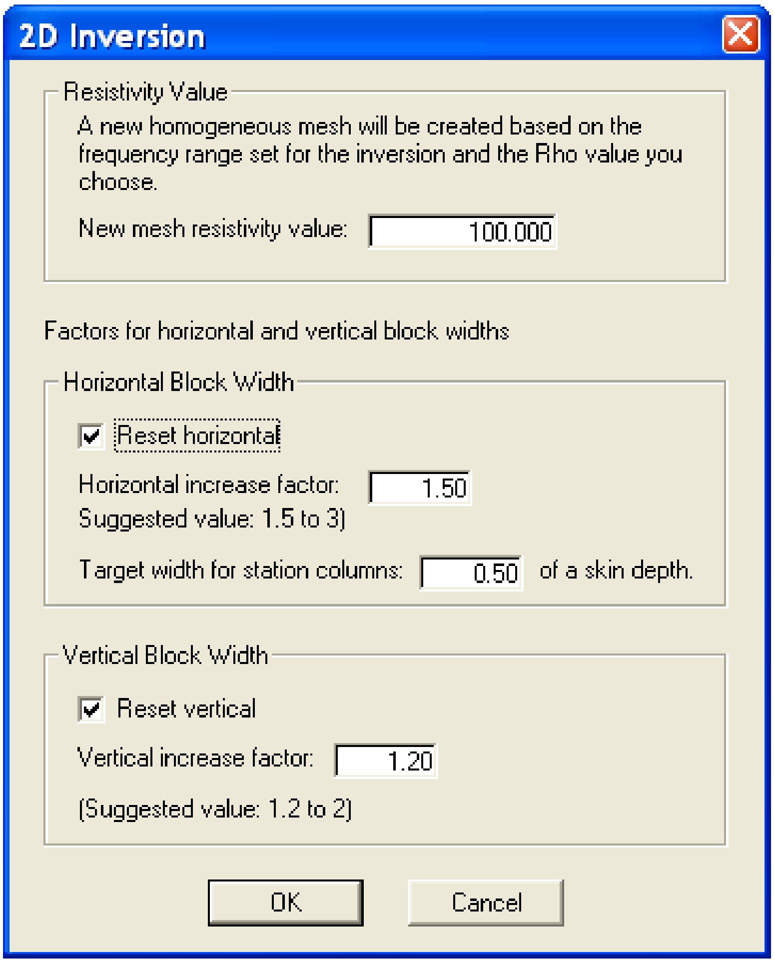
Note: All non-water cells are set to the resistivity value set in the second field; it is not possible to reset only the water resistivity value.

Default Mesh Generation

Whenever a new model is created, a coarse gridding default mesh is generated. To set a new default for mesh generation, select the **Edit | Reset Mesh to Default** menu command, then choose either **Coarse** or **Fine** according to your needs.

Warning: Resetting the mesh to default will reset the current model.

For more detailed control over mesh generation, select the **Edit | Reset Mesh to Default | User Defined…** menu command to open the mesh generation dialog box:

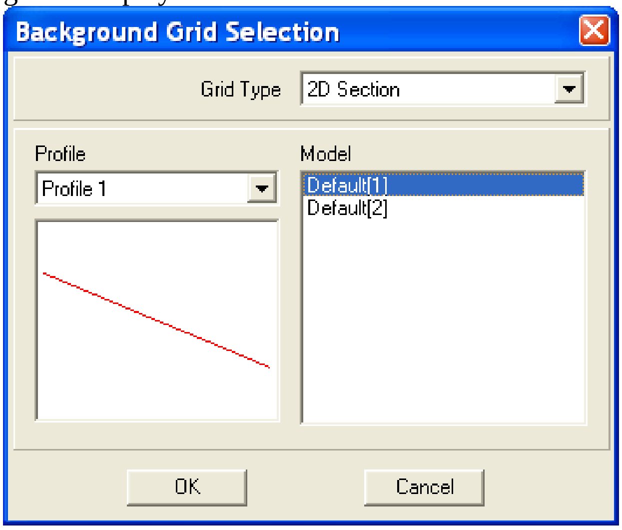


The value set in the **Resistivity Value** field is used as the value throughout the mesh. Existing values are overwritten.

Either or both block widths can be reset. You may optionally set the horizontal and vertical block widths in two separate operations by calling the **User Defined** menu command twice. Note, however, that the mesh resistivity values are reset to the specified value each time the block widths are reset.

Extracting a Mesh from an Existing Resistivity Section

1. If not already in Edit Mesh mode, switch to this mode by clicking the  button.
2. Select the **Edit | Background** menu command and choose the **Set background grid** option**.** Select the background resistivity grid to display:



Note: The selected background grid will NOT be displayed immediately on the screen.

3. Again select the **Edit** | **Background** menu command, and now choose the **Show background grid** command
4. Select the **Edit** | **Background** menu command, and choose the **Extract Mesh from background grid** command

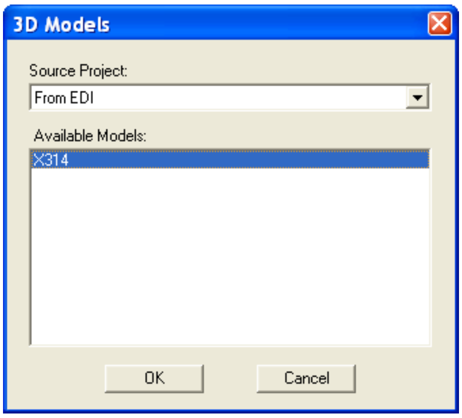
For each mesh cell where a grid value is available, a mesh resistivity value is extracted. Remaining cell values are extrapolated

Filling a Mesh with Values Extracted from a 3D MT Mesh

Meshes can be extracted from any 3D MT mesh in the current database along the active profile and the resistivity values contained therein used as fill values for the current mesh.

To Fill a Mesh with 3D MT Mesh Values:

1. Select the **Edit | Fill Mesh with 3D Mesh Values** menu command. A warning message appears indicating that, depending on 3D model dimensions, the time to extract the mesh may be on the order of minutes. This is particularly true if the original 2D mesh contains topography information.
2. After acknowledging the warning, the 3D Models selection box opens:



Select the project and desired mesh and click the **OK** button to initiate extraction and the subsequent filling of mesh cells.

During this process, 3D mesh cells located along the profile are extracted from the 3D MT mesh and used as fill for the current mesh. Thus, if the mesh is not coincident with any part of the profile, no resistivity values are replaced in the original mesh.

2D meshes generated by the MT 2D Inversion program generally contain mesh columns which extend well beyond the profile as well as below the expected maximum depth of penetration to allow the inversion to incorporate the effects of nearby structures on the predicted impedances. The 2D meshes extracted from the 3D MT meshes, however, are limited to values along the profile. Thus, resistivity values of the original mesh which lie outside of profile are not extracted from the 3D mesh and remain unchanged.

In addition, if the resolution of the 3D mesh is exceedingly coarse relative to the original 2D mesh, it is possible that no values can be extracted from the upper layers of the 3D mesh. In this case, the original mesh values are retained.

Editing a Mesh

The MT 2D program includes a powerful interface for editing meshes. To access this interface, either click the  button in the toolbar or select the **Edit | Goto Edit Mesh** menu command. This places the program in the Mesh Edit mode.

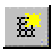
Upon entering this mode, both the toolbar and the Edit menu commands change and now include a number of commands and functions for performing mesh edit operations.

To exit Mesh Edit mode, select the **Edit | Exit Edit Mesh** menu command or reclick the now depressed Edit Mesh button. If any changes were made to the mesh while in Mesh Edit mode, the program prompts you to save the changes before returning to the standard program mode.

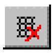
The individual mesh edit commands are described below.

Adding Rows or Columns to a Mesh

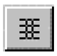
After entering Mesh Edit mode:

1. Click the  button and select **Add Row** or **Add Column**; the pointer changes to a cross hair.
2. In the model window, click the location in the mesh where the upper bound of the new row [left bound of the new column] is to be placed. This function effectively splits a row or column into two; the resistivity values on either side of the new row or column are identical.

Deleting Rows or Columns from a Mesh

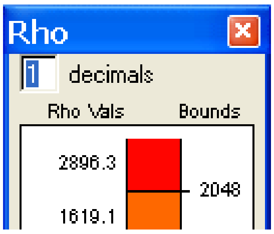
1. Click the  button and select **Delete Row** or **Delete Column**; the pointer changes to a cross hair;
2. In the model window, point in proximity of the upper bound of the row [left bound of the column] to delete and then click the mouse left button. When deleting rows, the current row is deleted and the area previously occupied by the deleted row is assigned the resistivity value of the next lower row. When deleting columns, the current column is deleted and the area previously occupied by the deleted column is assigned the resistivity value of the column to the right.

Sizing Rows or Columns

1. Click the  button and select **Row Height** or **Column Width**; the pointer changes to a cross hair.
2. In the model window, point in proximity of one of the row/column boundaries.
3. Click and hold down the button; drag the boundary to the new position and release the button.

Changing Cell Resistivities

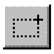
While in Mesh Edit mode, click the box in the Rho window which contains the new resistivity value you wish to assign to one or more mesh cells:



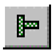
Then click:

 to change the resistivity of one cell at a time.

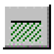
Click each cell to change the resistivity value.

 to change the resistivity of a group of adjacent cells.

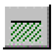
Click the mesh and drag the mouse to include all cells you wish to edit the resistivity of

 to change the resistivity of a row/column (or half row/half column):

1. select Row/Column selection
2. click the mesh, then click the row/column you want to change the resistivity of. To change a half row/column resistivity, click the cell the change should start from in the row or column.

 to change the resistivity of all the cells below a given row:

1. select Bottom Space Selection;
2. click the mesh, then click in proximity of the lower boundary of the row.

 to change the resistivity of all the cells left (or right) of a column:

1. select Left/Right Space Selection;
2. click the mesh, then click in proximity of the left/right boundary of the column.

When setting resistivity values, you are not limited to the values listed in the Rho window. To set a specific resistivity value, enter a value in the field provided at the bottom of the window. The appropriate cell color is selected automatically based on the resistivity values of the next higher and next lower color ranges.

Note: Changing the resistivity value associated with a color does not change the values of any cells in the mesh already assigned that color.

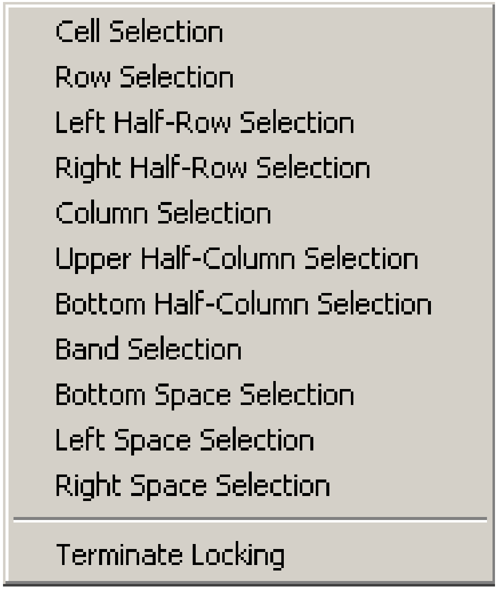
Locking/Unlocking Cell Resistivity

The resistivity of cells can be **locked** so that during the inversion process their resistivities are not changed.

To Lock Cell Resistivities:

1. Click the  button and select **Lock** or select **Edit |Lock Rho Value** to enable locking mode. While in this mode, the lock button remains depressed.

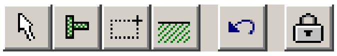
2. To lock cells, **either** select from the options which are listed when the mouse is positioned over the **Edit |Lock Rho Value** command:



then click the mesh with the mouse to lock cell values

or

use the buttons shown below to access the same commands:



From left to right, the buttons are used for cell selection, row/column selection, band selection, bottom/left/right space selection, undo locking operation, and terminate/enable locking.

Note: The first five buttons function as described above only when the locking button is depressed. Their function is otherwise dependent on the active edit mode, e.g. edit resistivities.

Locked items appear with a halftone screening.

To Unlock Cell Resistivities:

1. Click the  button and select **Unlock** or select **Edit | Unlock Rho Value** to enable unlocking mode. While in this mode, the lock button remains depressed.
2. Cells can be locked and unlocked using commands analogous to those used to lock cells. Likewise, the buttons described above can be used to unlock cells.

After either locking or unlocking cells in Edit Mesh mode and returning to inversion mode, the locked cells are displayed in the mesh with halftone screening and the inversion parameters are set to take into account any locked cells.

To Enable/Disable Cell Locking for a Model:

1. Select the **Inversion** | **Settings…** menu command.
2. The Parameter Settings dialog box opens. Select the **Fixed Parms** tab.

3. Select the **Fixed Parameters** check box.

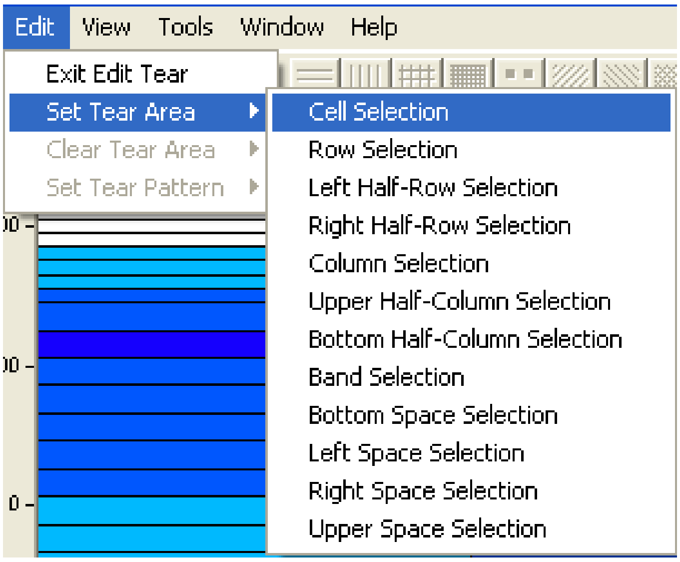
   Note: Halftone screening is used to display locked cells ***only*** when the **Fixed Parameters** check box is selected.

Defining Tear Areas

WinGLink includes functionality for specifying up to eight tear areas within a model. Using these zones, the user can define tears in the smoothing function, i.e. turn off smoothing across a boundary.

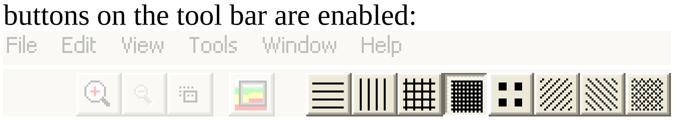
To Create a Tear Area:

1. After creating a model, select the **Edit | Goto Edit Tear** menu command to enter Edit Tear mode or click the  button.
2. Next, choose a selection mode on the **Edit | Set Tear Area** menu:



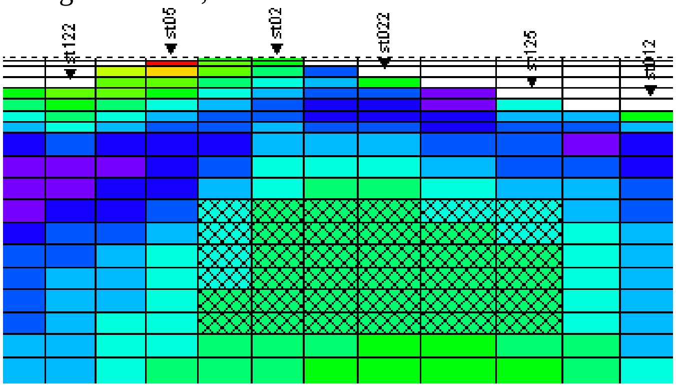
   Selection options are similar to those used elsewhere in the 2DMT module, e.g. for setting resistivities or locking cell values. Alternatively, you may use the selection buttons on the toolbar.

3. After choosing a selection option, the tear area menu options appear (**Edit | Set Tear Pattern…**) and the corresponding buttons on the tool bar are enabled:



   Each of the patterns corresponds to one of the eight different tear areas.

4. Using the mouse, select the cells to be included in the tear area:



   Although the function supports up to 8 different tear areas, a given cell can be associated with only one tear area.

5. After defining tear areas, exit Edit Tear mode with the **Edit | Exit Edit Tear** menu command or by again clicking the Tear button. Display of tear areas can then be toggled on and off with the **View | Tear** menu command.

Tear areas present in a model are included in any smooth inversion performed on the model.

Editing a Sharp Boundary Model

Editing Sharp Boundary Interfaces

Select the **Edit | go to Edit Resistivity Interfaces** menu command or click the  button to enter Edit Resistivity Interfaces mode.

The Edit Interfaces command buttons appear.

Click:

 to add an interface

Going from left to right, click with the mouse left button wherever you want a vertex of the polygonal interface to be added. When finished, right-click the last vertex you wish to add: the program automatically completes the right bound of the interface.

 to delete an interface

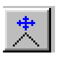
Left-click a segment of the interface to delete.

 to insert a vertex

Left-click the interface at the location where you want to insert a vertex.

 to delete a vertex

Left-click the vertex to delete.

 to move a vertex

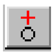
Left-click the vertex to move and drag it to the new location along the interface.

Setting Interface Resistivity Nodes

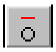
Select the **Edit | go to Edit Resistivity Interfaces** menu command or click the  button to enter Edit Resistivity Interfaces mode.

The Edit Interfaces command buttons appear.

Click:

 to add a resistivity node

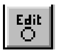
Left-click an interface where you wish to insert a node. A node will be inserted with a value of resistivity equal to the value currently selected in the Rho editor box.

 to delete a resistivity node

Left-click the node to be deleted.

 to move a resistivity node

Left-click on the node to be moved and move it along the interface.

 to edit the value of resistivity for a node

1. Left-click on the Rho editor box and select the value for the new resistivity.
2. Left-click each node to which you wish to assign the new resistivity.

Note: Each interface must contain at least one resistivity node.

Locking Interface Vertices and Resistivity Nodes

The positions of interface vertices and the values of resistivity nodes can be locked by selecting the **Edit | Lock** menu command while in Edit Resistivity Interface mode in the MT 2D program. After selecting the command, click each vertex or node to be fixed. Locked vertices are indicated by a red cross, locked nodes are indicated by a black circle around the node.

Be aware that locked vertices and nodes are not necessarily truly “locked”. Depending on the values of the Tau for Resistivities and Tau for Interfaces parameters, vertex and node values may actually be quite variant. These two parameters, which are set in the Sharp Inv. tab of the Parameters Setting dialog (**Inversion | Settings…**) control the stickiness of each of these values. A high value, e.g. 1E8, ensures relatively little change per iteration, whereas a low value, e.g. 10, restricts only slightly.

Locked vertices and nodes can be unlocked using the command provided under the **Edit | Unlock** menu command.

Inversion Settings

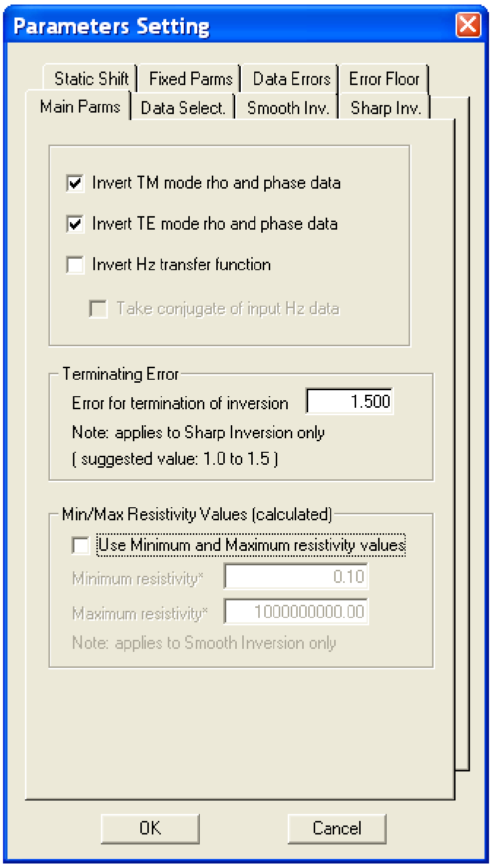
Overview

As described below the MT 2D provides a multi-paneled dialog window which provides extensive options for controlling the inversion process including, among others, the selection of modes and frequency ranges.

In addition to the options provided here, however, note also that the masking functions available in the MT Soundings and MT Pseudo-Section programs can be used to hide individual data points from the inversion routine. This provides an additional level of control over which data points are used in model inversions. Please refer to the respective chapters of manual for additional details.

Main Inversion Parameters

Select the **Inversion** | **Settings...** menu command to open the Parameter Settings dialog box:



Click the **Main Parms** tab**,** and select the desired options. Available for selection are:

_ Invert TM rho and phase data

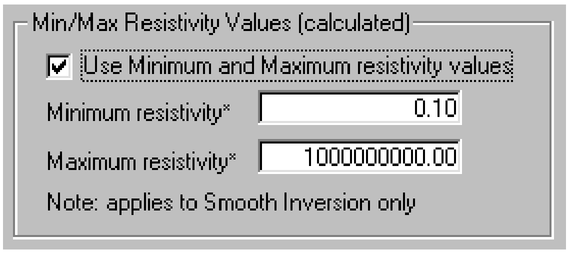
_ Invert TE rho and phase data

_ Invert Hz transfer function (*)

(*) The program is written assuming an e^{-iωt} time dependence. Many processing routines assume e^{+iωt}. If this is the case, then the data will need to be conjugated to be consistent with the program. Check the "Take conjugate of Hz data" box to do so.

Note: While in the Main Parameters dialog box, it is possible to enter an error value that will terminate the inversion. This last option is only available for the Sharp Boundary inversion routine.

In the Min/Max Resistivity Values (calculated) area of the Main Parms window, select the **Use Minimum and Maximum resistivity values** checkbox to force the ***smooth*** inversion code to restrict the resistivity values calculated by the code to values between the specified minimum and maximum values:



Specifying the Data for Which the Inversion is to be Performed

Select the **Data Select.** tab of the Parameter Settings window and make the appropriate settings:

Min Frequency	Specifies the lower bound of the observed curve's frequency range over which fitting is tried. If you want to fit all frequencies of the observed curve, specify a very low value, e.g. 10 E^-5
Decades	Specifies the upper bound of the observed curve's frequency range on which fitting is tried; it is equal to **Min Frequency** * 10 ^(**Decades**). Values can be set independently for each of the modes.
Use Station Data	Will force the program to use the observed curve data samples. This could lead to a long calculation time when the data points of the observed curves are not ordered in the frequency domain. In this case, you may want to select **Use Interpolated Data** to cut the inversion time considerably.
Interpolated Data	Uses a resampling of the observed curves. The **#Freqs in Decade** parameter specifies how many samples per decade must result from the interpolation. This option is useful for shortening the inversion time when the data points of the observed curves are not ordered in the frequency domain and/or when dealing with curves with many data points.
Use Smoothed Curves	By default, the observed curves used for the inversion are the edited curves saved by the Soundings program. Selecting this option will force the program to use the smoothed curves instead of the edited curves.

Smooth Inversion Parameters

The NLCG algorithm attempts to minimize an objective function that is the sum of the normalized data misfits and the smoothness of the model. The tradeoff between data misfits and model smoothness is controlled by the regularization parameter tau. The NLCG algorithm does not try to automatically determine tau to reach a target misfit. Rather, the user must run several inversions using different values of tau, and then run the algorithm until convergence to determine what value of tau will give the smoothest model and the target misfit. It is possible with real field data that one would never be able to reach the desired target misfit. In this case, one would want the tau that gives the smaller RMS error and the smoothest model. The following parameters are available to the user for inversion control:

Select the **Smooth Inv.** tab of the Parameter Settings window and make the appropriate settings:

Smoothest model or variations	The program can solve for the smoothest model or the smoothest variations away from the apriori (starting) model. Choosing smoothest variations is a useful way to do hypothesis testing for different model parameters or features.
Regularization Laplacian	It is possible to specify either a uniform grid Laplacian (which assumes for the purposes of computing the regularization function that the dimensions of the model are all equal) or standard Laplacian on the actual model mesh. The uniform grid Laplacian may produce smoother models, but at the expense of smearing features both vertically and horizontally. The standard Laplacian may produce a rougher-looking model, but the definition of smoothness is consistent with the model dimensions. Unless there are reasons to do otherwise, we recommend using the standard Laplacian.
Regularization Order	Option to minimize the gradients ($|\nabla m|^2$)of the model, or the Laplacian ($(\nabla^2 m)^2$) of the model. Variations of this parameter, equal or unequal grid Laplacian, and the smoothest model or the smoothest variations give the user a great deal of flexibility for generating different types of constraints on the inversion. It is recommended that you use Laplacian smoothing as the default. Smoother models can also be obtained with increasing tau
Tau for Smoothing operator	This is the regularization parameter that controls the tradeoff between fitting the data and adhering to the model constraint. Larger values cause a smoother model at the expense of a worse data fit. The value of tau should optimally be chosen such that the RMS error for the inversion is between 1.0 and 1.5. A few inversion runs may be necessary to determine the best tau for each inverse problem. Additionally, noise in the data and non-2D data

may prohibit the program from reaching the target RMS error values. This is especially true for TE mode data. If you find that the rms does not drop below 1.5 for any value of tau, you should increase tau until the rms is 1.5 (or perhaps 1.2) times the smallest rms you can achieve. Alternatively, you can increase the error floors until you can achieve a minimum rms of 1.0 and then increase tau until the rms is 1.5. Values between 3 and 300 seem to be typical for most MT inversions and are good starting points.

Weighting function

The first option specifies the α factor to multiply horizontal derivatives. A value of 1.0 is recommended unless you want to increase the horizontal smoothness.

The second line specifies β in the weighting function. A value of 3.0 is recommended if minimizing $(\nabla^2 m)^2$ and a value of 1.0 is recommended if minimizing $|\nabla m|^2$. If you set the value to zero, then no weighting is applied to the regularization term.

The third line specifies the minimum block dimensions to be used for computing the weighting function. These are specified in **meters**. These are useful for setting a transition from a depth-independent regularization in the shallow part of the model to a depth-dependent regularization in the deeper part of the model. Values of 500 – 1000 meters are recommended as good starting points. If values of 0.0 are specified, then the model will be overly rough in the shallow part if using a standard grid Laplacian , or overly smooth if using uniform grid Laplacian.

The weighting parameters entered here are sent to the inversion code only if the **Weighting enabled** check box is selected. Otherwise, alpha = 1.0, beta = 0.0 and min block dimensions =0.0 are sent to the inversion program. .

Output sensitivity map

When selected, the sensitivity map is output by the inversion routine at the final iteration, regardless of the preconditioner used. The diagonal part of $A^T R_{dd}^{-1} A$ is output for each model parameter. This information can be used to determine those parts of the model that have the greatest sensitivity to that data and which parts of the model are relatively unimportant. This information is in the same format as the model files and can be viewed as a mesh by selecting the **Display | Sensitivity Map** command. **Note:** Depending on the input parameters, the calculation of the sensitivity map may be time intensive.

Save

When selected, the resistivity values calculated for

intermediate models	each model iteration by the inversion code are stored in the database. Upon completion of the inversion, you can view the meshes for any or all of the iterations by selecting the **Display | Model Development** command. The mesh at a given iteration number can be saved as a model by right-clicking the model and specifying a new model name. The intermediate models remain stored in the database for a given model until a new inversion is performed on the model

Error Floor

Select the **Error Floor** tab of the Parameter Settings window.

This is the error floor for the data. Input errors that are below this value will be reset to this value. Values greater than 1.0 for TM and TE mode data are recommended. (For small values, percents are equivalent to natural logarithm errors). The error floor for the phase should be entered in rho equivalent percent. For example, 1% in rho is equal to 0.29 degrees in phase.

For Hz data, the error floor is an absolute magnitude that should optimally be chosen relative to the quality of the Hz estimates. A reasonable starting point might be 0.01 or greater

Static Shift

Select the **Static Shift** tab of the Parameter Settings window.

Select Stations to Invert for Static Shifts

In this version of the program, the user can specify which data are to be included in the inversion for static shifts. In other words, you can invert for static shifts for just the TE mode data, TM mode data, or any combination for each station. If you want to invert for static shifts for all data, just select the **Invert for Static Shift** check box. However, to specify any combination of data, on the **Inversion** Menu, select the **Edit Static Shift** command. While this command is active, double-click on each station one at a time to select the static shift option for the selected station (TE only, TM only, TE&TM). The name of the station on the topography (normally black color) will change color according to the static shift settings:

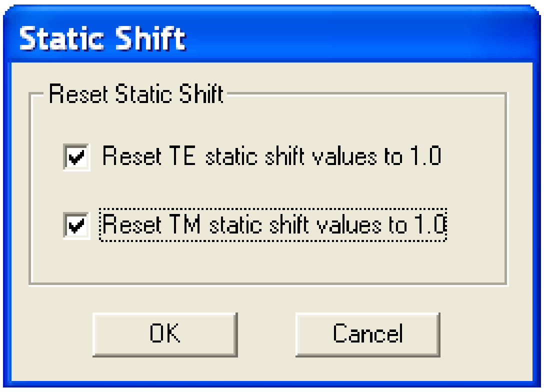
- None	BLACK
- TE only	RED
- TM only	BLUE
- TE & TM	PURPLE

To exit the static-shift edit mode, select the **Inversion Menu** | **Lock Static Shift** menu command**.**

The static shift values returned by the inversion routine are stored for each station in the database. Thus, these static shift values will be used in any subsequent inversions or forward calculations performed on the model, provided the static shifts are enabled for the respective mode.

Resetting Static Shift Values

The static shift values of all station used in a model can be reset to 1 using the **Tools | Reset Static Shift** command to open the Static Shift reset dialog box. The values are otherwise retained, i.e when editing meshes and when restarting inversions:



Static Shift Settings

Invert for static shift

The program can optionally invert for static shifts of the TM and TE mode apparent resistivities. The static shift values are included as parameters in the inversion, and the datum is that the sum of the ln of the static shifts should sum to zero.

Check the **Use screen settings for static shift** box to apply the static shift type for each station. When this box is not checked, static shifts apply to all stations for both TE and TM modes.

Note: After manually setting the static shift of individual stations, both the **Invert for static shift** and **Use screen settings for static shift** check boxes are automatically selected.

Variance for constraint

IF you choose to invert for static shifts, you are also asked to input a variance and a damping value. The variance is the variance in the requirement that the log of the statics sum to zero. Enter values in percent, like 5.0 or 10.0. Larger values mean the zero sum constraint is less important.

Damping for constraint

The other parameter you'll have to enter if inverting for static shifts is the damping term. Basically, this damps the inversion so that large changes are not made to the static shifts on the first few iterations. In other words, you might want to try to fit the data as good as possible before allowing changes to the static shifts. Every three inversion iterations, the program reduces the damping value by a third. Thus, the static shift parameters gain in influence as the inversion progresses. If you want the statics to remain fixed until later iterations, enter a large value here, like 1000 or 10000. If you want the statics to immediately be changeable, enter smaller values. A value of zero for the damping means that the statics are completely free to be changed.

Exporting Static Shift Values

Static shift values can be exported to external files after running an inversion by selecting the **Tools | Export | Import Files Used for Inversion command**. The static shift values are output to the files statics_vals.te and statics_vals.tm for the te and tm modes, respectively.

Data Errors

Select the **Data Errors** tab of the Parameter Settings window.

Use Data Errors if existing	Check this to use the errors of the observed curves.
(TE/TM) Rho, Phase	If the above option is not checked or the observed curves have no errors, these are the default standard deviation errors to use.

Fixed Parameters

If the resistivities of some model cells have been locked, use the options provided on the Fixed Parms tab to specify whether or not cell locking is to be used, and the parameter which specifies the amount which the fixed parameters may change over the course of the inversion.

Fixed Parameters	Certain model parameters can optionally be forced to remain fixed through the inversion. Do not mark this checkbox if you want to override all the locking performed on elements.
Tau for clamping fixed parameters	If the above option is chosen, then a tau value for keeping the parameters fixed must be input (1e5 is recommended - The smaller the value, the less damping).
Use Tears	Select this check box to enable tear functionality.

Note: After either locking or unlocking cells, the **Fixed Parameters** check box is automatically selected and **Tau** is set to a default value of 10.

Sharp Boundary Inversion Parameters

Select the **Sharp Inv. Parms** tab of the Parameter Settings window.

Damping Factor	This controls how damped the locked elements will be. The lower the value, the less the damping.
Tau for Resistivities/ Interfaces	This is the regularization parameter that controls the tradeoff between fitting the data and adhering to the model constraint. Larger tau values will result in smoother variations amongst the parameters, at the expense of a

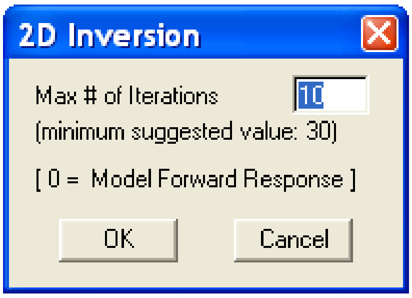
higher data misfits. Higher tau values mean that the interfaces have less variation, and the resistivity of each layer is more smoothly varying.

Running the Inversion

Running a Smooth Inversion

To Run a 2D Smooth Inversion on the Current Model:

1. As described in the previous section, select the **Inversion | Settings** menu command to open the Parameter Settings window and enter the appropriate inversion parameters.
2. Select the **Inversion** | **Run Smooth Inversion** menu command.
3. Enter the Max of iterations.



Enter **Max of iterations = 0** to run a forward computation and see the response of the current model.

Once the inversion is started, the algorithm iterates until the maximum number of iterations specified by the user has been reached or the program can reduce the RMS no further.

During the inversion, the progress can be monitored via the updated RMS error shown for each iteration in the **Running Smooth Inversion** progress window. Also displayed in this window is the expected time remaining for the inversion. In addition, the resistivity values for the mesh cells are updated following each iteration with the new values.

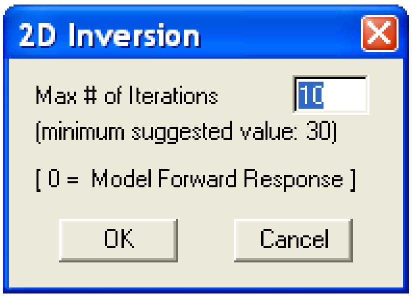
When the inversion is completed, the inverted model is shown as a mesh. Saving this model will overwrite the initial input model. If you wish to save the starting model, you should save it as a separate model before starting the inversion.

Running a Sharp Boundary Inversion

To Run a 2D Sharp Boundary Inversion on the Current Model:

1. As described in the previous section, select the **Inversion | Settings** menu command to open the Parameter Settings window and enter the appropriate inversion parameters, including the target RMS error.

2. On the **Inversion** menu, select **Run Sharp Bounds Inversion**.
3. Enter the Max # of iterations.



To run a forward computation and see the response of the current model, set **Max of iterations = 0**.

Once the inversion is started, the algorithm iterates until the target RMS error is reached or the user-entered maximum number of iterations is reached.

During the inversion, the progress can be monitored via the updated RMS error shown for each iteration.

When the inversion is completed, the inverted model is shown as a mesh. The resulting resistivity interfaces and resistivity nodes are superimposed upon this mesh.

Saving the output model overwrites the initial input model. If you wish to save the starting model, you should save it as a separate model before starting the inversion.

Running an Inversion Using CS/AMT Data

The inversion parameters are, by in large, the same as those used for regular MT data, though the options are somewhat limited. Note that only smooth inversions may be performed for CS/AMT data and that the inversion routine predates that used for regular MT data.

The MT 2D inversion module is plane-wave only, i.e. it is far-field as far as CSAMT is concerned. The bipole separations are, therefore, not used in the 2D calculations. Bipole separations are, however, used in the calculation of 1D models in the Soundings module.

Computing Synthetic Forward Modeling Results

Forward modeling results, corresponding to a given 2D model, can be calculated at any time by running an inversion with zero iterations. When the 2D model is saved, the results are saved as calculated data for the stations added to the model profile.

If no stations with observed data are available for a given profile, synthetic 2D forward modeling results can be calculated by:

1. Using the Maps program to insert new stations along the profile.
2. Editing a 2D model for the profile using the **2D Inversion** program.

3. Running a forward calculation by entering zero iterations for the inversion.

The computed results are saved as calculated 2D data for the stations of the profile. At this stage, the stations will still miss the observed data.

To store the forward response as station data, select the **Tools | Save model responses as station data** menu command in the 2D MT inversion program. .

Note: If using the **Save model responses as station data** command, any existing station data are overwritten. A warning message to this effect is displayed and must be acknowledged before the data are saved as station data.

Presentation of Results

Displaying Models as Sections

The **2D Models** are normally shown as a mesh showing the cell geometries and resistivity values.

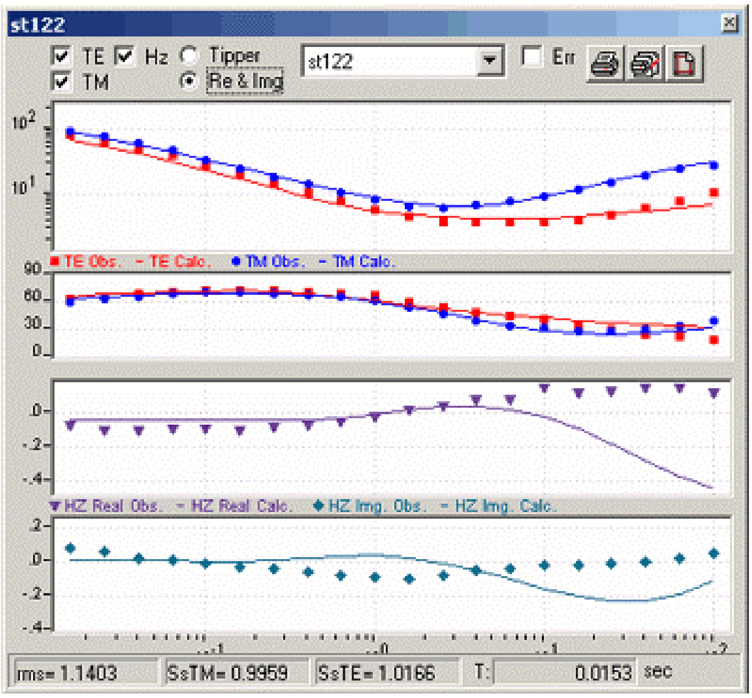
Each mesh can be gridded to obtain a section, which can be plotted or used as a background in other modeling programs.

On the **Display** menu:

- Select **Mesh**, to display the model as mesh (default).
- Select **Section**, to grid the mesh and display the model as a contoured section. Choose **Gridding** to interpolate the resistivity values the first time you select this display mode. For details on setting the gridding options, please refer to the "General Functions" chapter of this manual

Displaying Response Curves

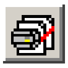
Response curves can be displayed for each station used in a model by double-clicking the column directly under a station name:

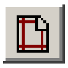


Provided data are available, curves can be displayed for the real and imaginary parts of TE, TM and HZ. For HZ data, the tipper can also be displayed. Error bars can be toggled on and off by selecting the **Err** check box. The drop-down list box can be used to open the response curve for a specific station without closing the window.

The three buttons in the upper right corner of the response curve windows are used to control print functions:

 print the current response curve to the default printer.

 open a selection dialog box from which you may select for printing any or all response curves for stations contained in the current model. Up to six response curves may be printed on a single sheet.

 open the printer settings dialog box.

The status bar in the response curve window contains the RMS and static shift values for the model, provided an inversion has been run. In addition, the position of the mouse along the time axis is also displayed.

Displaying Pseudo-Sections

One way to compare the inversion fitting is by displaying the pseudo-sections of the observed data together with the pseudo-sections of the calculated data. The MT 2D Inversion program assumes that the observed pseudo-sections have already been calculated by gridding the observed data in the **Pseudo-Sections** program (accessible via the WinGLink Shell).

To display the pseudo-sections, select the **Display | Pseudo-Sections** menu command.

The observed section is displayed above the calculated section. The calculated section is gridded using the same parameters as the observed section.

Displaying Multiple Models for Comparison

To Simultaneously Display Multiple Models:

1. Close the active model.
2. Select the S**pecial | Multiple Windows** menu command.
3. Select the models to be displayed for each profile.
4. Use the **Display | Mesh and Section** menu commands to toggle between the two different views.

Displaying and Exporting Sensitivity Maps

Provided the **Output sensitivity map** option was selected in the **Smooth Inv. tab** of the Parameter Settings dialog box prior to inversion, the sensitivity map generated by the inversion code can either be displayed on the screen or output (in the same format as a regular model) for use outside of WinGLink.

To display the sensitivity map, after running an inversion, select the **Display | Sensitivity Map** menu command. To return to the model view, select the **Display | Restore Model** command.

To export the sensitivity map, select the **Tools | Export | Input Files Used for Inversion** menu command. The sensitivity map will be exported to a file named **[Profile Name]_[Model Name].sens**.

Displaying Model Roughness

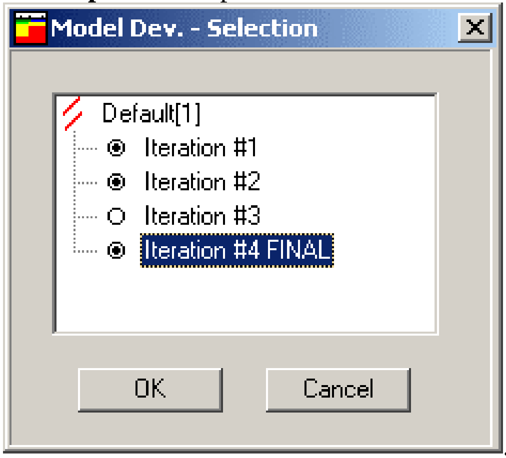
After running an inversion, the model roughness can be displayed by selecting the **Display | Model Roughness** menu command.

Displaying Model Development

The MT 2D Inversion program provides the option of storing and displaying model meshes at each iteration of an inversion. To enable and use this function:

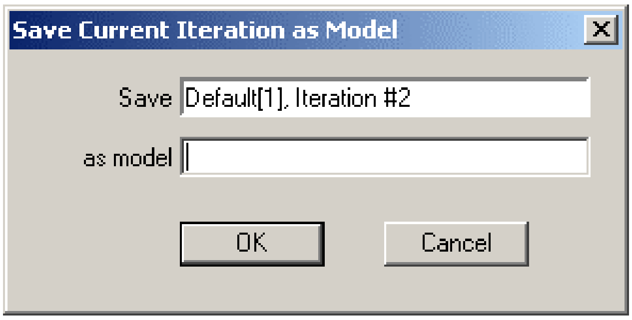
1. Select the **Inversion** | **Settings** menu command to open the Parameter Settings dialog box.
2. Select the **Smooth Inv.** tab. At the bottom of the tab, select the **Save intermediate models** check box.
3. Set the other inversion parameters as you normally would, then perform the smooth inversion.

4. Upon completion of the inversion, select **Display | Model Development** to open the selection window:



.

5. A given iteration can be saved as a model by right-clicking the model window:



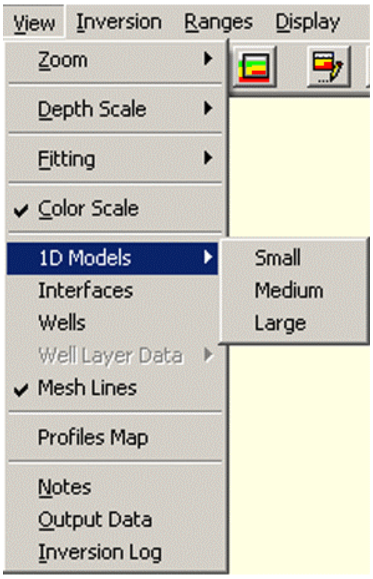
Models saved in this way are assigned the same inversion parameters as the original model

6. To return to the model, select **Display | Restore Model**.

Note: The model development function is available only for smooth models.

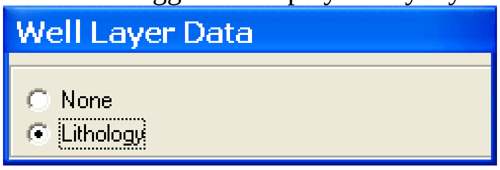
Displaying 1D Models for Stations

On the **View** menu, position the mouse over the **1D Models** command, then choose the most appropriate size for the model: small, medium, large.



Displaying Well Courses

The traces of any wells associated with the current profile can be superimposed on the current mesh by selecting the **View | Wells** menu command. When enabled, the Well Layer Data window appears. This can be used to toggle the display of any layer data associated with the wells:



Displaying Inversion Reports

In addition to the graphic display of inversion results, the parameters used in the inversion, inversion errors and calculated TE, TM and HZ values for each station and period can be viewed and stored as text files. The inversion reports for a given model remain available until another inversion is performed on the same model.

