

User Guide

ELETOM

Electrical Tomography

geo&soft[®]
international

Geo Soft di ing. G. Scioldo

Summary

Chapter 1 - Introduction to the program	1
Introduction to the program	1
Chapter 2 - System requirements and program installation	2
System requirements	2
Program installation.....	2
Chapter 3 - The program protection	7
Program registration.....	7
Chapter 4 - How to update the program	8
Procedure for updating the program	8
Autoupdating requirements and troubleshooting	9
Chapter 5 - User Interface	10
Usage Notations.....	10
User Interface and Data Entering.....	10
User Interface: Menu Bar and Menus	10
The Input Dialogue Windows	11
Data Input With Tables.....	12
Message Windows	13
Help On Line	13
Chapter 6 - Commands	14
Files Menu.....	14
New Command	14
Open Command.....	14
Save Command	14
Import readings Command.....	15
Printer Setup Command.....	15
Edit Menu	16
Parameters Command	16
The "Calculation Parameters" dialog box	16
Electrodes Command.....	16
The "Positioning of the electrodes on the ground" dialog box	16
Blocks Commands	17
The "Definition of homogeneous resistive blocks" dialog box	17
Layers Command.....	18
The "Definition of homogeneous resistive layers" dialog box	18
Readings Command	18
The "Resistivity values" dialog box	19
Titles Command	19
Interpretation Menu	20
Perform Calculations Command	20
Bad Readings Deletion Command.....	20
Print Menu	22
Graphics Command	22
The "Definition of colour scale" dialogue box.....	22
The <Manual Colour Setting> Button	23
The Graphical Output Preview Window.....	24

- Listings Command25
- Listings Preview Command25
- Listings Direct PrintCommand25
- Listings Export to DOC Format.....25
- Listings Export to TXT format25
- Listings Export to SLK format25
- Configure Command25
- Exit menu27

Chapter 1 - Introduction to the program

Introduction to the program

ELETOM is a software that performs the complete 2D inversion of surface electrical surveys with any measurement array (Wenner, dipole-dipole, user defined arrays, mixed arrays) with a maximum of 64 electrode positions that can be added as needed.

ELETOM uses the finite elements method (FEM) in two dimensions to model the subsurface electrical response.

The elements that define the grid are trapezoidal in form to enable to compensate any height difference between the electrode's positions.

The current dispersion in the third spatial dimension (perpendicular to section of interest) is considered by using Wannamaker's correction.

With ELETOM it is not necessary to specify any parameter to construct the grid. One needs only to enter the positions and numbering of the electrodes used in the survey's measurements. The iterative algorithm used to perform the inversion is based on the minimization of the mean square error between field data and the data simulated by the FEM, with the method proposed by Sasaki (1994).

The algorithm's convergence is obtained by estimating a stabilization (performed with the generalized reciprocal coefficients method) repeated during every iteration.

The reduction of the noise effect present in the data, is obtained with an iterative linear regression calculation (Ward, 1990) that enables to compensate for the fact that it is impossible to know the statistical variance of each measurement.

The result is given by a real resistivity section reconstructed with precision (in two dimensions) equal to the smallest halved inter-electrode distance, with the possibility to select a set of color shadings in order to enhance the anomalies present in the subsurface in the best way possible. The results can be rendered as iso-resistivity curves as well.

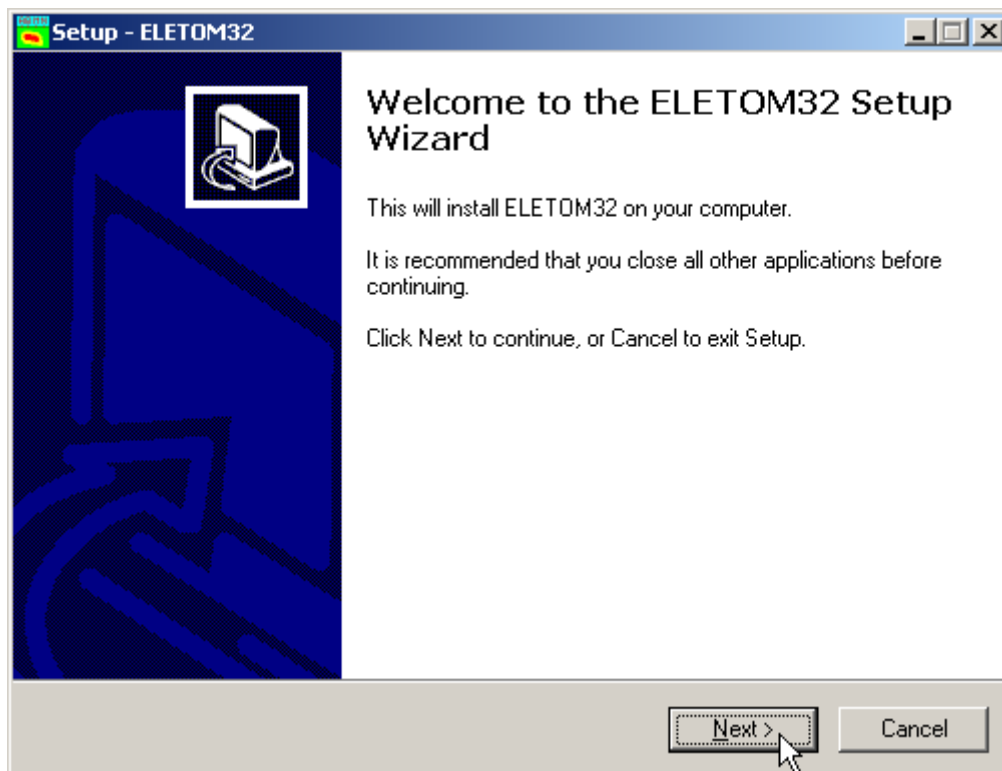
Chapter 2 - System requirements and program installation

System requirements

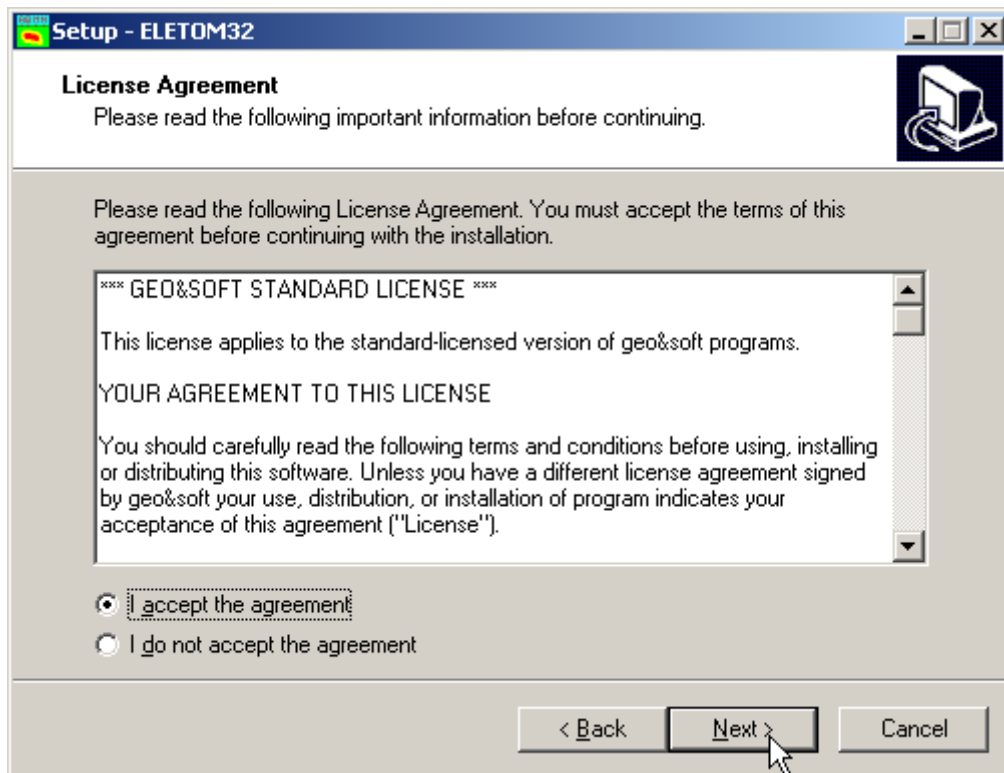
- Pentium® class processor
- Microsoft® Windows® 95 OSR 2.0, Windows 98, Windows Me, Windows NT®* 4.0 with Service Pack 5 or 6, Windows 2000, or Windows XP
- 64 MB of RAM (128 MB recommended)
- 100 MB of available hard-disk space
- CD-ROM drive
- A printer driver **must** be installed, even if the printer itself is not connected to the PC.