To view the parameters used in the inversion and the inversion errors, select the **View | Inversion Log** menu command.

To view the observed and calculated TE, TM and HZ values for all stations used in the model, select the **View | Output Data** menu command.

Note: For users of the stand-alone version of the inversion code, the data presented in the Output Data window are essentially the same as those output to **output_*.dat** files.

Taking Notes

Each model has associated with it a note pad, the contents of which are stored in the database together with the model. To open, select the **View | Notes** menu command. Notes stored here may optionally be saved to an external file by using the **File | Save (As)** command in the Notes window.

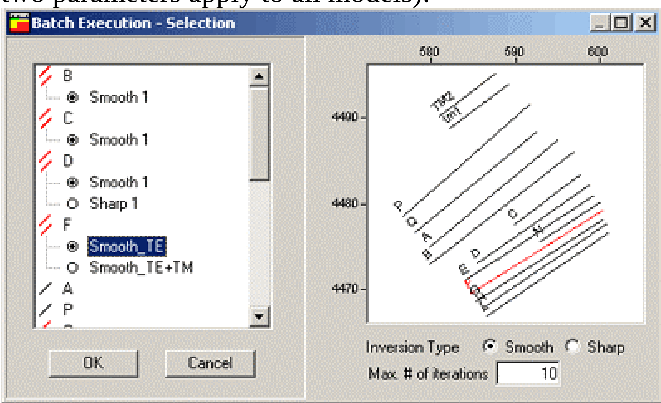
Batch inversion

Executing Batch Inversions

The MT 2D Inversion module can be used to successively execute inversions on a set of models. Inversions executed in this way make use of the inversion parameters set for each model in the Parameter Settings dialog box. All of the inversions to be performed must, however, be of the same type, i.e. either smooth or sharp, and they must have the same maximum number of iterations.

To Perform a Batch Execution:

1. Close any open models.
2. Select the **Special | Batch Execution** menu command
3. In the Batch Execution Selection window which opens, select the models to be inverted and specify the type of inversion and the maximum number of iterations to be performed (the last two parameters apply to all models):



   Select the models to be displayed for each profile and click OK to begin the inversions.

4. Unlike single inversions, the resistivity meshes are neither updated nor displayed during the inversion process. The current iteration numbers and RMS values are however, displayed on the screen. After all inversions have been completed, a summary window appears, listing the RMS values computed for each model and any problems encountered.
5. To view the inverted models, use the **File | Open Model** menu command as you would to open any model.

Exporting and Importing Models and Interfaces

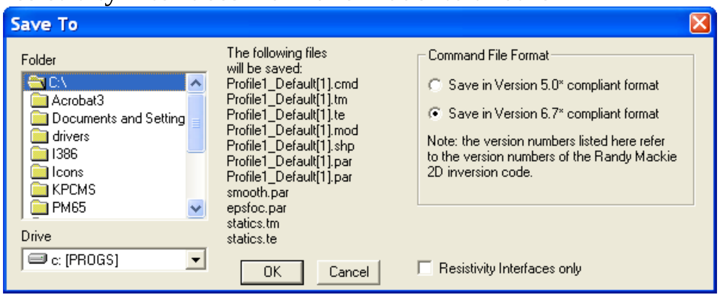
Exporting Models and Interfaces

Models created and used within WinGLink can also be exported to external files. In addition to the models themselves, all files necessary for running an inversion using the stand-alone version of the inversion code, including the command file for either version 5.0 or 6.7+ of the stand-alone code, are output. Models can optionally be output in GEMCOM format.

To Export a Model:

1. Select the **Tools** | **Export** menu command. Position the mouse over the menu command and select either **Input Files used for Inversion** or **Model in GEMCOM Format…**
2. When saving the files used for inversion, specify the directory to which the files should be output as well as the version of the inversion code with which the files are to be compliant.

   Instead of outputting all files used for the inversion, you may, for the case of sharp boundary models, instead output only resistivity interfaces by selecting the **Resistivity Interface only** check box. This function is useful for transferring complex resistivity interfaces from one model to another



When exporting models in the 6.7+ compliant format, the static shift values are exported to the files statics_vals.te and statics_vals.tm for the te and tm modes, respectively.

Importing Models and Interfaces

Models may be created or modified outside of WinGLink and imported into the MT 2D Inversion module. In order to import a model, both a command file (*.CMD) and a like-named model file (*.MOD) are required. The remaining parameter files, which are used by the stand-alone version of the inversion code and control parameters such as fixed cells, static shifts and weighting are optional. Command files must be compatible with either version 5.0 or newer of Randy Mackie's 2D inversion code. Either smooth or sharp models may be imported.

To Import a Model:

1. Select the **Tools | Inversion Model from External Files**… menu command.
2. The Import Model window opens. Browse the directories and select the desired .MOD file; click the **OK** button.

To Import an Interface:

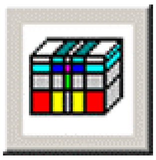
When working with sharp boundary models, it is possible to export and import resistivity interfaces – a convenient function when working with various models along the same profile (or profiles with similar structure).

1. Select the **Tools** | **Resistivity Interfaces…** menu command.

2. Before the import commences, the MT 2D Inversion module alerts you that any existing interfaces will be lost.
3. If you acknowledged the warning mentioned in step 2 with Yes, continue by selecting the interface file (*.PAR).

19: MT 3D Modeling

Overview



The 3D Modeling program is used to:

- Import, create, edit, print and export 3D meshes
- Create views, or slices, of 3D meshes
- Calculate the forward response of 3D meshes
- Export EDI files calculated for stations positioned on the mesh using field information returned following the forward calculation.

Used together with the Maps program for positioning stations and defining profiles and the X-Sections program, the 3D Modeling program is the cornerstone of WinGLink's 3D modeling functionality.

3D meshes are used in both the MT 3D modeling program. Though the functionality is similar in both, there are enough differences that a separate chapter is dedicated to each. This chapter is the MT 3D Modeling chapter.

Note: The 3D Modeling program is available with the MT_4 license option. For details on license options, please refer to the Modules page of the Geosystem website: http://www.geosystem.net. Prices available upon request.

How WinGLink Handles 3D Models

3D models are intended to add another dimension, so to speak, to WinGLink's palette of MT interpretation tools. The suite of 3D-specific functions included in several of the WinGLink modules provides the tools necessary for 3D MT modeling including mesh generation, 3D forward calculation and EDI export.

The 3D forward modeling code developed by Randy Mackie, MTD3FWD, is implemented in the 3D Modeling program. This forward modeling code computes magnetic and electric fields at the surface of a 3D electrical resistivity model illuminated by electromagnetic plane waves. Used as input are a 3D mesh in the format specified at the end of this chapter and several model parameters, which are also described in this chapter.

The fields computed by the forward modeling code are automatically converted to magnetotelluric impedances and tipper upon conclusion of the forward calculation. These can in turn be used to generate EDI files for stations contained in the mesh area.

Field values are generated for each column of data. Thus, EDI files can be generated for any cell in the mesh provided a station, synthetic or real, is positioned on the cell. The MT impedance and tipper values can be stored in the active database as well as stored in an external file and reused to generate EDI files should stations be added to the project at a later time.

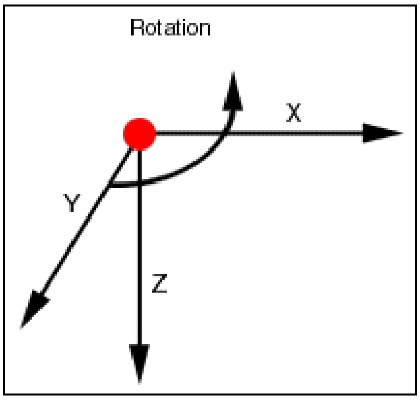
These EDI files, once imported into a WinGLink database, can be assigned to map profiles and used elsewhere in WinGLink as station data, i.e. in the MT 2D module.

About 3D Meshes

The 3D Mesh Definition

In order to be conformant with the forward modeling code, the 3D MT meshes must be in the slightly modified Randy Mackie format as defined at the end of this chapter. The aforementioned modification refers to the addition of georeferencing information, which is used to correctly position the mesh in the survey area.

The coordinate system of 3D MT meshes within WinGLink is illustrated below:



Note that the origin, or pivot point, of 3D MT meshes in WinGLink is the center of the top, left, rear mesh cell. Positive mesh rotation is performed counterclockwise about this point. For details on mesh rotation, refer to the Mesh Properties section of this chapter.

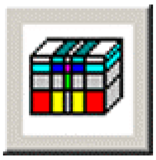
3D Meshes in WinGLink

3D meshes are specific to MT projects in WinGLink and may thus only be imported into projects of type MT. Once imported into a database, a 3D mesh remains associated with its project. The 3D Mesh Importer, which can be accessed both in the main WinGLink shell as well as in the MT 3D Modeling program, can be used to import 3D meshes from external files, the active database and other WinGLink databases.

In addition to its forward calculation feature, the MT 3D Modeling program also provides functionality for mesh creation, editing, importing, exporting and visualizing.

Once in a project, 3D meshes can be displayed in the Maps program. Here, stations can be positioned and profiles defined (refer to Chapter 5, Maps, for details). Map profiles defined on meshes can be used to extract vertical 2D sections in the X-Section program (refer to Chapter 7, Sections, for details). In the Maps program, it is also possible to extract from a 3D mesh maps at constant depth or elevation for resistivity/conductance. Meshes can be extracted along a profile in the MT 2D program and used as starting models for 2D inversions (refer to Chapter 8 for details).

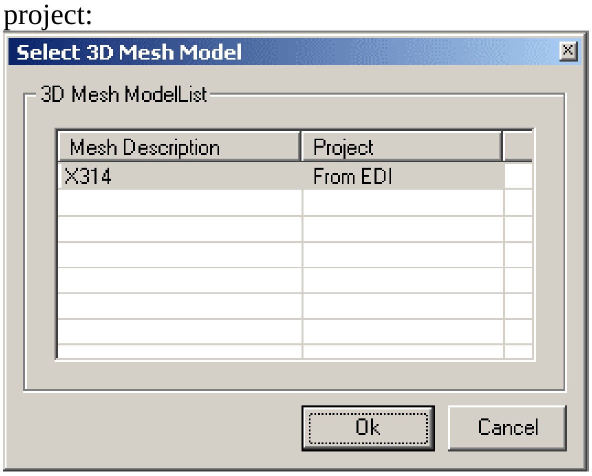
Using the MT 3D Modeling Program



The **3D Modeling** icon, shown to the right, is displayed in the WinGLink database shell whenever the project selected in the Project panel of the database shell is of type MT.

Selecting a Mesh

1. After clicking the **3D Modeling** icon, the following dialog window opens, listing all 3D meshes available for the current project:



   If no 3D meshes have yet been imported using either the WinGLink database shell or the MT 3D Modeling program or created within the MT 3D Modeling program, the table is empty. For details on importing 3D meshes into WinGLink, please refer to the next section or section “Importing 3D MT meshes” in Chapter 3 of the WinGLink manual. Instructions on how to create a mesh in the MT 3D Modeling program are provided later in this chapter.

2. Select the desired mesh with the mouse, then click the **OK** button to continue.

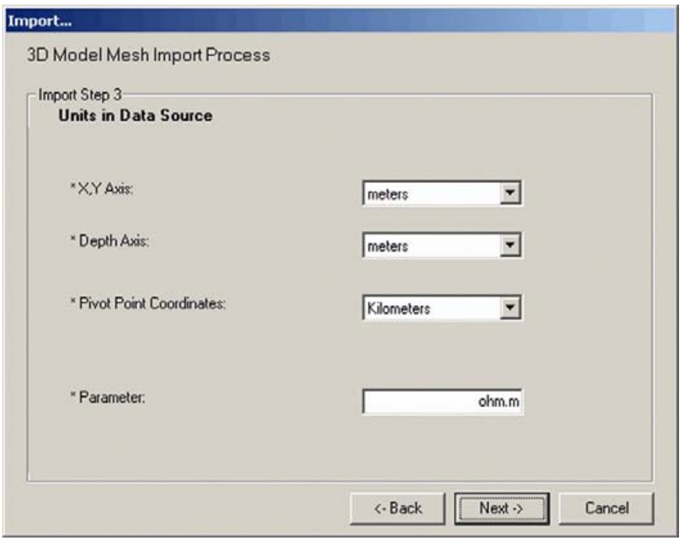
Importing Meshes

In addition to the mesh import function available in the WinGLink database shell, meshes can also be imported directly into the MT 3D Modeling program.

To import a mesh into an existing MT project from within the 3D Modeling program:

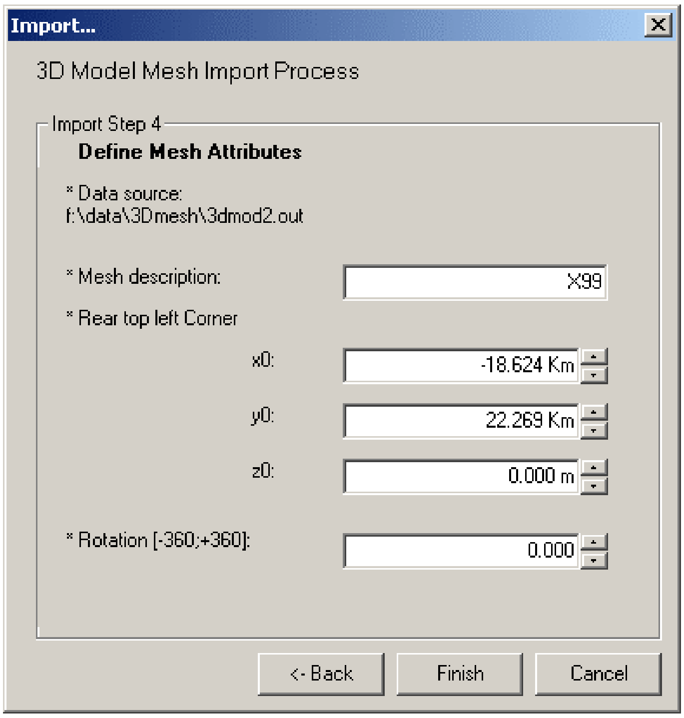
1. On the **Tools** menu, select the **Import 3D Mesh Model …** command.
2. The first of the 3D MT Mesh Import Wizard windows opens. Specify the data source: **from file**, **the current database**, or **an external database**.

   If you choose to import from either the current database or a different WinGLink database, additional dialog windows open prompting you to select the database name (if you selected external database) and to select the mesh to be imported.
3. In Import Step 2, select the source file. The file must be in the Randy Mackie format specified at the end of this chapter.
4. In step three of the import process, specify the units of the data source:



5. In Import Step 4, specify the mesh attributes, including mesh description – which also functions as the mesh name, location of the rear top left corner as well as the angle to which the mesh

is to be rotated:



6. After specifying the mesh properties, a window opens with a text entry box into which you can enter any notes you would like make regarding the mesh. These notes can be viewed and modified later in the mesh properties window (**File | 3D Model Mesh Properties**).

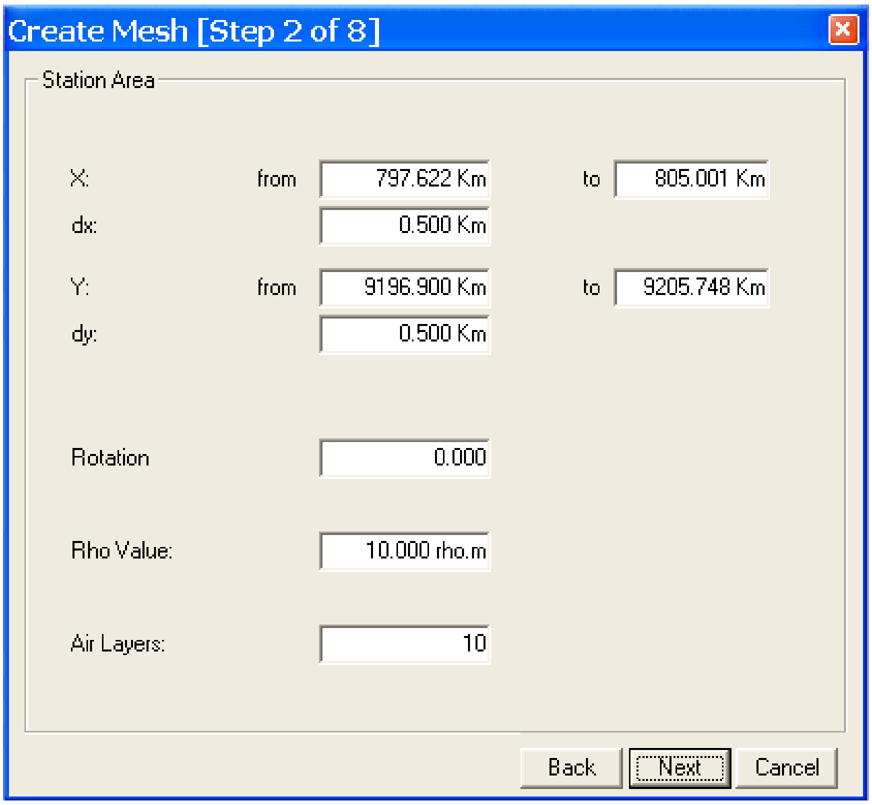
7. In the final step, WinGLink performs the import operation. Import progress and any encountered errors are displayed on screen.

Creating Meshes

The MT 3D Modeling program also provides functionality for creating meshes within WinGLink. The mesh generation wizard can be started with the **File | New** menu command or by actuating the **New Mesh** button, located at the far left end of the toolbar. As described below, this wizard presents several dialog windows for entering the parameters necessary for mesh creation.

In the first of the mesh creation dialogs, you are prompted to enter a mesh description, which is also used as the mesh name throughout the database. A notes field is provided in which any information relevant to the mesh can be entered.

Click the **Next** button to open the second of the dialog windows. Here, location-specific parameters and values for basic mesh properties can be entered:



X,Y,Z: mesh coordinates on the project datum

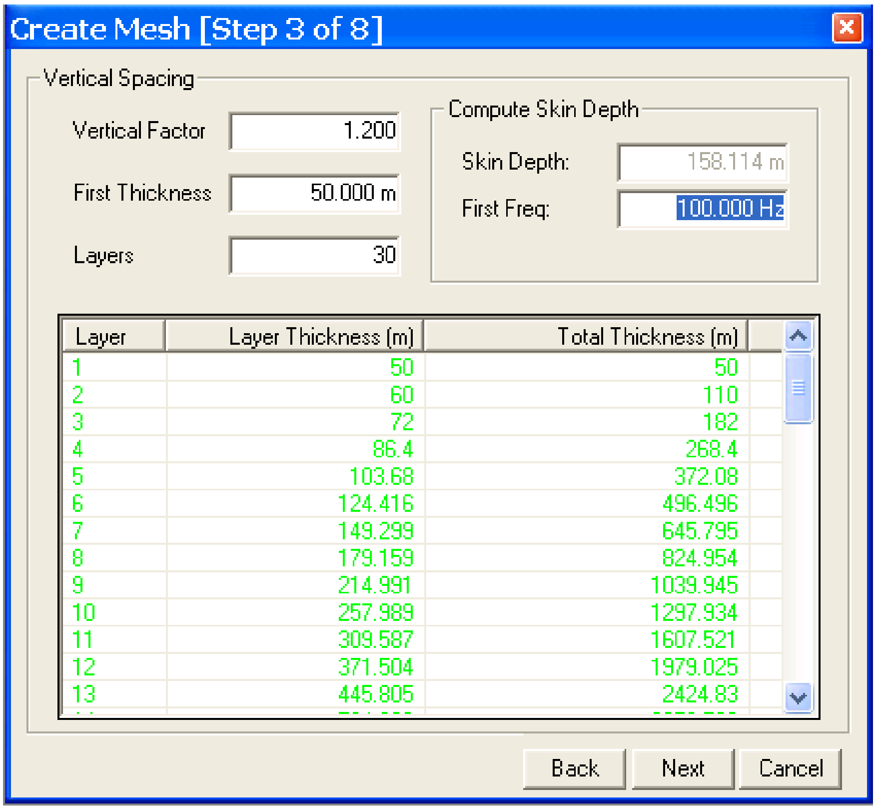
dx, dy: initial block dimensions in the x and y directions, respectively. The actual block dimensions will be set on the fourth and fifth wizard dialog screens using a padding factor.

Rotation: positive degrees of counterclockwise rotation about the top left, rear column

Rho Value: starting resistivity of the model. We suggest looking at the high-frequency apparent resistivities from several soundings in the project and using a representative value.

Air layers: number of air layers on top of the mesh (minimum: 7, ideal: 10)

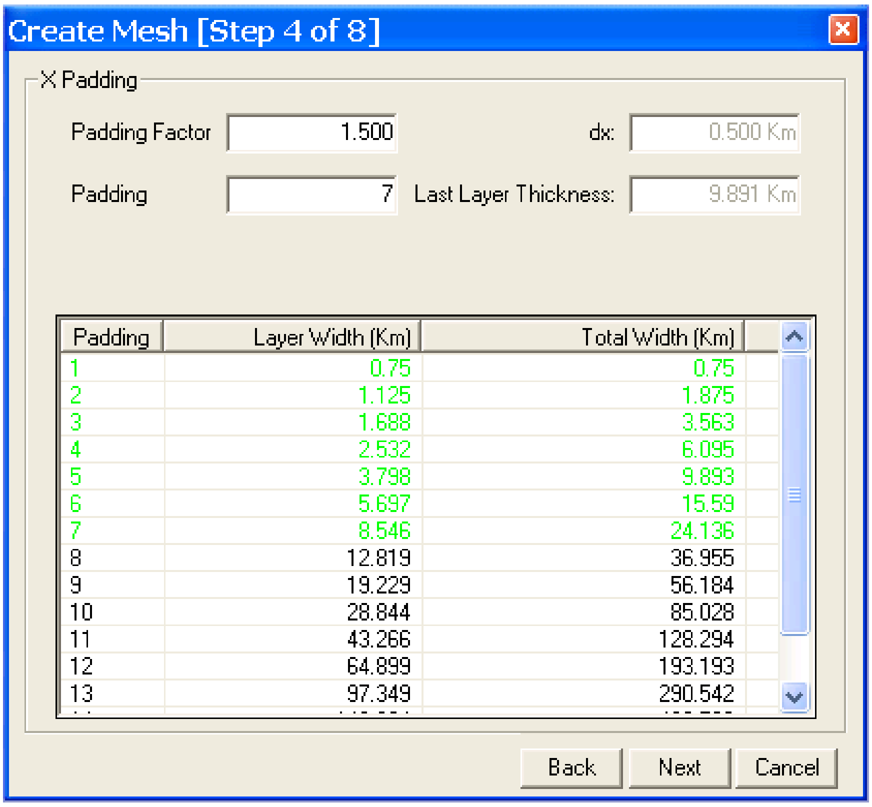
Click the **Next** button after entering the parameter values to open the third wizard dialog window.



First Thickness: Note that this is actually be the first layer under the topography, which is added later. First, compute the skin depth for the model at the highest frequency. This can be computed by entering the value of the highest frequency in the **First Freq.** field Recall that skin depth = 500*SQRT(rho/freq). If we take 100Hz to be the highest frequency, and rho to be 10 ohm-m, then the skin depth is 158 m. With a flat earth, we recommend a value of 1/10 the skin depth for the thickness of the first layer (which would be 15 m in this case). When you add topography, you will be adding several layers of the same thickness on top, and, depending on the amount of topography, you will need to use a larger value in order to keep the model size at a reasonable level. A value as large as 1/5 of skin depth (maybe 1/3 if there is a lot of topography) may be best. In the example shown above, the thickness of the first layer is set to 50 m, which is 1/3 of skin depth. If the number of layers of topography is large, you may need to compromise to reduce the number of layers to reduce processing time.

Vertical Factor and **Layers**: You can try various vertical factors to get a reasonable number of layers and a reasonable depth range. We recommend starting with a vertical factor of 1.2 and using 25-30 layers, as additional topography layers will be added later. We recommend that the thickness of the last layer to be approximately the same as the width of the first and last horizontal blocks. You can determine those widths by advancing to the next wizard dialog screen and entering the horizontal block size as the starting thickness and a padding factor of 1.5. After determining the width of these blocks, use the **Back** button to return to this wizard dialog screen. The number of layers to be contained in the mesh is set in the Layers field. Each of these layers is displayed with the corresponding layer thickness and the total thickness at each layer in green in the lower part of the window.

After entering the parameter values, click the **Next** button to open the fourth wizard dialog window, which is used to enter the parameters for determining the horizontal (x) block size:



dx: Initial block dimension along the x axis. This value was entered in the first of the wizard dialogs and cannot be edited here.

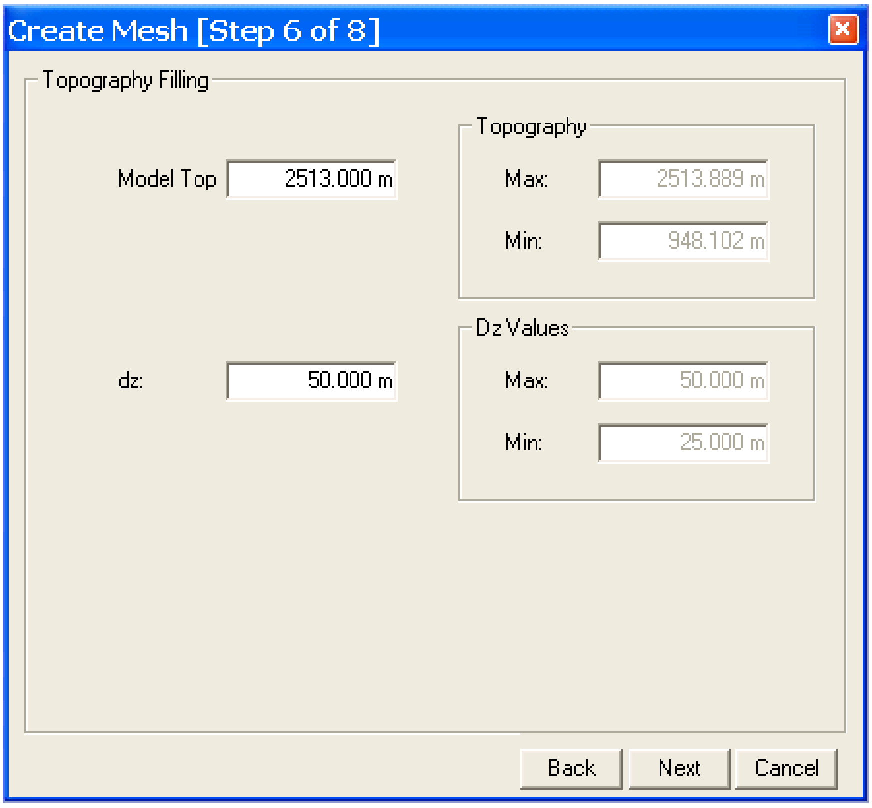
Last Layer Thickness: Thickness of the last layer used in the model. This value corresponds to the thickness of the layer specified in the **Layers** field on the previous wizard dialog and cannot be edited here.

Padding & Padding Factor: The number specified in the Padding field determines the number of padding mesh cells on either side of the first and last stations, respectively, along the x axis. The padding factor specifies the factor by which the width of the padding cells is increased as you move away from the mesh center.

You need to look at the minimum and maximum cell dimensions, and determine what block dimensions will allow you to have a block for each site (or most of the sites), and still give a reasonable sized model. In some cases, the block size will be such that two or more sites will fall on the same block. In that case, you'll have to eliminate all but one site. We recommend making the block size the same in both the x and y directions if possible.

After entering the parameter values for the mesh size in the x direction, click **Next** to open the fifth wizard dialog window. Here, the parameters for the mesh size in the y direction are entered. As mentioned above, we recommend making the block size the same in both the x and y directions.

Click **Next** to advance to the sixth wizard dialog. Here, the parameters related to topography are entered:

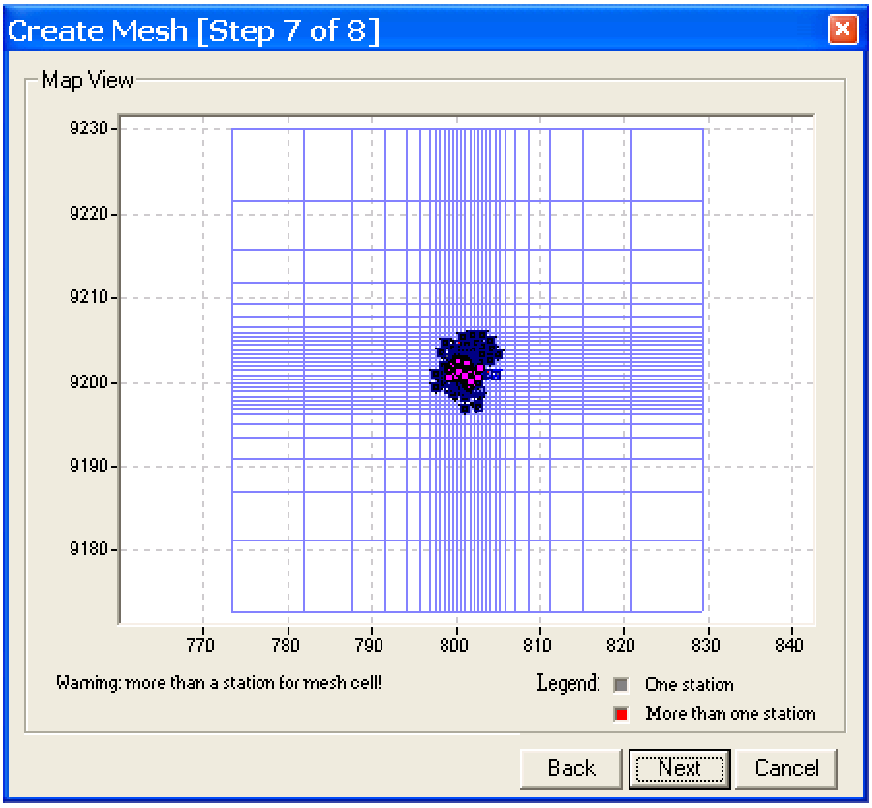


The **Topography Max.** and **Min.** values correspond to the maximum and minimum station elevations. The **Dz Max.** and **Min.** values are the minimum and maximum possible values which can be entered in the dz field. All four of these values are fixed and cannot be edited.

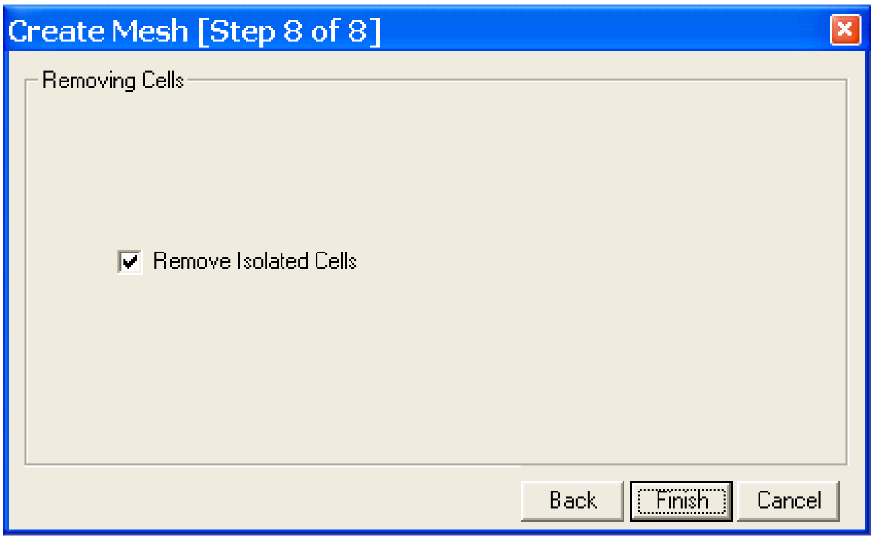
Model Top: Sets the elevation of the upper-most block. If the value is set equal to the **Max.** elevation specified in the **Topography** frame, the topography will include the actual topography of all stations. If the value entered for model top is less than the max. topography value, the mesh will extend upward only to this value, effectively slicing off the top of the mesh.

dz: Thickness of each of the topography layers. The min. and max. values are the values listed in the **Dz Values** frame of this dialog. Note that a small value will result in more layers and, therefore, additional processing time.

After entering these values, click **Next** to display the seventh screen of the dialog wizard. Here, a display of the mesh is presented from above:



Click **Next** to proceed to the final wizard screen. Here, you are given the option of removing any isolated cells:



An isolated cell is any top-level cell which is surrounded on all sides by air. This cells must be removed in order for the forward calculation to be completed properly.

Click **Finish** to complete the mesh creation wizard.

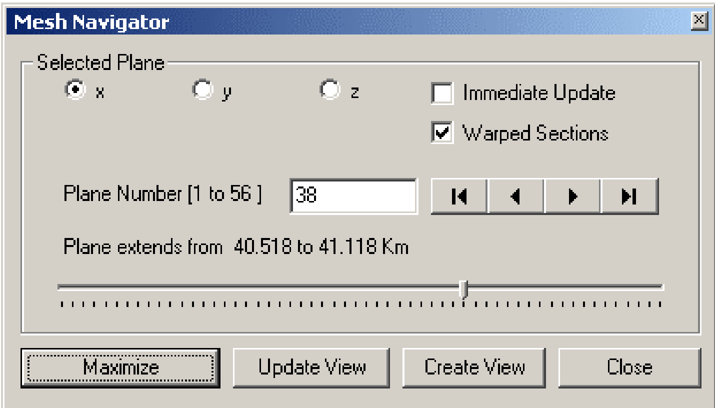
Exporting Meshes

Meshes can be exported from the MT 3D Modeling program to files in the modified Randy Mackie format. Files exported in this way can be reimported into WinGLink. In addition to the 3D MT mesh file, two additional files are also exported. A *.LCK file contains information about cell locking. This file is used by the stand-alone version of the forward calculation program. A file prefixed with SITES_ and post fixed with .DAT contains the mesh coordinates of each station, i.e. cell number and row number. This last file is currently used only for test purposes.

To export a 3D MT mesh, select the **Tools | Export 3D Mesh Model** command to open the 3D Model Mesh Export window. Specify a file name and click the **Save** button.

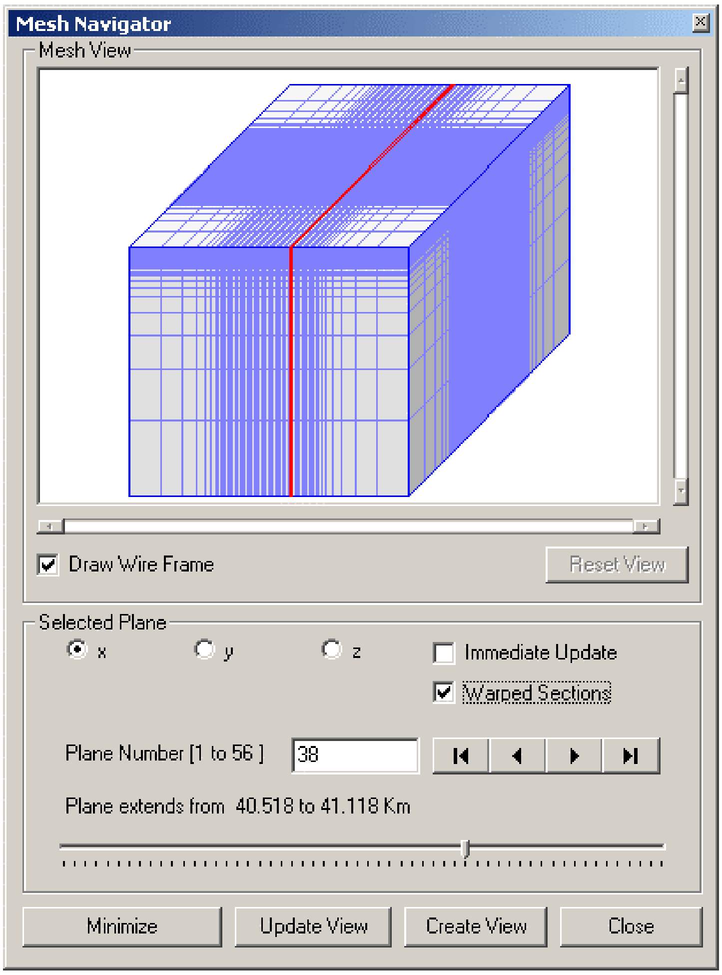
The Mesh Navigator

After selecting a 3D mesh, the Mesh Navigator opens in its minimized state, as shown below. The Mesh Navigator is used to extract and display planes from the mesh along any of the principle axes. These planes are referred to as ***Views*** within WinGLink.

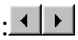


Once one or more views have been created, the MT 3D Modeling program’s suite of mesh editing functions are enabled. For details on mesh editing, please refer to the “Editing meshes” section of this chapter.

As mentioned above, the Mesh Navigator is initially displayed in its minimized state. Click the **Maximize** button to include a 3D display of the mesh in the Mesh Navigator:



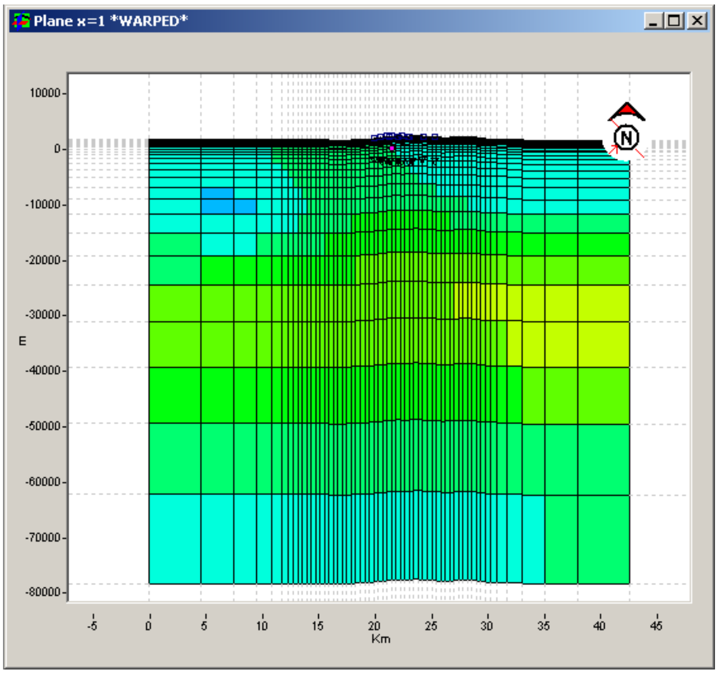
The Mesh Navigator serves to extract sectional views of meshes. The red slice displayed in the mesh corresponds to the currently selected plane. A specific plane can be selected for a given axial view by entering the appropriate value in the **Plane Number** field. Alternatively, the arrow buttons can be used to cycle through the planes along a given axis, whereby the double arrows (<< >>) position the selected plane at the respective outer mesh edge. The slider bar located at the base of the Selected Plane section of the window, which includes the width/height of the current plane, functions in a manner similar to the arrow bars with the added feature that the arrow buttons on the keyboard can be used to cycle through the layers. If the **Immediate Update** checkbox is selected, the view is updated immediately following a plane change. If this checkbox is not selected, the **Update View** button must be clicked to initiate the update.

As an alternative to the arrow buttons on the Mesh navigator, “Next Plane” and “Previous Plane” buttons located on the toolbar can be used to display each of the planes along a given axis: 

Mesh views may also include topography. To include the elevation data, select the **Warped Sections** checkbox. The section views created when this checkbox is selected are warped to fit the mesh to the topography.

To extract a planar view from a mesh:

1. Planar views can be extracted for the selected plane by clicking the **Create View** button:



2. Multiple views can be opened along any axis by pressing the **Create View** button multiple times. You may also change the mesh displayed in a given view window by first selecting the window to be updated, selecting a new plane in the Mesh Navigator, then clicking the **Update View** button.

It is possible to zoom in on a particular area of the mesh displayed in the Mesh Navigator by repeatedly clicking the area of interest on the mesh. Click the **Reset View** button in the Mesh View area of the Mesh Navigator to restore the mesh to its original size.

The compass displayed in the upper right corner of the view windows serves to provide a frame of reference for better user orientation. In the view window shown above, the mesh has been rotated +40 degrees as is indicated by the fine, red arrow. The compass display can be toggled on and off with the compass button on the toolbar:

Once closed, the Mesh Navigator can be reopened by selecting the **Mesh Navigator** button from the 3D Modeling toolbar:

About Views

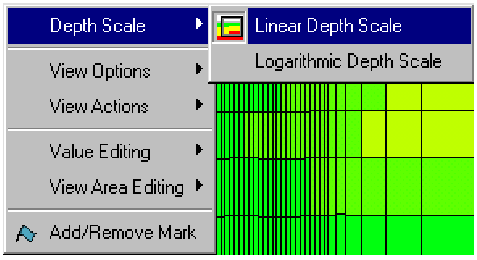
As mentioned above, views are planes extracted along one of the three principle axes. As is described in the Editing Meshes section of this chapter, editing operations on the mesh can be performed in these views, i.e. rows and columns can be added and deleted and resistivity values modified.

The MT 3D Modeling program provides several display options for view windows. These options, which are accessible via the toolbar and/or the menu bar or shortcut menu, are described below.

A miniature version of the 3D mesh, called the Navigator Preview icon, is displayed in the toolbar and helps in mentally visualizing the relative location of the active view in the mesh: 

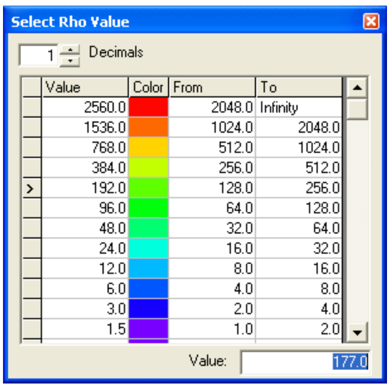
Button operations only apply to the currently active view window. A given button operation remains associated with a view window, even when simultaneously working with multiple windows, i.e. selecting the **Split Row** button in one view window does not enable the **Split Row** button for all other open view windows.

The vertical axis may be displayed with either a linear or logarithmic depth scale. To switch between the two scales for either x or y plane view windows, right click on the view window to open the shortcut menu and select the desired depth scale:



Show Rho Values

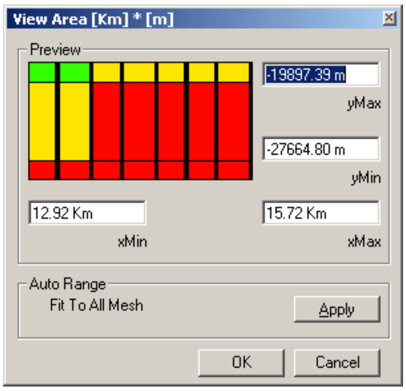
Use this button to open and close the Select Rho Value window, which is similar to the range window found in the other WinGLink modules. This window is used when editing to select the active resistivity value. The active resistivity range is indicated in the Select Rho Value window by an arrow in the left-hand column. A range may be selected by either clicking a line in the Select Rho Value window or by entering a value in the **Value** field. Note that when selecting a value with the mouse, the lower range value is used as the active resistivity value when editing. The number of decimals shown in the display can be set in the **Decimals** field at the top of the window:



The Rho Values window can also be toggled on and off using the **View | Show Rho Values** menu command.

Select View

Use the **Select View** button to open the View Area window, which is used to define the bounding rectangle of the mesh displayed in the view:

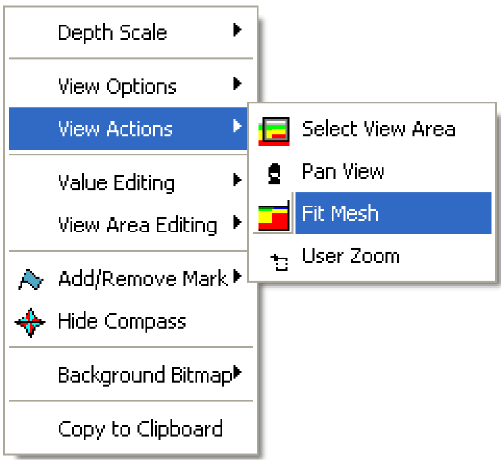


Changes made to the bounding rectangle in this window are reflected immediately in the Preview area. To fit the entire mesh within the window, click the **Apply** button next to the **Fit To All Mesh** label in the AutoRange area of the window .

Fit Mesh

The **Fit Mesh** button serves the same function as the **Apply** button in the View Area window: Click the button to fit the entire mesh to the active window.