Program installation

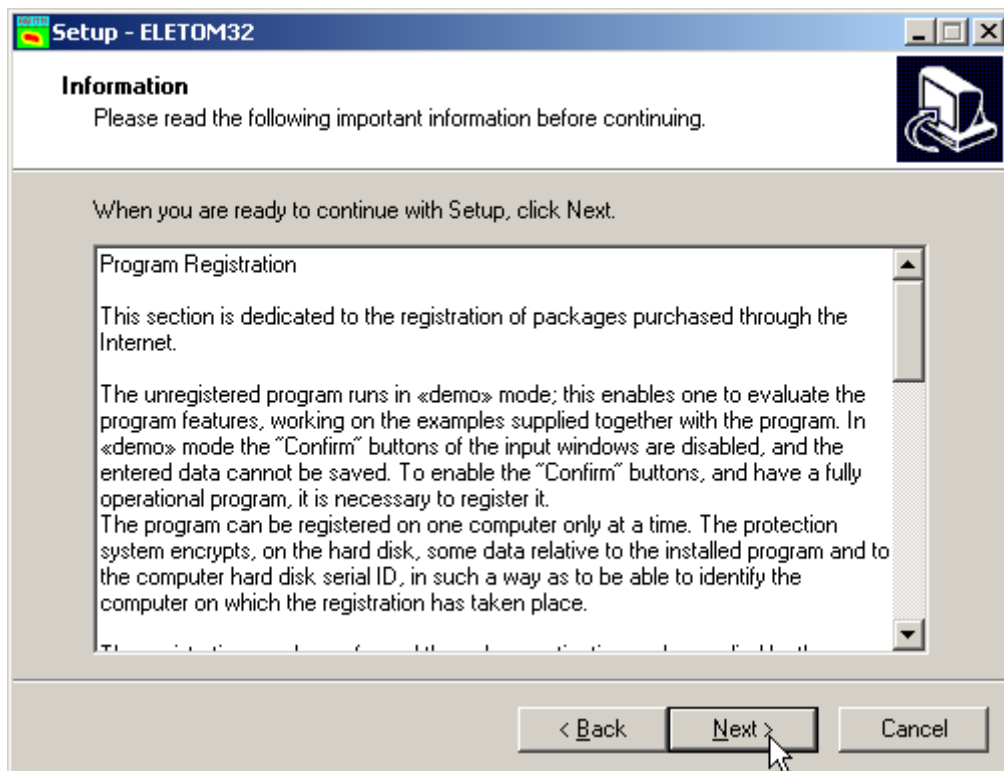
To install the ELETOM program, run **ELETOM32SETUP.EXE** from the CD-ROM or from the folder where you downloaded and saved the setup program.



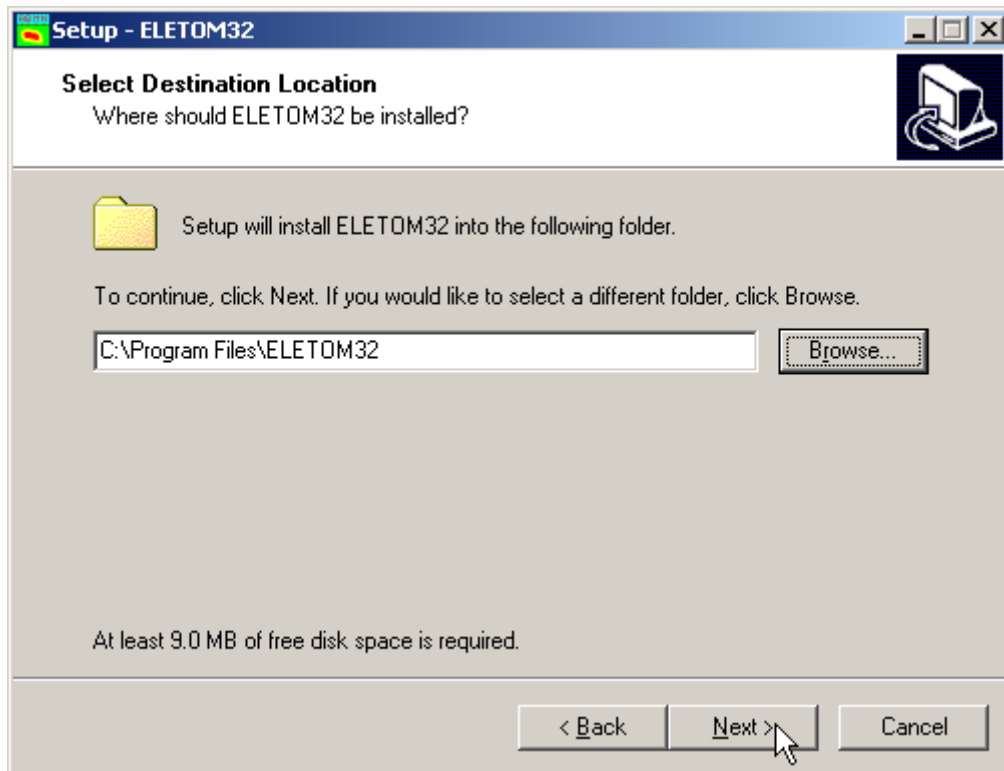
Press the **NEXT** button to start the installation.



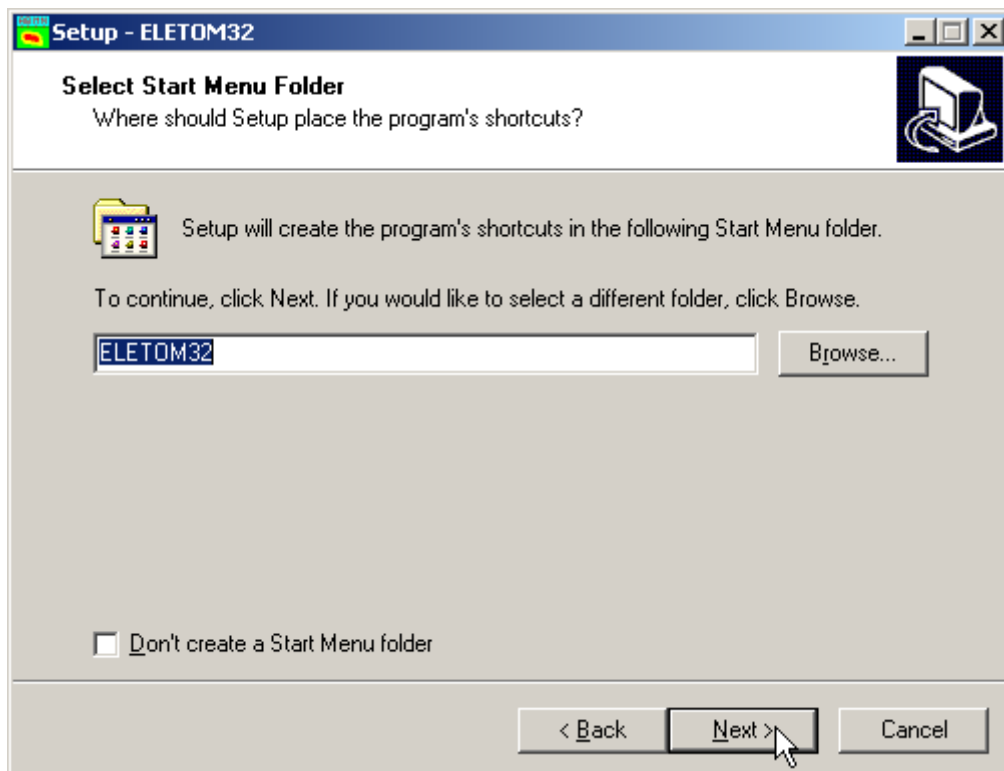
Carefully read the License Agreement and press the **NEXT** button.



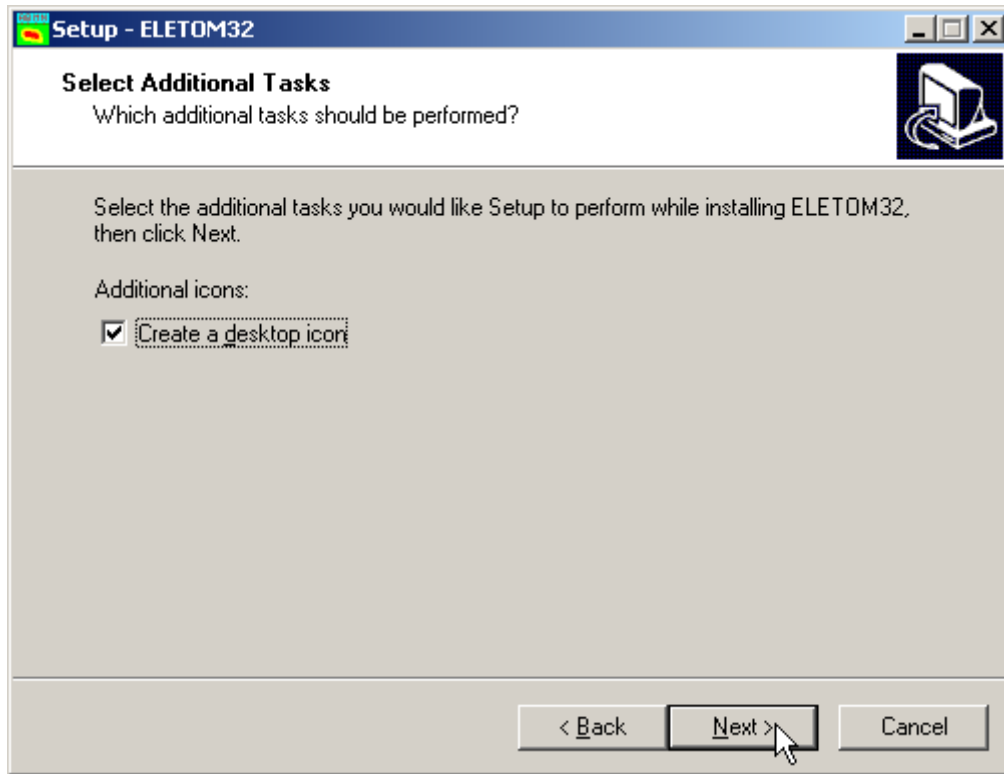
Carefully read the informations about the registration of the software and press the **NEXT** button.



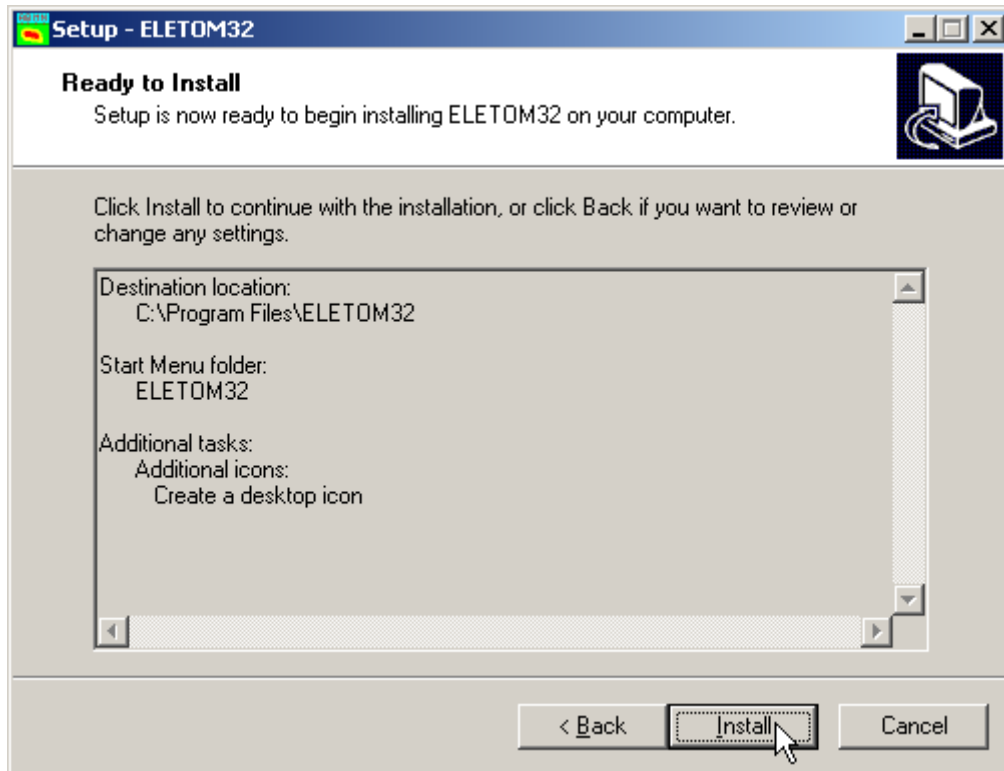
Select the folder where the **ELETOM32** directory would be created and press the **NEXT** button.