The fit mesh function, as well as many of the functions described in this section, can also be accessed via the shortcut menu, which can be opened by right-clicking a View Area window:



Hide Mesh Lines

Hide Fill Colors

Zoom In/Out

When the **Magnifying Glass** button is selected, the degree of zooming can be controlled by clicking the left and right mouse buttons: left click to zoom in, right click to zoom out.

User Zoom

When the **Crosshair** button is selected, use the mouse to select an area of the view window. This selected area is then magnified to fill the entire window.

Numerical Zoom

To exit the User Zoom mode, click the **Magnifying Glass** button. Click this button again to exit Zoom In/Out mode. You can now select predefined numerical zoom values from the dropdown list. Selecting the User Zoom option disables the numerical zoom function and automatically selects the **Crosshair** button.

All Zoom functions can also be selected using the **View | Zoom View** menu command.

Pan View

Use the **Pan View** button to move the area of the mesh which is displayed in the active View window. While in this mode, the cursor is displayed as a hand. The cursor returns to an arrow after the mouse has been used to shift the mesh position once, another toolbar button has been clicked, or another shortcut menu command selected.

Editing Meshes

Mesh editing can be performed in the MT 3D Modeling program provided one or more views are open. For details on creating views, please refer to the Mesh Navigator section of this chapter.

Certain edit operations affect the entire mesh. For example, if a row is added in the X plane, the Y plane, if displayed, is automatically updated to reflect the addition. Other operations, such as edits to the resistivity values, affect only the current view.

In MT 3D Modeling, edit operations are initiated by clicking the edit buttons and shortcut menu commands described below. Note that most edit functions do not have a corresponding command in the main menu.

Undo/Redo

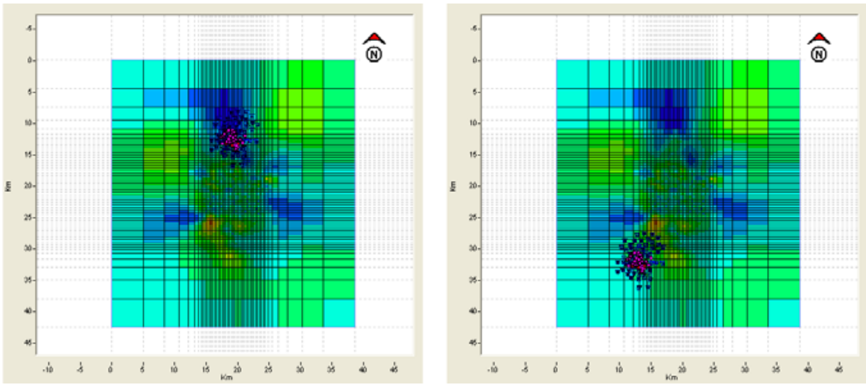
The **Undo** button, the arrow button at the far left, remains disabled until an edit operation is performed. Once enabled, it can be used to stepwise reverse all actions performed with any of the edit functions. For example, if the Set Value function was used to change the resistivity values of five cells, then columns were deleted, the **Undo** button can be used to undo each of changes.

The **Redo** button, located to the right of the **Undo** button, remains disabled until the **Undo** button has been used at least once. It can be actuated to reverse the effect of the Undo function. Thus, it can only be actuated as many times as was the **Undo** button.

The Undo and Redo functions can also be executed with the commands available on the **Edit** menu.

Move

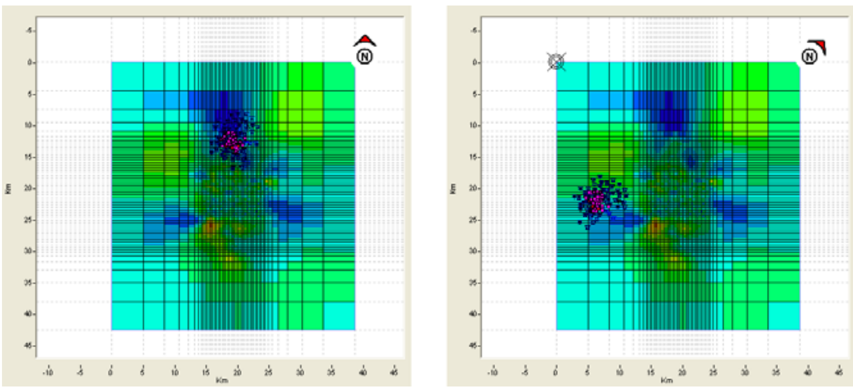
The **Move** button is used to reposition the mesh relative to the real world coordinates. Because this function operates on the horizontal plane, the button is enabled only when a Z-plane is the active view window. To move the mesh, click the button; the cursor changes to a cross hair with a diagonal red arrow. Click the mouse anywhere in the mesh area and drag the mesh to the desired location:



Deactivate the move function by again clicking the **Move** button. The new mesh coordinates are reflected in the mesh properties window (**File | 3D Mesh Properties**) as well as any open Z-plane view window.

Rotate

Like the **Move** button, the **Rotate** button is use to reposition the mesh relative to the rear world coordinates. Here, however, the mesh is rotated around the pivot point of the mesh, the rear, left corner in the Z-plane. To rotate the mesh, click the button; the cursor changes to a cross hair surrounded by a circular arrow. Click the mouse anywhere in the mesh area and drag the mesh around the pivot point to the desired rotation angle:



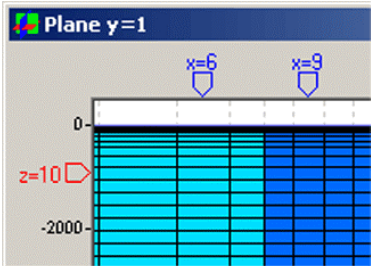
Deactivate the move function by again clicking the **Rotate** button. The angle of rotation is reflected by the compass symbol, if activated, in the upper right corner of the view windows, in the mesh properties window as well as any open view window.

Flag

Using the flag option, rows or columns in any plane can be marked for easy reference. Once a flag has been set, a marker appears in all view windows which contain the given row or column

To use, click the **Flag** button and select from either of the two displayed options:

Then click the mouse on the row or column to set the flag:



Flags can be toggled on and off by successively clicking the respective row or column.

Whereas the **Flag** button in the toolbar can be used to mark rows or columns in all view windows containing the selected rows or columns, the **Add/Remove Mark** command, accessed in the shortcut menu (right-click), serves the same function as the toolbar button.

Compass

As mentioned earlier in this chapter, the compass displayed in the upper right corner of the view windows serves to provide a frame of reference for better user orientation. In the view window shown above, the mesh has been rotated +40 degrees as is indicated by the fine, red arrow.

Lock/Unlock

The **Lock** button is used to lock and unlock resistivity values. Note, however, that the locked/unlocked status of cells is ***not*** used during the forward calculation performed within WinGLink. Locked cells are used only by the standalone version of the forward code.

Users of the standalone version of Randy Mackie's 3D forward code can make use of this feature by locking cells as described below, then exporting the 3D mesh model to an external file. The export function actually exports three files: a file which contains the mesh model (*.out) a file which contains the lock state of each cell (*.lck) and a third file (sites_*.dat), currently used only for test purposes, which contains the names of the stations located on the mesh. To export the mesh model and lock file, select the **Tools | Export 3D Mesh Model** command.

The resistivity values of locked cells remain fixed during the forward calculation only when using the standalone version of the code.

Locked cells are displayed in the view windows with a dotted grid. To lock/unlock cells, click the **Lock** button to enter locking/unlocking mode. Immediately after clicking the button into the down state, a shortcut menu appears with two menu commands: **Lock** and **Unlock**. Click the appropriate command. The four buttons to the right of the **Lock** button, which are described below, can now be used to lock/unlock individual cells, rows, columns and regions on the mesh.

Note that the locking/unlocking functions apply only to the current view in the active view window. You cannot, therefore, lock all cells in a given column by, for example, locking a cell in the top Z plane.

Cut on Topography

The **Cut on Topography** function is used to set the resistivity of cells positioned above the topography to the fixed resistivity value for air.

Set Value

To set the resistivity values of individual cells, select the **Arrow** button. Cells clicked while the **Arrow** button is down are assigned the resistivity value active in the Rho Value window. Similarly, if locking/unlocking is enabled (**Lock** button down), use the **Arrow** button to lock cell values.

Mesh ranges can be edited as elsewhere in WinGLink by using the Range Editor, which can be opened by selecting the **Edit Ranges** command on the **Range** menu. Refer to the Common Functions chapter of this manual for details.

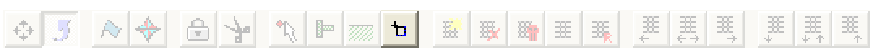
Row/Column Selection

Click the **Row/Column** button to open a shortcut menu which lists several selection options. As long as the button is in the selected state, cell locking/unlocking or setting of resistivities is performed by row or column.

Wide Selection

The **Wide Selection** button is used to lock/unlock cells or set resistivities for all cells to the left, right or below the selection point on the view window. The behavior of the **Wide Selection** button is dependent on the menu option selected from the shortcut menu which appears when the button is clicked.

Band Selection



The band selection locks/unlocks cells or sets cell resistivities for a region of the mesh defined by a rectangle created by clicking and dragging the mouse across the view window.

Split Row/Column

Click the **Split** button to open a shortcut menu which lists options for rows and columns. After making the appropriate selection, move the mouse over the active view window. Position the mouse pointer, which is now displayed as a pair of scissors with a miniature crosshair, at the point where the row or column to be split and click. After clicking, the row or column is split in two at the crosshair. Both halves retain the resistivity values of the original row or column.

Delete Row/Column

As with the **Split** button, after clicking the **Delete** button, a shortcut menu opens which lists options for rows and columns. Make the appropriate selection and position the mouse over the active window. Place the mouse pointer, which is now displayed as a box with a red cross and a miniature crosshair, on the row or column to be deleted. The row or column under the crosshair is deleted after clicking the mouse.

When deleting rows, the row directly under the deleted row is extended upward. When deleting columns, the column to the immediate right of the deleted column is extended towards the left. The exception to these rules are the deletion of the bottom row or right-most column. In these cases the row above the bottom row is extended downward or the column to the left of the right-most row is extended rightward.

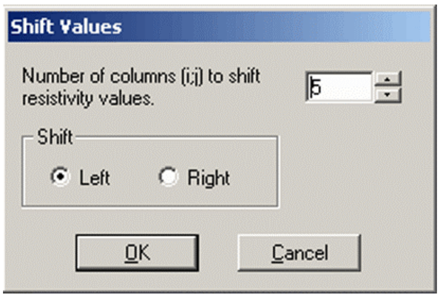
Redim Row/Column

Row and column dimensions can be modified by clicking the **Redim** button. As with the **Split** and **Delete** buttons, a shortcut menu opens which lists row and column options. After selecting an option, position the mouse over the active view window. The mouse pointer changes to either a horizontal or vertical double-ended arrow when positioned on a mesh line. By redimensioning the right-most column and bottom-most row in a view window, it is possible to resize the mesh.

Value shifting

The mesh values displayed in a given view window can be shifted to either the left or right using the horizontal shift buttons at the right end of the toolbar. The **Shift Left** and **Shift Right** buttons, located at either side of the **Shift Values** (horizontal) button with the double arrows, move all cells by one column to the left or right, respectively. When shifting to the left, the far-right column retains its values. Likewise, when shifting to the right, the far-left column remains unchanged. The Shift Up and Shift down buttons on either side of the Shift Values (vertical) button at the far end of the toolbar can be used to shift values up and down, respectively.

The horizontal and vertical **Shift Values** buttons can be used to shift the mesh by any number of columns to either the left or right / up or down:

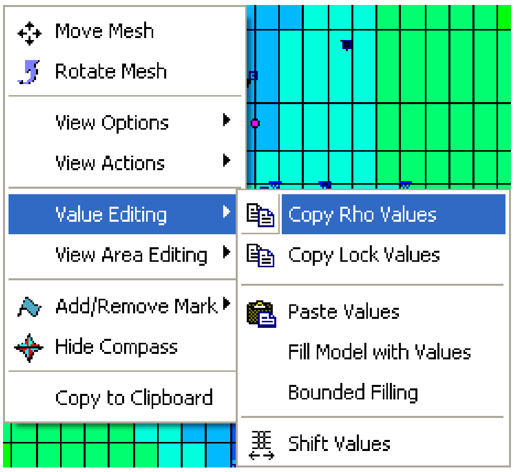


The Shift Values windows can also be accessed on the shortcut menu by right-clicking and positioning the mouse over the **Value Editing** option.

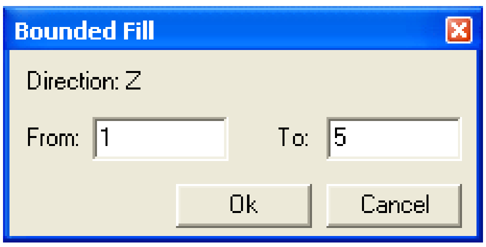
Note that values are shifted only on the selected plane. To shift resistivity values for all planes, the operation must be repeated throughout the model.

Value Editing

The **Copy Rho Values** and **Copy Lock Values** commands can be used to copy all rho or lock values from one view window to another view of the same plane, to a range of planes or for the entire model. These commands are accessed on the **View Window** shortcut menu, which is opened by right-clicking a view window, then positioning the mouse over the **Value Editing** command:



To copy either rho or lock values from one view window to another, select the respective command, open a new view window of the same type, i.e. X, Y or Z plane, then select the **Paste Values** command from the shortcut menu in the target window. The copied rho or locked values can be copied to all like-type planes in the model by selecting the **Fill Model with Values** command. Use the Bounded Filling command to specify a range of planes over which the fill command is to apply:



View Area Editing

The view area displayed in one view window can be copied to another with the **View Area Editing** commands. Resistivity and lock values are not copied when using this command, rather the actual mesh view area is copied from one view to another of the same type.

The **Copy View Area** command can be accessed on the View Window shortcut menu by positioning the mouse over the **View Area Editing** command to open a submenu. To copy the view area from one window to another, select the **Copy View Area** command in one window, then select the **Paste View Area** command from the shortcut menu in the target window.

Anchor Function 

The anchor button, which is enabled for Z-plane and map views, is used to control how the mesh position is recalculated when deleting rows and columns. If, for example, an anchor is placed on a column with index i and the you use the Destroy Column function to delete a column with an index > i, the locations of the columns with an index < i remain fixed. Thus, this tool gives the user control in how columns and rows are deleted. Note that this function is not available for the X- and Y-planes.

Editing Meshes: Differences Between the MT 2D Inversion and MT 3D Modeling Programs

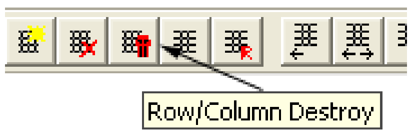
The edit functions in the MT 3D Modeling program are similar, however not identical, to those available in WinGLink's 2D Inversion program. The principle difference is the way in which rows and columns are added and deleted.

Adding rows/columns:
In the 2D Inversion program, rows and columns are actually added. New rows and columns are assigned the resistivity value currently selected in the Rho window. In MT 3D Modeling, on the other hand, rows and columns are not added, but rather split. A given row or column is divided into two: both halves retain the resistivity value(s) of the original row/column.

Deleting rows/columns:
To delete a column in MT 2D Inversion, you must click the line which defines a given row or column. In MT 3D Modeling, you instead position the crosshair on the row or column which is to be deleted.

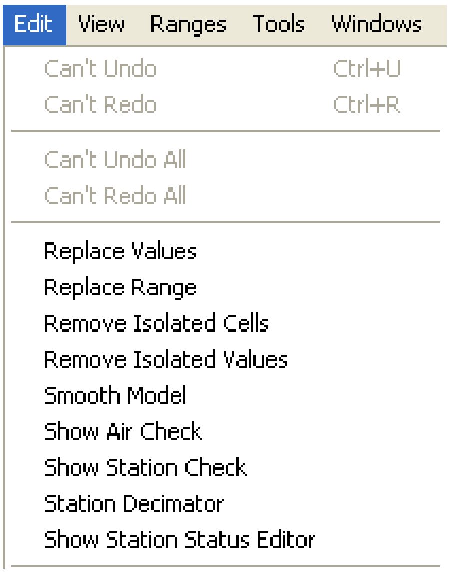
Destroying rows/columns

Unlike the Delete rows/columns function, which effectively merges two adjacent rows or columns, the Destroy functions actually remove the selected row or column from the mesh. Thus, execution of the Destroy commands, results in a change in the mesh dimensions. To execute the commands, press the **Row / Column Destroy** button (10th button from right in the toolbar) and then select appropriate function (Row or Column):



Additional edit functions: the Edit menu

A number of additional edit functions can be accessed via the Edit menu:



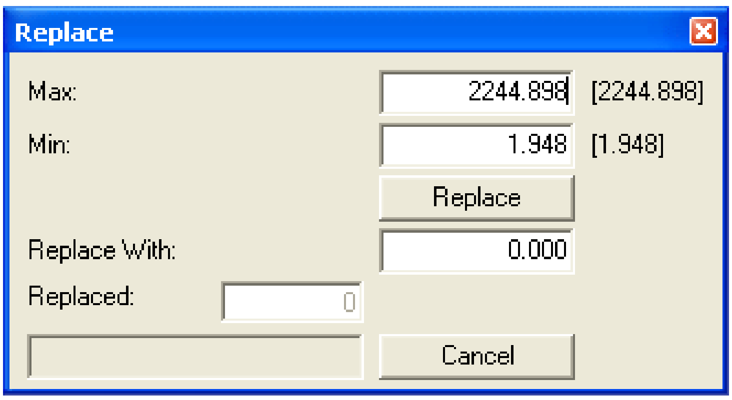
Replace Values

Use **Replace Values** to swap all cells with a given resistivity value with a specified resistivity value:

The number of replaced resistivity values is displayed in the **Replaced** field.

Replace Range

This function is similar to the **Replace Values** function, the difference being that min. and max. values are specified. Resistivity values between and including these two limits values are replaced by the specified replacement value:



Note that the min. value cannot be less than and the max. value cannot be greater than the respective min. and max. resistivity values in the model. These values are displayed to the right of the **Min.** and **Max.** fields in parenthesis.

Remove Isolated Cells

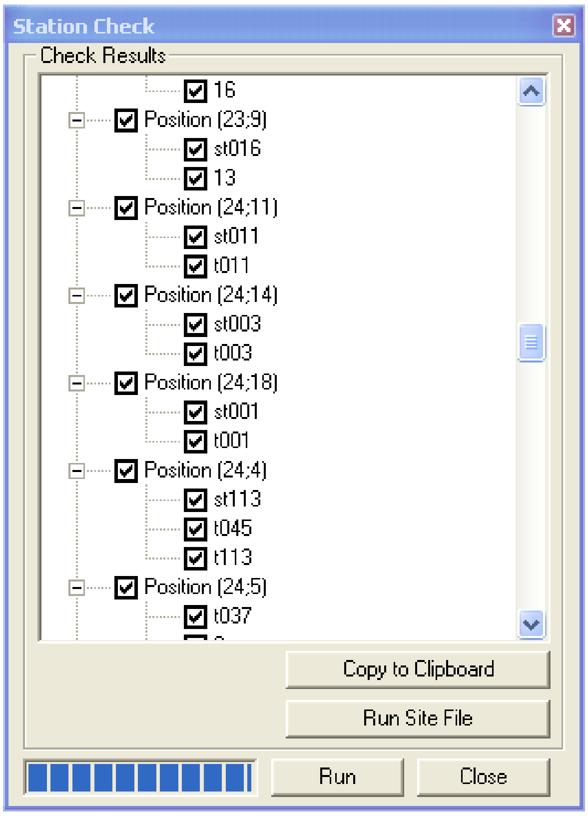
In order for the forward modeling code to function properly, there must not be any isolated mesh cells, i.e. cells without neighbors, which protrude from the top of the mesh. These can occur during the creation of relatively coarse meshes when using topography. To remove all such cells from the model, select the **Remove Isolated Cells** command.

Show Air Check

This function runs a check to ensure that there are no air cells located below the model topography.

Show Station Check

After running a forward calculation, it is possible to generate an EDI file which contains the forward response at a given cell. While it is possible to position more than one station on a given cell, the EDI files generated for all stations on a particular cell will be identical. To ensure that there is not more than one station on any cell, execute the Station Check command:



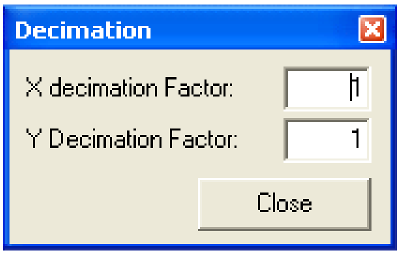
All cells which contain multiple stations are listed. Either individual stations or all stations on a given cell can be deactivated.

Exported site files can be read into the MT 3D Modeling program via the **Run Site File** button. By working with various site files, it is possible to quickly change the stations which are used with a given forward calculation.

Station states can be exported to so-called site files with the **Tools -> Export Site File** menu command.

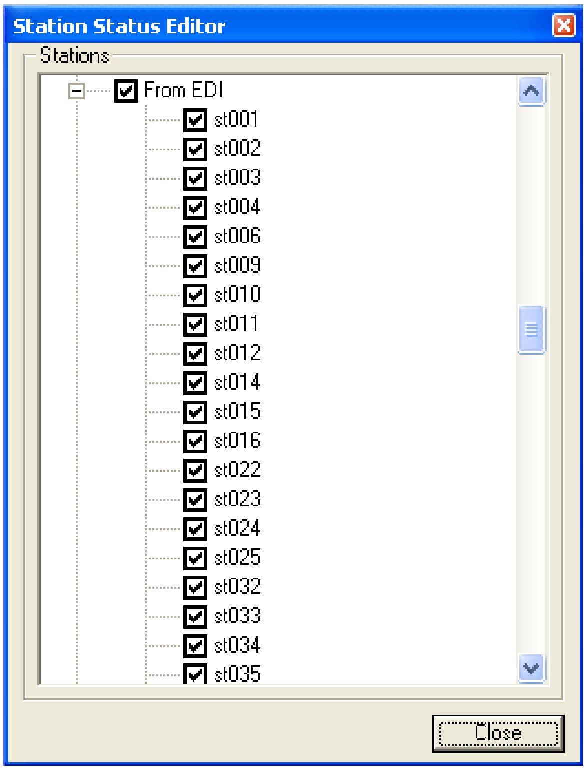
Station Decimator

This function gives you the option of extracting a subset of soundings from all available soundings. Use the X and Y direction fields to specify the decimation factor along the respective axes. Every nth sounding is extracted according to the values specified here:



Show Station Status Editor

Unlike the Station Check window, which can be used to edit stations in cells which contain multiple stations, the Station Status editor can be used to edit the states of all stations:



Station states can be exported to so-called site files with the **Tools -> Export Site File** menu command and reimported into WinGLink in the Station Check window as described under Station Check.

Remove Air Layers

Use the **Edit | Remove Air Layers** command to remove all planes which contain only air.

Remove Isolated Values

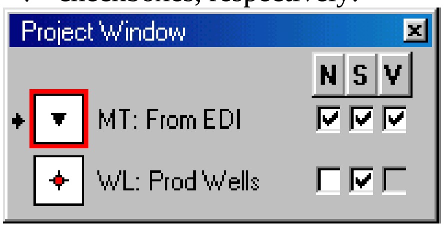
The **Edit | Remove Isolated Values** command is used to remove cells with isolate values (with respect to the surrounding cell values).

Station Data Options

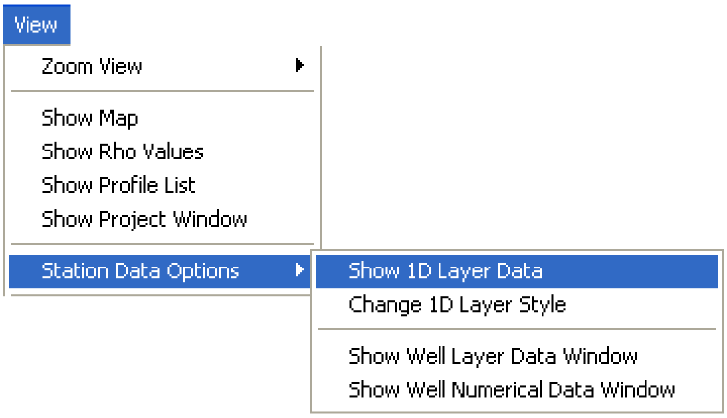
1D models for MT stations as well as layered/numerical data may be superimposed on vertical sections, i.e. X and Y views.

To Display 1D Models for MT stations

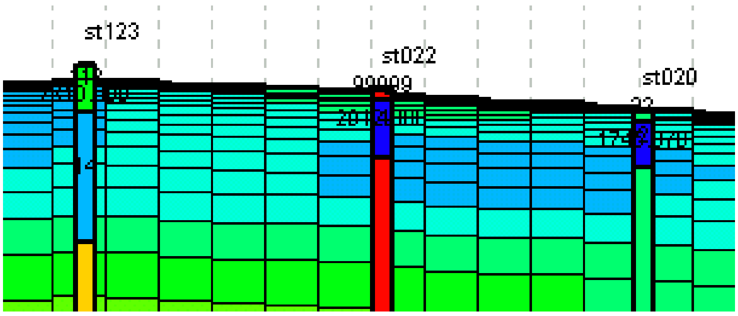
1. In order to display 1D models, first open the Project Window (**View | Show Project Window**) and check the "**S**" checkbox of any projects for which the 1D models are to be displayed. To display station names and resistivity values, check the "**N**" and "**V**" checkboxes, respectively:



2. Display the 1D models by selecting the **Show 1D Layer Data** command from the **View | Station Data Options** menu extension:



When the **Show 1D Layer Data** option is enabled, the 1D model of any MT station for which a model has been created is superimposed on the 3D mesh, provided the station is located in the current view window:

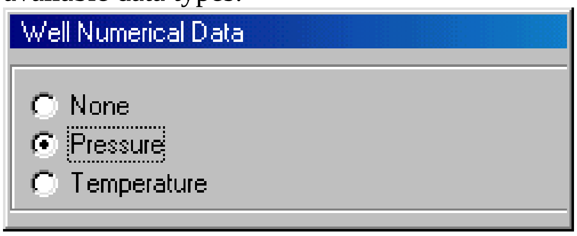


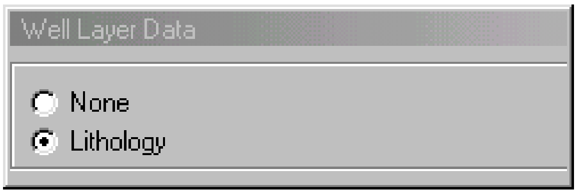
3. To toggle between the colored resistivity values in the 1D models and a vertical line with simple tic marks which indicate

changes in resistivity value, select the **Change 1D Layer Style** option from the **Station Data Options** menu extension.

To Display Wells, Layered Data and Numerical Data

1. As with 1D models, in order to view the well data, the "**S**" checkbox in the Project Window (**View | Show Project Window**) must first be selected for the project which contains the well data. Well traces can be toggled on and off by selecting/unselecting the "**S**" checkbox; well station names and values can be toggled on and off with the "**N**" and "**V**" checkboxes, respectively.
2. Next, make the appropriate selection from the **View | Station Data Options** menu extension. After selecting either **Show Well Layer Data Window** or **Show Well Numerical Data Window**, the respective data window opens which lists the available data types:





The selection is immediately reflected in the view window.

To Export Data in txrx Format

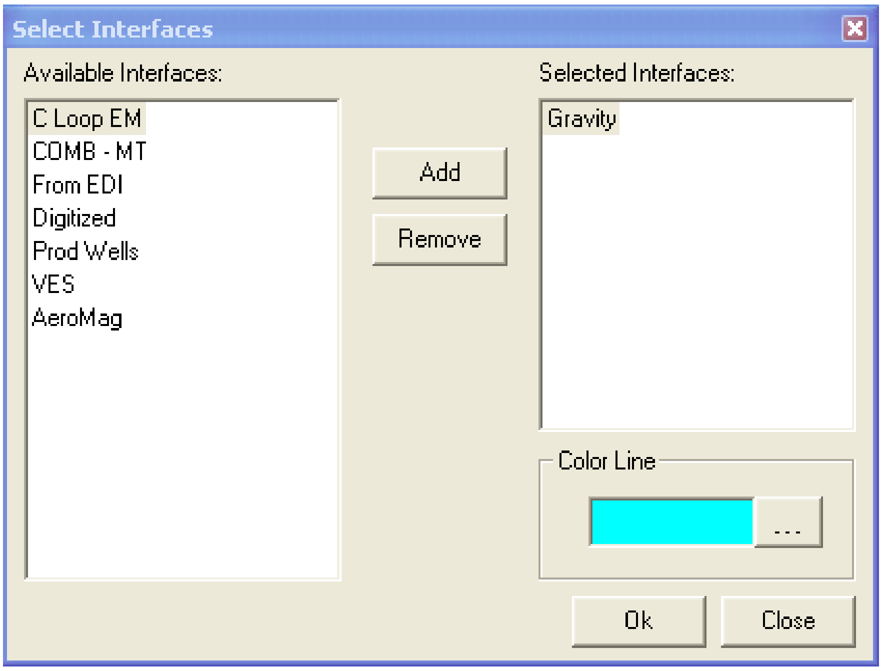
Stations can be exported in the Randy Mackie txrx format: by executing the Tools | Export rx and tx files menu command:

Additional Display Functions

Several other display functions are available under the View menu item in addition to those listed above:

Select Interface Grid

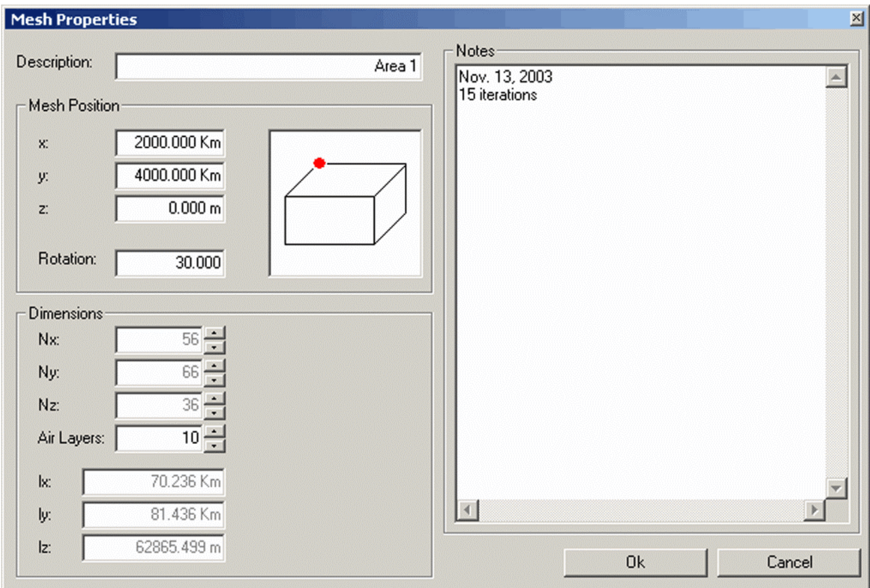
The **View | Select Interface Grid** menu command is used to superimpose a profile extracted from a selected elevation grids. Upon selection of the menu command, the following dialog box opens:



Here, you can the projects from which the elevation grid is to be extracted as well as the color with which the extracted elevation profile is to be displayed on the mesh. Extracted profiles are displayed only on vertical sections.

Mesh Properties

Mesh properties, including name, position and number of air layers can be edited in the Mesh Properties window. To open this window, select the **File | 3D Model Mesh Properties** command:



Description: Mesh name

Mesh position (x,y,z): Coordinates on the project datum

Rotation: Positive degrees of counterclockwise rotation about the top left, rear column

Air layers: Number of air layers on top of the mesh (minimum: 7, ideal: 10)

lx, ly, lz: Actual mesh dimensions

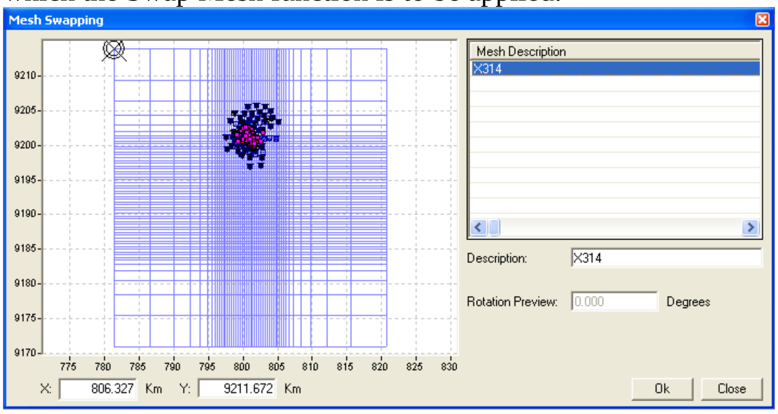
Notes: Any information you would like stored together with the database

Swap Mesh Function

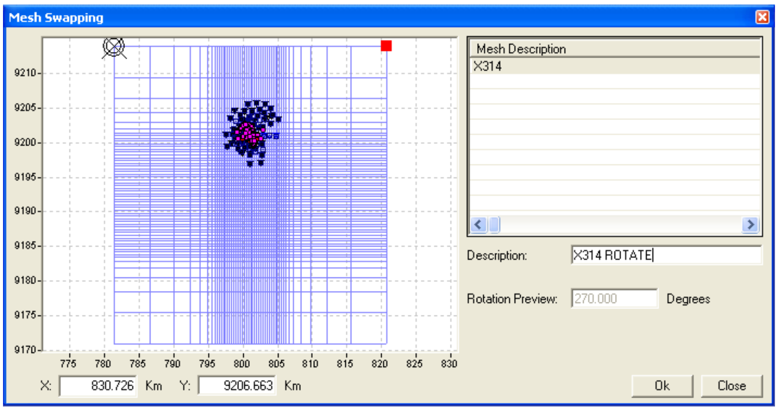
With the **Swap Mesh** function, which is located on the **File** menu, you can change the corner which is used as the pivot point for the model i.e. change which corner be used as the upper left corner of the model. When applied to a mesh, a copy of the mesh is translated in such a way that the area covered by the final mesh is the same as that covered by the original mesh.

To Swap a Mesh:

1. Select the **Swap Mesh** function on the **File** menu. If a mesh is currently open, the program prompts you to close the mesh.
2. In the Mesh Swapping window which opens, select the mesh to which the Swap Mesh function is to be applied:



3. With the mouse, click the corner which is to be defined as the new pivot point. When selected, a red square appears on the corner:

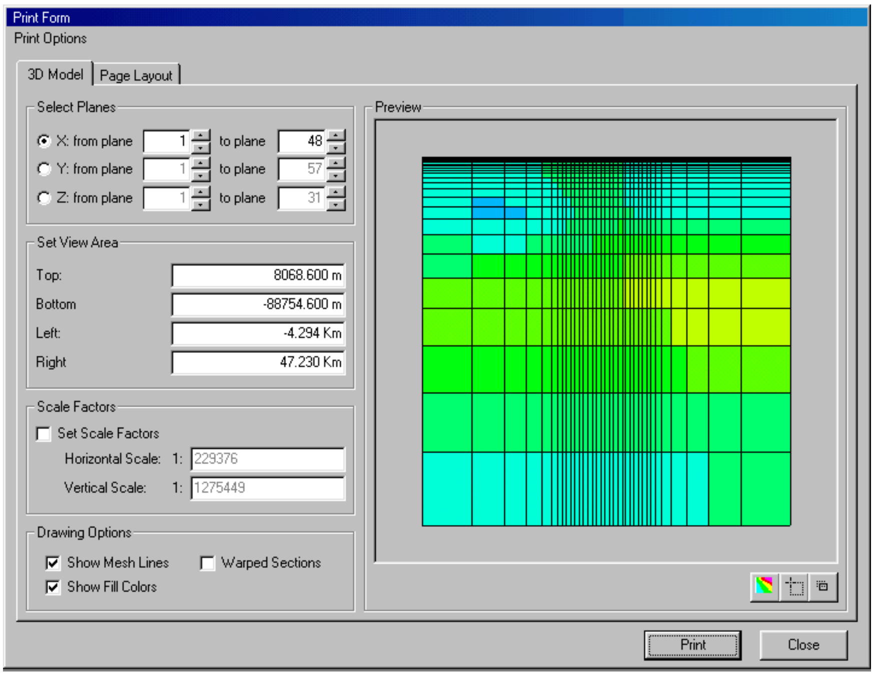


4. After selecting the new pivot point, enter a new mesh name in the Description field.
5. To translate the mesh to the new pivot point, click the **OK** button. A progress window appears in the status bar of the main window. Once the translation process has completed, the Mesh Swapping window closes.

6. The translated mesh covers the exact same area as the original. The only difference is the position of the pivot point. This is indicated by the a corresponding change to the compass direction. Any rotation operations now performed on the mesh now rotate around the new pivot point.

Printing Mesh Views

The MT 3D Modeling print form is called up with the **File | Print** command:



Individual or ranges of mesh planes can be printed. In addition to the meshes themselves, legends, text boxes and other relevant information can also be printed.

Before printing, specify the planes to be printed as well as the area of the mesh to be printed. The view area to be printed can be set either manually in the **Set View Area** section of the window, or using the **Fit All**, **User Zoom** and **Pan View** buttons located in the lower-right corner of the window. If desired, horizontal and vertical scaling factors may be set in the **Scale Factors** section of the window.

Mesh lines and fill colors are printed by default. To disable these, or enable mesh warping, make the appropriate selections in the **Drawing Options** section at the bottom of the window.

On the **Page Layout** tab of the Print Form window, you can select and position various display elements including, among others, mesh, color ranges and text descriptions. The number of units printed on each page can also be selected here.

Even though up to four meshes can be printed per page, you can quickly generate more printouts than intended if the planes to be printed are not selected correctly on the **3D Model** tab of the dialog box.

Forward Model Calculation

Overview

The forward model calculation performed in the MT 3D Modeling program makes use of the MT3FWD code written by Randy Mackie:

Developed in 1999, MT3FWD is a program to compute magnetic and electric fields at the surface of a 3D electrical resistivity model illuminated by electromagnetic plane waves. It is a modified version of D3MTFWD2 released by Mackie and Madden in 1997. The MT 3D modeling algorithm uses the integral form of Maxwell's equations to derive a finite difference approximation for the magnetic field that is second order. Non-divergence of the magnetic field is enforced by evaluating the magnetic and electric fields on grids that are staggered relative to one another. The resulting linear system is solved by pre-conditioned conjugate gradient relaxation. Convergence is considerably enhanced by explicit correction of residual failure of the non-divergence condition. The algorithm is discussed in Mackie, Smith & Madden* (1994). However, the program does not employ the coordinate transformation that converts an isotropic model on a non-uniform grid to an anisotropic model on a uniform grid.

The program assumes that 2D structure parallel to each edge continues uniformly to infinity in the direction normal to the edge. The tangential magnetic fields are assigned using 2D calculations for each edge. The edges parallel to the source magnetic field will have electric current flowing normal to them and so TE mode calculations are done. The edges perpendicular to the source magnetic field will have current flowing parallel to them. In these cases, the tangential magnetic fields are zero.

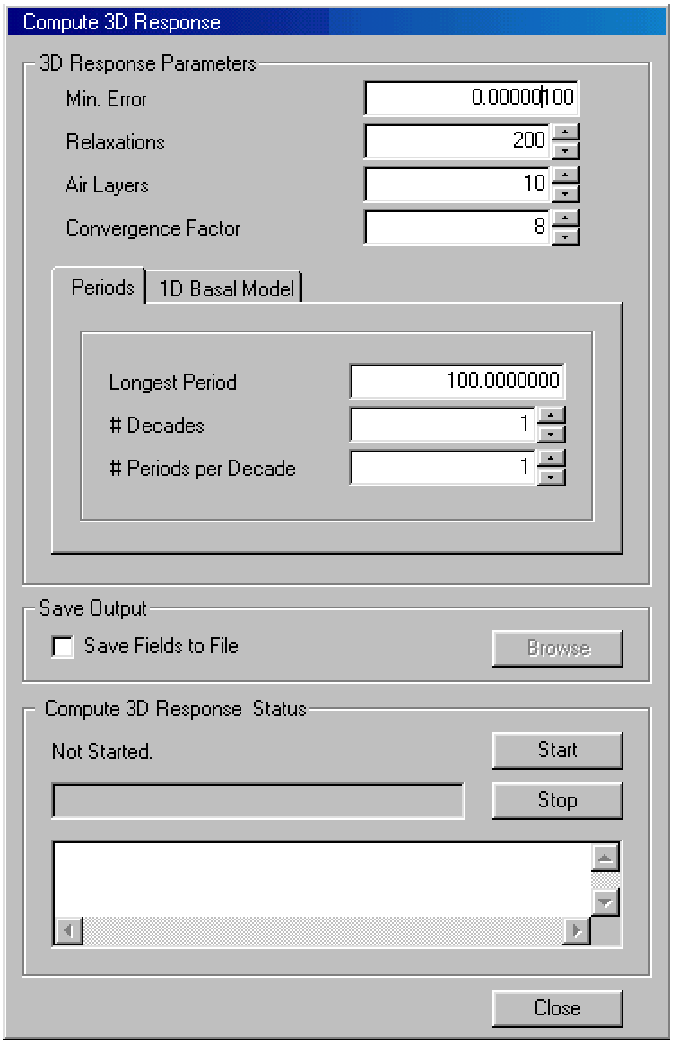
Relaxation is faster if a good estimate of the solution is provided at the start. The program interpolates the vertical boundary fields to form the initial guess in such a way that the fields should already be a solution to the interior fields if the model is 2D rather than 3D.

*[Mackie, R.L., Smith, J.T. and Madden, T.R., Three-dimensional electromagnetic modeling using finite difference equations: the magnetotelluric example. Radio Science, 29, 923-935, 1994.]

Forward Calculation

Required for input is a 3D MT mesh in the slightly modified Randy Mackie defined at the end of this chapter. All 3D MT meshes which are imported into or created in WinGLink fulfill this requirement.

To set the parameters for the forward calculation for the loaded 3D MT mesh, select the **Tools | Forward 3D Model Mesh** menu command. The 3D Forward Process dialog box opens:



3D Forward Parameters

Min. Error: Error level at which the forward calculation is stopped. (1E-4 - 1E-6 recommended)

Relaxations: 50-100 recommended

Air layers: Min. 7, 10 recommended

Convergence factor: Provides control over the speed at which conversion is reached. A higher value, e.g. 8, *may* reduce the number of iterations required for convergence at the expense of requiring additional memory.

Periods:

Min Period: Specifies upper bound of frequency range over which fitting is tried

#Decades: Number of decades to be used. Maximum period used is thus equal to the minimum period *10^number of decades

#Periods per Decade: Number of periods per decade

1D Basal Model:

The 1D basal model is used to set the bottom impedance condition that relates the E and H fields at the bottom of the model. It can be a homogeneous half-space model or a layered 1D model.

Thickness: Layer thickness in meters

Resistivity: Layer resistivity in ohm.m

Save Output

The forward calculation returns field values which are then transformed into MT impedances for the purpose of generating EDI files. The field values are automatically stored in the database when the mesh is saved. The values may also be written to an external file.

Save to File: Saves the field values to an external file (*.rslt). When saving to a file, click the browse button to specify the name and path of the file to which the fields are to be written. Files stored using this option can be opened in the EDI Export window and used to export EDI files for any MT station located on the mesh.

Time Requirements

The time required to perform the forward calculation varies dramatically (approximately cubically) depending on the mesh size. During testing, a forward calculation performed on a sample 20x20x20 mesh using 1 decade and 3 periods on a 1.4GHz, 512MB RAM computer required 2 minutes. A 35x35x35 mesh, on the other hand, required nearly 20 minutes. A progress bar and message box in the 3D Forward Process window provides information on the status of the calculation.

Starting the Forward Calculation

To start the forward calculation, click the **Start** button in the 3D Forward Process window. If the **Save to File** checkbox is selected, the **Start** button is not enabled until a file has been selected using the **Browse** button.

The error of each iteration is displayed in the text box in the lower area of the 3D Response window.

The **Stop** button can be used to terminate the calculation.