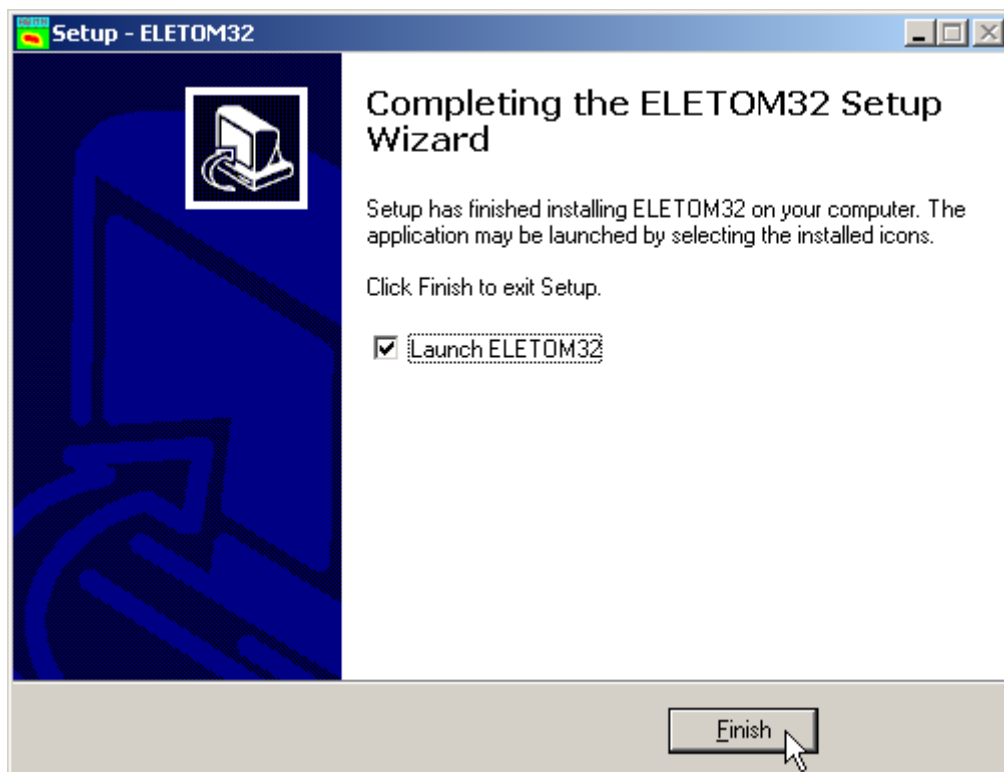
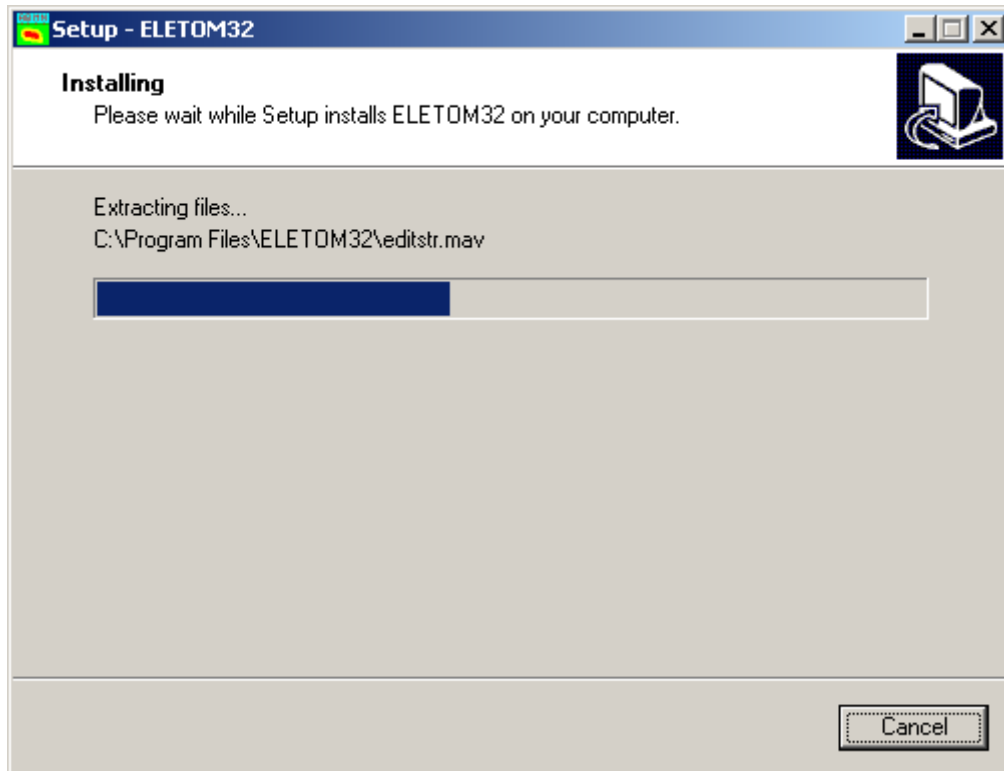
Press the **NEXT** button to create the **ELETOM32** group in the Start menu.



Choose if you want an icon to be created on your desktop, or not.



Press the **INSTALL** button to continue the installation procedure ...



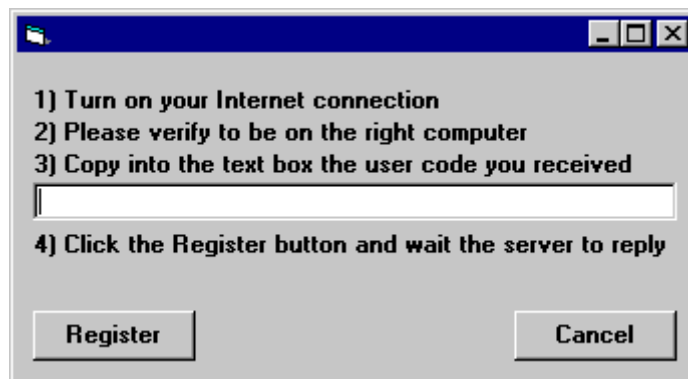
Once completed the installation, press the **END** button to exit the setup procedure and to start **ELETOM32**.

ELETOM32 can be run by clicking the **Start** button, then selecting the **ELETOM32** group in the **Programs** menu, and clicking **ELETOM32**.

Chapter 3 - The program protection

Program registration

This section is dedicated to the registration of packages purchased through the Internet.



The unregistered program runs in «demo» mode; this enables one to evaluate the program features, working on the examples supplied together with the program. In «demo» mode the “Confirm” buttons of the input windows are disabled, and the entered data cannot be saved. To enable the “Confirm” buttons, and have a fully operational program, it is necessary to register it.

The program can be registered on one computer only at a time. The protection system encrypts, on the hard disk, some data relative to the installed program and to the computer hard disk serial ID, in such a way as to be able to identify the computer on which the registration has taken place.

The registration can be performed, by the manufacturer or supplier, with an **activation code**.

We would like to stress that the encrypted protection data on the hard disk are bound in a unique way to the hard disk itself: subsequently, one can no longer use the activation code to install the program on other computers.

Let us describe the procedure for the registration of the software:

1. select the **Registration** command from the Help menu.
2. turn on your Internet connection
3. copy into the box the activation code you received
4. click the <Registration> button and wait the server to replay

Chapter 4 - How to update the program

Procedure for updating the program

We believe that the best advertising is a client who shows to the colleagues a state-of-the-art product, so we have always made available the free updates of our packages.

Now a new service is available for free: **AutoUpdater** automatically searches and downloads the new components from our web site **www.geoandsoft.com**, saving the older versions in a backup folder.

If you prefer, you still can download the updated full setup from our web site, of course.

To download the updated components with AutoUpdater, follow this procedure:



1. Start the program
2. Click the "**Update**" button at the bottom of the first window.
3. Wait for the program to verify if some updated component is available on our servers. If the program finds new components, it shows the list: each component has a check box which allows to choose if a single component has to be downloaded, or not. AutoUpdater automatically checks the boxes of the components whose download is suggested, and leaves unchecked the files which could have been modified by the user, like the colour configuration files.
4. Select the files you want to download and click the "**Update**" button.
5. Once the files have been installed, AutoUpdater runs the updated program.

AutoUpdater can also be configured to automatically start at each run of the program.

Simply check the "update at startup" check box, and AutoUpdater will automatically search for new components each time the program is run.



Autoupdating requirements and troubleshooting

The autoupdating procedure requires an active Internet connection.

The search for new components is directly performed by the program, while the download is performed by AutoUpdater: for this reason, both the main program and AutoUpdater need the right to access to the Internet.

All the autoupdating procedures use the HTTP protocol, port 80, and automatically recognize if a proxy server is used on the local network to access the Internet.

If you experience some problems with the autoupdating procedure, they may be due to a FIREWALL: ask your System Administrator to give the right to access the Internet to the executable of the program (*32.exe) and to the AutoUpdater.exe module, in the program folder. No "act as a server" right is needed.

If particularly hard security rules disallow giving those access rights, remember you can download the updated full setup, from our web site, through your web browser.

Chapter 5 - User Interface

Usage Notations

Some typographical notations and keyboard formats are used in this manual to help locate and interpret information more easily.

Bold print is used to indicate command names and related options. Characters appearing in bold print should be typed exactly as printed, including spaces.

Words written in *italics* indicate a request for information.

CAPITAL letters are used to indicate computer, printer, directory, and file names.

User Interface and Data Entering

The user interface is designed to be easy to use and powerful and is supported by complete on-line help. This help contains practical hints and the theoretical background, where applicable. It should reduce the requirement of frequently consulting the printed manuals.

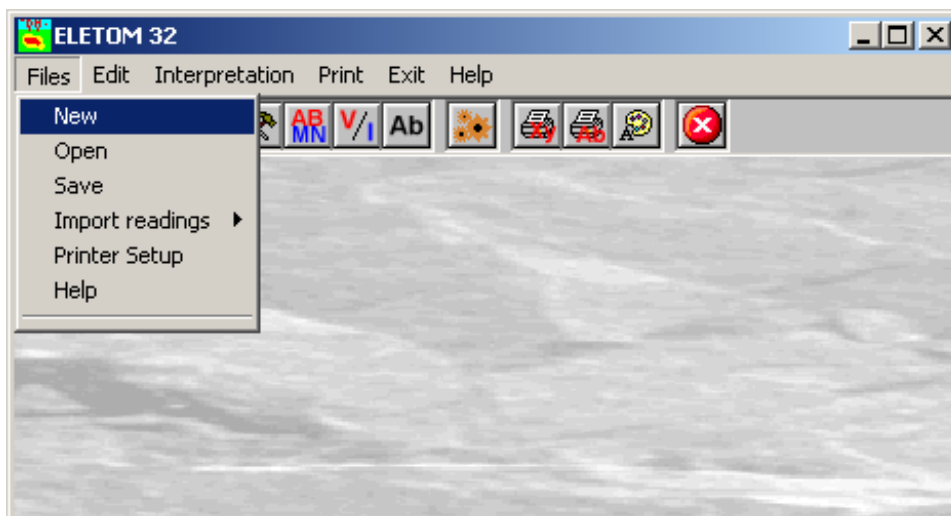
All the commands are located inside a menu bar. Each menu contains a list of commands that one can select with the mouse or the keyboard. The arrangement of the menus, designed with ergonomic criteria, follows the logical order of the operations, inhibiting the access to further operations until all the necessary data have been entered.

The interface layout is maintained in all of our programs, to simplify, as much as possible, the transition from one program to another to avoid having to learn different commands and procedures for similar functions (such as entering data or managing files).

Let us examine the general components that are available in the user interface of geo&soft programs.

User Interface: Menu Bar and Menus

The Menu Bar manages the access to all the program commands. The goal of the menu design is to offer an ergonomic, simple, and understandable arrangement of the commands.



The menus used to perform a complete operation are normally ordered left to right and top to bottom. When possible, the following scheme is used: definition of the name of the project, entering the required data, performing the calculation, and generating the output as a preview or final print.

The programs automatically disable some menus or commands when the execution of such menus or commands is not yet feasible. Typical examples are: a calculation menu is disabled when not all the data needed for the calculation have been completely entered, or a print menu is disabled if not all the calculations have been executed.

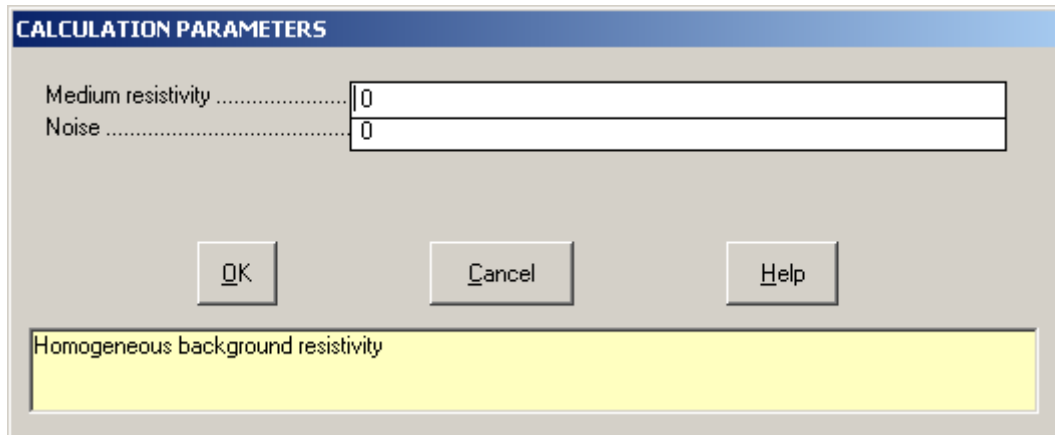
To choose a command from a menu, use the following procedures:

- ☞ Point the menu that is to be activated, click the left button of the mouse, then point the command to be activated, and click the left button of the mouse again. If one does not wish to select any command from the menu that is open, exit the area of the menu before releasing the mouse button.

The Input Dialogue Windows

The different menu commands can perform an immediate action, or display a dialogue window in order to input or edit the various data sets.

When a dialogue window is visible, all the actions that do not pertain to it are ignored. Hence, it is necessary to close the dialogue window to resume the normal use of the program.



Some fundamental tools are used inside the dialogue windows: data fields such as text and list boxes, and buttons. The text boxes are used to input numerical values and text strings. Most of the editing keys (HOME, END, INS, DEL, etc.) can be used inside these fields. If one sees a small arrow pointing down on the right side of the data field, one has a list box.

To scroll the list box and select an item, do as follows:

- ☞ click the arrow, and then click the item to be selected.

There are three buttons in the dialogue window:

- <Ok> - this button saves the entered information and goes on to the next phase.
- <Cancel> - this button closes the Dialogue window without saving the just entered data, or without executing the command.
- <Help> - this opens a window that contains general information on how to use the dialogue windows.