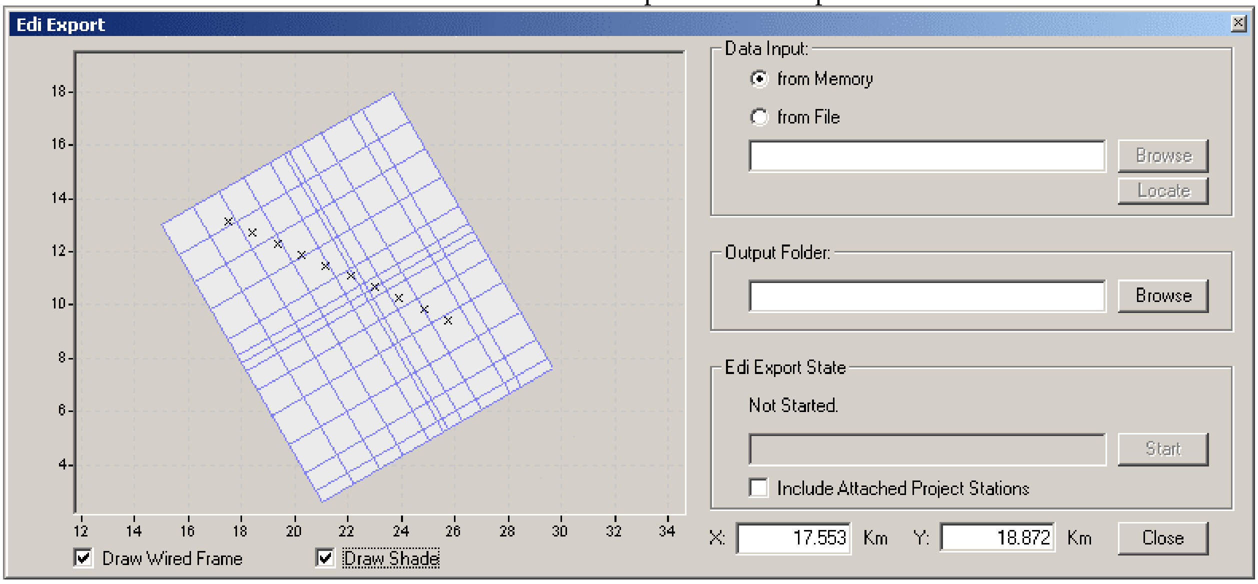
Exporting EDI Files

The MT impedance and tipper values calculated from the field values returned by the forward calculation can be used to generate EDI files. These impedance and tipper values, located either in memory or in an external file, can be used, in principle, to generate EDI files for any mesh column. The EDI export is, however, limited to columns containing at least one MT station. The EDI files of stations associated with the same column are identical except for the station names.

Stations can be added and removed, as well as activated and deactivated, in the Maps program. (Refer to Chapter 4, Maps, for details.) Note that stations do not need to be active or even exist when calculating the forward model, i.e. they can be added after performing the forward calculation.

To export EDI files for stations associated with a mesh:

1. On the **Tools** menu, select the **Export EDI File** command.
2. The EDI Export window opens:

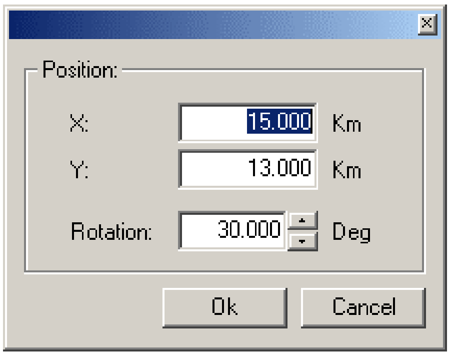


3. In the Data Input area of the window, specify the location of the impedance/tipper values.

   If you have performed a forward calculation during the current MT 3D Modeling session or if a forward calculation has been performed using the current mesh and the impedance values were stored in the database, the **from Memory** option is active in the Data Input area of the window. If not, only the **from File** option is available. External files must be files containing impedance values generated by the MT 3D Modeling program, i.e. result files generated by the stand alone version of the 3D forward program cannot be used.

   If the **from File** option is selected, the EDI export cannot be started until the file has been specified.

4. Click the **Locate** button in the Data Input area of the window to open the Locator window:



   Use the **X** and **Y** fields to adjust the position of the mesh relative to the stations. The **Rotation** field may be used to change mesh rotation. Changes made here are not permanent and are thus not reflected in the mesh properties window or elsewhere in WinGLink.

5. Displayed in left portion of the window is a view of the top plane of the 3D mesh including all stations contained in the project. Use the two checkboxes **Draw Wired Frame** and **Draw Shade,** located at the base of the window, to change the mesh display.

   The current position of the mouse pointer on the mesh is indicated in the X and Y fields at the bottom of the window.

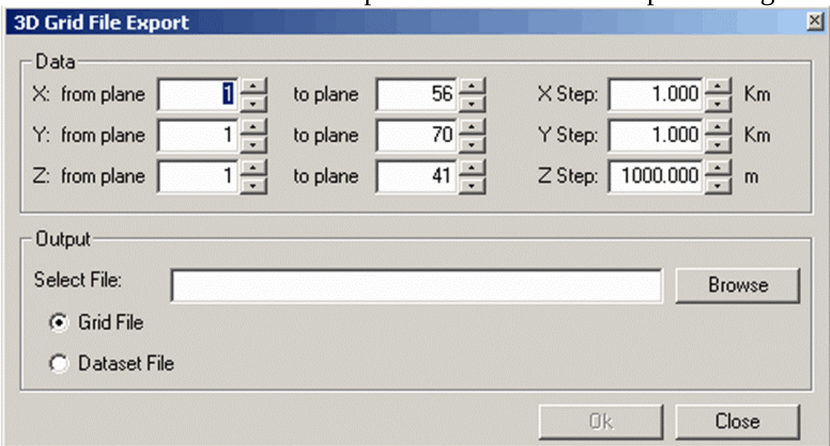
6. Select the **Include Attached Project Stations** checkbox to export EDI files for all MT stations positioned on the mesh which are contained in projects attached to the current project in addition to those contained in the current project.

   For details on attaching one project to another project, please refer to section "Attaching a Project to another Project" in Chapter "Getting Started".

7. After specifying a destination folder for the EDI files in the **Output Folder** field, click the **Start** button to begin the export process. The export status is indicated by a progress bar.

Exporting 3D Grid Files

Not only can 3D MT meshes be output to mesh files which conform to the Randy Mackie 3D MT mesh specification, but they can also be output to uniform 3D grid files in either the Slicer format or as Dataset files. To export a 3D mesh to a uniform 3D grid, select the **Tools | Export 3D Grid File** menu command to open the 3D Grid File Export dialog box:



Specify the planes to be included in the file and the spacing to be used in each of the planes. After specifying the file type and an output file name in the **Output** section of the dialog box**,** click **OK** to export the file.

Note: 3D Grid files exported using the **Export 3D Grid File** command cannot be reimported into WinGLink.

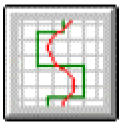
Randy Mackie 3D Mesh Specification

The mesh format is that specified by Randy Mackie plus an additional data block, located at the end of the file, which contains georeferencing information.

```
NX, NY, NZ (#’s of blocks) [Nair (layers)], [MAP.
VAL[UES]]
X block sizes (NX values in meters)
Y block sizes (NY values in meters)
Z block sizes (NZ values in meters)
1 (layer #)
NX*NY codes or values in free format: x varies
fastest
-----
-----
NZ (layer #)
NX*NY codes or values in free format: x varies
fastest
0. resistivity(for code 1) resistivity(for code
2) ....
resistivity(for largest code) [resistivity for
sea water]
end (optional termination text)
-----
-----
WINGLINK
ABC (site name)
I J (block numbers)
0000.000 0000.000 (real world coordinates)
0 (rotation)
```

20: TDEM Soundings

Overview



The Soundings icon is displayed in the program menu whenever a sounding project is selected.

When a sounding project (MT, TEM, DC, etc.) is selected and the program launched, the stations of the selected project are loaded together with their original datasets.

The discussion in this chapter is limited to the TDEM Soundings program.

For each station, this program can be used to:

- Display sounding curves, either as resistivity or voltage values vs. time
- Edit sounding data
- Compute a 1D smooth and layered inversion model
- Export data (TEM format)

As described at the end of this chapter, many of the functions available for individual soundings can also be performed in batch mode on any number of soundings contained in the current project.

About Editing

The results of any editing on sounding data will be saved in the *"edited data"* section of the database. Original data will not be overwritten and can always be restored using the menu command: **File | Reload Original Data**.

The edited data saved by the Soundings program are used by other WinGLink programs:

- Smooth and layered 1D models are used to calculate **imaged sections** and **cross sections,** respectively.
- Curves and 1D models are also used to **extract parameter maps** by the **Maps** program.

Sounding Data Which are Saved in the Database

For each station of the current project, the following datasets are saved in a WinGLink database:

Original Data

Apparent resistivity as imported from external files. Original curves can always be recovered to restart the editing process using the Soundings menu command **File | Reload Originals**.

Edited Data

Edited curves are initially constructed by assigning to each datapoint the same value of the original curves. At the end of each editing session, the edited values are saved in the database, separately from the original values.

Smooth 1D Model

One smooth model is saved for each sounding. This provides a preliminary guess model for the layered inversion routine and is also used to produce imaged sections along profiles.

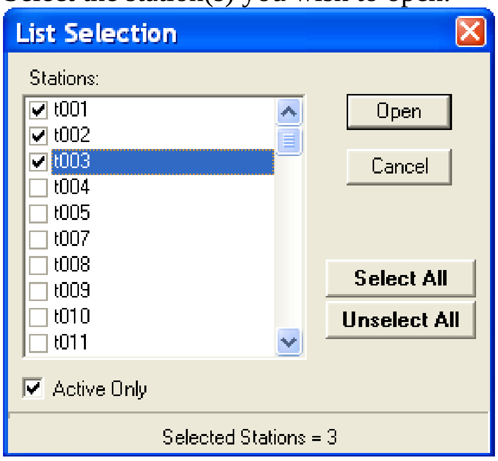
Layered 1D Model

One layered model is saved for each sounding. These models are displayed along profiles by the *Cross Sections* program. The 1D curve used to calculate the layered model is the same curve used for the smooth model.

Opening, Saving and Printing Sounding Data

To Open Soundings from the Station List

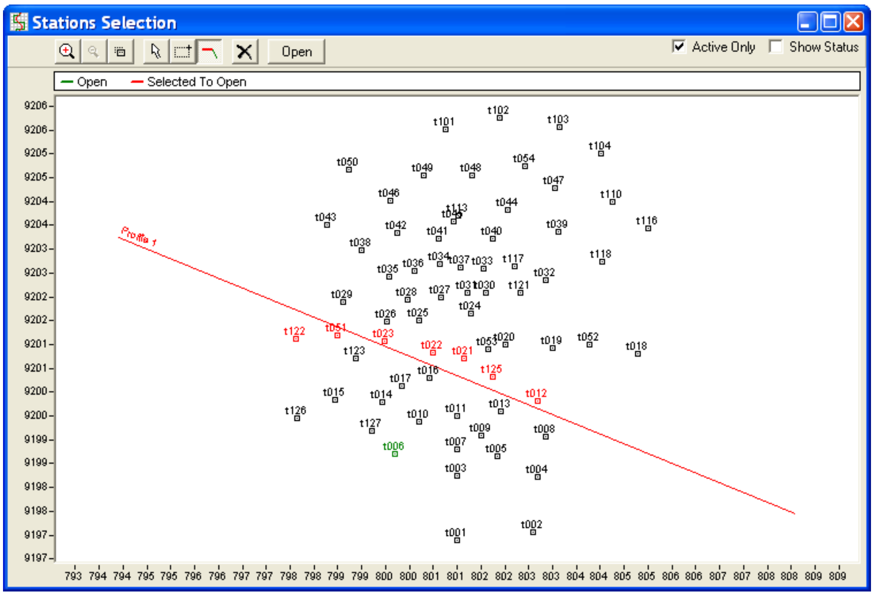
1. Select the **File | Open from List** menu command.
2. Select the station(s) you wish to open:



3. If you want to display only the active stations, select the **Active Station** check box.
4. Click the **Open** button.

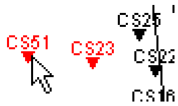
To Open Soundings on the Station Map

Select the **File|Open from Map** menu command; a simplified location map for the project station is displayed:



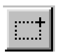
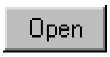
Open Soundings by Clicking Individual Stations:

1. Click the  button.
2. Click the stations you wish to open; the selected stations turn to red:

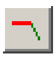
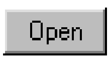


3. Click the  button.

Open All Soundings Within a Given Area:

1. Click the  button.
2. Left-click the mouse on a corner of the area of interest and drag it to define a rectangle on the map. When releasing the mouse button, all stations in the rectangle will turn to red.
3. Click the  button.

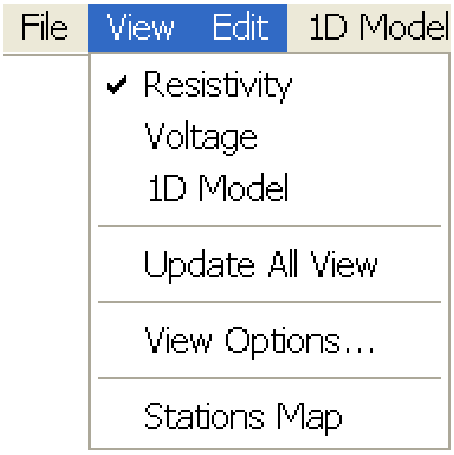
Open All Soundings Associated with a Profile Trace:

1. Click the  button.
2. Left-click the mouse on the profile or on a segment of the profile trace of interest. All stations previously added to that profile trace will turn red, indicating selection.
3. Click the  button.

Display Options

Each window can be set to show the resistivity/voltage curves or the 1D model for each sounding.

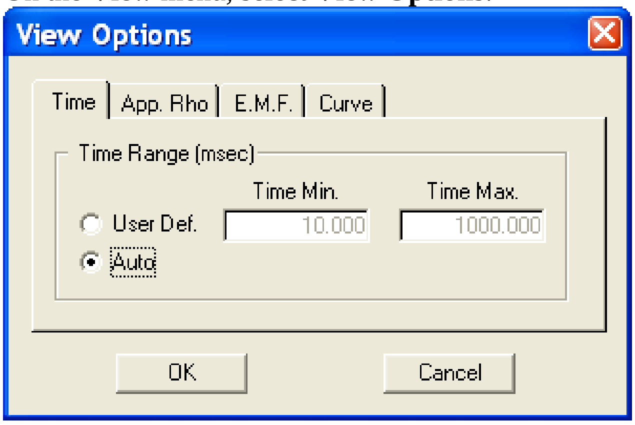
On the **View** menu, select from the available display options:



Setting Diagram Ranges and Scale

To Set the Ranges and Scale for the Plot Diagrams:

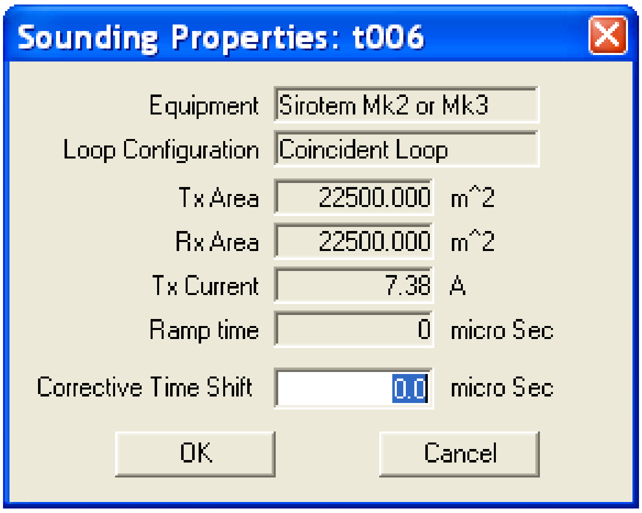
1. On the **View** menu, select **View Options**:



2. For each parameter, set the automatic range option or enter the max. and min. values to be plotted. The axis scale can be set to linear or logarithmic for many plots.
3. The settings made here apply to all window.

Sounding Properties

An overview of the sounding properties can be displayed by selecting the **File | Sounding Properties** menu command:



With the exception of the **Corrective Time Shift** field, all fields displayed here are read only. The **Corrective Time Shift** field can be used to enter a corrective time shift, e.g. in the event that the value was entered incorrectly during data acquisition.

Printing Curves and 1D Models

To Print Station Curves

1. Select the station by clicking on its window;
2. Select the **File | Print** menu command.
3. Select the printing options and click the **Print** button.

Exporting TEM-Format Files

To facilitate the exchange of TEM soundings between projects and users, station soundings can be exported to TEM-format files. The TEM format is a text-based format specified by Geosystem originally created for the purpose of simplifying the import of TEM data into WinGLink, regardless of data type, e.g. Sirotem, Geonics or Zonge.

To export a TEM sounding to a TEM-format file, select the **File | Export** command from the Soundings menu.

TEM files, created either using the TemMerge program or exported form the Soundings module, can be imported into WinGLink as described in the “Data Import” chapter of this manual.

Note regarding inconsistencies between input and output TEM files:
Due to data normalization procedures, exported TEM files may not necessarily be identical to the TEM files used for input. This is particularly true of Geonics soundings which may contain up to three curves. In this case, the second and third curves are normalized to the current, TX area, and RX area of the first curve.
In addition, as not all descriptive information contained in the input TEM file is stored in the database it is not included in the output TEM file.

Editing Sounding Data

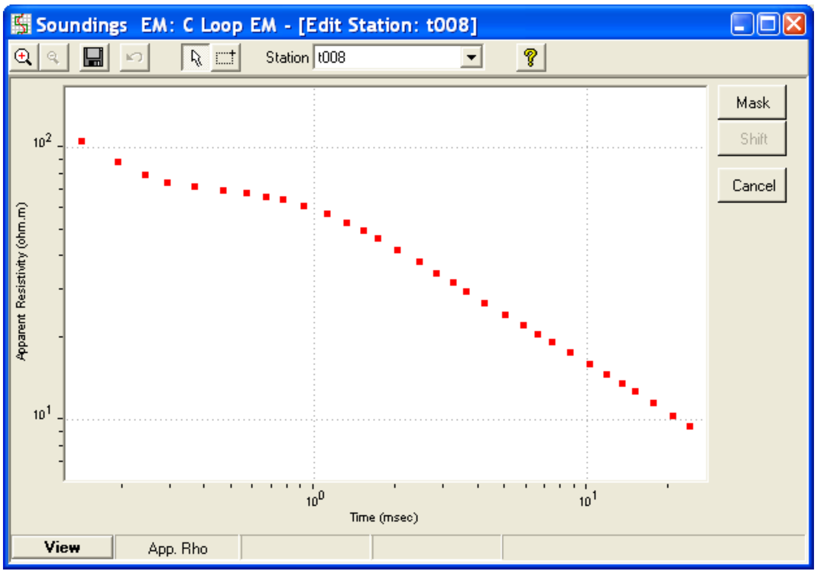
The Edited Curve

When sounding data are imported into a WinGLink database, they are saved as “original data” in a reserved area of the database and are never modified unless new data are imported. At the same time, a duplicate dataset is generated and saved in a different area of the database for use by editing functions. When an editing session is carried out on the sounding data, it is carried out on this second dataset, which is referred to as the “edited curve” of the sounding.

Saving changes following an editing session means saving an edited curve. This curve is used by default in calculating 1D inversion models and pseudosections.

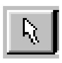
Interactive Sounding Editing Form

TDEM sounding data can be edited in the interactive sounding editing form, which is opened with the **Edit** menu command:



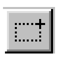
The form includes commands for performing various editing functions:

Selecting Data Points



Single point selection

Click to select/unselect for editing single data points on the curve.



Multiple points selection

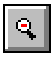
Click to select/unselect for editing all data points included in a user-defined range: click and drag the mouse to define the area.

Zooming in on the Curve

Zoom in

1. Click a point in the curve diagram and release the button.

2. Move the mouse to define the zoom area.
3. Click and release the left mouse button.

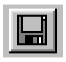


Zoom out

Click to reset the previous view.

Note: this button is available only after the **Zoom in** button has been used. It remains available until the original curve size is restored.

Saving Editing Changes



Save button

Saves the editing changes. These changes must be confirmed when closing the station window in the main menu in order to store them in the database.

Editing Data Points

Masking data

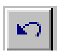
1. Click and select the points by single or multiple selection (the masked points will turn gray).
2. Click the **Mask** button again to confirm the changes.
3. The points will be hidden from the display
4. Click the **Mask** button again to restart the masking/unmasking process.

Shift data points

1. Click and select the data points to be shifted by single or multiple selection.
2. Move the data points to the desired positions and click the left mouse button.
3. Click the **Shift** button again to confirm the action.

Note: The Shift button is enabled only if the sounding curve is plotted as voltage vs. time. Use the View button at the bottom of the Edit window to toggle between resisitivity and voltage views.

Cancel the current action



Undo button

Undo all the editing changes made since the last time the curve was saved. All editing steps can be undone provided the station has not yet been saved since editing.

Reloading Original Curves

This action overwrites the edited curves of a station with the original curves.

To reload original curves:

1. Select the station by clicking its window.
2. Select the **File |Re-Load Original** command.

After reloading, the smoothed curves and the 1D models need to be recalculated.

1D Inversion

1D Inversion: Overview

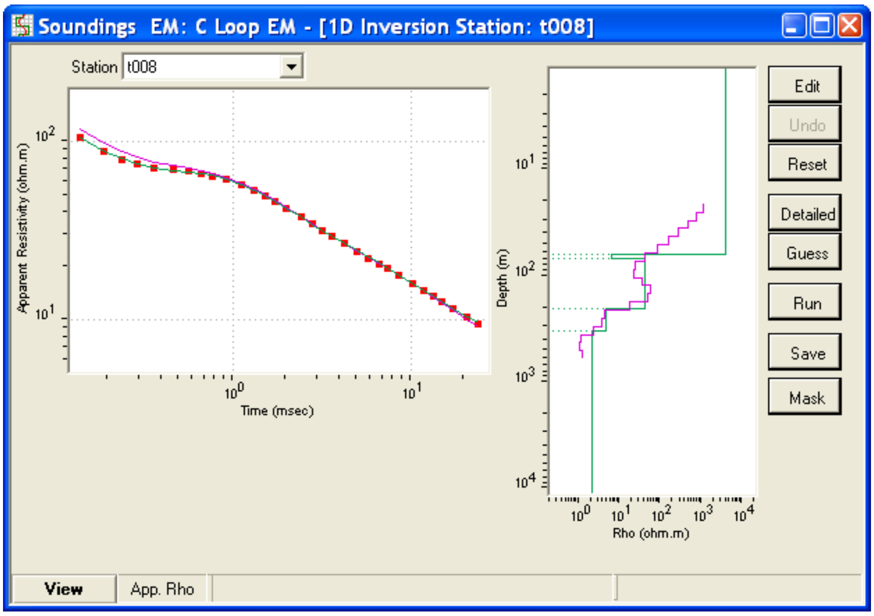
The 1D Model command starts the inversion program for the station whose window is currently active.

The main steps involved in the inversion process are:

Set curves to use for inversion	The inversion can be run using the edited or smoothed curve. The curve selection button, located on the bottom bar, allows the user to change the selected curve at any time. The models must be recalculated when the curve to be used is changed.
Calculate the smooth inversion model	For each station, one smooth model is calculated and stored in the database. This model will be used to produce imaged sections for the project.
Guess the layered inversion model from the smooth model	A layered model can be guessed from the calculated smooth models. The number of layers is entered by the user by editing the inversion settings or by editing the model with interactive graphic commands.
Edit the initial model using the graphic model editor (optional)	An interactive graphic editor allows the editing of layers thicknesses and resistivities. The response is displayed in real time.
Run the inversion and save the model	The inversion process can be run with a given number of iterations and predefined RMS fitting degree. Resistivity and/or thickness values of one or more layers can be kept fixed.

The 1D Inversion Window

The following figure shows the 1D inversion window:



Curve Selection

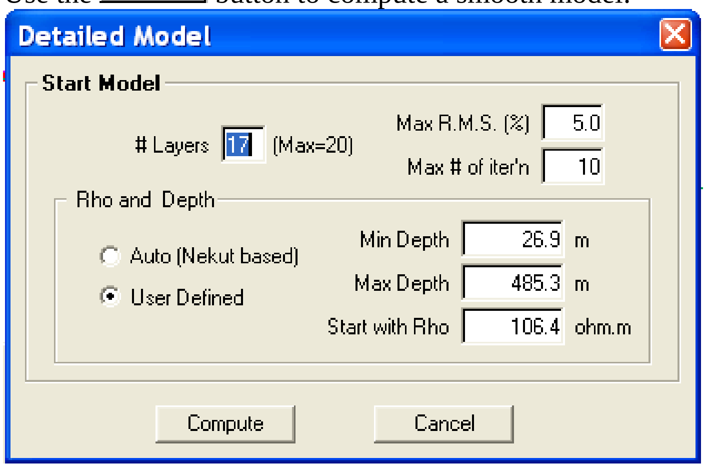
1. Use the **1D Model** command to enter the inversion mode for the active sounding.

2. Use the  button, located at the bottom of the form, to display the apparent resistivity or voltage curve.

To Calculate Smooth Models

1. Use the **1D Model** command to enter inversion mode for the active sounding.

2. Use the  button to compute a smooth model:



To Guess the Layered Model from the Smooth Model:

1. Use the **1D Model** command to enter the inversion mode for the active sounding.

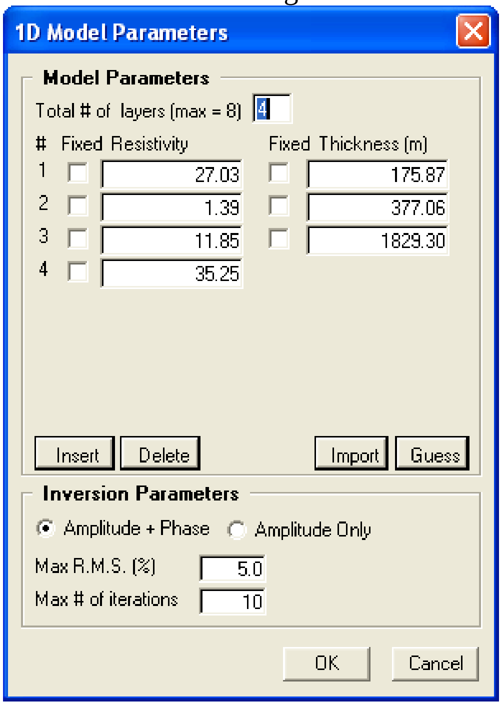
2. Calculate the smooth model (magenta lines), if you have not already done so.

3. Click the  button.

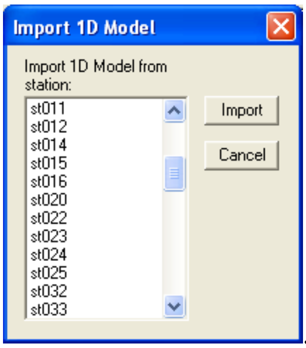
4. The layered model is guessed (green lines).

To Import a Model Created for a Different Station

1. In 1D Modeling mode, click the **Edit** button to open the 1D Model Parameters dialog:



2. Here, click the **Import** button to open a dialog box which contains all available stations:



.

3. Select the desired station and click Import. Note that any changes to the imported model are *not* reflected in the original model.

To Run the Layered Inversion:

1. Use the **1D Model** command to enter the inversion mode for the active sounding.
2. Click the 

 button.
3. The inversion is started using the model shown with green lines as the starting model and the inversion parameters set for the sounding.

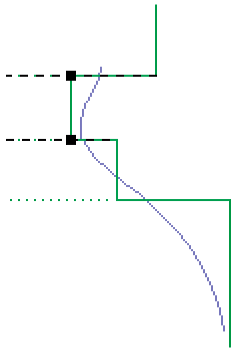
Graphic Editor for 1D Layered Models

The layered model can be edited graphically using drag-and-drop operations.

The resulting resistivity curve is updated in real-time, allowing you to compare the fit with the observed curve.

To Graphically Change the Resistivity of a Layer:

1. Click the vertical line which denotes the layer's resistivity. Two black dots appear on its edges indicating that this line can be moved:



2. Release the mouse.
3. Move the mouse pointer to the selected line; the arrow pointer changes to a double-headed arrow.
4. Left click and hold down the button; drag the line to the new layer resistivity delimiter position.
5. Release the mouse button. The resistivity curve coming from the layered model is recalculated and redrawn.

To Graphically Change a Layer's Thickness

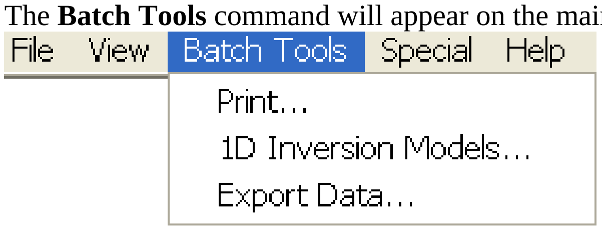
1. Repeat the procedure used to change a layer's resistivity, but instead select one of the horizontal lines which delimits the layer and move it to the desired position.

Batch Commands & Quick View

Running Operations in Batch Mode

A group of batch procedures are available for performing selected operations on a set of user-selected stations. To start a batch procedure:

1. Close all open stations.
2. The **Batch Tools** command will appear on the main menu:



3. Select the desired operation.

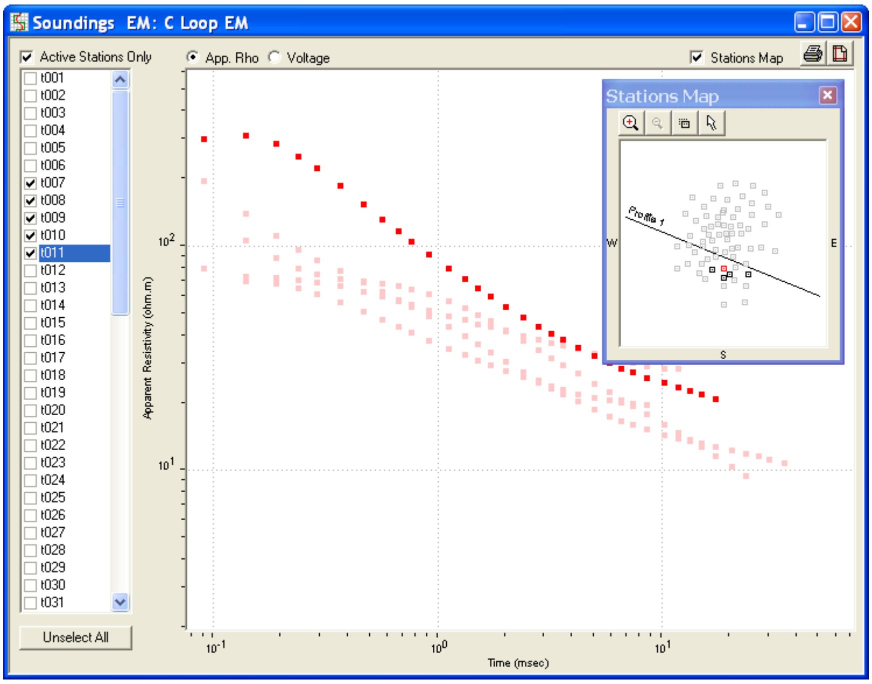
4. Enter the parameters as requested.

Operations available in batch mode include:

- Printing
- 1D inversion modeling
- Data export; batch window includes the 1D model worksheet function.

Quick View

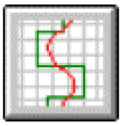
The Quick View option, which is available on the **Special** menu item when no soundings windows are open, can be used to display multiple soundings in a single window. To open the Quick View window, select the Special | Quick View menu command:



The Stations Map window, which can be toggled on and off with the corresponding check box in the upper right corner of the window, indicates the positions of the stations which are currently open (black squares) as well as the position of the active station (red square).

21: DC Soundings

Overview



The Soundings icon is displayed in the program menu whenever a sounding project is selected.

When a sounding project (MT, TEM, DC, etc.) is selected and the program launched, the stations of the selected project are loaded together with their original datasets.

The discussion in this chapter is limited to the DC Soundings program.

For each station, this program can be used to:

- Display sounding curves as resistivity vs. AB/2
- Edit sounding data
- Compute a 1D smooth and layered inversion model
- Import sounding data

As described at the end of this chapter, many of the functions available for individual soundings can also be performed in batch mode on any number of soundings contained in the current project.

About Editing

The results of any editing on sounding data will be saved in the *"edited data"* section of the database. Original data will not be overwritten and can always be restored using the menu command: **File | Reload Original Data**.

The edited data saved by the Soundings program are used by other WinGLink programs:

- Smooth and layered 1D models are used to calculate **imaged sections** and **cross sections,** respectively.
- Curves and 1D models are also used to **extract parameter maps** by the **Maps** program.

Sounding Data Which are Saved in the Database

For each station of the current project, the following datasets are saved in a WinGLink database:

Original Data

Apparent resistivity as imported from external files. Original curves can always be recovered to restart the editing process using the Soundings menu command **File | Reload Originals**.

Edited Data

Edited curves are initially constructed by assigning to each datapoint the same value of the original curves. At the end of each editing session, the edited values are saved in the database, separately from the original values.

Smooth 1D Model

One smooth model is saved for each sounding. This provides a preliminary guess model for the layered inversion routine and is also used to produce imaged sections along profiles.

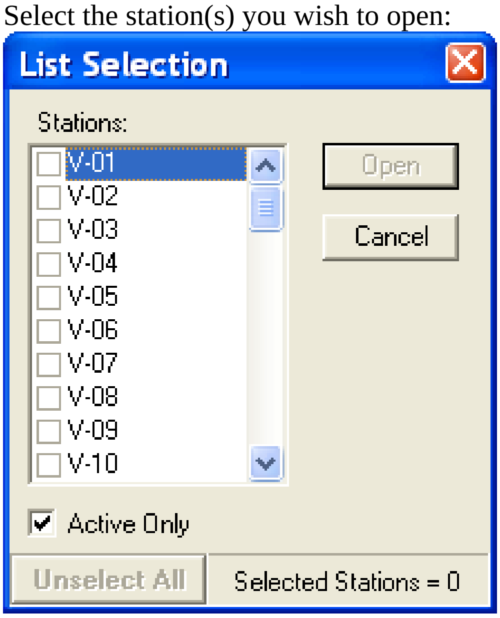
Layered 1D Model

One layered model is saved for each sounding. These models are displayed along profiles by the *Cross Sections* program. The 1D curve used to calculate the layered model is the same curve used for the smooth model.

Opening, Saving and Printing Sounding Data

To Open Soundings from the Station List

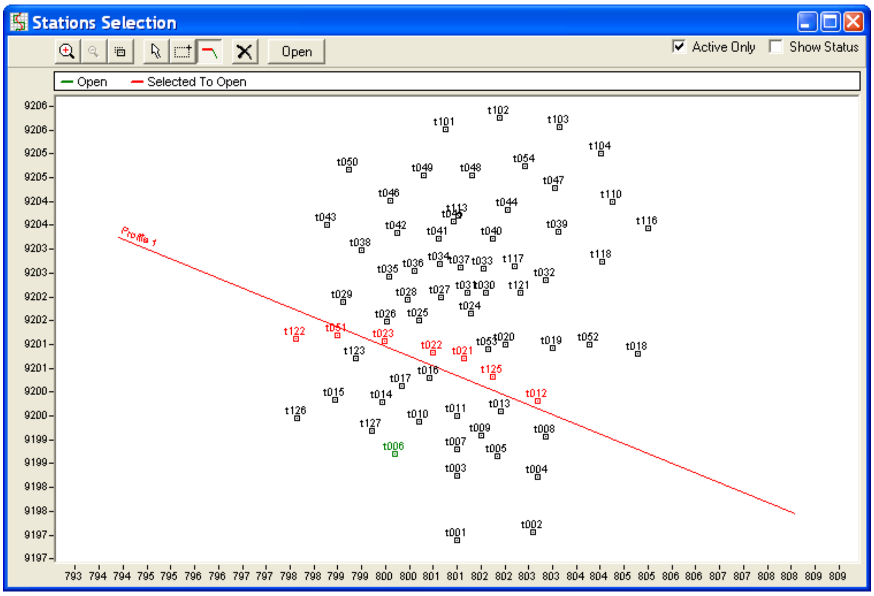
1. Select the **File | Open from List** menu command.
2. Select the station(s) you wish to open:



3. If you want to display only the active stations, select the **Active Only** check box.
4. Click the **Open** button.

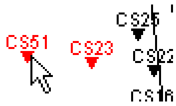
To Open Soundings on the Station Map

Select the **File|Open from Map** menu command; a simplified location map for the project station is displayed:



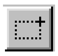
Open Soundings by Clicking Individual Stations:

1. Click the  button.
2. Click the stations you wish to open; the selected stations turn to red:

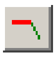


3. Click the  button.

Open All Soundings Within a Given Area:

1. Click the  button.
2. Left-click the mouse on a corner of the area of interest and drag it to define a rectangle on the map. When releasing the mouse button, all stations in the rectangle will turn to red.
3. Click the  button.

Open All Soundings Associated with a Profile Trace:

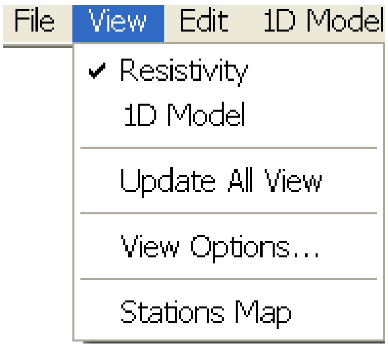
1. Click the  button.
2. Left-click the mouse on the profile or on a segment of the profile trace of interest. All stations previously added to that profile trace will turn red, indicating selection.

3. Click the  button.

Display Options

Each window can be set to show the resistivity/voltage curves or the 1D model for each sounding.

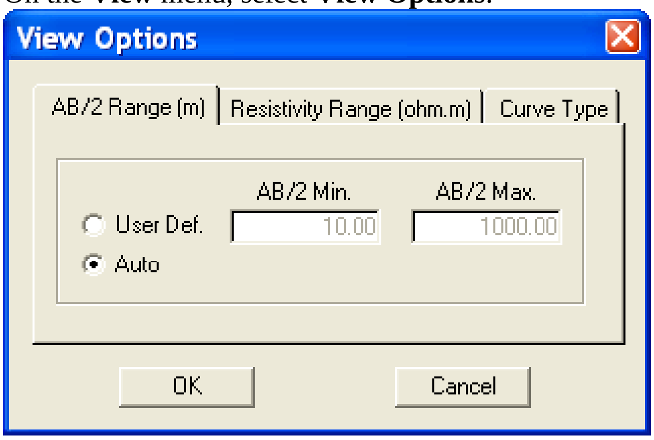
On the **View** menu, select from the available display options:



Setting Diagram Ranges and Scale

To Set the Ranges and Scale for the Plot Diagrams:

1. On the **View** menu, select **View Options**:



2. For each parameter, set the automatic range option or enter the max. and min. values to be plotted. The axis scale can be set to linear or logarithmic for many plots.
3. The settings made here apply to all window.

Printing Curves and 1D Models

To Print Station Curves

1. Select the station by clicking on its window;
2. Select the **File | Print** menu command.
3. Select the printing options and click the **Print** button.

Importing Sounding Data

In general, sounding data are imported into WinGLink using the import functionality provided in the WinGLink shell (see the “Data Import” chapter of this manual for details).The DC Soundings module includes an option, however, for importing new data into an existing sounding.

Import functions are available for:

- Text files
- Keyboard entry

Note: This function cannot be used to import new stations into the current project.

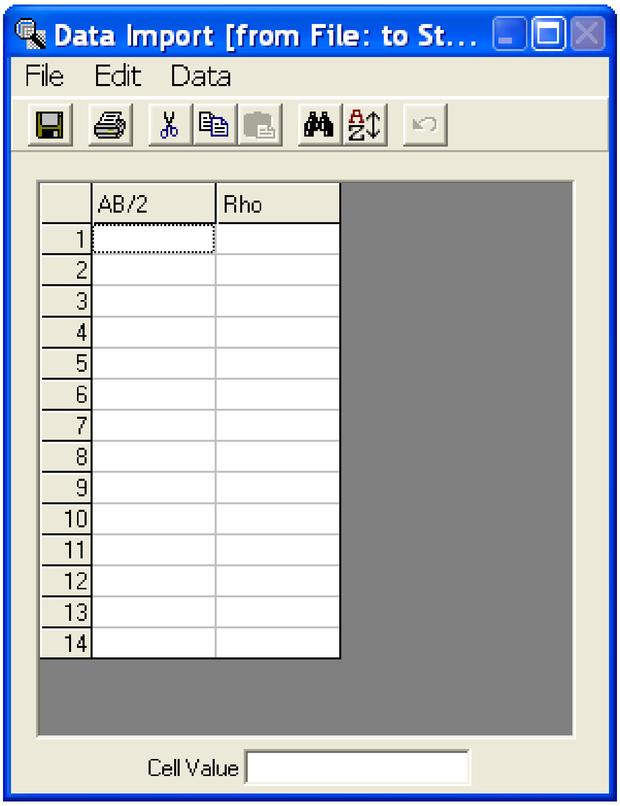
To Import Datasets from Text Files:

1. Open the sounding into which the data are to be imported
2. Select the **File | Import new data | From Text File…**menu command
3. Follow the instructions given by the Import Wizard (refer to the “Data Import” chapter of this manual for details.)

Note: Any previously saved sounding data are lost when new data are imported.

To Enter Data From the Keyboard:

1. Open the sounding into which the data are to be imported
2. Select the **File | Import new data | From Keyboard…**menu command
3. The program alerts you that any existing data will be lost.
4. In the Data Import window, enter the data in the appropriate columns:|



When finished, use the **File | Save** menu command to store the data in the database

Editing Sounding Data

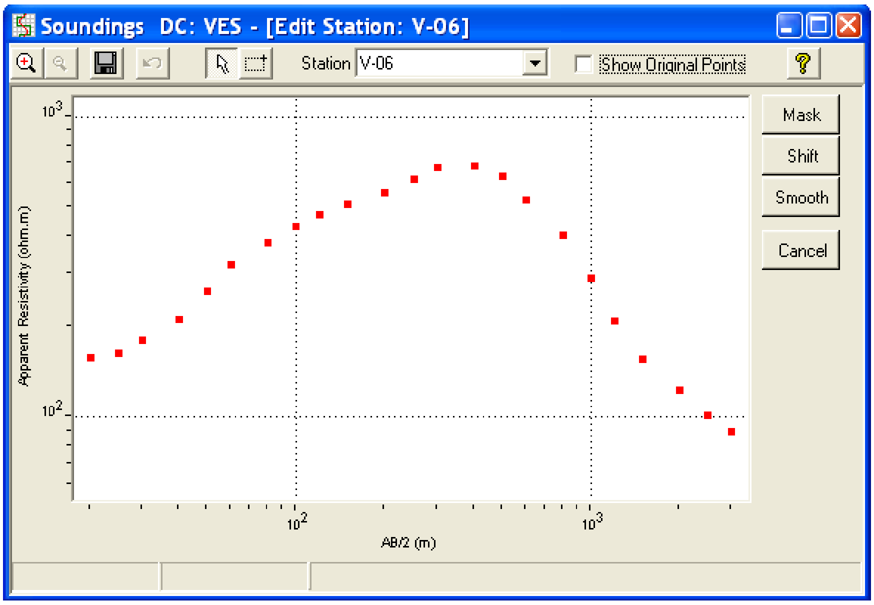
The Edited Curve

When sounding data are imported into a WinGLink database, they are saved as "original data" in a reserved area of the database and are never modified unless new data are imported. At the same time, a duplicate dataset is generated and saved in a different area of the database for use by editing functions. When an editing session is carried out on the sounding data, it is carried out on this second dataset, which is referred to as the "edited curve" of the sounding.

Saving changes following an editing session means saving an edited curve. This curve is used by default in calculating 1D inversion models and pseudosections.

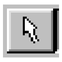
Interactive Sounding Editing Form

DC sounding data can be edited in the interactive sounding editing form, which is opened with the **Edit** menu command:



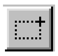
The form includes commands for performing various editing functions:

Selecting Data Points



Single point selection

Click to select/unselect for editing single data points on the curve.



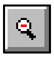
Multiple points selection

Click to select/unselect for editing all data points included in a user-defined range: click and drag the mouse to define the area.

Zooming in on the Curve

Zoom in

1. Click a point in the curve diagram and release the button.
2. Move the mouse to define the zoom area.
3. Click and release the left mouse button.

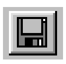


Zoom out

Click to reset the previous view.

Note: this button is available only after the **Zoom in** button has been used. It remains available until the original curve size is restored.

Saving Editing Changes



Save button

Saves the editing changes. These changes must be confirmed when closing the station window in the main menu in order to store them in the database.

Editing Data Points

Masking data

1. Click and select the points by single or multiple selection (the masked points will turn gray).
2. Click the **Mask** button again to confirm the changes.
3. The points will be hidden from the display
4. Click the **Mask** button again to restart the masking/unmasking process.