To use the buttons:

- ☞ click the button

To edit the data inside the dialogue window, use the following keys:

- TAB - moves the cursor to the next field; when the cursor is inside the last visible field the cursor goes to the first control button of the Dialogue window. To return to the previous field, press SHIFT+TAB.
- ENTER - moves the cursor to the next field. If the cursor is positioned on one of the window buttons, the corresponding command is executed.
- BACKSPACE - cancels the last character that has been entered.
- DEL - cancels the character to the right of the cursor.
- ESC - closes the Dialogue window without saving the entered values or without executing the command. The <Cancel> button will do the same.
- UP ARROW/ DOWN ARROW - these are used in multiple fields, or those fields that have a list box.
- LEFT ARROW/ RIGHT ARROW - moves the cursor to the previous or next field.
- HOME - moves the cursor to the beginning of the field.

- END - moves the cursor to the end of the field.

A yellow box with a short text that explains the meaning of the value to be entered, can be seen in the lower part of the window. If the text is not completely visible, click the yellow box to read the complete text.

Data Input With Tables

Tables are used to enter long sequences of numerical values and/or text strings. The keys to be used are the following:

N.	xmin [m]	xmax [m]	ymin [m]	ymax [m]	rho [ohm]
1					
2					
3					
4					
5					
6					
7					
8					
9					
10					
11					
12					
13					
14					
15					

minimum abscissa of the block

OK Cancel Insert Delete Help


- TAB - moves the cursor to the first button in the window. When one presses this key again, the cursor is moved to the next button.
- SHIFT+TAB - moves the cursor to the previous button.
- ENTER - moves the cursor into the next input field. If the cursor is positioned on one of the window buttons, the corresponding command is executed.
- PAGE UP - moves the cursor up 15 lines.
- PAGE DOWN - moves the cursor forward 15 lines.
- UP ARROW - moves the cursor to the input field directly above.
- DOWN ARROW - moves the cursor to the input field directly below.
- LEFT ARROW / RIGHT ARROW - moves the cursor to the field respectively to the left or to the right of the current position.
- HOME - moves the cursor to the beginning of the line.
- END - moves the cursor to the end of the line.
- F2 - copies the field contents to the extended editing field under the title bar in order to facilitate the editing of the long strings. When working in this editing field, please **REMEMBER** to press ENTER, even before clicking the <Ok> button. Double-clicking a field has the same effect as pressing the F2 key.

The tables have two additional buttons:

- <Insert> - creates an empty line before the one in which the cursor is positioned.
- <Delete> - deletes the line in which the cursor is positioned.

IMPORTANT SUGGESTION: you can copy data to or from other programs such as Microsoft Excel:

The data entered in the table can be copied in order to be pasted into another table.
To copy the table's contents:

 press the key combination CTRL+C. The contents will be copied into the Clipboard of Windows.

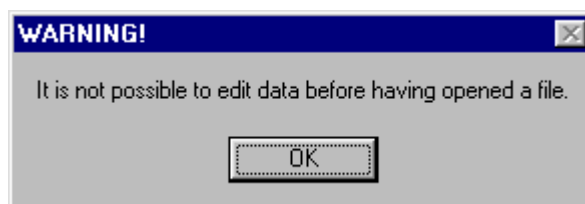
To paste the Clipboard contents into the table:

 press the key combination SHIFT+INS, or CTRL+V.

A yellow box with a short text that explains the meaning of the value to be entered, can be seen in the lower part of the window. If the text is not completely visible, click the yellow box to read the complete text.

Message Windows

The function of these windows is to give information to the user concerning the system status, as in the case of an error due to an improper use of the program.



Help On Line

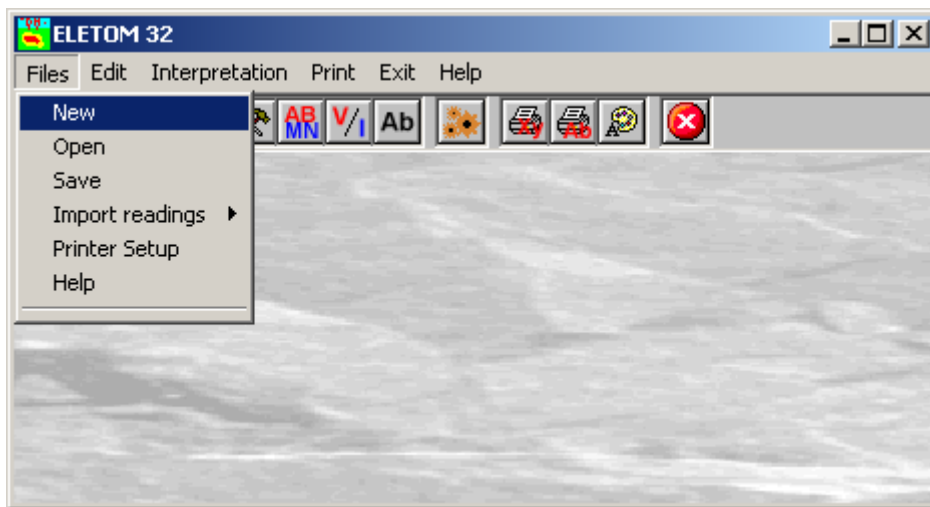
A complete Help On Line is available. It is possible to ask for information or suggestions related to the commands or the use of the program. In order to access the help on line, proceed as follows:

1. Position the cursor on a field of a dialogue window and press F1.
2. Use the Summary from the Help menu.

Chapter 6 - Commands

Files Menu

This menu contains the commands for opening and saving files. In these all the information relative to the project is stored.



New Command

To open a new file:

- ☞ from the menu choose New by first clicking on the name of the menu then on the New command.

Open Command

To be able to work with the program one should, first of all, create a new file or open a pre-existing file.

To open a file:

- ☞ from the menu choose Open by first clicking on the name of the menu then on the Open command. In the File Name field type the name of the file to be opened or double click on the name of the desired document which can be found in the list underneath the File Name field.

The name of the file being used is reported inside the title bar of the main window and inside a window referable by selecting the Help menu.

The name of the file should not contain spaces and/or punctuation marks. It is not necessary to indicate extensions, since the extension ".ELR" is added automatically: for example, the given name "TEST" will have the file name "TEST1.ELR"

Save Command

During the work session the program uses a copy of the file, which is temporarily, loaded into the computer's RAM. To save work on the hard disk it is necessary to use the Save command; otherwise all the data introduced will be lost when one exits from the program. If an electrical problem or any other type of problem interrupts the computer's functioning all the data not yet saved will be lost, so it is advisable to save the work often.