Shift data points

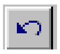
1. Click and select the data points to be shifted by single or multiple selection.
2. Move the data points to the desired positions and click the left mouse button.
3. Click the **Shift** button again to confirm the action.

s

Calculate and display a smoothed curve

Click the button to display the smoothing options

Cancel the current action



Undo button

Undo all the editing changes made since the last time the curve was saved. All editing steps can be undone provided the station has not yet been saved since editing.

Reloading Original Curves

This action overwrites the edited curves of a station with the original curves.

To reload original curves:

1. Select the station by clicking its window.
2. Select the **File |Re-Load Original** command.

After reloading, the smoothed curves and the 1D models need to be recalculated.

1D Inversion

1D Inversion: Overview

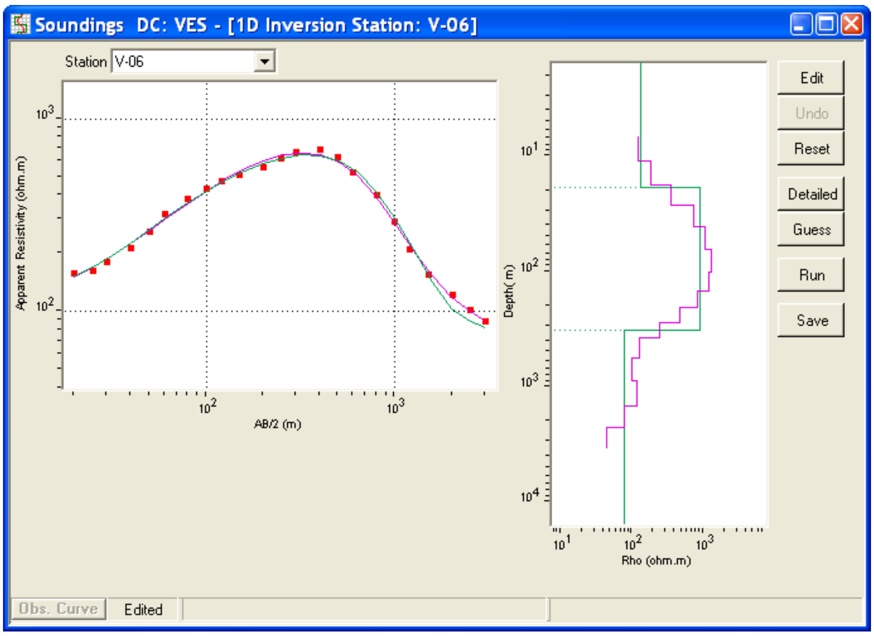
The 1D Model command starts the inversion program for the station whose window is currently active.

The main steps involved in the inversion process are:

Set curves to use for inversion	The inversion can be run using the edited or smoothed curve. The curve selection button, located on the bottom bar, allows the user to change the selected curve at any time. The models must be recalculated when the curve to be used is changed.
Calculate the smooth inversion model	For each station, one smooth model is calculated and stored in the database. This model will be used to produce imaged sections for the project.
Guess the layered inversion model from the smooth model	A layered model can be guessed from the calculated smooth models. The number of layers is entered by the user by editing the inversion settings or by editing the model with interactive graphic commands.
Edit the initial model using the graphic model editor (optional)	An interactive graphic editor allows the editing of layers thicknesses and resistivities. The response is displayed in real time.
Run the inversion and save the model	The inversion process can be run with a given number of iterations and predefined RMS fitting degree. Resistivity and/or thickness values of one or more layers can be kept fixed.

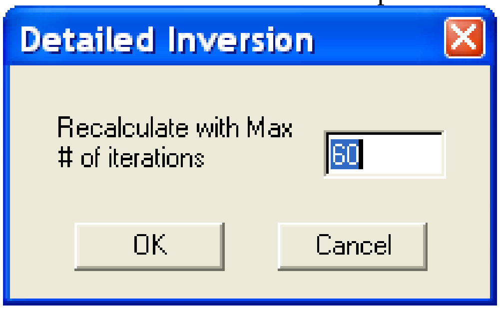
The 1D Inversion Window

The following figure shows the 1D inversion window:



To Calculate Smooth Models

1. Use the **1D Model** command to enter inversion mode for the active sounding.
2. Use the  button to compute a smooth model:

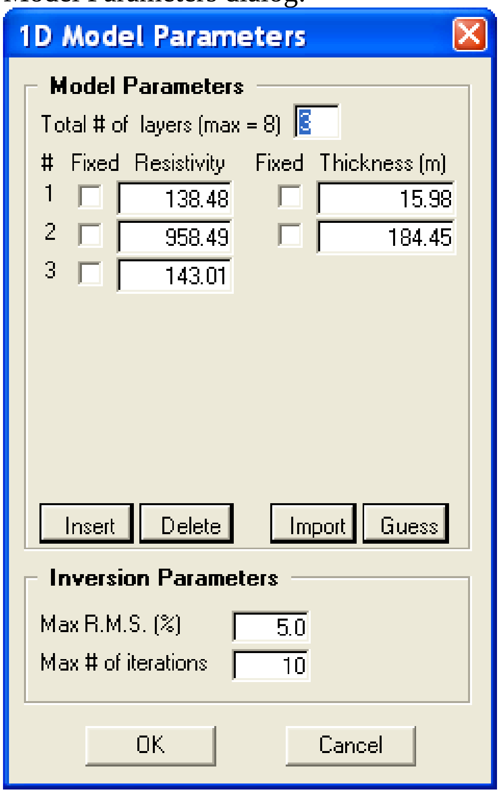


To Guess the Layered Model from the Smooth Model:

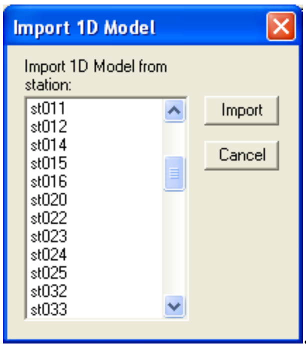
1. Use the **1D Model** command to enter the inversion mode for the active sounding.
2. Calculate the smooth model (magenta lines), if you have not already done so.
3. Click the  button.
4. The layered model is guessed (green lines).

To Import a Model Created for a Different Station

1. In 1D Modeling mode, click the **Edit** button to open the 1D Model Parameters dialog:



2. Here, click the **Import** button to open a dialog box which contains all available stations:



3. Select the desired station and click Import. Note that any changes to the imported model are *not* reflected in the original model.

To Run the Layered Inversion:

1. Use the **1D Model** command to enter the inversion mode for the active sounding.
2. Click the  button.

3. The inversion is started using the model shown with green lines as the starting model and the inversion parameters set for the sounding.

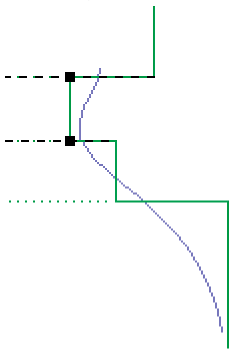
Graphic Editor for 1D Layered Models

The layered model can be edited graphically using drag-and-drop operations.

The resulting resistivity curve is updated in real-time, allowing you to compare the fit with the observed curve.

To Graphically Change the Resistivity of a Layer:

1. Click the vertical line which denotes the layer's resistivity. Two black dots appear on its edges indicating that this line can be moved:



2. Release the mouse.
3. Move the mouse pointer to the selected line; the arrow pointer changes to a double-headed arrow.

4. Left click and hold down the button; drag the line to the new layer resistivity delimiter position.
5. Release the mouse button. The resistivity curve coming from the layered model is recalculated and redrawn.

To Graphically Change a Layer's Thickness

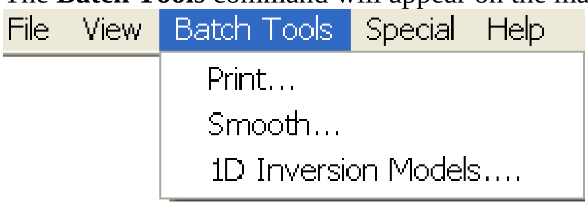
1. Repeat the procedure used to change a layer's resistivity, but instead select one of the horizontal lines which delimits the layer and move it to the desired position.

Batch Commands & Quick View

Running Operations in Batch Mode

A group of batch procedures are available for performing selected operations on a set of user-selected stations. To start a batch procedure:

1. Close all open stations.
2. The **Batch Tools** command will appear on the main menu:



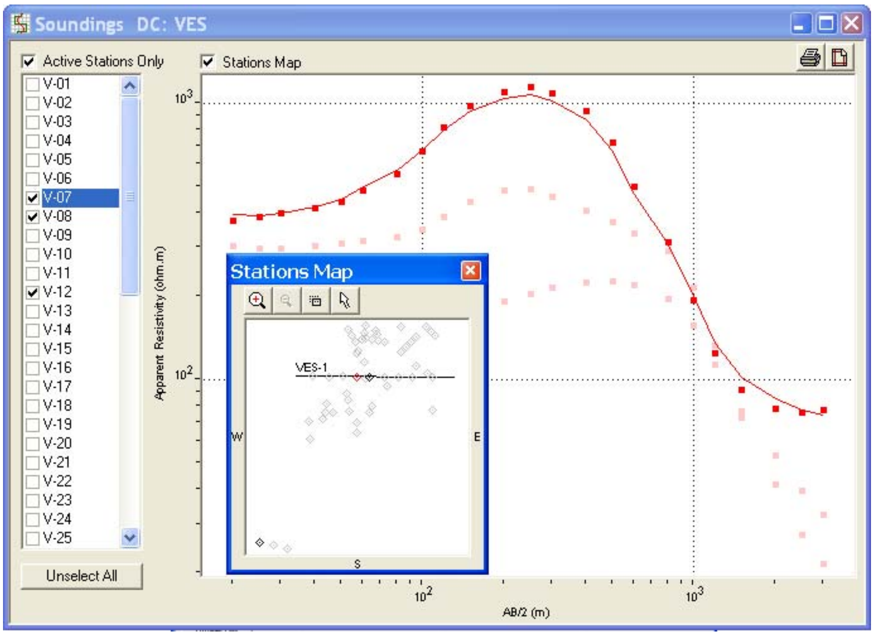
3. Select the desired operation.
4. Enter the parameters as requested.

Operations available in batch mode include:

- Printing
- Smooth curve calculation
- 1D inversion models

Quick View

The Quick View option, which is available on the **Special** menu item when no soundings windows are open, can be used to display multiple soundings in a single window. To open the Quick View window, select the Special | Quick View menu command:



The Stations Map window, which can be toggled on and off with the corresponding check box in the upper right corner of the window, indicates the positions of the stations which are currently open (black squares) as well as the position of the active station (red square).

22: EM/DC Pseudo-Sections

Overview

The Pseudo-Sections program reads the apparent resistivity vs. measuring parameter values for each station associated with a profile, then interpolates them to display a section showing the lateral variation along the profile.

Each pseudo-section is defined by a profile trace and its associated stations, which supply the datasets with the apparent resistivity values vs. the measured parameter (i.e.: AB/2, time, or frequency, depending on the data type of the project).

With the Pseudo-Sections program, multiple sections can be open at any time. Pseudo-sections may also be opened along multiple profiles. Each section is displayed in a separate window.

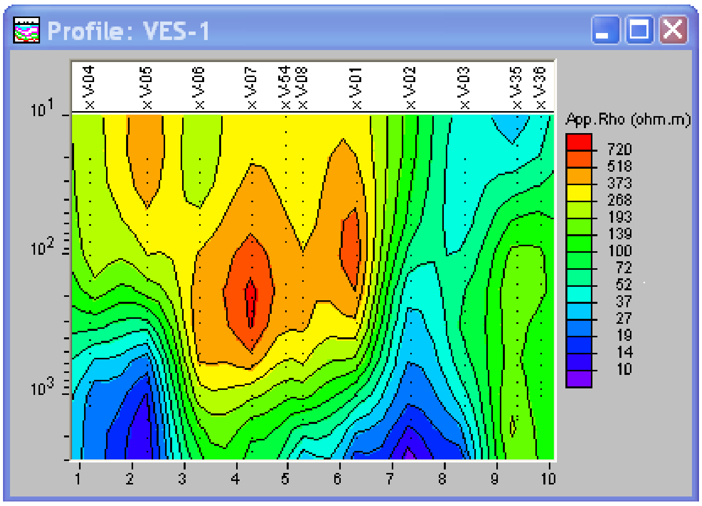
The curves used to generate the pseudo-sections can be viewed and edited, one at a time, by clicking the vertical dotted line below each station on the section.

Creating a New EM or DC Pseudo-Section

To create a **new** EM or DC pseudo-section:

1. Construct a new profile, as described in the “Maps” chapter of this manual.
2. Make sure you have imported or entered the field data for each station of the new section. You can use the **Soundings** program to check that the stations have data. The Soundings program will use the edited data, if available, or the smoothed data, if requested. Alternatively, the original field data can be recalled.
3. Start the **Pseudo-Sections** program.
4. In the Profile Selections window, which opens immediately after starting the program, select the profile created in the Maps program for which the section is to be created.
5. On the **Gridding** menu, select **New Grid** and enter the appropriate parameters.

6. The section is displayed. The section can now be edited, printed and stored in the database for future use:

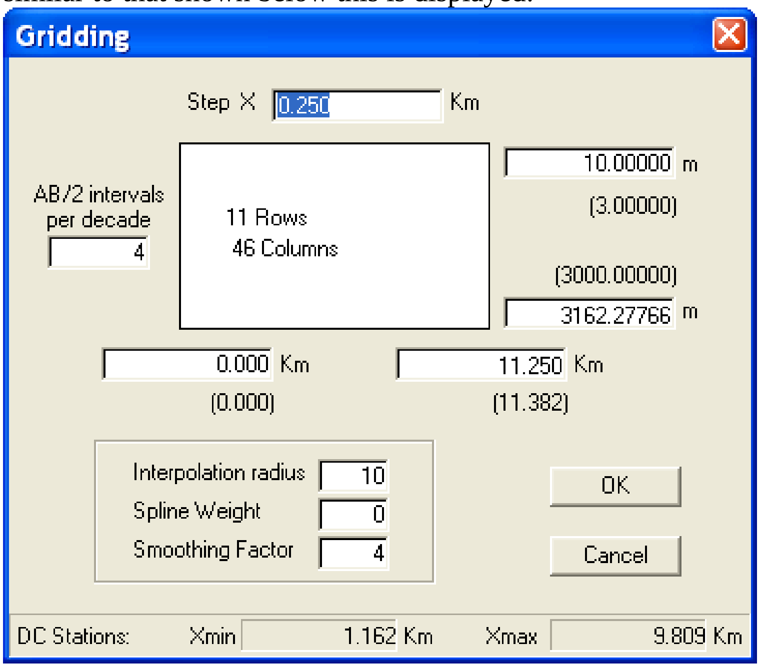


Gridding the Profile Data

The data of the stations used to generate each pseudo-section must be gridded in order to display contours and color ranges.

To Grid the Profile Station Data

1. Select the section you want to grid.
2. Select the **Gridding | New Grid…** menu command. A form similar to that shown below this is displayed:



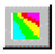
3. The numbers in parenthesis on the gray background show the maximum and minimum values of the data; the current value to be used to define the grid extent must be entered in the field-box close to each number.

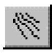
4. In the **Step X** field, enter the step in the specified units. This number determines the number of columns in the grid, which will be shown in the center white box, representing the section.

5. In the **T intervals per decade** field, enter the number of grid rows per decade.
6. Fill the fields with the values for the interpolation radius, spline weight and smoothing factor.
7. Click the **OK** button.

Note: As the interpolation radius increases (values >5), the gridding algorithm attempts to fill any holes present in the data by interpolating between relatively distant data points. When creating difference pseudo-sections (see below), we therefore recommend using a low interpolation radius (2) to prevent the creation of misleading differences.

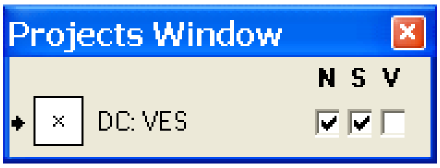
To Display Color Ranges and Contour Lines

Click the icon  to display/hide color ranges.

Click the icon  to display/hide contour lines.

Displaying Station Dataset Values

On the **View** menu, select the **Projects Window**, if not already displayed:



Check the **V** box to display the values of the dataset at each station.

To update the display of the section, click the section with the mouse.

Editing Data Points

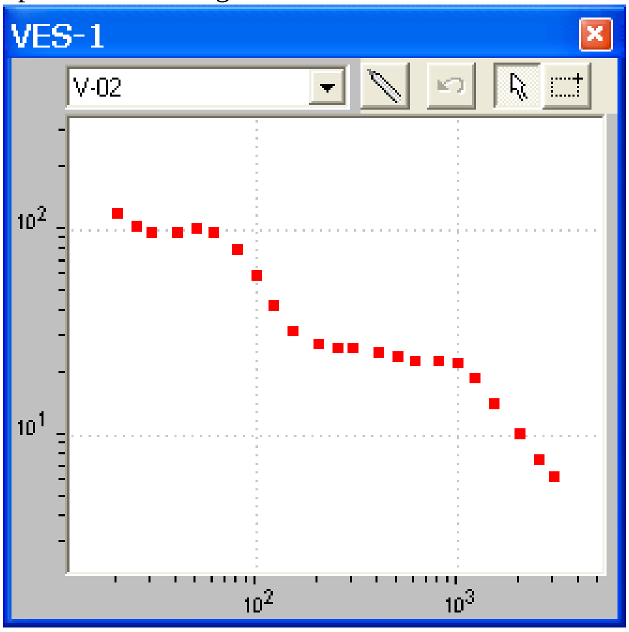
A number of edit operations can be performed on the data associated with each station. These include masking samples, static shift and station state.

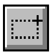
Edits made here affect the stations elsewhere in the database. Thus, a station disabled in the Pseudo-Sections program is not available in the MT 2D Inversion program.

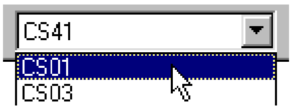
To Edit Data Points:

1. Double-click the dataset of the sounding you wish to edit in the section (the data are shown as dots below each station). This

opens the sounding window:



2. To enter edit mode, click the  button. Depending on whether the  button or the  is enabled, data points can be masked either individually or as groups.
3. Click the  button again to terminate masking.
4. The masking operation just performed can be undone by clicking the  button.
5. Other stations can be selected for editing from the dropdown list at the top of the editing window.



To Change the State of a Station

1. Select the **Station | Edit Status** menu command.
2. Click the station symbols to toggle the current state. When disabled, the stations are displayed in light gray.

Station states may only be altered if the current project is a single project, i.e. this function cannot be applied to stations contained in an integrated project.

Note: In order to see the effects of changes or edits made to the stations on the section, you must select the **Gridding | Regrid** menu command.

23: X-Sections

X Sections

Overview

The X-Sections program provides a tool for displaying multiple sections for a specific type of geophysical data side-by-side. Using this program, it is possible, for example, to display in a single application sections created for 1D, 2D and 3D models on separate (or the same) profiles.

In order to use this program for a given project, the project must contain a profile and there must be stations associated with that profile. Sections can only be generated if the corresponding data have been prepared using the appropriate modules in WinGLink, e.g. Soundings, MT 2D Inversion, etc. For further information on creating profiles, please refer to the “Maps” chapter of this manual.

Well courses and layer data belonging to stations assigned to the active profile can be displayed in a section, regardless of project type.

Before starting, note the data sources for the various section types:

- 1D models: these can be created from DC, EM or MT projects. Before creating sections from 1D models, make sure you have used the respective **Soundings** program to calculate the smooth and layered resistivity models.

  For MT projects, sections can be created using Bostick and Occam 1D models. Layered 1D models can be superimposed upon both of these. Note that sections cannot be created for layered 1D MT models.

  2D models: these can be created from MT or CS projects. The models used in creating this type of section are those created and used in the 2D Inversion programs. Sections can be extracted from both standard terrestrial 2D models as well as from marine models.

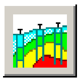
- 3D models: these models can be created from MT projects which contain 3D meshes. X-Sections generates sections of this type by interpolating along a profile which transects an area of a 3D mesh. Before a section of this type can be created, it is necessary to first create one or more profiles in the MT project which contains the 3D mesh. As sections produced for 3D meshes are generated by interpolating cell values along the profile using the resistivity values stored in the mesh – and not station values, it is not necessary to assign stations to this profile.

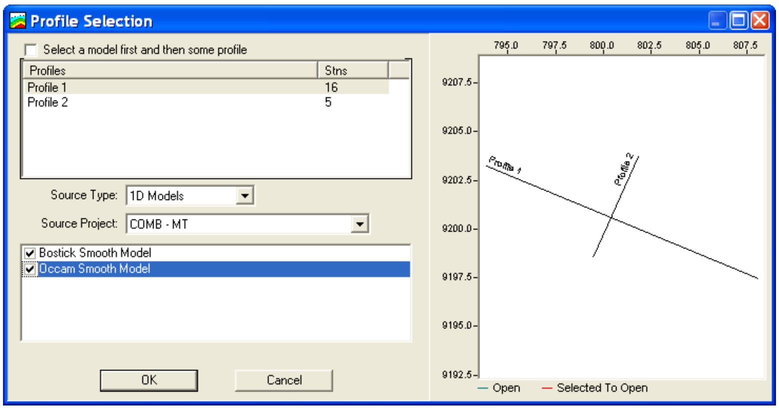
- Wells: this type of section is created using the layer data imported for each well station. It is possible, for example, to import layer data containing temperature or pressure at various depths along a well course. X-Sections interpolates these values to generate sections. In addition to creating sections using layer data in the form of numerical values, X-Sections can also display well courses and lithologic information, displayed as user-defined fill patterns along the well courses.
- Vertical data: values measured at depth not associated with well courses can be used to create sections provided the stations have been assigned to a profile.

Creating a New X-Section

Described in the following is the general procedure for creating sections of all project types in the X-Section program. It is assumed here that a profile has been created and that stations which the corresponding data have been associated with the profiles and that models have been prepared.

To Create a New Section:

1. Select the desired project in the WinGLink Shell and start the **X Sections** program by clicking the  icon in the program window.
2. The first window displayed by X-Sections after the program has been started is the Profile Selection form:



Listed in the Profiles section of the window, in the upper left corner, are the profiles available for the selected project and the number of stations assigned to each.

Below this, for projects of type DC, EM or MT, is a drop-down list containing the model types available for the given project type, i.e. 1D, 2D and 3D. In the window below the drop-down are the models available in the database for the selected type on the profile selected for opening.

For projects of type WL (wells), this drop-down list lists ***all*** types of numerical data which have been imported for the all well stations in the database, i.e. all types of numerical data in all well projects. Note that sections can be created using numerical values only if data exist for at least two stations on the selected profile.

When creating sections extracted from 3D models (MT), the **Source Project** drop-down list in the Profile Selection dialog lists all projects in the database. Select any of the projects to display the 3D models contained in the given project. As there is no direct relationship in a WinGLink database between projects and 3D models, sections extracted from 3D models can be opened even if the 3D model is not present in the current project. For informative purposes, however, 3D models present in the active project are indicated by an asterisk. If the model is present in a different project, the name of that project listed next to the 3D model name. If the 3D model is not present in any project in the database, there is no additional information displayed.

Sections can be extracted along profiles on 3D models not containing any stations.

Make the desired selections and proceed to the X-Section workspace by clicking the **OK** button.

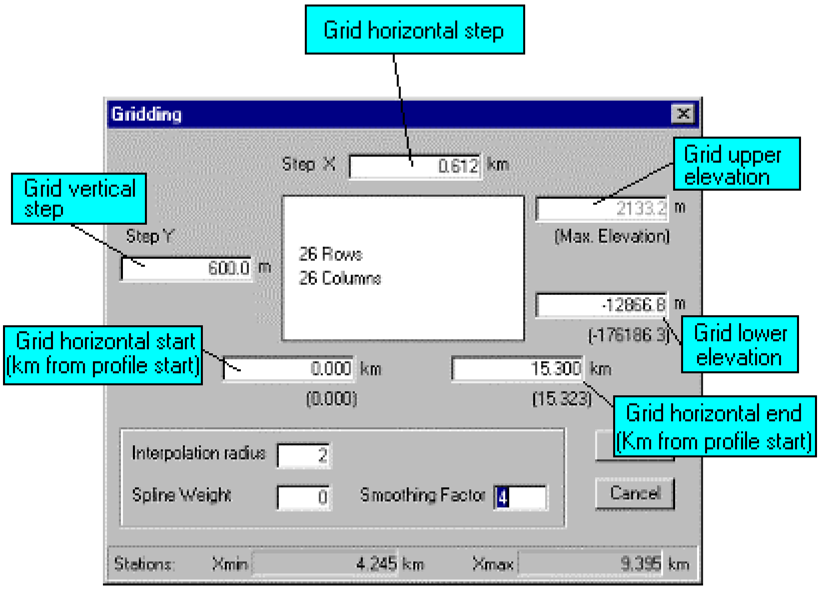
3. The section is displayed with both smooth and layered models, if available.
4. If viewing a section for which no grid has yet been created, a window is displayed prompting you to select the gridding options.

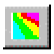
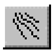
Gridding the Station Datasets

Each time a modification is made to the station’s data or status, the section must be re-gridded to update the display and show the latest changes.

To Grid the Station Data:

1. Select the window containing the section.
2. Select the **Gridding | New Grid…**menu command
3. Enter the grid boundaries and steps:



4. Enter the interpolation radius, spline weight and smoothing factor.
5. Click the **OK** button.
6. Click the  button to display color ranges
7. Click the  button for contour lines.

To Export a Grid

To export a grid created in the X-Sections program, select the **File | Export Grid** menu command. Grids can be exported in most standard grid formats, including Geolink USGS, Surfer and others.

Activating/Deactivating Stations

The data of any station on the section profile can be excluded from the processing by the user.

This is done by editing the status of the station. To do this, select the **Stations** | **Edit Status menu** command, then click the station symbol on the section. By clicking the symbol on the station, the user can change its status from **On** or **Off** and vice versa.

When a station is disabled, its name and symbol are displayed in gray, no data points are shown below the station on the section, and the station data are not used when a new grid is calculated.

Note: This function is available in the Pseudo-Sections and X-Sections programs. Stations enabled /disabled in any Sections module are enabled/disabled in all section programs, including 2D Inversion.

Station states may only be altered if the current project is a single project, i.e. this function cannot be applied to stations contained in an integrated project.

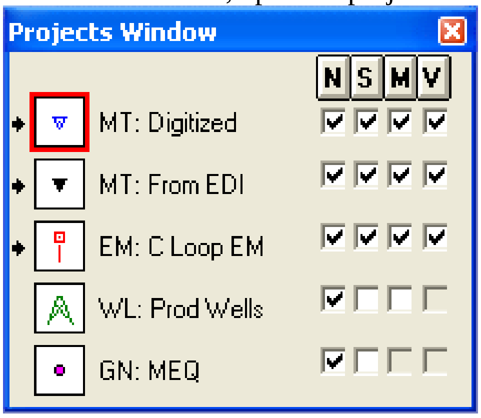
To toggle the enabled / disabled status of a station:

1. Select **Stations | Edit Status**.

2. Click the symbol of the station(s) you wish to disable (if not shown in gray) or re-enable (if shown in gray).
3. When finished, select **Gridding | Regrid** to update the display according to the new station settings on the section.

Displaying Features and Parameter Values

On the **View** menu, open the projects window if not already visible:



Check the corresponding box to display stations

N = Names

S = Symbols

M = Layered models

V = Parameter values

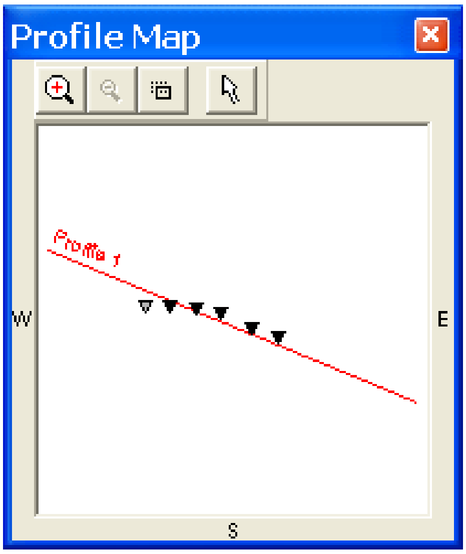
The buttons located at the top of each column can be used to toggle all station values for the respective column on and off. After clicking a button, a dialog window appears prompting you to specify whether the respective value should be displayed or hidden.

To update the display of the section after selecting or deselecting checkboxes, click the section with the mouse.

Displaying the Profile Location Map

A window can be opened in the X-Sections program (or any section program) which contains a map with the locations of all profiles and the stations associated with the profile in the current project. The active profile (on which the section currently being edited is located) is displayed in red. The locations of stations associated with the active profile are also shown.

To open the ancillary location map window, select the **View | Profile Map** menu command:



Profile Intersections

The points at which profiles intersect one another can be displayed on pseudo-sections with the **Profile Intersections** function, which is enabled using the **View | Crossing Traces** menu command. Profile intersections are indicated with red labels along the top of the section.

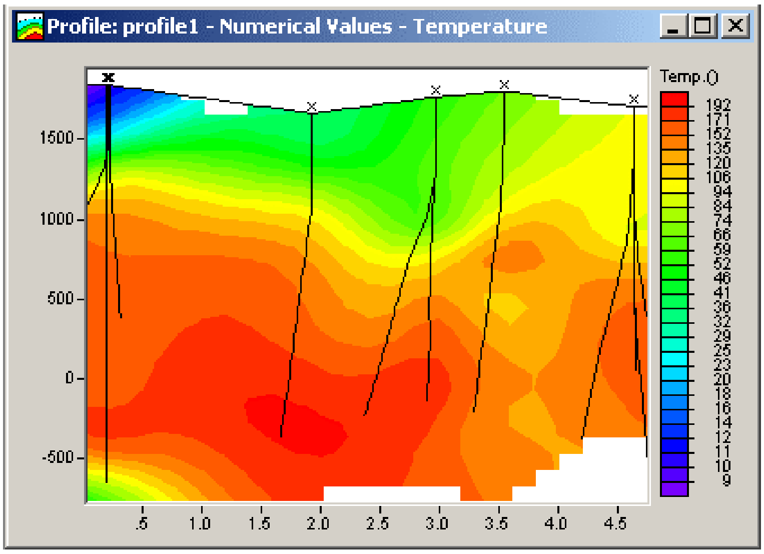
Section Topography

Section topography is extracted according to the following rules:

1. From the area topography grid as defined in the WinGLink shell.
2. From the project topography grid.
3. From the elevations of the stations attached to the profile.

Viewing Well Courses and Well Layer Data in X-Sections

Well courses and associated layer data can be displayed in section programs X-Sections and 2D Inversion. Any well station assigned to the active profile is automatically displayed when the section is opened:



Depending on program settings, the well courses may or may not be displayed initially.

In the X-Sections program, well traces and layer data can be toggled on and off using the Projects Window and the Well Layer Data window.

If layer data have been imported, e.g. lithologic data, you may use the Well Layer Data window to toggle the display of the layer data on and off. As is described in the “Common Functions” chapter of this manual, layer data can be superimposed onto well courses and represented as user-specified fill patterns using WinGLink’s Category Editor.

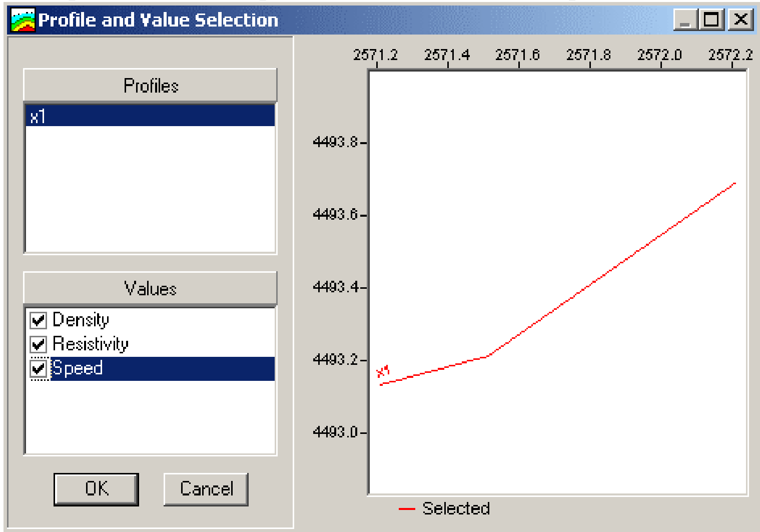
Working with Vertically Distributed Data in X-Sections

After importing vertically distributed data, as described in the “Data Import” chapter of this manual, and assigning the stations to a profile using the Maps program, X-Sections can be used to grid the values.

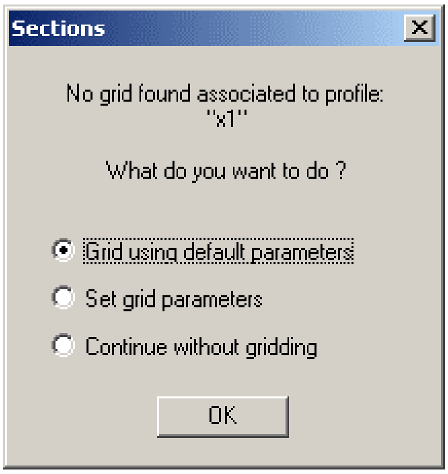
To Work with Vertically Distributed Data in X-Sections:

1. Select a project of type Vertical Data [**VD**].

2. Click the X-Sections program icon. The Profile selection dialog box opens. Select the profiles and values to be opened:



3. If the grids have not yet been created for the value you have selected, you will be prompted to select the type of gridding to be used:



4. Make the appropriate selection and click the **OK** button. Regardless of which option is chosen here, the grid parameters can be modified after the grid window has opened.

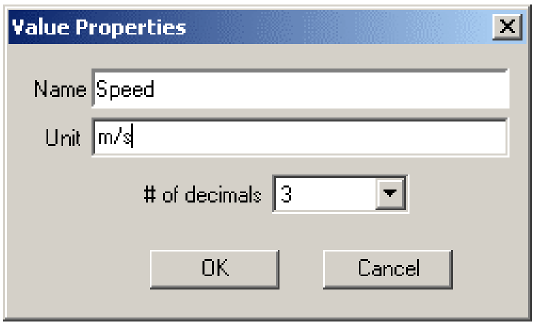
In addition to the functions available throughout WinGLink, e.g. color ranges and contours, and those generally available for sections, e.g. gridding, area selection etc, the X-Section program offers functionality for performing arithmetic operations on a given set of values to create a new value.

To Create a New Value:

1. Select the **File | New Value** menu command.
2. The New Value dialog box opens. Select the value to be used as the source from the drop down list, which contains all of the values available for the profile. Make the appropriate selections in the remaining fields and click the **OK** button.
3. To open a grid window for the new value, select the **File | Open** menu command to open the Profile and Value Selection

dialog box.

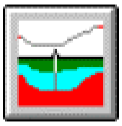
4 The properties for a new value or those for an existing value can be modified by selecting the **File | Value Properties** menu command.



Edit the **Unit** and **# of decimals** fields appropriately and click **OK**. The **Name** field cannot be modified.

24: Gravity and Magnetic 2.75D Modeling

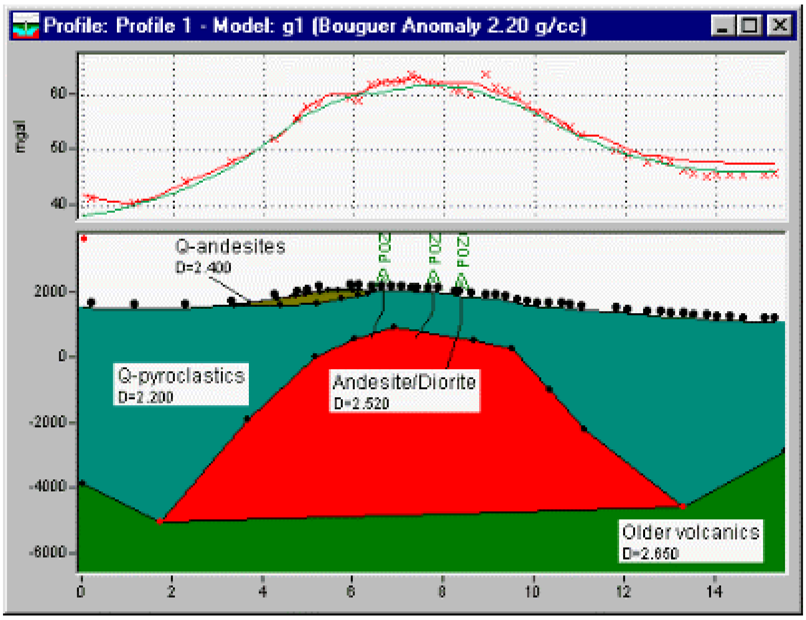
Overview



The Gravity and Magnetic Modeling program is used to create, edit and manage **2.75 D models** of **rock density** (gravity projects) or **rock susceptibility** (magnetic projects).

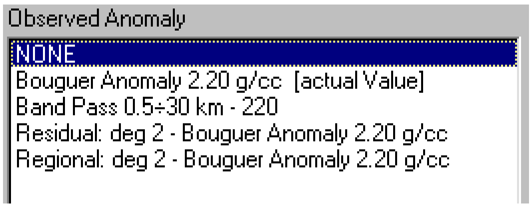
Each model is attached to a profile of the database. One or more models can be attached to each profile. The profiles must have been previously defined using the Maps program. The observed anomaly values to be fitted are automatically associated to each defined profile.

Several models can be displayed simultaneously, each one in a separate window. An example is shown below:



The upper frame shows the observed anomaly curve (red) and the calculated anomaly curve (green). The lower frame is used to display and interactively edit the model. As the editing goes on, the calculated curve is updated automatically in order to show in real time how the changes affect the calculated curve and check the fitting of the model.

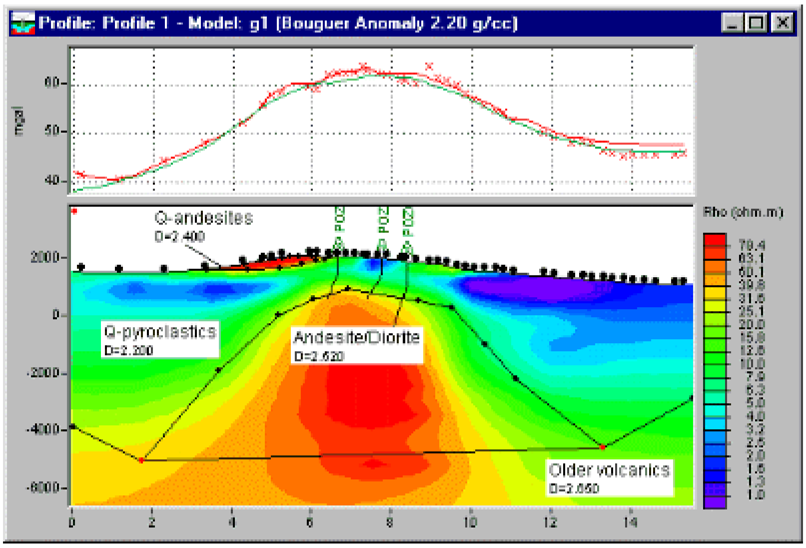
The values for the observed anomaly curve can be chosen from any of the values of the project stations. A selection window, which lists all values available for the selected projects, is available for value selection:



Choosing NONE will allow a forward modeling to be performed with no anomaly value.

Finally, it is possible to display a background section from any project (imaged or cross section), thus integrating gravity with any other information available in the area.

In the example shown below, the deep high resistivity complex is modeled as the source for the gravity high.



Observed Anomaly Values

The observed anomaly curve can be represented in two ways:

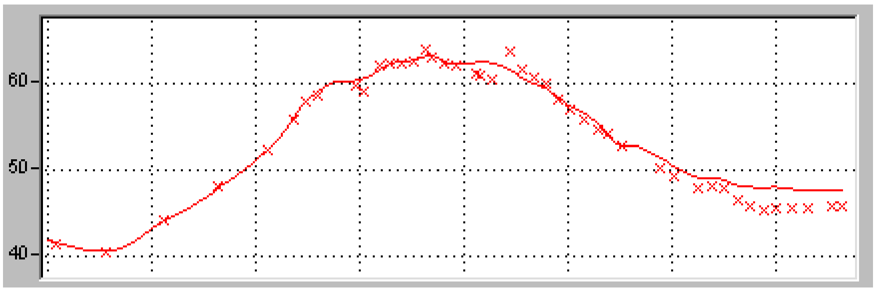
Anomaly Curve with Measured Values

This curve is obtained by using the observed value associated with each station of the profile.

For each model, the values used to construct the curve can be selected from the values available for the stations of the project. These can be raw, edited or processed values (i.e.: measured, filtered, etc.).

Anomaly Curve with Interpolated Values

This curve is obtained by extracting the values along the profile from a map grid, using a fixed sampling step. The resulting curve is a smooth curve that may show values in no-data areas, due to the interpolation process.



The above picture shows an example of observed anomaly curves, with both station values and interpolated values. The red crosses are the values of the stations attached to the profile.

The continuous red line is the line extracted from the map of the same values, sampling the grid at a regular step. The shape of the curve depends on the gridding parameters that were used in the **Maps** program. This curve will not be displayed if the selected station values have not been gridded in the **Maps** program.

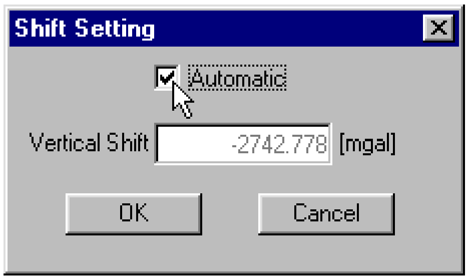
Selecting the Anomaly

This option lets you change the source of the observed curve.

1. Select the window containing the model.
2. Select the **File | Load Anomaly Values** menu command.
3. Select the desired value type.

The new observed anomaly curve is displayed. If you cannot see the calculated curve, do the following:

1. Select the **Tools | Set Vertical Shift** menu command.
2. Select the **Automatic** check box and click the **OK** button.

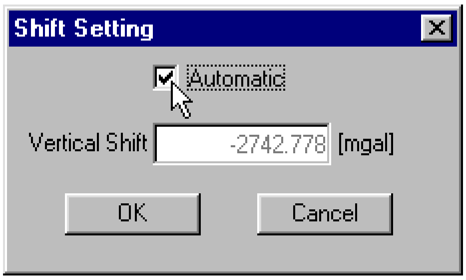


Changing the Vertical Shift

The vertical shift brings the calculated anomaly curve into the observed anomaly range by applying a constant value to the whole calculated anomaly curve.

1. Select the **Tools** | **Set Vertical Shift…**menu command

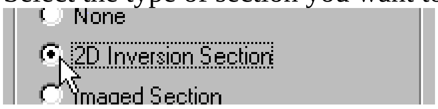
2. Select the Automatic check box or enter the value and click the **OK** button.



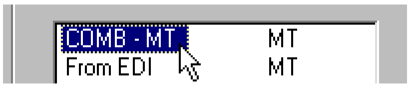
Displaying a Section in the Background

With this option, a gridded section from any other project of the database or bitmap image can be set as the background of the 2.5D gravity model.