To save a file:

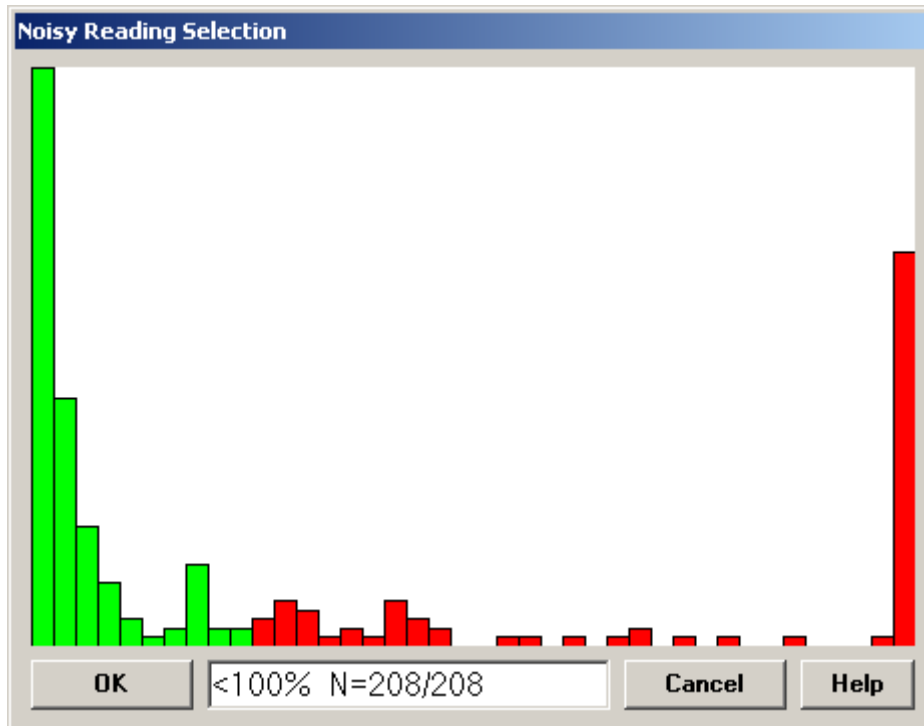
- ☞ select Save from the file menu by clicking first on the name of the menu and then on the Save command. In the File Name field, the name given to the file when it was originally opened appears. To confirm that name click on the <Ok> button. To assign a new name to the file, type the new name in this field.

The program will save the file in the current disk unit and directory unless specified otherwise. To save the file on a different disk unit or in a different directory, type the full path in the File Name field. For example, to save the document in the root of disk C: type *c:\filename* or select a different unit or directory.

Import readings Command

This command is used to import the files produced by instruments which record the noise level, like the 16G PASI, a ".TXT" extension is expected.

Before importing the file, the program verifies the quality of the readings, and shows an histogram of the noise/signal ratio. Moving the mouse in the window, the percentage of the n/s ratio is shown, together with the number of readings which have a quality better or equal to the value shown.



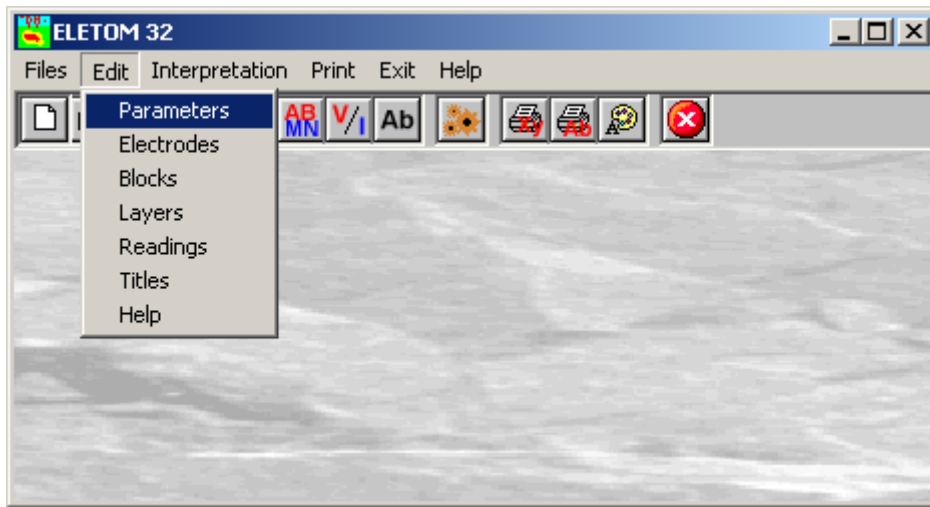
By clicking the histogram, the maximum value of noise/signal ratio is chosen, and the most noisy readings are discharged.

Printer Setup Command

By selecting this command one goes to "Print" dialog box. Here one can verify and modify the Default printer's Setup or select another printer from those installed one ones system.

Edit Menu

Through this menu one can enter the data necessary for a calculation.



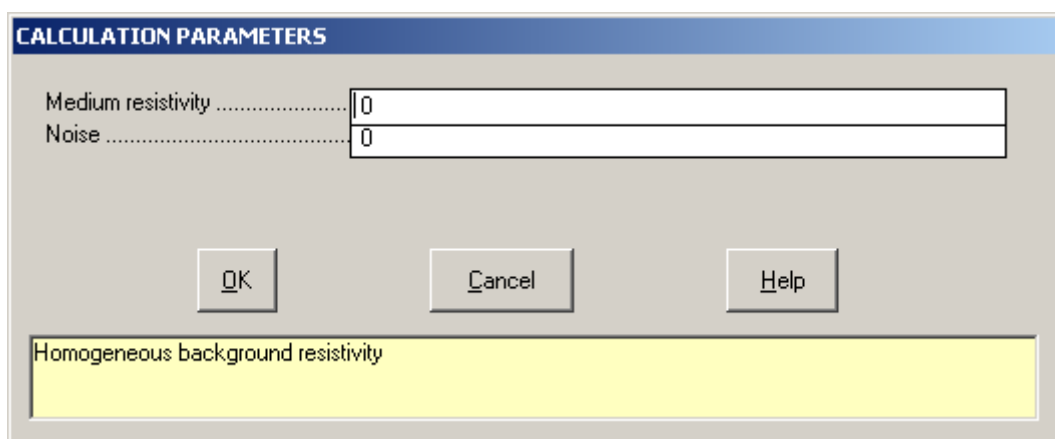
Parameters Command

This is used to enter the mean value of the terrain being surveyed. These parameters allow a filtering of the data in order to speed up the characterization of the terrain.

The “Calculation Parameters” dialog box

In the window with the caption “Calculation Parameters” one should enter:

- *Medium resistivity*: homogeneous background resistivity
- *Noise*: percentage of estimated noise in the measures. This is given by the proportion, expressed as a percentage, between the average of a set of repeated readings with the same electrode configuration and their standard deviation. When it is not possible to determine the amount of noise, one ought to remember that when the noise level increases the precision of the calculation diminishes; it diminishes when the noise estimated is lower than in reality. Thus, when it is not possible to estimate the noise level it is best to repeat the interpretation, starting with high noise values (e.g. 20%) and to decrease this gradually.