1. Create or open a model for your profile.
2. Select the **Tools** | **Set Background** menu command.
3. Select the type of section you want to display:



4. Select the project from which the section is to be taken and click the **OK** button.

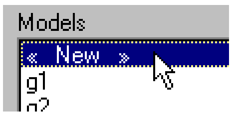


Creating and Defining Models

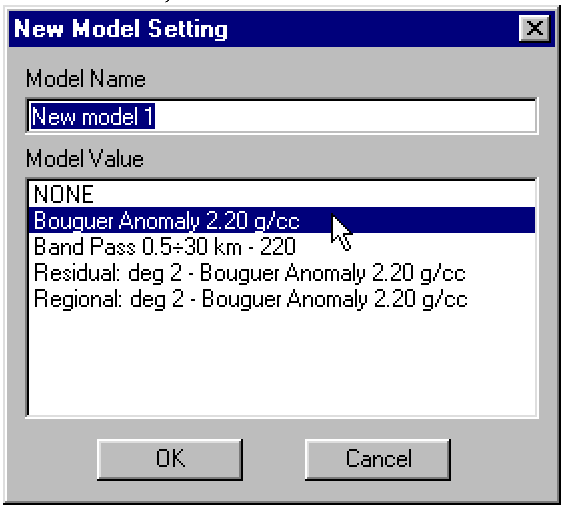
Creating a New Model

1. Select the **File | Open Model** menu command.
2. Select the profile with which the new model is to be associated:

3. In the models frame, click << **New** >>**,** then click the **OK** button:

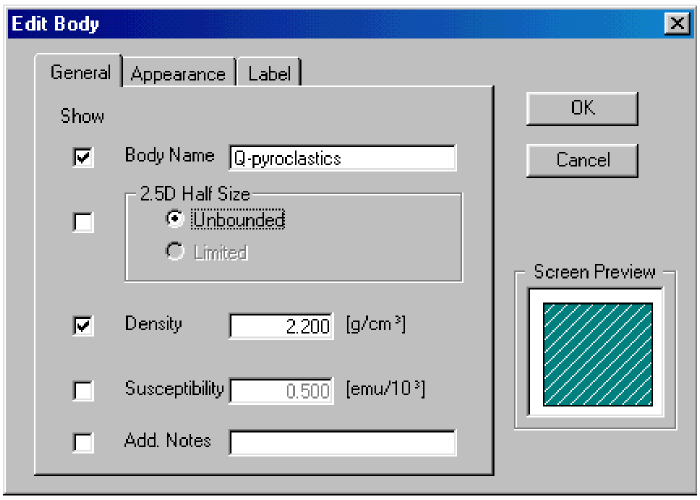


4. Enter the model name and choose the observed anomaly from the values list, then click the **OK** button**:**

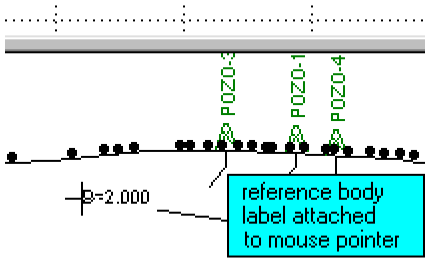


5. The listed anomaly values are the values available for the stations of the current Project. The anomaly values can be selected and changed anytime.

6. When creating a new model, an open reference body, which lies directly underneath the topography, is automatically added to the model. This body represents an uniform earth, prior to adding other bodies to the model. Enter the density (and any other parameters) for this body in the Edit Body form, then click the **OK** button.



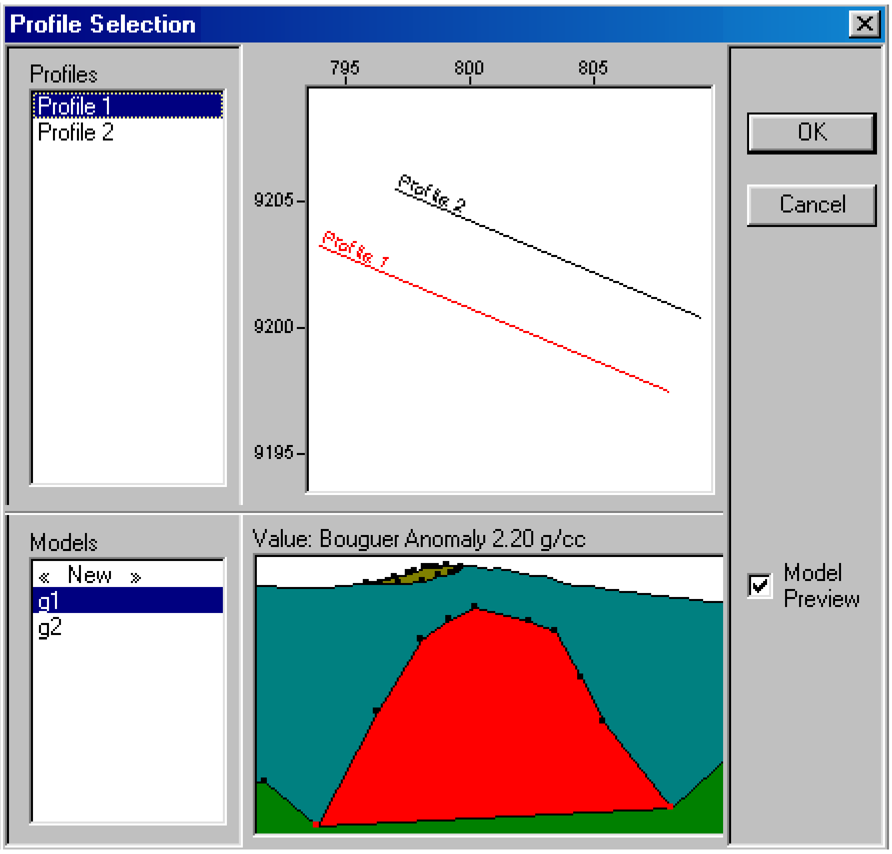
7. Drag the reference body label to the desired location and drop it by clicking the left mouse button:



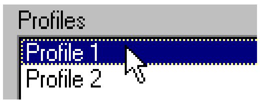
8. Other bodies can now be added to the model.

Opening a Model

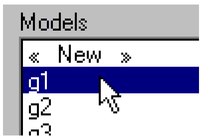
Select the **File | Open Model** menu command**:** a window opens, listing the available profiles, a simplified profile location map, and an optional preview of associated models:



Select the profile by clicking its name:



Select the model by clicking on its name; select the Model Preview check box to display a simplified sketch of each model.



Click **OK**.

Deleting a Model

1. If the model you want to delete is open, close its window.
2. Select the **File | Delete Models** menu command.
3. Select the model you wish to delete.

Editing Models

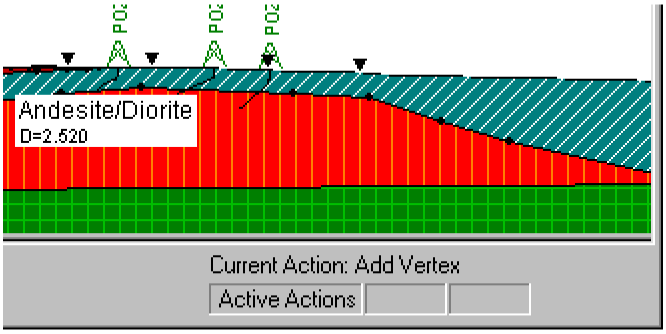
Editing Commands and Icons

Models can be edited using menu commands or edit icons:

- To edit a model using commands, on the **Model** menu, the select **Body, Vertex, or Label** menu command to display the corresponding editing commands, or
- Click the  button on the toolbar to display the Editing Icon Bar.

Important warning: When an editing command (for example: **Add Vertex**) is started, this command remains active after executing the action, allowing the same command to be applied repeatedly. To terminate the editing command, click the mouse right button or press the **Esc** key.

The currently active action command is indicated in the bottom right corner of the program window:



Adding Bodies to a Model

- Select the **Model** | **Body** menu command and then choose the type of body to be added or
- Click the  button to display the editing icons. Click:
   to add a body open on both sides
   to add a body open on the left-hand side
   to add a body open on the right-hand side
   to add a closed body

To define the polygonal section of the added body:

1. Move the mouse pointer into the model window, then left-click where you want the first vertex of the body to appear.
2. Repeat the above step to add at least two more vertices.
3. When finished, right-click the mouse: the program will complete the body geometry according to the initial body definition.

4. The Edit Body form appears.
5. Edit the body as detailed in paragraph Editing body properties.
6. Right-click or press <**Esc**> to stop adding bodies.

Deleting Bodies

- Select the **Model** | **Body menu command**, and then **Delete**, or
- Click the  button on the Editing toolbar:

When the mouse pointer becomes cross-shaped: left-click inside of the body to delete the body. Confirmation is requested before the body is deleted.

Right-click or press <**Esc**> to stop deleting bodies

Moving Bodies

- Select the **Model** | **Body** menu command, and then **Move**, or
- Click the  button on the Editing toolbar.

When the mouse pointer becomes cross-shaped: left-click inside of the body. To move the body, drag it to the new position and left-click to drop it.

Right-click or press <**Esc**> to stop moving bodies.

While moving the body, all clipped vertices remain fixed.

Important: Before moving the body, make sure the **Automatic Vertical Shift** option is turned off, otherwise it is calculated and applied at every move.

Splitting a Body

- Select the **Model** | **Body | Split** menu command, or
- Click the .button on the Editing toolbar.

To define the polygonal section of the new resulting body:

1. Move the mouse pointer into the model window, then left-click on a vertex of the source body (where you wish to start splitting it).
2. Continue to left-click to add vertices inside the source body.
3. Right-click the mouse on another vertex of the source body (where you wish to stop splitting it).
4. The Edit Body form for the new resulting body is then shown.
5. Edit the body as detailed in paragraph Editing body properties
6. Right-click or press <**Esc**> to stop splitting bodies.

Notes:

- Both closed and open bodies can be split, including the reference body.
- When splitting a body, the common vertices are automatically clipped and displayed in red.
- When splitting the reference body, a new body is added to the model. Vertices on the topography are clipped to the topographic profile.

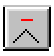
Editing Bodies in a Model

To Add Vertices to a Body:

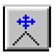
1. Select the **Model** | **Vertex** | **Add** menu command, *or* click the  button on the Editing toolbar.
2. When the mouse pointer becomes cross-shaped, left-click near the line where the vertex is to be added.
3. Move the mouse pointer to drag the new vertex and left-click to drop it.
4. Continue adding vertices as desired. The anomaly curve is updated accordingly.
5. Right-click to **stop** adding vertices.

- Select the **Model | Vertex** | **Add** menu command, *or*
- Click the  button on the Editing toolbar.

To Delete Vertices from a Body

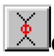
1. Select the **Model** | **Vertex** | **Delete** menu command, *or* click the  button on the Editing toolbar
2. When the mouse pointer becomes cross-shaped, left-click on each vertex to delete.
3. Right-click to **stop** deleting vertices.

To Move the Vertices of a Body:

1. Select the **Model** | **Vertex** | **Move** menu command, *or*
2. Click the  button on the Editing toolbar.
3. When the mouse pointer becomes cross-shaped, left-click on the vertex to move
4. Move the mouse pointer to drag the vertex and left-click to drop it.
5. Right-click to stop picking vertices to move

Important: Before moving the vertex, you may want to make sure the **Automatic Vertical Shift** option is turned off, otherwise it is calculated and applied at every move.

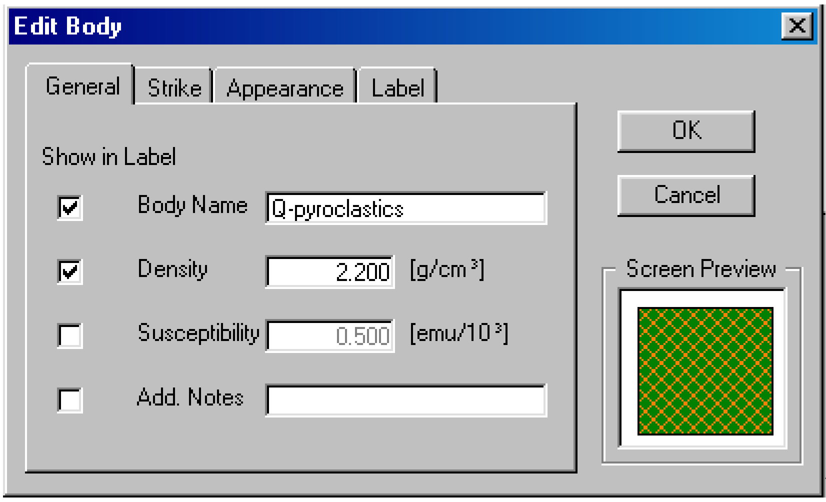
To Clip Two or More Vertices:

1. Select the **Model** | **Vertex | Clip** menu command, *or* click  on the Editing toolbar.
2. When the mouse pointer becomes cross-shaped, left-click the vertex to clip.
3. Move the mouse pointer to drag the vertex on the partner vertex, and left-click to drop it.
4. The clipped vertices turn red.

Setting the Properties of a Body

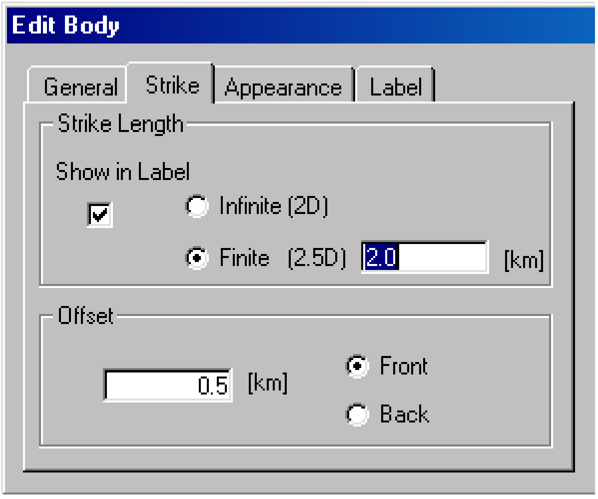
To edit the properties of a body:

1. Select the **Model** | **Body | Properties** menu command, or click the  button on the Editing toolbar.
2. Double-click with the mouse inside the body to be edited.
3. The Edit Body form opens. Select the desired property tab, and enter the settings as appropriate:



When body properties are entered or edited, check the box beside each property to display it in the label attached to the body.

For example, if you wish to display the strike length and offset of a body, select the check box <**Show in Legend**> in the Edit Body form:



The label for a **new** body is automatically created by the program when closing the **Edit Body** form

Each time the body properties are edited, the contents of the Body Labels are automatically updated.

Moving Body Labels

1. Click the  button on the Editing toolbar, then click the label and drag it to its new position or simply on the label and move it.
2. A straight line will connect the label to its old position. To edit the line, click on one edge, and drag it to the desired location.

Formatting Body Labels

1. Select the **Model** | **Format Labels** menu command or right-click on one label (the format settings are applied to all labels).
2. Enter the desired format setting.

25: Interpreted Views and Montage

Overview

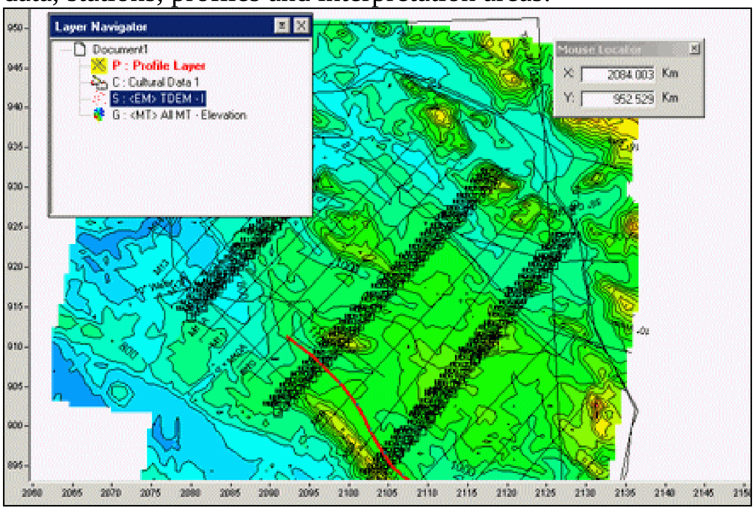
The Interpreted Views and Montage (IVM) program provides a tool for displaying multiple layers of geophysical data in a single document. As a set of medical transparencies might be used to display skin over muscles over bones, IVM is used to superimpose various types of geophysical data onto one another, e.g. cultural data and well logs over a 2D model of magnetotelluric data acquired along the same profile.

IVM is designed to prepare for presentation data which have been processed and modeled using the various interpretation modules contained within WinGLink. In addition to displaying raw and processed data, the program contains several sets of tools for annotating documents. IVM itself contains no data analysis functionality.

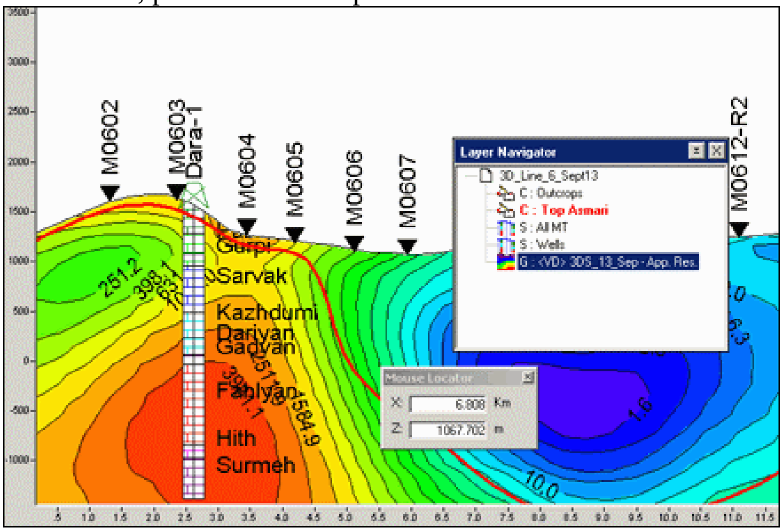
Document Types

The IVM program can be used to create two types of documents:

- **Maps**: these documents may contain, among other features, maps created in the Maps program in WinGLink, cultural data, stations, profiles and interpretation areas:



- **Sections:** these documents may contain sections created in WinGLink, e.g. grids and meshes created in the 1D Sections and 2D MT Inversion modules, as well as cultural data, station data, profiles and interpretation areas:



In both types of documents, the different features are placed on separate layers. Each layer is treated as an independent object and has associated with it an independent set of display properties, i.e. symbol name, font size, contour colors, etc. Layers can be shown or hidden, and, to use the medical transparency analogy again, can be moved above or below other layers.

Both of these document types are stored in the active WinGLink database as enclosed objects.

Plate Mode

Documents stored in the database can be opened in IVM **plate** mode, which provides you with an interface for positioning and annotating documents with legends, scales, text boxes, and other features on a single *plate*. Plates can then be sent to an available printer or to EMF or CGM files. Plates are stored in the active WinGLink database and, like map and section documents, can be opened and edited at a later time in IVM.

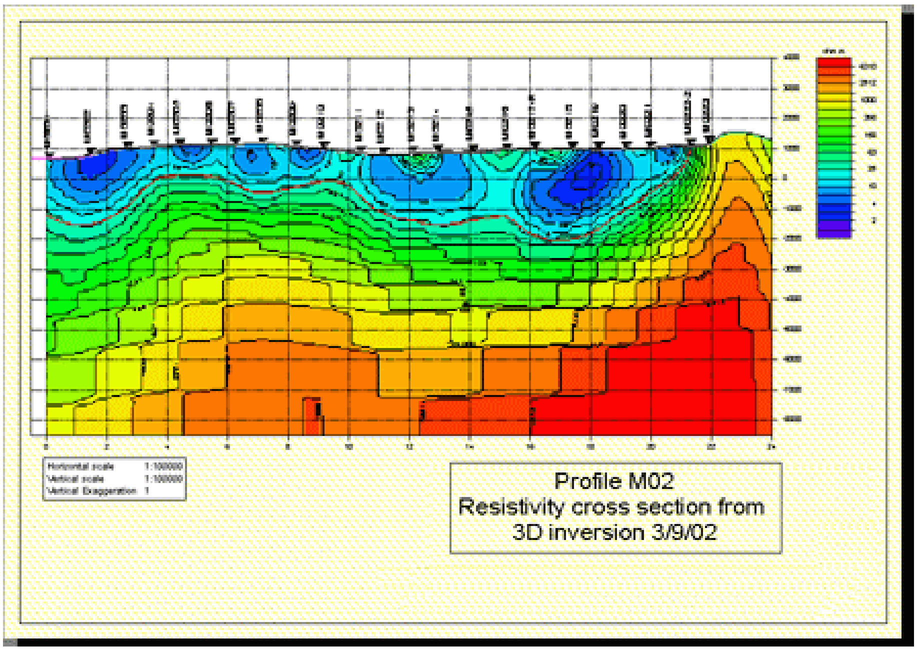
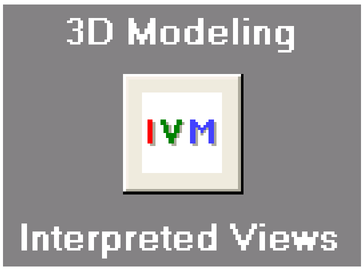


Figure 1 Example of a plate containing a section document, scale box, title box and color scale.

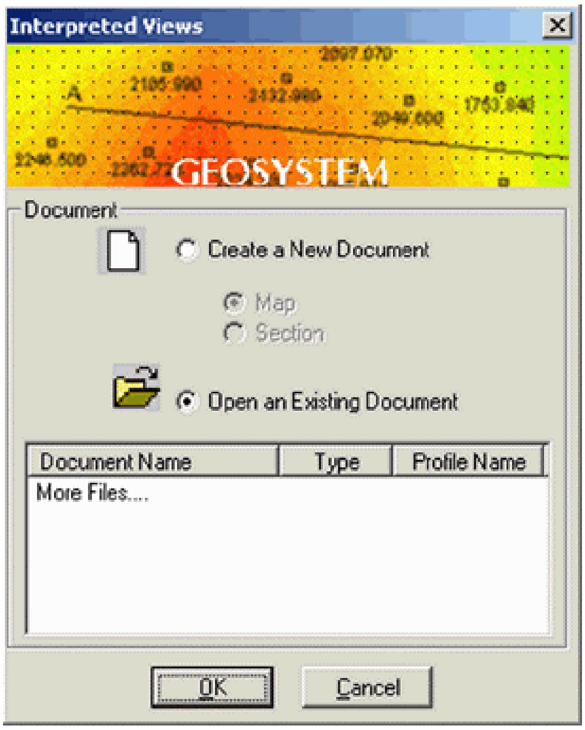
Getting Started with IVM

Note: IVM can be started only if your WinGLink license agreement includes the Interpreted Views and Montage option and your dongle has been correctly configured. Please contact Geosystem for further details.

To open IVM, click the **Interpreted Views and Montage** button located at the right side of the main database window in WinGLink:



The program opens with the Document Editor dialog box:

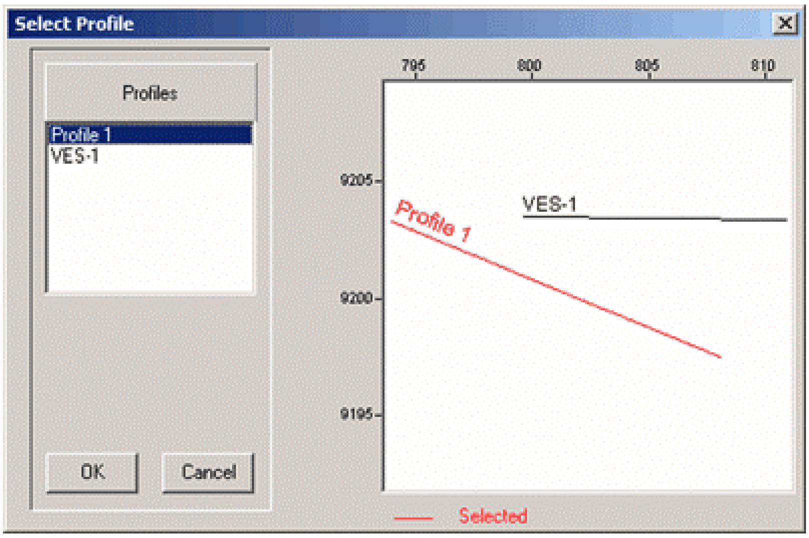


Recently opened IVM documents, if any, contained in the active database, their types, and the profiles with which they are associated are listed in the box in the bottom half of the dialog box. To view a list of all IVM documents contained in the database, click the **More Files…** label in the text box. This opens an additional dialog box which lists for selection all IVM documents contained in the database.

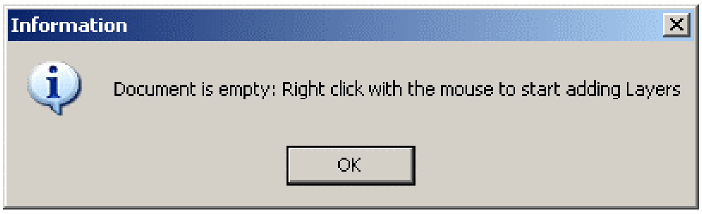
Note: Documents are stored within the WinGLink database, not as external files.

To continue, either select an existing document or create a new document by clicking the **Create a New Document** option button and either the **Map** or **Section** option button.

If you choose to open a section document, you will be prompted to select a profile from the profiles contained in the WinGLink database:



After clicking **OK** for section documents, or immediately if you are creating a new map, the document window opens with the following message:



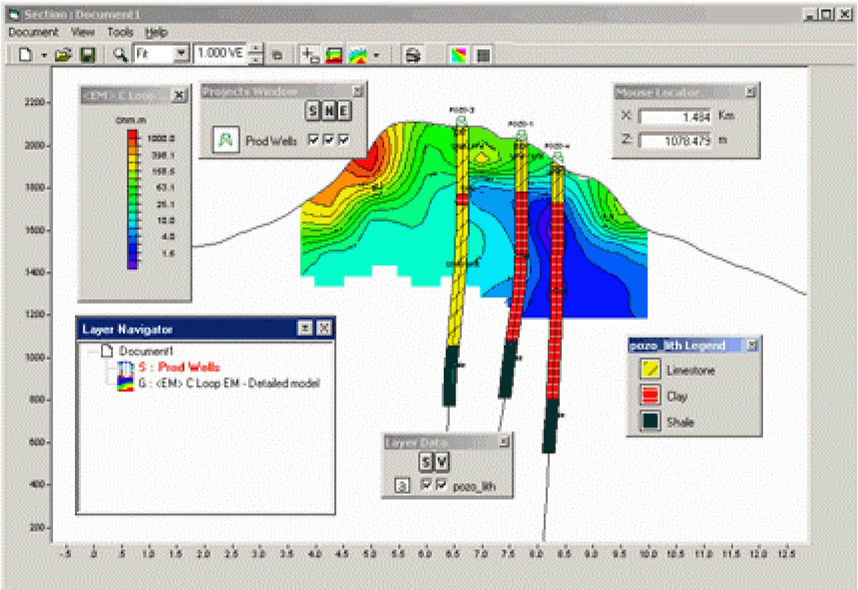
After clicking **OK**, IVM opens either the Map or Section editor, both of which are described below.

Understanding How IVM Works

Regardless of whether you are working with a map or a section document, the core functionality is, in principle, the same. Although the types of layers which can be added to each document type differ, how they are added to a document, most menu options and dialog boxes available to the two document types are the same.

Using a section document as an example, this section will introduce the IVM features and functions common to both document types.

Shown below is a section window which contains a multilayer document, the toolbars and legends associated with those layers, the Layer Navigator window, and the Mouse Locator window:



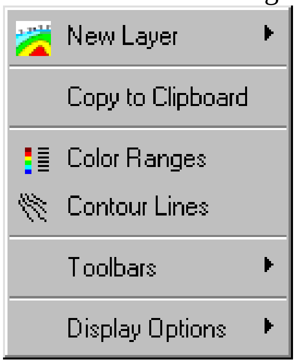
The Document Window

This is the principle area of both the Map and Section editors. Layers, as they are added, are displayed here in the specified Z-order, the order in which the layers are stacked on top of one another.

The menu options contained in the **View** menu item and the corresponding buttons in the main toolbar can be used to adjust the area of the document displayed on the screen as in other WinGLink modules. Document can be enlarged, reduced, panned, or a specific area selected.

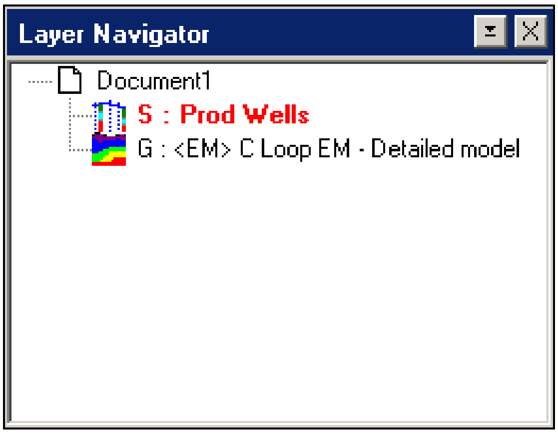
Changes in the view area are immediately reflected in the scales displayed along document window.

The entire area of the document window is mouse sensitive. When the right mouse button is clicked anywhere on this window, a shortcut menu similar to the following appears:



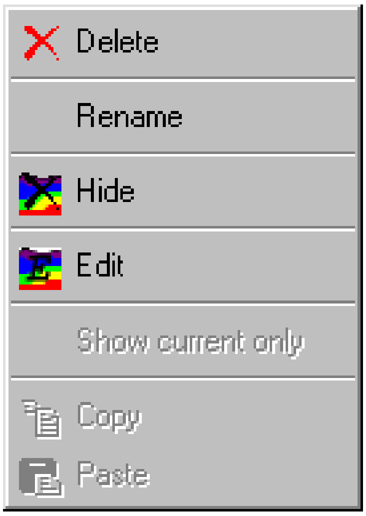
Which menu commands are listed in the shortcut menu is dependent on the type of document, the layers contained in the document, if any, as well as which layer is currently active. The functions listed in this shortcut menu can, for the most part, also be accessed in either the main menu bar or from the toolbar. As the submenu command names imply, these commands are used to add new layers and change the display characteristics of the individual layers in the document. The function of each of these and other submenu commands are described in detail later in this chapter.

Layer Navigator:



The Layer Navigator window is used to manage the layers contained in the document. The first layer listed below the document name (in this example Document1), is the top-most layer in the document, i.e. has the lowest *Z-order*. In the example shown above, the layer **Prod Wells** precedes the layer **C Loop EM**. If the Z-order of these two layers was reversed, the wells layer would be all but hidden by the grid. To change to the Z-order of a document, click the name of the layer which is to be repositioned with the left mouse button, and drag to the new position in the Layer Navigator. After repositioning a layer, double-click the layer name to activate the layer and redraw the window.

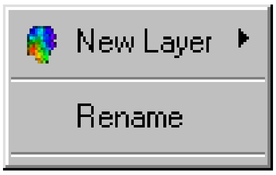
The currently active layer is displayed in red. A layer can be activated by either double-clicking the layer name with the mouse or by clicking the right mouse button anywhere within the Layer Navigator window to open the Layer Navigator shortcut menu and selecting the **Edit** menu command:



Note: Although the available commands vary depending on what type of layer is active, the four active menu commands shown above are available for all layer types. When a layer is active, the **Edit** command changes to **Exit Edit**; when a layer is hidden from view, the **Hide** command changes to **Show**. Before a layer can be activated, it must be visible.

Menu commands **Show current only**, **Copy** and **Paste** are not available for all layer types.

The document name, the top item in the Layer Navigator, has associated with it a different shortcut menu:

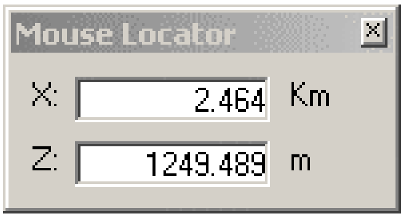


This shortcut menu can be used to add new layers to the document or to assign the document a new name.

The toolbars, legends and available menu commands change depending on what type of layer is active. For example, when the active layer is of type grid, the display window shortcut menu contains commands for changing the color ranges and contour lines whereas for a layer of type station containing well data, this menu might contain, among others, a command for editing well logs.

The Layer Navigator window is reduced to a title bar by clicking the title bar with the left mouse button. When the mouse pointer is positioned over the title bar, the window is automatically restored to its original size. Likewise, when the mouse pointer is moved off of the Layer Navigator, the window is again reduced to a title bar. To switch off this auto-close functionality, click anywhere within the Layer Navigator window, and the window will remain open. Use the  button, located in the main toolbar, to toggle the Layer Navigator window on and off.

Mouse Locator



The Mouse Locator window displays the mouse position on the on the map or section. This window can be toggled on and off using the  button located in the main toolbar.

Zoom

IVM provides several zoom functions, all of which are available for both map and section documents:

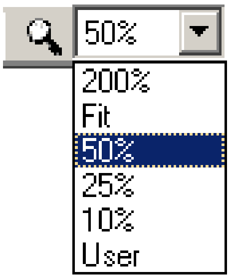
Zoom In/Out

When in this mode, click the left mouse button to zoom in on the document area below the mouse, click the right mouse button to zoom out. To enter Zoom In/Out mode, click the magnifying glass button to put the button into the down state:

While in this mode, the mouse pointer is displayed as a magnifying glass. IVM remains in zoom in/out mode until the magnifying glass button or another toolbar button is clicked.

Other Zoom Options

Open the dropdown list box to the right of the magnifying glass to list several zoom options:



Use the numerical options to specify a zoom size.

The **Fit** option fits the entire document onto the window.

The **User** zoom option is used to zoom in on a selected area of the document. To use, select the **User** option. Immediately after selecting, the mouse pointer changes to a crosshair with a small rectangle in the lower right corner of the crosshair. Click the mouse at the desired location on the document and, keeping the mouse button depressed, draw a rectangle over an area of the document. This area of the document is then enlarged to fill the screen.

Note: while the Zoom In/Out button is depressed, only the zoom in/out functions are available.

Vertical Exaggeration

Use the vertical exaggeration option, available in section documents, to change the scaling of the vertical axis relative to horizontal axis. For example, with a vertical exaggeration of 10, one unit length on the horizontal axis is equivalent to ten unit lengths on the vertical axis.

To set the vertical exaggeration, either set a numerical value in the vertical exaggeration box, located in the toolbar, or use the scroll bars to adjust the exaggeration to a desired value:

Crossing Traces

Use the View | Crossing Traces menu command to display the points at which the current profile intersects any other profiles (sections only).

Toolbars

As discussed above in the Layer Navigator section, different layer types have associated with them different toolbars. These toolbars are floating windows, each of which contains a set of controls or legends specific to the given layer.

For example, layers containing station data have associated with them a Projects Window toolbar, which is used to control the display properties of the stations contained in a given layer. Depending on the station type, other toolbars may also be available. Well log stations may have associated with them toolbars for layer data (lithologic data) – which can be used to toggle on and off stratigraphic features, as well as additional toolbars for controlling the display of numerical data and depth tags, and legends which provide details about the patterns used to describe the stratigraphy.

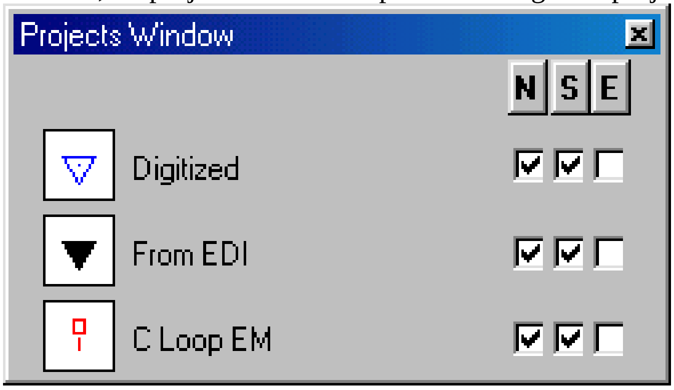
To Hide and Display the Toolbars Associated with a Given Layer:

1. Activate the layer associated with the toolbar by double-clicking the layer name in the Layer Navigator with the mouse.
2. Right click the mouse anywhere on the display window.
3. In the shortcut menu which opens, select the **Toolbars** command. All toolbars, if any, associated with the layer are listed in a secondary menu. Those currently displayed in the document have next to them a light gray button.
4. To toggle a toolbar on or off, click the toolbar name in the shortcut menu.

Each of the toolbars is described below:

Projects Window toolbar:

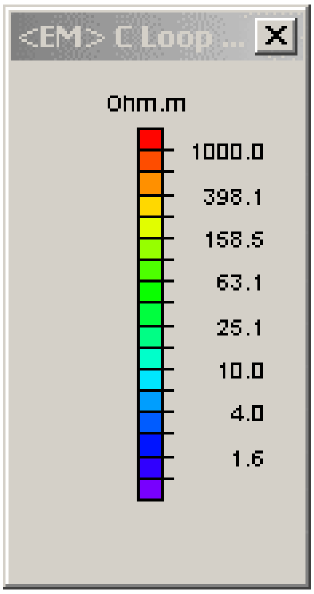
Layers containing station or profile data have associated with them a Projects window which can be used to change how the station data are displayed in the document. The Projects window serves the same function in IVM as the Project panel in the main WinGLink database window. When individual projects are selected in IVM, the Projects window contains a single project. However, when an integrated project is selected, all projects which comprise the integrated project are listed:



Consider, for example, the Projects window shown above for an integrated project. The **[N], [S] and [E]** buttons toggle on and off the station name, symbol and elevation, respectively. Project windows for other types contain similar options, e.g.**[T]** for toggling on and off profile traces (map documents only).

Double-click the symbol to the left of the station type to open the Select Options dialog box to specify colors, fonts, text positions and other display properties.

Color scale toolbar:



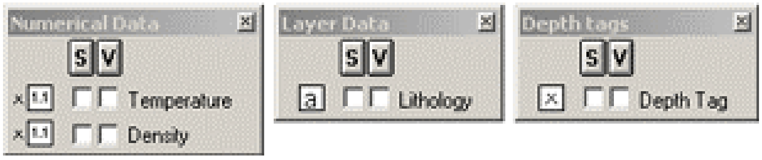
The Color Scale window shows the range scale for layers containing mesh and grid data. Color ranges are set and edited in IVM using the Range Editor as they are elsewhere in WinGLink. For full details on how to use this feature, please refer to the Common Functions chapter of the WinGLink manual.

To Toggle the Color Scale Window on and off for a Given Layer:

1. Activate the grid layer by double-clicking the layer name in the Layer Navigator.
2. Right-click anywhere on the display window.
3. Click the **Color Scale** command from the **Toolbars** submenu.

Other toolbars

As mentioned above, different toolbars are available for different layer types. In principle, however, they all function in the same way as the Projects Window toolbar described earlier. Shown below are three toolbars available to layers containing well log data:



Clicking the **[S]** button or checking the check boxes below the **[S]** button toggles the symbols for the respective value; enabling or disabling the checkboxes below the **[V]** button toggles the value display. Double-clicking the icons to the right of the checkboxes opens the Select Options dialog box which provides options for setting various display options.

Legends

Layers containing information too detailed to be displayed in the Section window itself may also contain a legend window. For example, the pozo_lith Legend, shown below and in the Section window at the beginning of this section, contains a legend for the stratigraphy data for the respective well log stations.

Note: Legends are automatically created when well log stations which include layer (stratigraphic) data are imported into IVM. Bear in mind, however, that automatically created legends contain only blank pattern blocks and the names of the individual layers.

Fill patterns can be created for each of the items contained in a legend with the Pattern Editor, described later in this chapter.

Creating and Working with Layers

Map and section documents may contain the following types of layers:

- Grids
- Stations
- Profiles (maps only)
- Cultural data
- Interpretation areas

Note: With the exception of cultural data and interpretation areas, which are created within the Maps and Sections editors themselves, all geophysical data must exist in the active WinGLink database in its final, processed form. No data processing is possible within the Interpreted Views and Montage program.

While the Maps and Section editors both generally support the same layer types, the actual data which can be displayed in the two editors is, of course, quite different. The Maps editor is used to display horizontally distributed data, whereas the Section editor is used to display vertically distributed data. Despite this fundamental difference, the individual layers types are handled in the same way in both editors.

A second major difference between the two document types is the availability of data. When creating a map document, you may select from all (horizontal) grids, i.e. all map data, and all station data contained in the active database. Section documents, on the other hand, may only contain meshes or grids created using stations located along the selected profile as well as any stations assigned to that profile.

This section describes how to create and work with each of the layer types listed above. The differences between the two editors, i.e. what types of physical data can be represented in each of the layer types, will be indicated at the appropriate points.

Grids

Grid layers in the map and section documents may contain most types of gridded data stored in the active WinGLink database as follows:

Map documents: all maps created in the WinGLink Maps program which are stored in the active WinGLink database can be placed on grid layers within a map document, e.g. elevations, gravity, apparent resistivity at depth, etc.

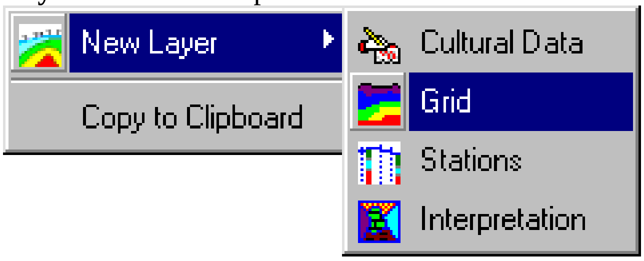
Section documents: all section data stored in the active WinGLink database can be placed on grid layers within a section document. These include:

- Sections extracted from 3D MT meshes (using X-Sections)
- 2D MT meshes and grids (2D MT Inversion module: MT data)
- 1D model images(Imaged Sections module, Vert: EM + MT data)

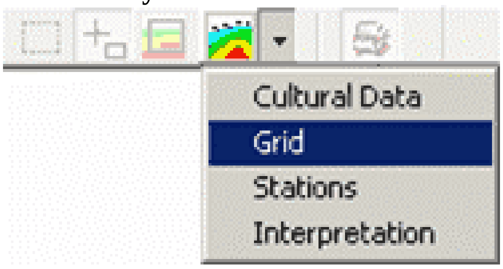
Creating a Grid Layer

To create a grid layer in a new or existing document:

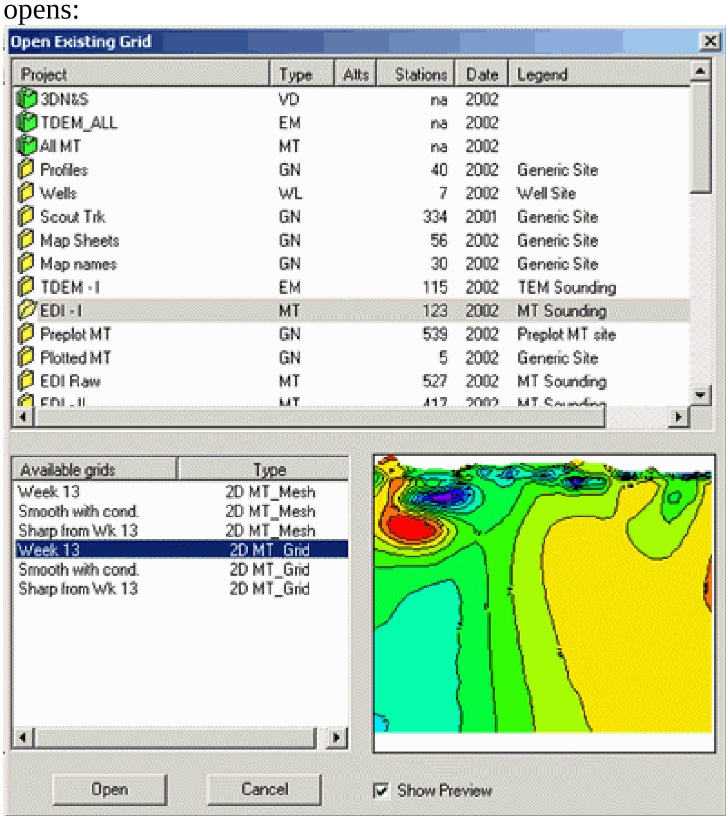
1. Right-click the mouse anywhere on the document window or right-click the document title in the Layer Navigator window to open a shortcut menu. Then position the mouse over the **New Layer** command to open the submenu shown below:



Alternatively, you can open the layer-type submenu by clicking the **Add Layer** button in the main toolbar:



2. Click the **Grid** command; the Open Existing Grid dialog box opens:



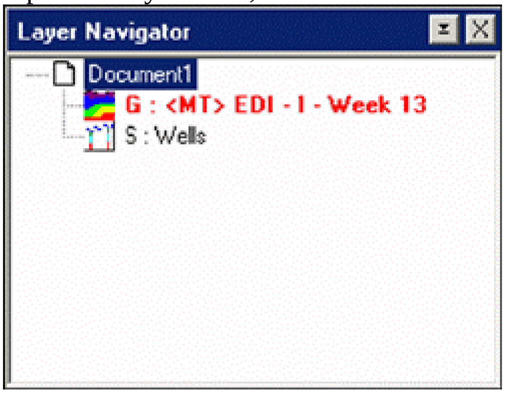
Listed in the upper half of the window are all projects contained in the active WinGLink database. Select a project by clicking it with the mouse.

Listed in the lower left part of the window are the grids available for the project selected in the upper half of the screen. Select a grid by clicking it with the mouse.

Click the **Show Preview** checkbox at the bottom of the screen to display a preview of the selected grid in the lower right corner of the window.

Note: When working with section documents, only grids created along the active profile are available for selection.

3. Click **Open** to create the grid. The new layer is automatically selected and assigned the lowest Z-order, i.e. is placed at the top of the layer stack, as can be seen in the Layer Navigator:



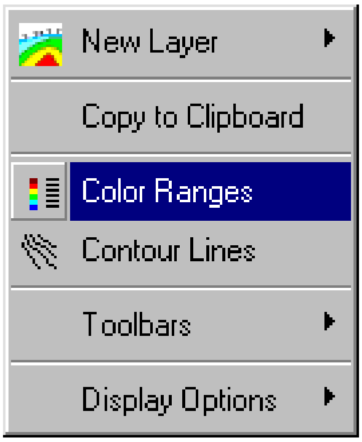
4. Reposition the layer in the Layer Navigator as desired by clicking the layer name with the mouse and dragging to a new position in the layer list.

Working with Grid Layers

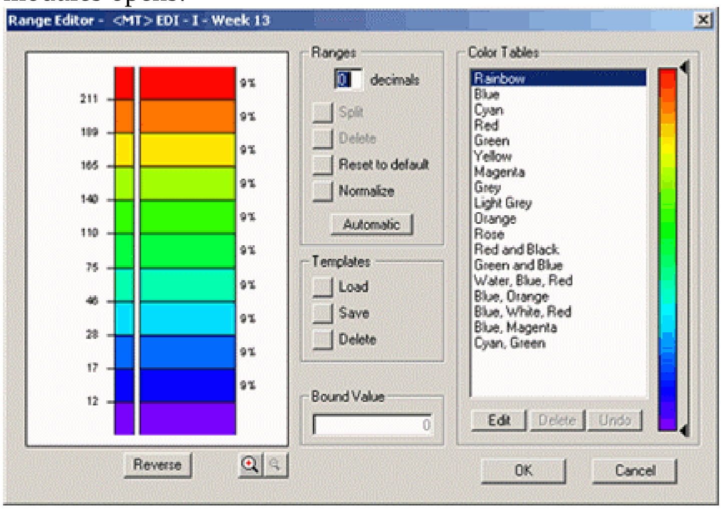
The same display functions available to grids within the WinGLink modules which support grids are also available in IVM. The Range Editor window can be used here to redefine the range distribution of colors used in the mesh. The display properties of contour lines can be modified using the Contour Editor.

To Open the Range Editor:

1. Activate the grid layer to be edited by double-clicking the layer name in the Layer Navigator or by right-clicking the layer name in the Layer Navigator and clicking the **Edit** command from the Layer Navigator shortcut menu.
2. Right-click the mouse anywhere on the document window to open the document window shortcut menu; select the **Color Ranges** command:



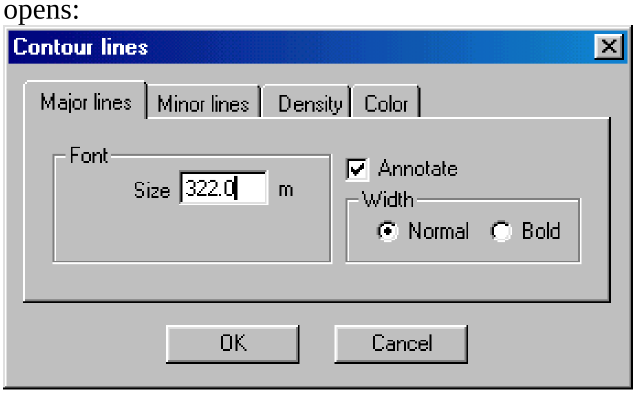
3. The Range Editor window familiar from other WinGLink modules opens:



For a detailed description on how to use the Range Editor, please refer to Chapter 5 of the WinGLink manual, “Common Functions”.

To Open the Contour Editor:

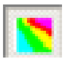
1. Activate the grid layer to be edited by double-clicking the layer name in the Layer Navigator or by right-clicking the layer name in the Layer Navigator and clicking the **Edit** command.
2. Right-click the mouse anywhere on the document window to open the shortcut menu; select the **Contour Lines** command.
3. The Contour Lines window found elsewhere in WinGLink opens:

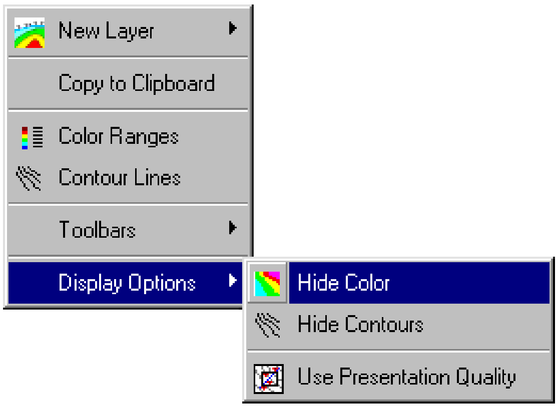


Note that the size of the labeling in current database units is specified in the Size field.

For a detailed description on how to use the Contour Editor, please refer to Chapter 5 of the WinGLink manual, “Common Functions”.

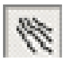
Showing/Hiding Fill Colors

To show or hide fill colors, activate the grid layer, then either click the  button or right-click the mouse anywhere on the document window to open the shortcut menu; click the **Display Options | Hide/Show Colors** command:



The **Hide Color** command toggles to **Show Color** when the colors are hidden.

Showing/hiding contour lines

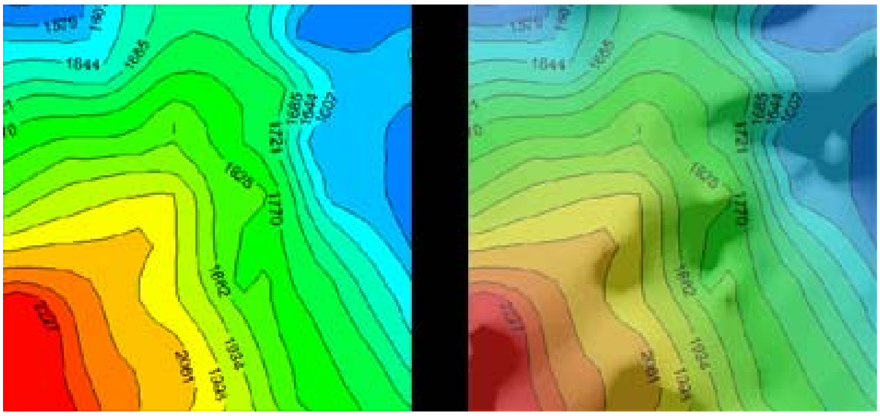
To show or hide contour lines, activate the grid layer and click either the  button or right-click the mouse anywhere on the document window to open the shortcut menu; click the **Display Options | Hide/Show Contours** command: The **Hide Contours** command toggles to **Show Contours** when the contours are hidden.

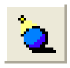
Changing the display quality

Two display qualities are available for grid layers: draft and presentation. Presentation quality produces a more exact representation of the data and is, thus, more time intensive. To change the display quality, activate the grid layer and right-click the mouse on the document window to open the shortcut menu. Select the **Display Options | Use Presentation Quality/Use Draft Quality** command. The default presentation quality is **Draft**. The menu command changes between **Use Draft Quality** and **Use Presentation Quality** depending on the current selection.

Topography Shadowing

A virtual light source can be directed onto elevation maps to generate shadows:



To toggle the light source on and off, use the shadow button:  located in the toolbar.

Note: The shadow button is only visible for elevation map grids.

Stations

Station layers are used to display station locations for a given project on an IVM document. Although some station data are displayed in the same way in both document types, there are some notable differences.

First the similarities. It is possible to display for stations in both document types the station name, a station symbol and the station elevation. For most station types, these values can be toggled on and off using the Projects Window described earlier in this chapter.

The value data associated with stations, e.g. resistivity, Bouguer anomaly, are, however, generally available only in map documents. This is due in large to the fact that maps are defined in WinGLink to be sets of horizontally distributed data. These sets of data consist of the value data which can be viewed in the WinGLink Maps module. While it is possible to interpolate between these values to create horizontal grids, even at depth, it is not yet possible to use WinGLink to transform the values at depth to vertical sections.

Stations for earthquake projects (EQ) are plotted at true elevation.

WinGLink considers sections, on the other hand, to be sets of vertically distributed data or models associated with a profile trace, for example well traces, 1D models or 2D models created along profile.

Creating Station Layers

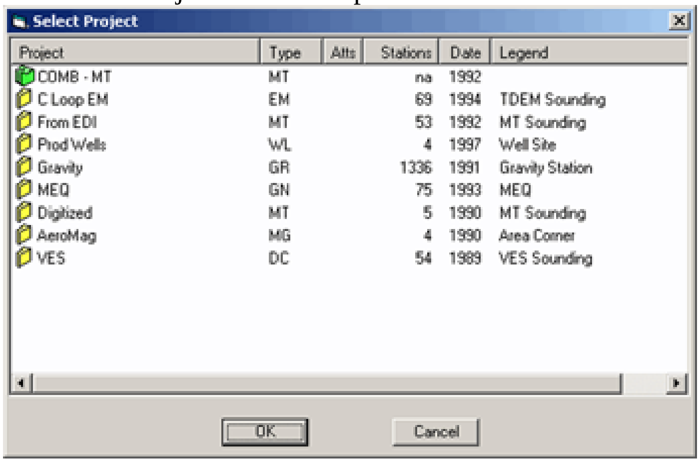
Display differences aside, station layers are handled the same way in both map and section document types.

To create a station layer:

1. Right-click the mouse anywhere on the document window or right-click the document title in the Layer Navigator window to open the shortcut menu. Then position the mouse over the **New**

Layer command to open the submenu; select the **Stations** command.

2. The Select Projects window opens:



Listed here are all projects contained in the database, the type and number of stations available for selection in each project.

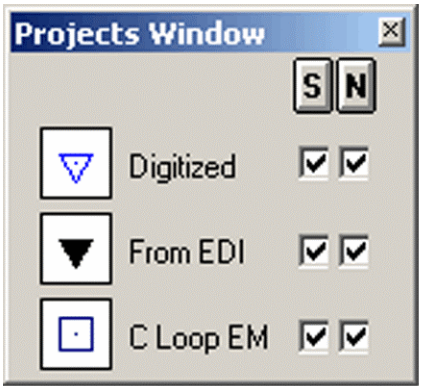
Station layers created for map documents will contain all stations in the selected project. Station layers created for section documents, on the other hand, will contain only those stations positioned along the active profile. If there are no stations along the profile, **na** is listed in the Station column.

Select the desired project and click **OK**.

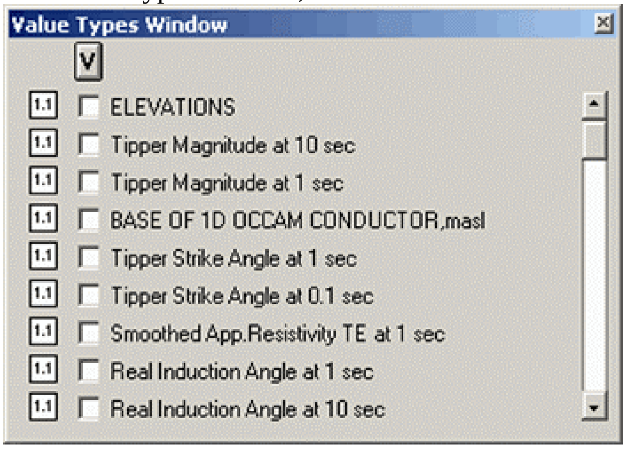
3. As with other layer types, reposition the station layer in the Layer Navigator with respect to other layers. if necessary by clicking the layer name with the mouse and dragging to a new position in the layer list.

Working with Station Layers

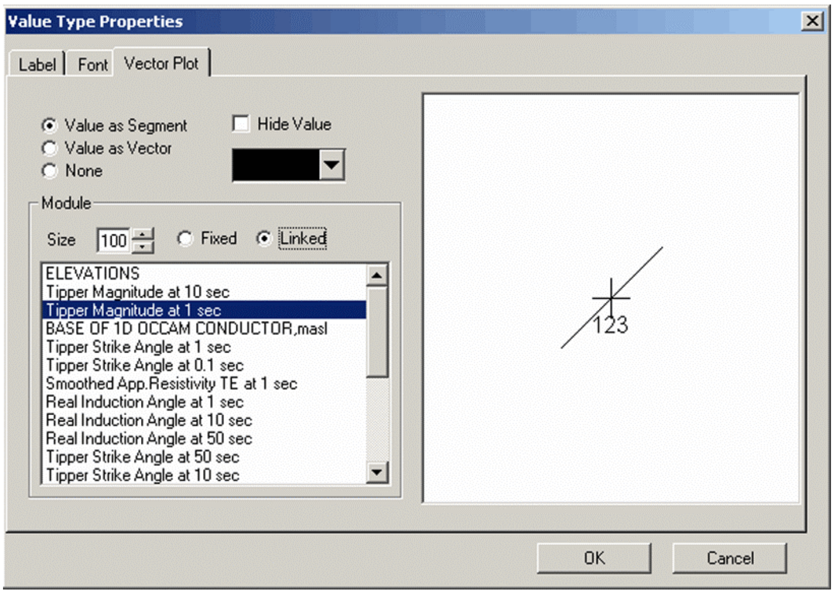
Station layers containing any of the data types supported in WinGLink have associated with them a Projects toolbar, which is used to set the appearance of intrinsic station properties on the screen (name, symbol, elevation as well as format and location of text labels):



Depending on the type of data the station contains and the type of document, i.e. map or section, the station may also have associated with it a Value Types window, which lists all values available for the stations:



Select the check box(es) to the left of the value names to display the respective value(s). Click the **[V]** button located above the check boxes to display all values. Double-click the symbol to the left of the check boxes to open the Value Type Properties window for modifying the display properties of the value data, i.e. font size, text location with respect to the symbol, vector plots etc:

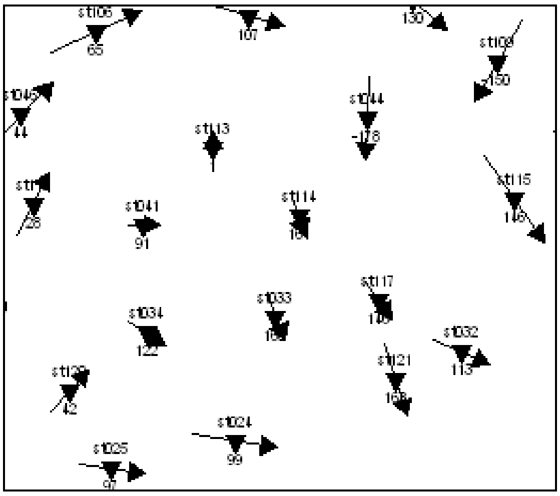


The Value Types window is generally available only in map documents, the exception being well stations displayed in section documents. This exception aside, the values listed in the Value Types window correspond to the values available in the WinGLink Maps module.

The function of the **Label** and **Font** tabs is straightforward. The function of the **Vector Plot** tab, shown above, is, however, worth a closer look. Using the options provided on this tab, vector arrows can be drawn for MT parameters which are expressed as complex quantities, e.g. tipper. Because the real and imaginary components of complex quantities are stored separately within WinGLink, the real portion of the vector must be linked to the imaginary portion. It is up to the user to link the correct real quantity to a given imaginary quantity.

Use the radio buttons in the upper left corner to specify how the angular values are to be displayed: as segments, vectors (segments with directional arrows), or not at all. Select the **Hide Value** check box to hide the measured value.

The **Module** area of the Value Type Properties window contains options for specifying how the segments are displayed. The **Fixed** option (default) specifies that the directional arrow be displayed with a fixed length for all stations as specified in the **Size** field. By selecting the **Linked** option and selecting one of the values listed in the values field, it is possible to assign a magnitude to the directional quantity. In this way you can, for example, link the tipper magnitude at 1 sec. to the tipper strike angle at 1 sec, thereby constructing the vector quantity:

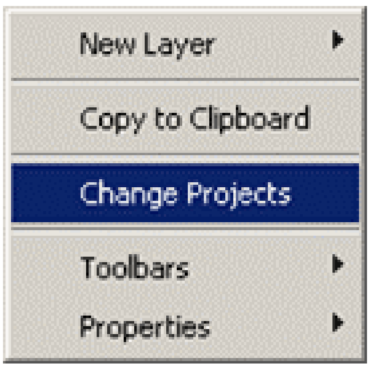


When linking an angular quantity to a magnitude, the value specified in the **Sizes** field serves to “amplify” the magnitude value.

The Vector Plot tab is available only for MT stations on Maps documents. Moreover, it is available only for the angular component of a given complex quantity.

Changing projects

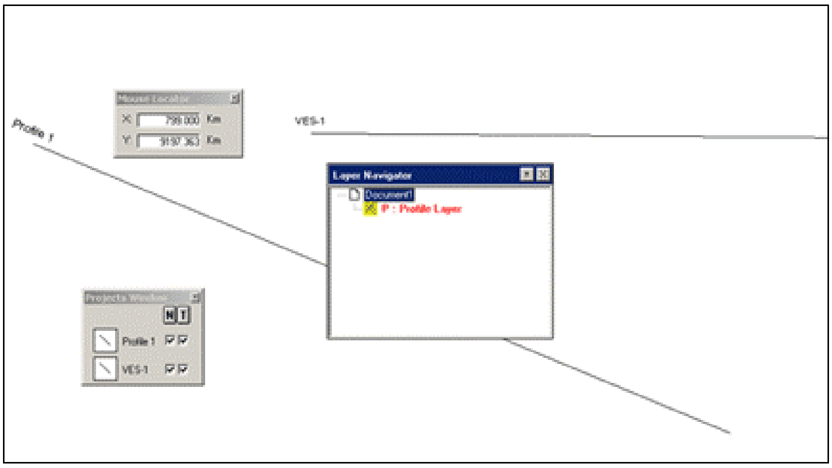
The shortcut menu for station layers includes a command for changing projects:



Clicking this menu command opens the Select Projects window shown in Step 2 of the **Creating station layers** section of this chapter. The project selected in the Select Projects window replaces the current project. This function can be useful when working with complex maps or sections containing cultural data or interpretation areas and you would like to create similar documents for various projects.

Profiles

Profile layers are available only in map documents. As shown below, the display is similar to that seen in the WinGLink Maps module:



When a profile layer is first created, all profiles in the active WinGLink database are plotted on the map document. The Projects Window, seen in the lower left above, can be used to toggle on and off the individual profile traces and profile names.

Click the symbol to the left of the trace names to open the Profile Properties window, which can be used to modify text and profile display properties.

Creating Profile Layers

To create a profile layer, right-click anywhere on the document window or right-click the document title in the Layer Navigator window to open the shortcut menu. Then position the mouse over the **New Layer** command to open the submenu; select the **Profile** command.

Cultural Data

Cultural data layers are used to display local cultural features. Several display elements are available for this purpose:

- **Labels:** text labels
- **Text boxes:** text, with extended formatting options and the option to place a frame around the text.
- **Shapes:** choose from rectangles and ovals; shapes may be assigned colors and/or filling patterns.
- **Polylines:** may be assigned width and color; polyline definitions may also be exported to and imported from external files. May optionally be smoothed.
- **Free shapes:** polygonal shapes. Shapes may be assigned colors and/or filling patterns. May optionally be smoothed. May be exported to and imported from external files.
- **Images:** external bitmap images. Images may be resized and rotated.

- **Symbols:** select from rectangles, circles, crosses and arrows. Symbols may be assigned color, width and rotation.

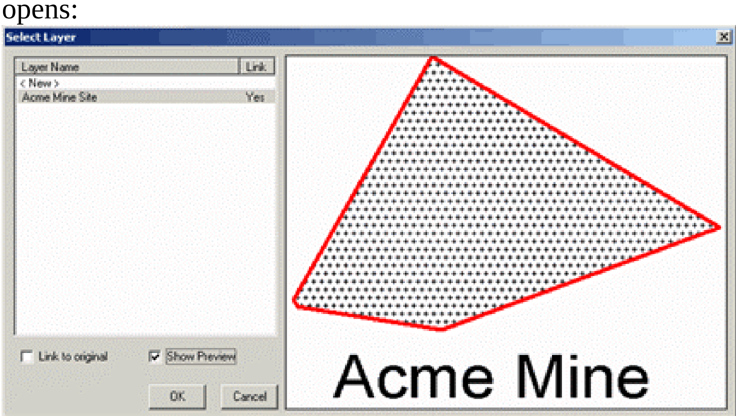
With the exception of the cultural data layer type in IVM, cultural data do not exist in any form elsewhere in WinGLink databases. These features must, therefore, be created manually using the set of tools provided for this purpose and described below. Cultural data layers are available in map documents as well as in section documents.

Several cultural data elements may exist within a given cultural data layer. Thus, this layer type actually represents a document within a document. The Z-order of the individual elements within a cultural data layer can be modified, allowing you to position one element on top of another. A cultural data layer can also be stored in the active WinGLink database and reused in other IVM documents.

Creating Cultural Data Layers

To create a cultural data layer:

1. Right-click the mouse anywhere on the document window or right-click the document title in the Layer Navigator window to open a shortcut menu.
2. Then position the mouse over the **New Layer** command to open the submenu
3. Select the **Cultural Data** command. The Select Layer window opens:



4. Listed in this window are all cultural data layers contained in the active WinGLink database.
5. To create a new cultural data layer, select the **<New>** item in the Layer Name column and click **OK**. Likewise, an exiting layer can be inserted into the document by selecting the layer name and clicking **OK**.

When inserting an existing cultural data layer into a document, you may insert either the original layer or a copy of it. If you select the **Link to Original** checkbox when inserting an existing layer into a cultural data layer, any change made to the layer will be reflected in other documents/layers which use the layer, provided they, too, use the layer in linked mode. If the **Link to Original** checkbox is not selected when inserting an existing layer into a document, a new cultural data layer is created and will be listed in the Select Layer window the next time a cultural data layer is created.

The names of linked cultural data layers are displayed in bold text in the Layer Navigator when the document is next opened.

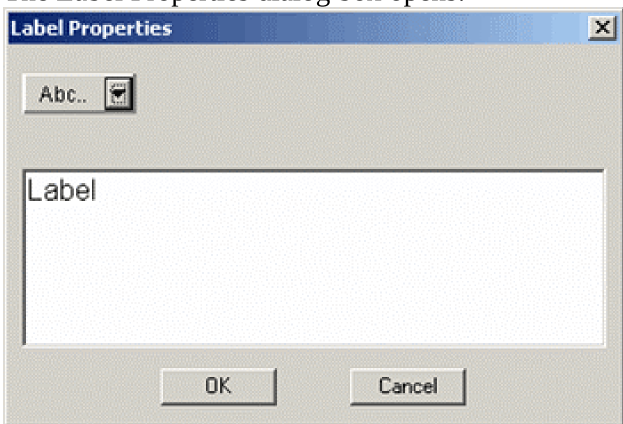
Note: To prevent confusion with layer names, when inserting an existing layer into a document without checking the **Link to Original** check box, it is recommended that you rename the layer in the Layer Navigator. The same name will otherwise be used for two different layers the next time the Select Layer window is opened.

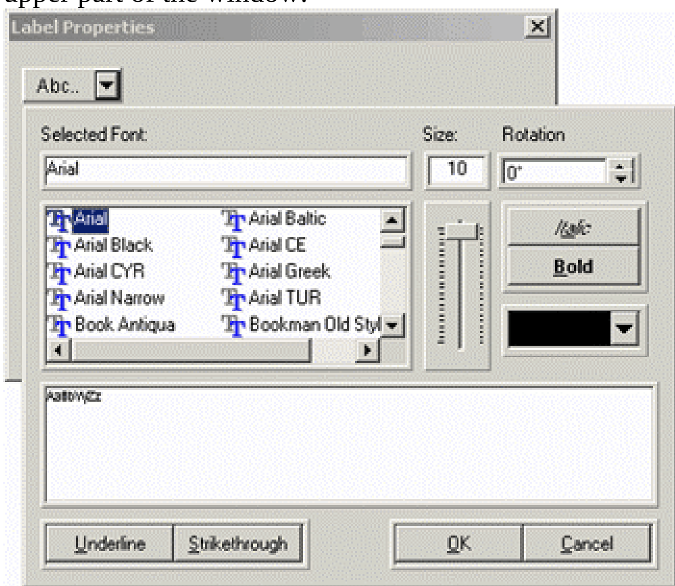
Instructions on how to create each of the cultural data types follow:

Labels

To create a new label:

1. Click the label button on the IVM toolbar:


2. The mouse pointer changes to a crosshair with a question mark. Click the mouse at the location on the document at which the label is to be inserted.
3. The Label Properties dialog box opens:


4. Enter the desired text in the text field. The text display properties can be edited by clicking the **Abc…** button in the upper part of the window:


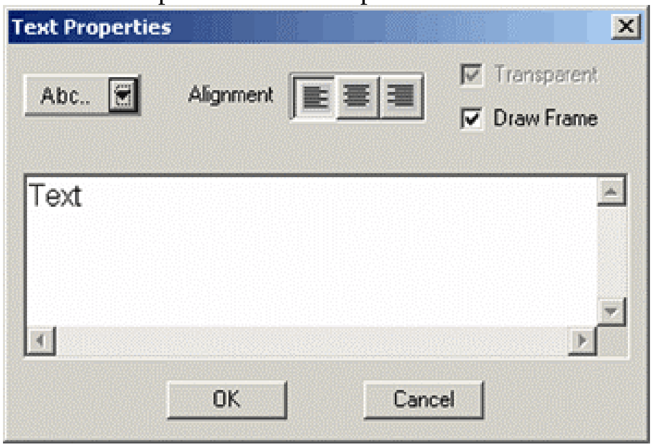
In addition to standard text formatting features, the Text Format dialog box includes an option for rotating the label.

5. After entering the text and setting any formatting properties, click **OK** in the Label Properties dialog box to close the window. The label is positioned on the document at the desired location with the specified format properties.

Text boxes

To create a new text box:

1. Click the text box button on IVM toolbar:

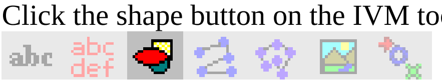

2. The mouse pointer changes to a crosshair with a question mark. Click the mouse at the location on the document at which the text box is to be inserted.
3. The Text Properties window opens:


While similar to the Label Properties window, the Text Properties window includes additional options for text alignment, drawing a frame around the text and adding transparency to the element.

   Enter the desired text in the text field. Formatting properties can be assigned to the text by clicking the **Abc…**option in the upper part of the window, which opens the same Text Format dialog box used for label elements.
4. After entering the text and setting any formatting properties, click **OK** in the Text Properties dialog box to close the window. The text is positioned on the document at the desired location with the specified format properties.

Shapes

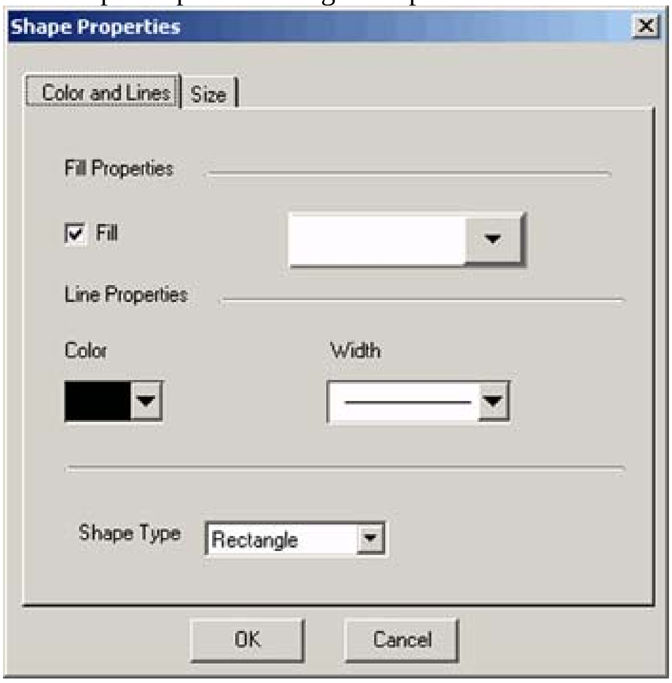
To create a new shape:

1. Click the shape button on the IVM toolbar:


2. The mouse pointer changes to a crosshair with a question mark. Click and hold down the mouse at the location on the document at which the shape is to be inserted and, keeping the mouse

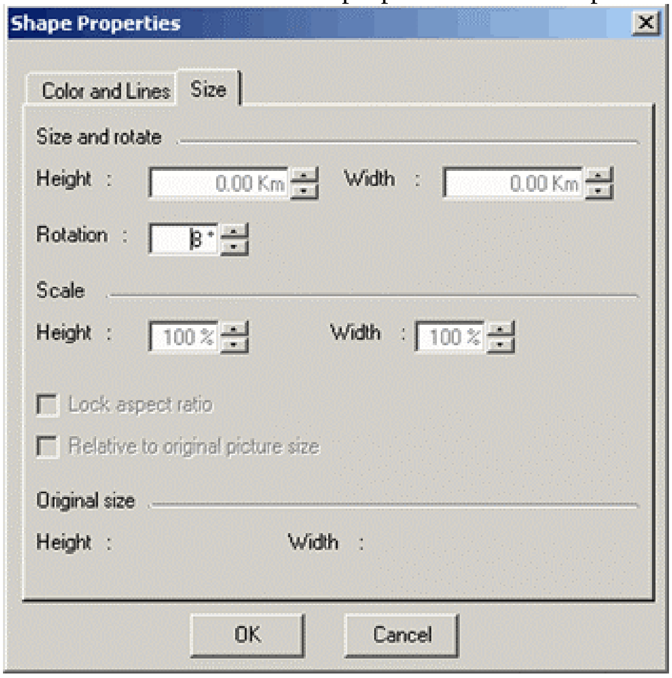
button depressed, drag the mouse to create a box of the desired size.

3. Upon releasing the mouse, a shape of the default type appears (rectangle). The shape properties can be edited by
 - double-clicking the shape
 - right-clicking the shape to open the shortcut menu; when open, select the **Properties** command.
 - clicking the **Edit Object Properties** button on the IVM toolbar:

The Shape Properties dialog box opens:


In the Color and Lines tab of the Shape Properties dialog box, set the display properties for the object and select the shape type (rectangle or oval). Click the **Fill** check box to enable the **Fill** button. The IVM fill feature, which is described in the Tools section of this chapter, can be used to fill the shape with a standard- or user-defined fill pattern.

4. Click the Size tab to set size properties for the shape:



The **Height** and **Width** fields are not active for shape objects. Positive rotation is counterclockwise about the upper left corner.

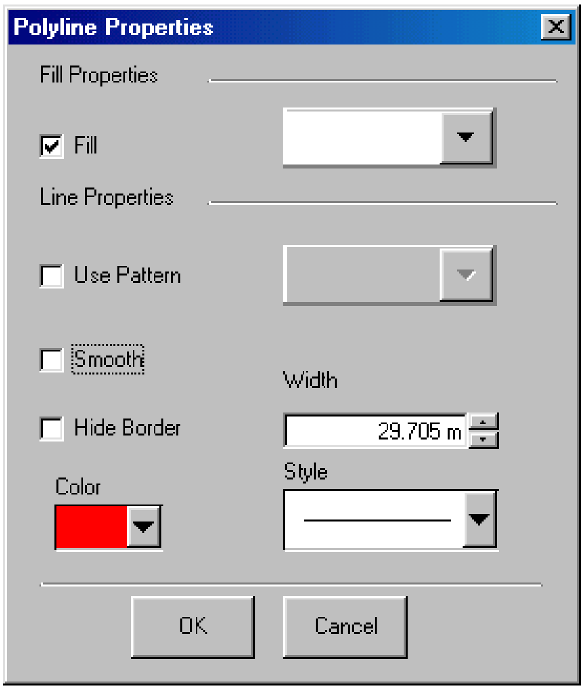
Polylines

To create a new polyline:

1. Click the polyline button on the IVM toolbar:

2. The mouse pointer changes to a crosshair with a question mark. Click the mouse at the location on the document at which the polyline is to begin. Move the mouse to the location of the next vertex and click again. Continue this procedure until the desired polyline has been created. Double-click the mouse to set the final vertex.

3. Immediately after setting the final vertex, the Polyline Properties window opens:



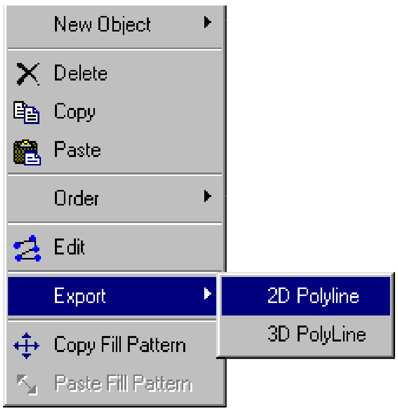
Set display properties as desired and click **OK**. The value entered in the width field specifies the border width in database units.
Note: The IVM fill and pattern features are described in the Tools section of this chapter. The **Use Pattern** option in the Line Properties section of the window can be used to augment the defining line with a pattern, e.g. x marks, circles, triangles.

4. A polyline object can be scaled or resized by first selecting it with the mouse, then grabbing one of the yellow handles with the mouse pointer and pulling it to the desired size.

Polylines can also be exported as external text files and then as well as reimported into other documents.

To export a polyline:

1. Click the polyline object to select it.
2. Right-click the mouse on the document window to open the shortcut window:

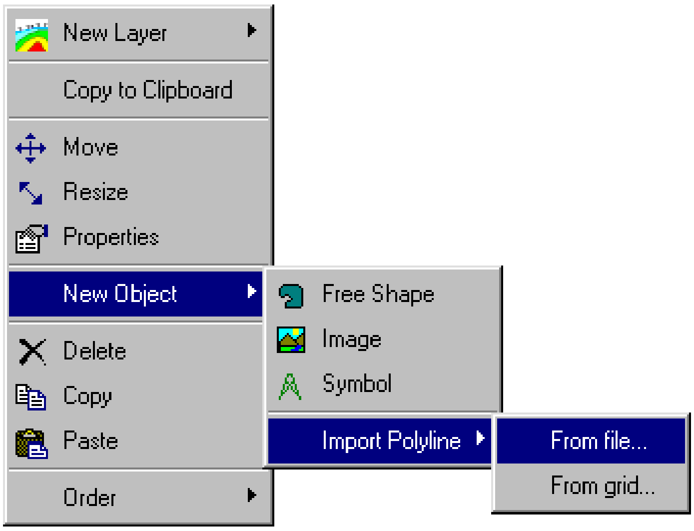


Select from 2D and 3D polylines. 2D polylines are defined by distance along the profile and depth and can be reimported into other IVM documents. 3D polylines contain actual geographic coordinates and cannot be reimported.

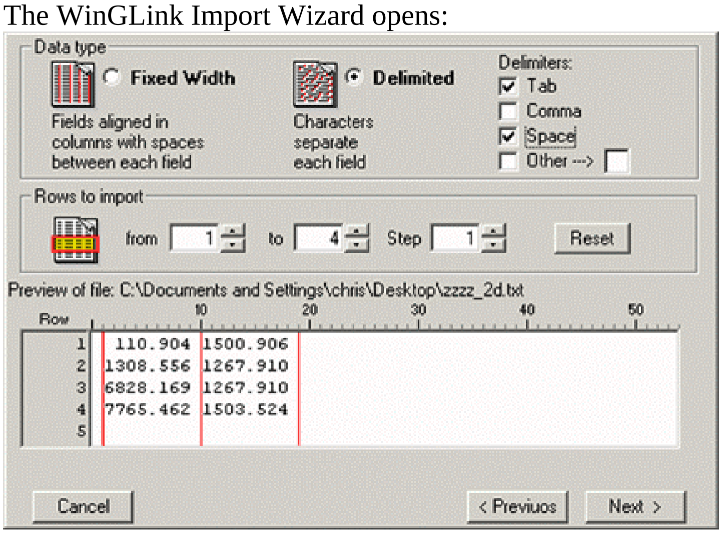
3. Select the Export command to open the standard Windows dialog box for saving files.

To import an external polyline definition:

1. Make certain that a Cultural Data layer is enabled for editing in the Layer Navigator.
2. Right-click the mouse on the document window to open the shortcut menu:



3. Click the **Import Polyline** command to open the standard Windows dialog box for opening files. Browse the directories until you have found the desired file, then click **Open**.
4. The WinGLink Import Wizard opens:



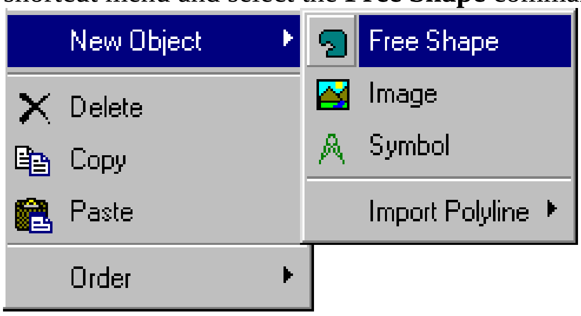
The Import Wizard, which is described in detail in Chapter 3 of the WinGLink manual, guides you through the import process. After providing the requested information, the polyline data will be imported from the external file and the object created in the active cultural data layer.

Closed polylines:

Closed polylines are similar to open cousins, the primary difference being that closed polylines automatically connect the last node to the first node, resulting in a closed element.

To create a closed polyline:

1. Either click the Closed polyline button on the IVM toolbar:


or right-click the mouse on the document window to open the shortcut menu and select the **Free Shape** command:


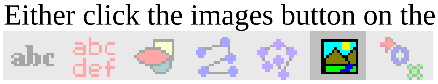
2. The mouse pointer changes to a crosshair with a question mark. As with polyline elements, click the mouse at the location on the document at which the free shape is to begin. Move the mouse to the location of the next node point and click again. Continue this procedure until the desired shape has been created. Double-click the mouse to set the final vertex and automatically connect the last vertex to the first vertex.
3. After setting the final vertex, the same properties window as for polylines opens. Set the properties as desired, then click the **OK** button.

As with polylines, closed polylines can be exported to external files and reimported. Refer to the description of polylines above for details.

Images

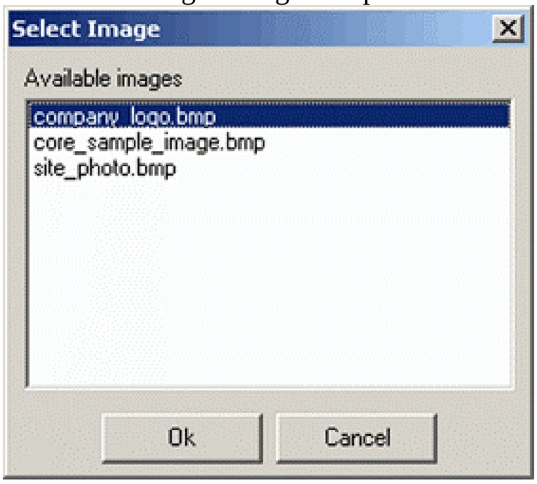
External graphics files can be imported into cultural data layers. Most standard graphic formats are supported, including BMP, JPEG, GIF, PNG, TIFF, EXIF , WMF and EMF. Before they can be imported into IVM, however, they must first be copied into a folder named **images** which is created automatically by IVM the first time the images button is clicked. This folder is located in the same directory as the current database on your hard disk. Only images located in the **images** folder are available for import.

To place an image in a cultural data layer:

1. Either click the images button on the IVM toolbar:


or right-click the mouse on the document window to open the shortcut menu and select the **Image** command.

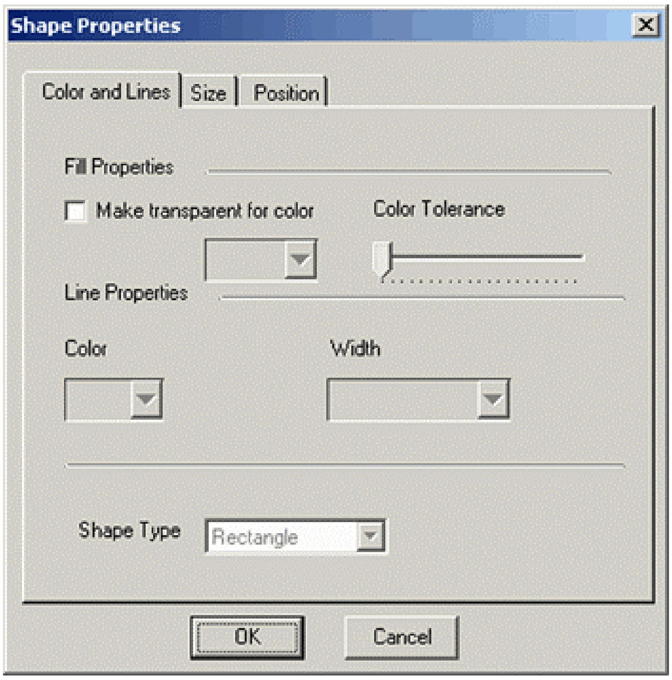
2. The Select Image dialog box opens:



This dialog box lists all images contained in the Image folder mentioned above. Select the desired image and click **OK**.

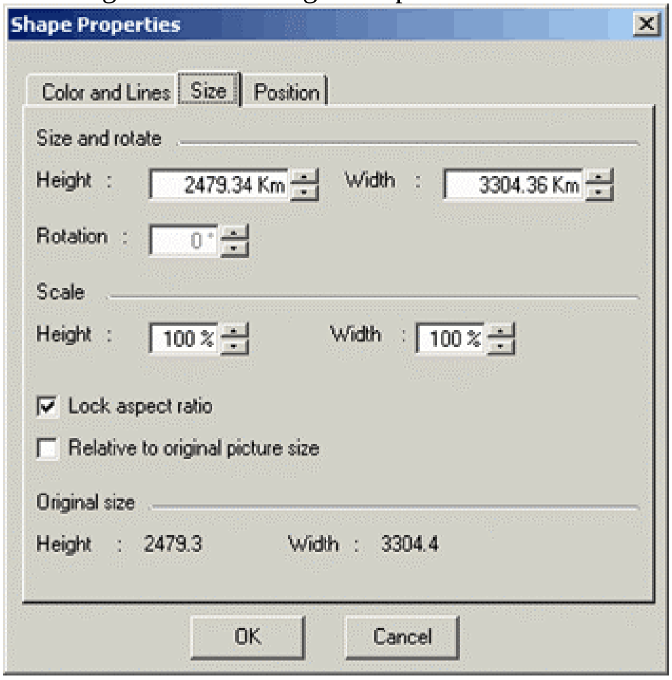
3. The mouse pointer changes to a crosshair with a question mark. Click the mouse at the location on the document at which the image is to be inserted.

4. Open the Shape Properties window by double-clicking the image or by using one of the other previously mentioned methods for opening property windows. The Shape Properties window consists of three tabs:



In the **Colors and Lines** tab, only the options in the Fill Properties section of the tab are available.

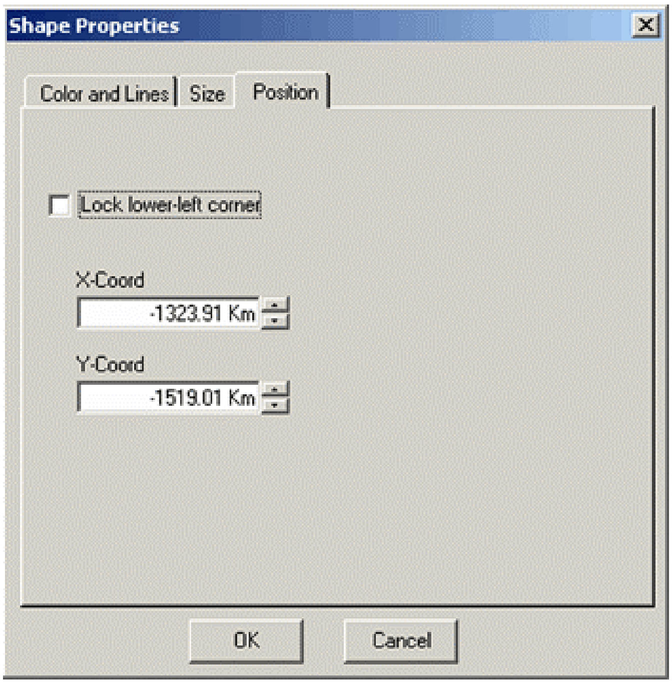
The **Make transparent for color** option is used to make a specific color or range of colors transparent. Enable this function by clicking the checkbox. Select the desired color by clicking the arrow button. You can specify the sensitivity of the transparency function with the **Color Tolerance** scale. With a low tolerance value (arrow to the left), only colors closely matching that in the color picker are made transparent. Higher values, on the other hand, make a broader range of colors transparent. This function can be used, for example, to make the background of an image transparent.



The options in the **Size** tab are used to specify image size in either absolute units or to scale the image to a relative size. The values entered in the upper set of **Height** and **Width** boxes set the image to the specified size, where the units are the same as those used in the document axes. The lower set of **Height** and **Width** boxes scales the image to the specified percent value. When the **Lock aspect ratio** checkbox is enabled, the height and width values change in proportion to one another, preventing the image from becoming distorted as the size is changed.

When the **Relative to original picture size** checkbox is enabled, the scale values displayed are relative to the original image size, which is displayed at the bottom of the window. If not enabled, the scale values are relative to the current image size, i.e. size of the image when the Shape Properties window was opened.

The **Rotation** option is not available for image elements. The origin of images is the lower-left corner.



You may set the position of the image to a specific value on an IVM document using the fields provided in the **Position** tab. The **X-Coord** and **Y-Coord** fields refer to the lower left and lower bottom corners of an image, respectively.

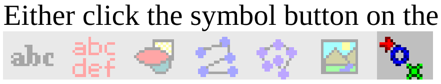
If the **Lock lower-left corner** checkbox is enabled, the image location is fixed on the screen. While the image can still be resized, the position of the lower-left corner does not change.

5. Shape objects can be scaled or resized either by using the options provided in the Shape Properties dialog box or by using the mouse. To manually resize a shape element, select it by clicking with the mouse, then grab one of the yellow handles with the mouse pointer and pull to the desired size.

   Note: Options set in the Shape Properties window remain in effect when manually changing the image size and location. For example, when the **Lock aspect ratio** checkbox is enabled, the image proportions are retained when the image is resized with the mouse.

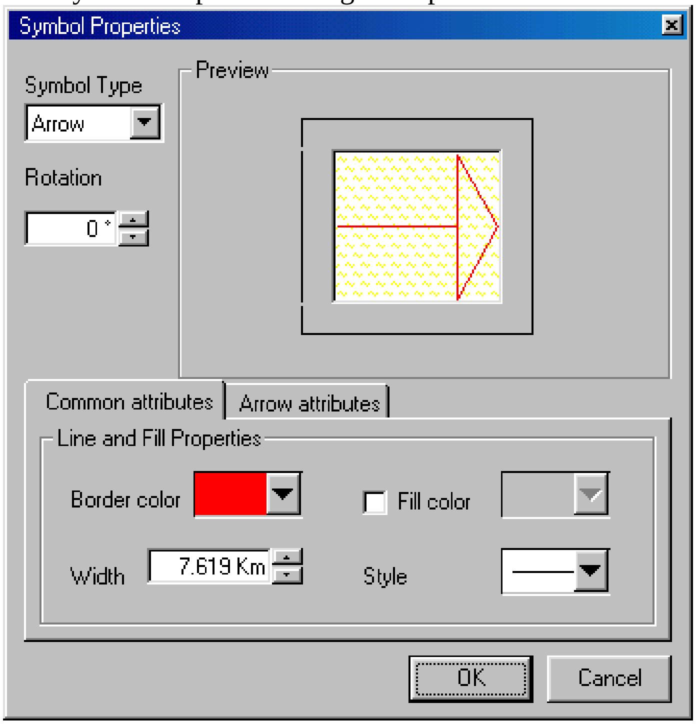
Symbols

To place symbol in a cultural data layer:

1. Either click the symbol button on the IVM toolbar:
   
   or right-click the mouse on the document window to open the shortcut menu and select the **Symbol** command.
2. The mouse pointer changes to a crosshair with a question mark. Click and hold down the mouse at the location on the document at which the symbol is to be inserted and, keeping the mouse button depressed, drag the mouse to create a box of the desired size.
3. As with shape elements, a symbol of the default type appears (arrow). The symbol properties can be edited by

- double-clicking the symbol
- right-clicking the symbol to open the shortcut menu; when open, select the **Properties** command.
- clicking the Edit Object Properties button on the IVM toolbar:

4. The Symbol Properties dialog box opens:



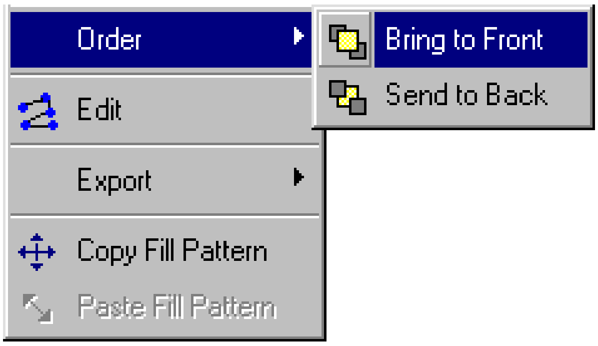
Chose the symbol type from the dropdown list. Symbols currently available are: arrows, rectangles, circles and crosses. Set the remaining parameters as desired. Click **OK**.

Working with Cultural Data Layers

As mentioned earlier in this section, cultural data layers can themselves be considered a type of document. They may consist of multiple elements, each of which is located on a separate layer. They may be stored in the current WinGLink database and reused in other IVM documents.

Z-order
To change the Z-order of cultural data elements within a cultural data layer:

1. Click the cultural data element which is to be repositioned.
2. Right-click the mouse on the document window to open the shortcut menu. Position the mouse over the **Order** command and select either the **Bring to Front** or **Send to Back** command:



Bring to Front places an element on the top of the cultural data layer, **Send to Back** places an element on the bottom.

Reusing cultural data elements
Cultural data elements can be duplicated within a cultural data layer by using copy and paste functions.

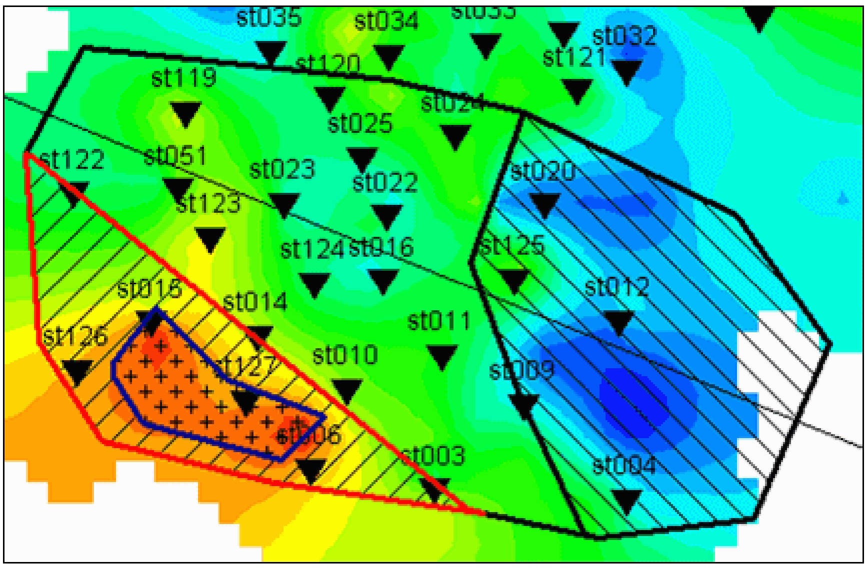
To create a copy of a cultural data element:

1. Click the cultural data element which is to be copied.
2. Right click the mouse on the document window to open the shortcut menu. Select the **Copy** command.
3. **Again** click the cultural data element which is being copied to select it.
4. Right click the mouse on the document window to open the shortcut menu. Select the **Paste** command. A copy of the data element appears on the document.

Note: It is not possible to copy a cultural data element from one layer to another.

Interpretation Areas

Interpretation area layers are used to denote specific areas of interest on a map or section document. This layer type is similar to the cultural data layer type in that the layers can be saved in the WinGLink database and reused on other documents:



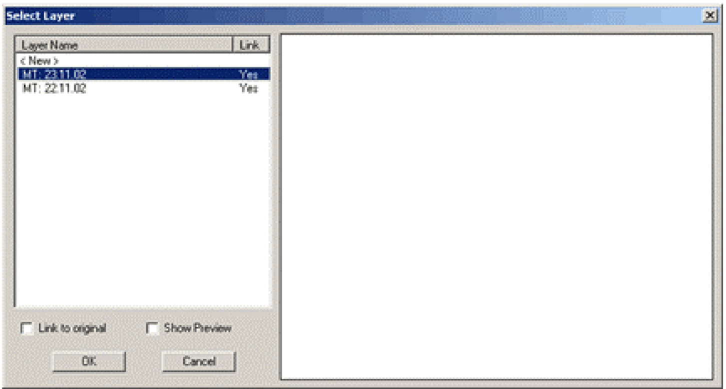
These layers consist of objects similar to the free shapes used in cultural data layers. Interpretation areas are drawn on a document in the same way as shapes. They are provided, however, with additional functionality. You can:

- add, move or remove vertices
- change the shape of an interpretation area by moving or deleting segments
- assign colors and fill patterns
- split interpretation areas and assign different properties (colors, fill patterns) to each part of the area
- position interpretation areas within interpretation areas

Creating Interpretation Areas

To create an interpretation area layer:

1. Right-click the mouse anywhere on the document window or right-click the document title in the Layer Navigator window to open a shortcut menu. Then position the mouse over the **New Layer** command to open the submenu; select the **Interpretation** command.
2. The Select Layer window opens. Listed here are all Interpretation areas which are stored in the active WinGLink database:

   To create a new interpretation area layer, select the **<New>** item in the Layer Name column and click **OK**. Likewise, an existing layer can be inserted into the document by selecting the layer name and clicking **OK**.

   When inserting an existing interpretation area layer into a document, you may insert either the original layer or a copy of it. If you select the **Link to Original** check box when inserting an existing layer into an interpretation area layer, any change made to the layer will be reflected in any other documents / layers which use the layer, provided they, too, use the layer in linked mode. If the **Link to Original** check box is not selected when inserting an existing layer into a document, a new interpretation area layer is created and will be listed in the Select Layer window the next time an interpretation area layer is created.

3. If creating a new interpretation area, IVM displays an info box instructing you to insert an interpretation area. Confirm by clicking **OK**.
4. The mouse pointer changes to a crosshair with a question mark. Click the mouse at the location on the document at which the interpretation area is to begin. Move the mouse to the location of the next vertex and click again. Continue this procedure until the desired interpretation area has been created. Double-click the mouse to set the final vertex and automatically connect the last vertex to the first vertex.

Working with Interpretation Areas

A set of commands are available for working with interpretation areas. These can be accessed either from the IVM toolbar or via the shortcut menu. The commands are available only when an interpretation area is the active layer.

- Split an existing body 
  Use this command to split an interpretation area into two parts. May be used successively to further subdivide an area.

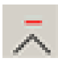
- Create a closed body 
  Use this command to create a new body on the current interpretation area. New bodies must be entirely contained within an existing body and may not intersect any other bodies existing in the body into which the new body is being inserted.

- Body union 
  Use this command to unite two bodies. These may be either two halves of one body or a body enclosed with another. The two bodies to be united must be clicked. The entire area will take the properties of the second body which is clicked.

- Move vertex 
  Use this command to drag a vertex to a new position.

- Move segment 

- Delete vertex 

- Delete segment 

After clicking any of the above command buttons or after selecting one of the corresponding shortcut menu commands, IVM is in edit mode. The respective command remains active until you:

- Click a different command button.
- Right-click the mouse on the document window to open the shortcut menu and select the **Exit Edit** command.

- Activate a different layer in the Layer Navigator.
- Create a new layer.

GEOTiff Maps

What are GEOTiffs?

The GEOTiff specification provides a mechanism which can be used to embed georeferencing information in TIFF images. By incorporating a series of tags in a TIFF image, it is possible to correctly position a map, for example, onto a given coordinate system. While WinGLink cannot yet be used to create GEOTiff images, images created using other software packages can be correctly positioned onto map documents in the IVM program.

For details on the GEOTiff specifications, see: http://remotesensing.org/geotiff/spec/geotiffhome.html

Numerous third-party software houses have developed and sell software packages which can be used to generate GEOTiff images.

Creating GEOTiff Map Layers

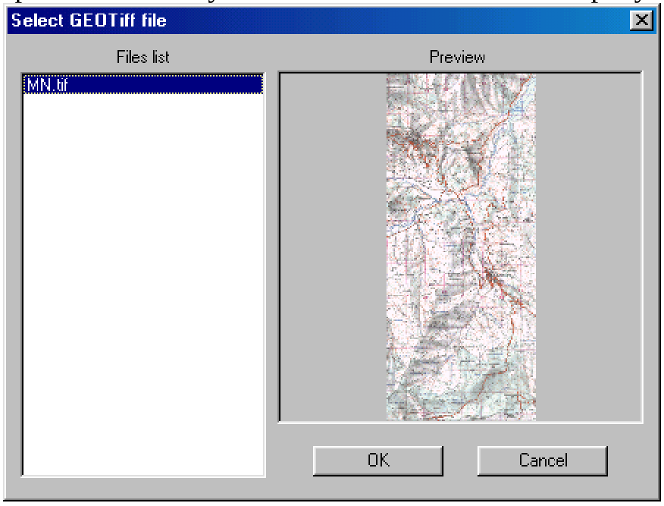
As the name implies, this type of layer can be inserted only Map documents only. Theses layers are created much in the same way as other layers:

To create a station layer:

1. Right-click the mouse anywhere on the document window or right-click the document title in the Layer Navigator window to open the shortcut menu. Then position the mouse over the **New Layer** command to open the submenu; select the GEOTiff Map item:

2. The GEOTiff selection window opens. All files with a .TIF or .TIFF extension which are located in the images folder located in the same directory as the current database are listed. Note, however, that only .TIF images which fulfill the GEOTIFF

specifications may be inserted into a GEOTiff Map layer:



3. After clicking **OK**, a progress window opens indicating that the georeferencing data are being read from the tiff file. Once complete, the image is positioned at the correct coordinates on the map.

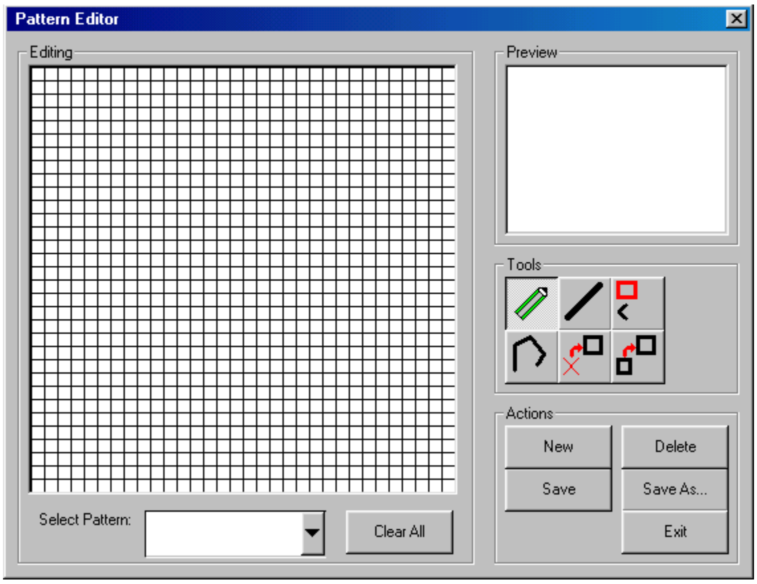
Tools: Pattern and Category Editors

The Pattern Editor

The pattern editor is used to create and edit fill patterns. Theses patterns can be used, for example, to fill shapes used in cultural data layers or as fill patterns to describe layers of vertically distributed data. Patterns created in the fill editor are available in the property windows of all elements which support fill patterns, both in IVM and in other WinGLink modules. In the main WinGLink modules, this feature available only for sections which contain well courses, e.g. X-Sections.

To open and use the Pattern editor:

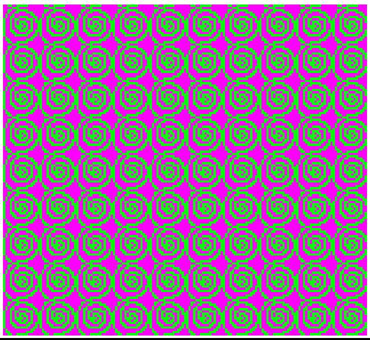
1. Select the **Tools | Pattern Editor** command to open the Pattern editor:



2. To create a new pattern, position the mouse over the Editing window, then click and hold down the left mouse button. Slide the mouse across the edit window to create a pattern. You may alternatively click individual squares to create a pattern. When finished, click the save button.

3. To edit an existing pattern, open the **Select Pattern** dropdown list. Make any changes to the pattern, then click the **Save** button to overwrite the pattern with your changes, or **Save As…** to save your changes to a new pattern.

Note: The only color available in the pattern editor is red. When using patterns as a fill, you may, however, select both foreground and background colors.



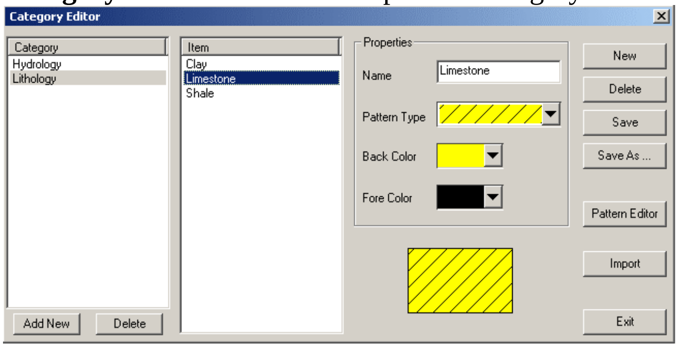
The Category Editor

The Category Editor is available in IVM as well as the X-Sections, 2D Inversion and Well Editor modules. The categories are centrally stored in the database. Thus, regardless of the program in which categories are edited or new categories created, the modifications are reflected throughout the database.

Once categories, or types of layers, have been created, any time another layer of that type is encountered in a section within the database, WinGLink automatically performs filling.

To Assign Fill Patterns to Layer Data:

1. On the **Tools** menu of the X-Sections window, select the **Category Editor** command to open the Category Editor:



Displayed in the Category frame are all existing categories. Categories are created automatically when importing layer data into WinGLink and can also be created using the **Add New** button located at the bottom of the left column.

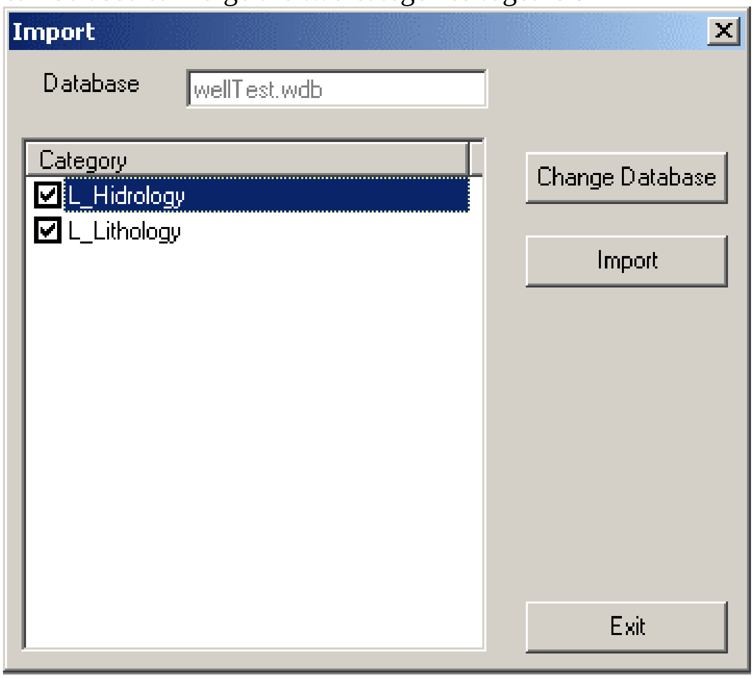
The elements in the Item frame are the description elements contained in layer-data files which were linked to the given category type during import. New items can be created using the New button located in the upper right corner of the Category Editor.

2. Select a category and item with the mouse. The current pattern type is displayed in the **Pattern Type** dropdown box.

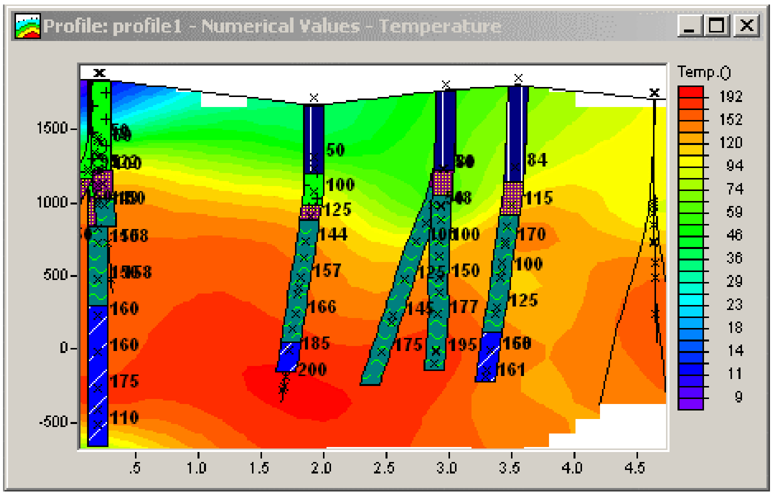
3. Click the arrow at the right end of the **Pattern Type** dropdown box to open a box containing all available fill patterns. Choose back color and fore color for the filling using the provided drop-down boxes.

4. The Pattern Editor, which can be opened using either the button provided in the Category Editor or by selecting the **Tools | Pattern Editor** command on the main menu, can be used to create new patterns.

The Import function, which is called up using the **Import** button in the Category Editor, provides a function for importing entire categories from other databases. If you import a duplicate category, the Category Editor can be used to merge the two categories together.



Upon closing the Category Editor, well courses are filled with the appropriate fill patterns:

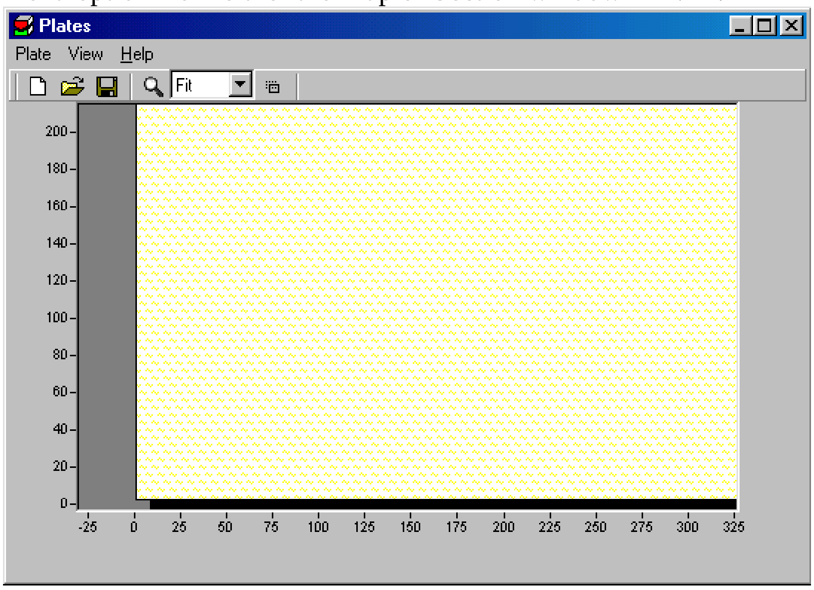


The IVM Plates Module: Preparing Documents for Printing

After either map or section documents have been created and saved in Interpreted Views and Montage (IVM) they exist within the current WinGLink database as independent objects. Using the IVM **Plates** module, these documents can be combined with other IVM display elements, e.g. scales, legends, text boxes, or even other maps or sections to annotate documents for plotting. Once completed, plate documents can be sent to either an installed printer or to a file (CGM or EMF).

Getting Started with the Plates Module

To open the Plates module, select the **Document | Go To Plate Mode…** menu option from either the Map or Section window in IVM:

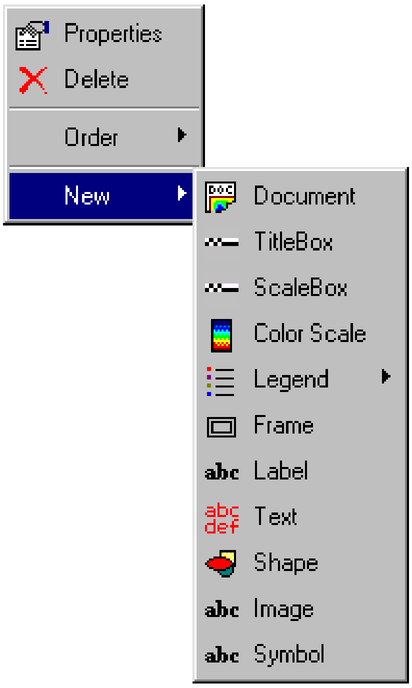


The yellow area in the center of the can be thought of as the layout area: documents, legends, texts etc. are printed as they are positioned in this area.

The size of the layout area, which is identical to the page size, can be set by selecting the **Plate | Page Setup** command. In the Page Setup dialog box which opens, select from a range of standard paper sizes. The page size is reflected in the plate axes and the size of the layout area.

To Display a Document Object on a Plate:

1. Right-click the mouse anywhere in the layout area to open a shortcut menu:

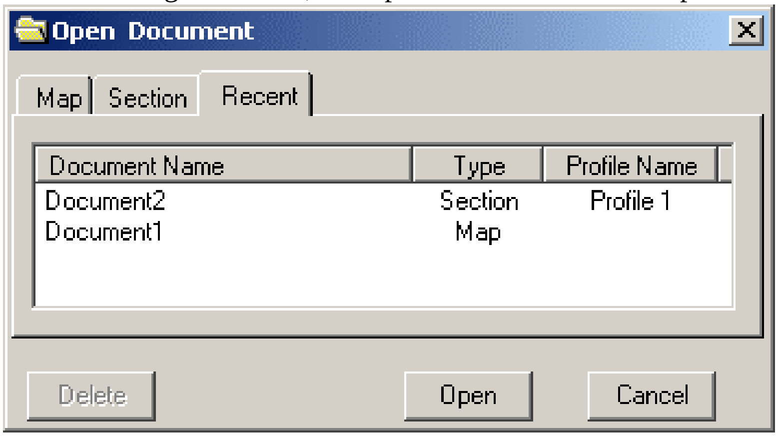


2. Position the mouse over the **New** command to open a secondary menu containing all display elements. Click the desired type to select it.

3. Depending on the type of element selected, the mouse pointer may change to a crosshair, indicating that you are to use the mouse to create the area on the plate into which the element is to be inserted, or you may be prompted to select an image.

4. To insert a map or section document, select the **Document** command. The mouse switches to a crosshair with a question mark. Position the mouse pointer at the intended location of one of the document corners; click the left mouse button and drag the mouse across the screen, creating a black-framed box in the process. When the box is approximately the correct size, release the mouse. The box changes to a blue-framed box with an x though it. This box may be repositioned by clicking the mouse anywhere within its boundaries and dragging it to a new location. It may also be resized by clicking any of the yellow handles and pulling to a new location on the screen.

5. After releasing the mouse, the Open Document window opens.



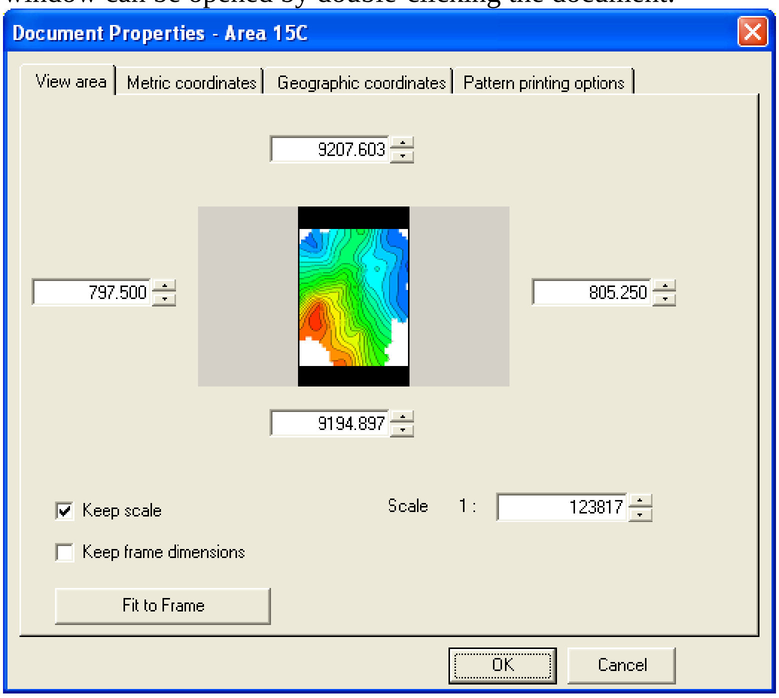
Select the desired document and click the **Open** button.

Note: The most recently viewed documents are listed in the Recent tab. All other IVM map and section documents contained in the current WinGLink database are available for selection in the respective Map and Section tabs.

6. The document appears on the screen in the boundaries defined by the blue-framed box in Step 4. The document may be repositioned or resized using the same methods described in Step 4.

Adjusting Document Properties

Depending on the size of the defining rectangle, the document may not be displayed in its entirety. The view area, as well as other document display properties, can be adjusted in the Document Properties window. This window can be opened by double-clicking the document:



The View area tab is available for both map and section documents. For section documents, the two center tabs are Distances and Depths; for map documents they are Metric coordinates and Geographic coordinates. Use the settings in the two center tabs to set display features such as font, grids, colors and frames. The Pattern printing options tab contains options for scaling the size of any patterns used in the document. The Axes Legends tab, which is available for Section documents only, can be used to set legend texts and other display parameters.

Note: In order to display the axis legends, the **View tickmark** checkboxes in the Distances and Depths tabs of the Document Properties window need to be enabled.

To adjust the view area:

Use the four fields on the sides of the preview window and the scale field(s) to adjust the display area. Note, however, that the function of these fields varies depending on the selection states of the **Keep scale** and **Keep frame dimensions** check boxes. Maps documents contain only one scale field; section documents contain scale fields for both the horizontal and vertical scales.

If the **Keep scale** and **Keep frame dimension** checkboxes are selected, changes to the view area fields serve to "pan" the document within the bounding rectangle. Changes to the scale fields zoom in or out on the document (sections only), effectively adjusting the vertical exaggeration. Click the **Fit to Frame** button to automatically center the document in the bounding rectangle.

If the **Keep frame dimension** checkbox is *not* selected, scale is maintained as the view area fields are changed. The size of the bounding rectangle is, however, changed. Click the **Fit to Frame** button to automatically center the document in the bounding rectangle.

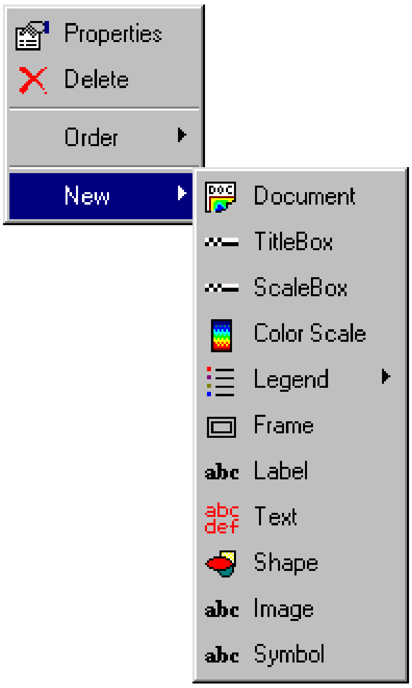
If the **Keep scale** checkbox is not selected, the bounding rectangle remains a fixed size. Changes to any of the view area fields are reflected in the scale fields in the lower right corner. Likewise, changes made to the scale fields are reflected in the view area fields. Click the **Fit to Frame** button to fit the entire document in the bounding rectangle.

Other Display Elements

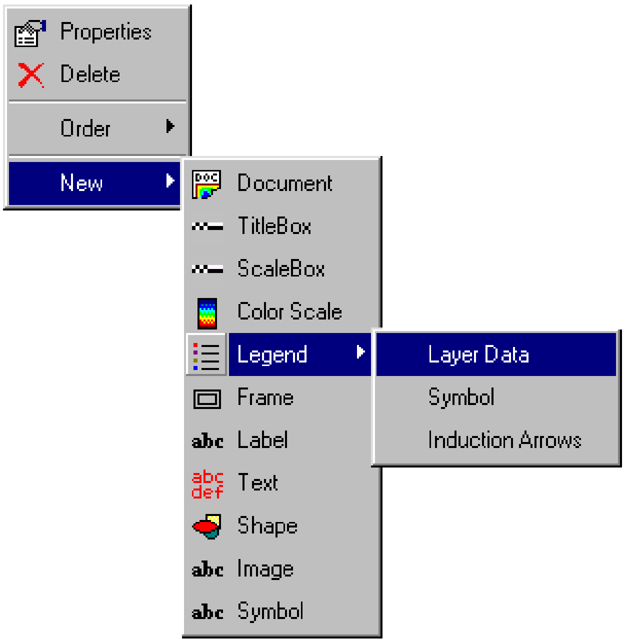
The other types of display elements which can be added to a plate are typically associated with a single document. For example, a Scale Box display element reflects the scaling used for a specific Document element.

To create (non-document) display elements and assign them to specific document elements:

1. As with document elements, right-click the mouse anywhere in the layout area to open a shortcut menu:

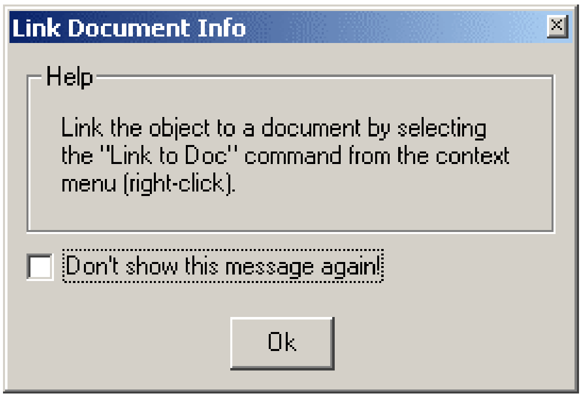


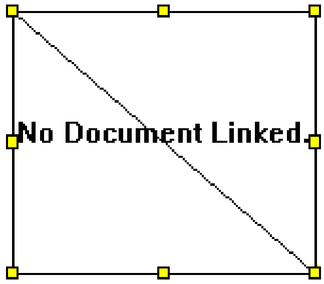
2. For example, to position a legend for layer data contained in a section, i.e. well data, select the **New | Legend | Layer Data** command from the shortcut menu:



3. After selecting a display element, the mouse switches to a crosshair with a question mark. Use the same procedure as was used to position a new document element on the screen to create a box into which the legend (or other display element) is to be placed.

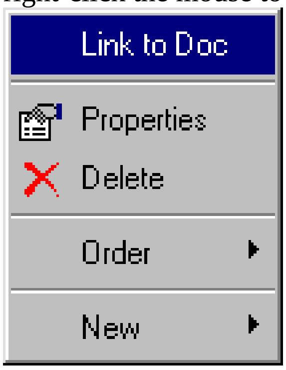
4. Immediately after releasing the mouse, an info box opens indicating that the element has not yet been linked to a document:





The Link Document Info message box serves only to inform you that the element must be linked to a document before it can be filled with the appropriate information. The lower box, which contains the text "No Document Linked", defines the rectangle into which the element will be placed once the element has been linked to a document.

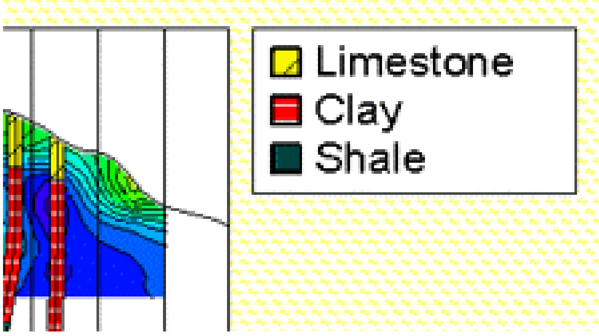
5. Position the mouse within the boundaries of the new box and right-click the mouse to open a shortcut menu:



Select the **Link to Doc** command.

6. The mouse pointer changes to a plain crosshair. Now, select the document with which the given display element is to be associated by clicking the document with the mouse. If the document supports the element, it should now be displayed

correctly on the screen:



7. The display properties, i.e. text color/size, frame width etc., for each element can be modified by selecting the element with the mouse, then right-clicking to open the shortcut menu. Select the **Properties** menu item to open the Properties window, which contains standard formatting options. You may alternatively double-click a display element to open the Properties window.

   Elements which are not supported by the document to which they are linked are displayed as an empty, white box. These can be deleted by right-clicking the element and selecting the **Delete** command from the shortcut menu.

Other display elements which must be linked to documents include:

- scale box: provides information about scaling, and, for section documents, information about vertical exaggeration
- color box: displays the same color range information as is displayed in the original document (color range window)
- legend: layer data: provides a visual link between fill types and layer names for vertically distributed data
- legend: symbols: provides a visual link between symbols used to indicate measurement points for numerical data and the corresponding symbol names for vertically distributed data, i.e. temperature or pressure measurement points along well courses

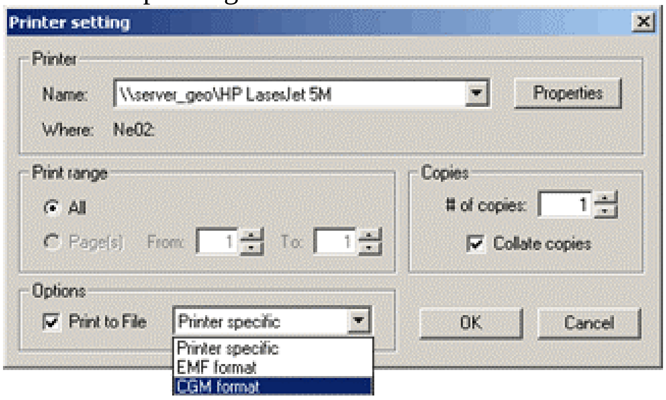
The remaining display elements, e.g. title box and frame, are for document-independent annotation purposes and, thus, are not associated with a specific document.

Printing Plates

Plates created within the IVM plates module are printed much as any other document would be in a standard Windows program. One notable difference, however, is the fact that documents can be printed to files of types EMF and CGM.

To print to a file or to a printer other than your default printer:

1. On the **Plate** menu, select the **Printer Setup** option to open the Printer Setup dialog box:



2. To print to a file, click the **Print to File** checkbox in the Options area of the screen, then select the type of file to which you would like to print.
3. To change other printing options, e.g. printer name, print range, proceed as you would with any other Windows program.
4. To send the plate to the selected printer or file, select the **Plate | Print** command.

Important:
When printing to EMF of CGM files, note the following: Although output is not sent to a physical printer, the paper properties of the printer selected in the Printer section of the Printer Settings dialog are used. Each output file can be considered to be a page of printer output. Because the entire output is sent to a single file, however, you must ensure that the paper size selected for the printer is large enough for the entire print area, i.e. fits within the red border which defines the first page of output in the Print Preview screen. Any information outside of the first page will be clipped in the output file.

26: Appendix A: TGF files

The TFG Format

In order to import vertically distributed data, including lithologic data, into WinGLink, they must be contained in either columnar text files or in TFG-format files. This description describes the TGF specification and how to work with TGF files.

The TGF format is an expandable data format specified by Geosystem for the purpose exchanging geophysical data across applications. A given TGF file may contain station data for one or more stations, e.g. name and coordinates, as well as data values for one or more values, e.g. temperature and density.

TGF files may also link to other TGF files. In this way it is possible to create a batch file of sorts. For example, the top-level file might contain station data and links to several files containing different value types, e.g. temperature, pressure and density.

Organizing Station Data in TGF-Format Files

TGF files are organized by block using markup tags in a way similar to those used in standard markup languages such as HTML. Within the tags, data are organized primarily as tabular data.

Example of a TGF file structure:

```
<TGF100>
; this is a comment

<TAG1=param1 param2>
 (user data…)
<TAG1>

</TAG2>
 (user data…)
</TAG2>

<TAG2>
 (user data…)
</TAG2>

<TAG_A>
 (user data…)
</TAG_A>
```

```
<INCLUDE="Stations.tgf">

<END>
```

The file MUST begin with the string `<TGF100>`. The 100 indicates version 1.00 of the file format, allowing for future changes.

A TGF file consists of a series of `<TAG>…</TAG>` pairs, which surround user data. All opening and closing tags, i.e. `<TAG>` and `</TAG>`, must stand alone on a line. Opening tags may contain parameters, i.e. `<TAG= param1>`.

Every TGF file must end with an `<END>` tag. Anything after this tag is ignored.

There are two types of tags:

1. Multi-line tags, which act as data containers, primarily of tabular data
   Example:
   ```
   <DATA=STAT_COORDS>
   …
   </DATA>
   ```
2. Single-line tags, which are used for less complex data. No ending `</TAG>` is used for this tag type.
   Example:
   ```
   <INCLUDE="stations.tgf">
   ```

Valid Tags

TGF100

File type/version, must be on the first line. Without this tag the file will be rejected.

Example:

```
<TGF100>
...
...
(rest of file)
```

WAIT

This tag instructs the importing program to stop interpretation of the file until the user acknowledges a message displayed on the screen.

Example:

```
<WAIT=Press a key to continue> ; this will wait for user
```

DATA=*NAME*

This is a general user data/table container. The type of data is specified by the name. These are multi-line tags and must be formatted as follows:

- One or more spaces or tabs must separate user data columns.
- The decimal separator is "." (dot); no thousands separator is allowed.
- Numbers can be expressed in scientific notation.
- Text data must be enclosed in " " (quotation marks).
- Blank lines are ignored.

DATA=STAT_COORDS

Defines a group of stations

Table column definition is:

1. "Station name"
2. X (northing) [Km]
3. Y (easting) [Km]
4. Elevation [m]

Example:

```
<DATA=STAT_COORDS>
; This table builds station locations
"st 01"    34.456     21.451     500.2
"st 02"    31.456     13.455     534.7
"st 03"    24.456     43.552     400.9
"st 04"    14.461     65.952     840
"st 05"    74.556     28.457     510
</DATA>
```

DATA=VSAMPLE_TYPES

Defines a table of vertical sample names, with their attributes, and unit of measure and number of decimals to display.

Table column definition is:

1. NAME
2. Unit
3. Ndec

Example:

```
<DATA=VSAMPLE_TYPES>
; This section defines the names of used vertical
vector types
```

```
"Resistivity"    "rho"           3
"Temp"           "C"             2
"Pressure"       "Kpa/cm"        3
</DATA>
```

DATA=VSAMPLE

Defines a vertical sample, its sample type and the reference for Z coordinate.

`ZMODE`="Elev" means Z = 0 at sea level, positive upward

`ZMODE="Depth"` means Z = 0 at station surface, positive downward

`STATION="Name"` name of station to which station is to be associated

`STYPE="Name"` join sample to sample type "name"

Table column definition is:

1. Z [m]
2. value [unit of vector type, if any]

Example:

```
<DATA=VSAMPLE>
ZMODE="Depth" ;depth or elev
STATION="st 02"
STYPE="Resistivity"
122.34     45.223
234.23     167.332
455.43     123.23
1120.34    145.223
1234.23    171.332
1457.43    231.23
2126.34    252.223
2634.23    167.332
2795.43    283.23
3120.34    458.223
3234.23    676.332
3455.43    532.23
</DATA>

<DATA=VSAMPLE>
ZMODE="Depth"
STATION="st 02"
STYPE="rho"
```

```
122   48.223
238   117.332
457   123.23
1100  175.223
1234  171.332
1437  231.23
2128  212.223
2654  117.332
</DATA>
```

INCLUDE

This tag allows the interpretation of another file before continuing with the current one.

It is useful for keeping homogenous data in separate files.

Processing will continue with the external file and will resume execution from the next line

Included files cannot call each other (circular reference). If "file A" includes "file B" then "file B" can't include "file A"

Be careful!

If the included/called file has some tag present also in the calling file, the data of the last tag read will be kept.

The include tag can be in any part of the file, but must be outside of other tags.

Example:

```
; External links to other files
<INCLUDE="stations.tgf"> ;file stations.tgf will be read and..
<INCLUDE="vectors.tgf"> ; then file vectors.tgf
```

END

This tag ends interpretation of file. It must be present

Example:

…

…

```
<END>

These lines are ignored
These lines are ignored
These lines are ignored
These lines are ignored
```

```
These lines are ignored
```

(Physical end of file)

Importing TGF Files into WinGLink

TGF data are imported into WinGLink using the **Importing station data** command located in the main Database window.

To import TGF files into WinGLink:

1. On the **File** menu of the Database window, select **Importing station data**, then specify **External Files**.
2. Next, select the project into which the data are to be imported. For vertical data, the project must be of type **Vertically Distributed Data [VD],** for wells of type **Wells [WL]**
3. In the <**Files of Type**> box, select **TGF Files**.
4. Browse the directories until you find the file(s) to be imported.
5. Select the name of each file to be imported
6. Click **Next**.
7. Click each of the stations which are to be imported.

Note: If no stations are listed, either no stations are contained in your TGF file or your file is not conformant with the TGF specifications.

27: Index

C

D

E

G

I

M

P

S

T

V

W

X

Z