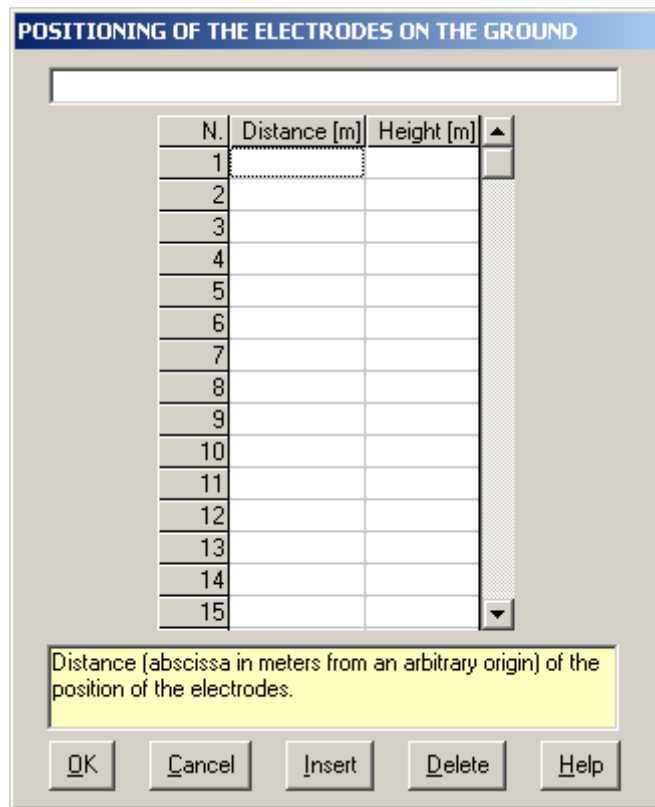


Electrodes Command

With this command it is possible to enter the geometry of the terrain under survey.

The “Positioning of the electrodes on the ground” dialog box

In the window with the caption "Positioning of the electrodes on the ground" one can enter:



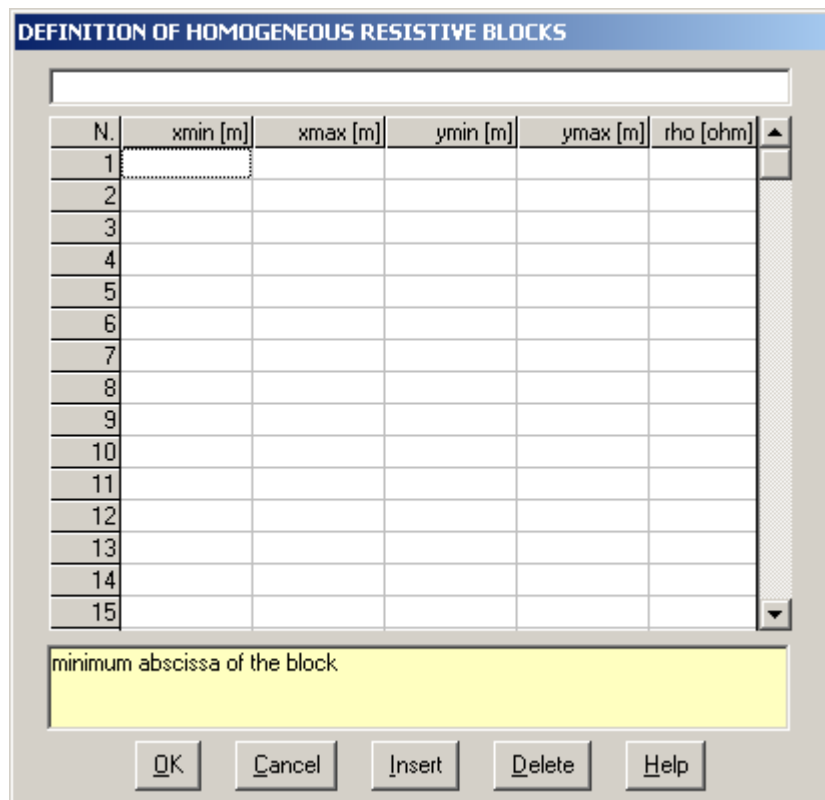
- *Distance [m]*: distance (abscissa in meters from an arbitrary origin) of the position of the electrodes.
- *Height [m]*: height (vertical distance in meters from an arbitrary horizontal plane) of the electrodes.

Blocks Commands

With this command one can define the areas with constant resistivity, in order to speed up calculations.

The “Definition of homogeneous resistive blocks” dialog box

In the window with the caption "Definition of homogeneous resistive blocks" one can enter:



- $xmin [m]$: minimum abscissa of the block
- $xmax [m]$: maximum abscissa of the block
- $ymin [m]$: minimum ordinate of the block
- $ymax [m]$: maximum ordinate of the block
- $\rho [ohm]$: block resistivity

The coordinates ought to be given in meters from an arbitrary origin. This origin ought to coincide with the one used to define the electrodes' position.

Layers Command

With this command one can define strata with constant resistivity, this to speed up calculations.

The "Definition of homogeneous resistive layers" dialog box

In the window with the caption "Definition of homogeneous resistive layers" one should enter the X,Y coordinates of the endpoints of the segment that represent the top of the discontinuity.

N.	xmin [m]	xmax [m]	ymin [m]	ymax [m]	rho [ohm]
1					
2					
3					
4					
5					
6					
7					
8					
9					
10					
11					
12					
13					
14					
15					

Minimum abscissa of the layer

OK Cancel Insert Delete Help

- $xmin [m]$: minimum abscissa of the layer
- $xmax [m]$: maximum abscissa of the layer
- $ymin [m]$: minimum ordinate of the layer
- $ymax [m]$: maximum ordinate of the layer
- $\rho [ohm]$: layer resistivity

Also in this case the origin ought to coincide with the one used to define the electrodes' position. The resistivity assigned to this segment will be assigned to the underlying column of rock or soil.

Readings Command

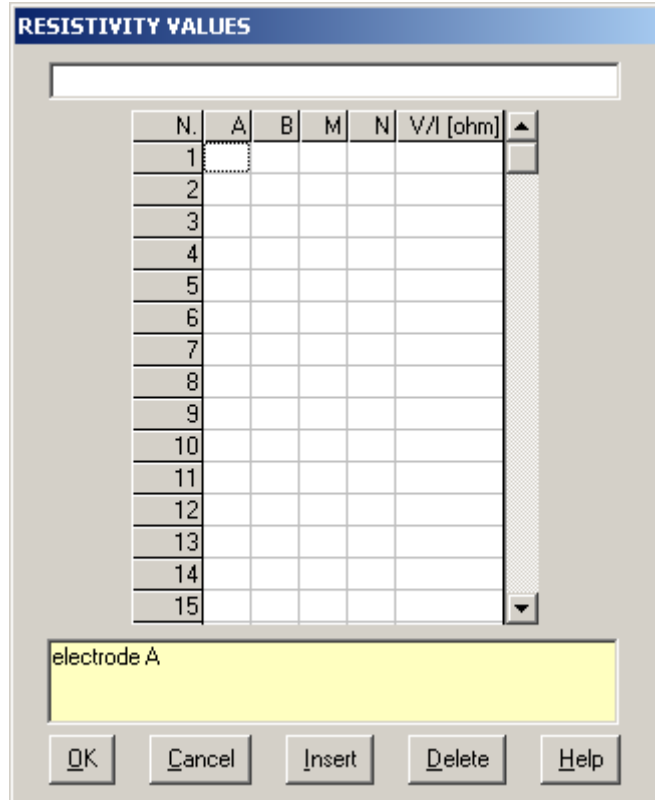
With this command one can enter, for each reading, the position inside the electrode array, of the current electrodes A and B and the potential electrodes M and N, hence the resistivity value.

The "Resistivity values" dialog box

The program can interpret readings with any electrode layout, from the simple Wenner and Schlumberger configurations, to the dipole-dipole, user defined and mixed arrays (for example Wenner and dipole-dipole), with a maximum of 64 electrode positions.

Note: the readings should be entered as a positive value for Wenner or Schlumberger configurations, and as a negative value for dipole-dipole configurations.

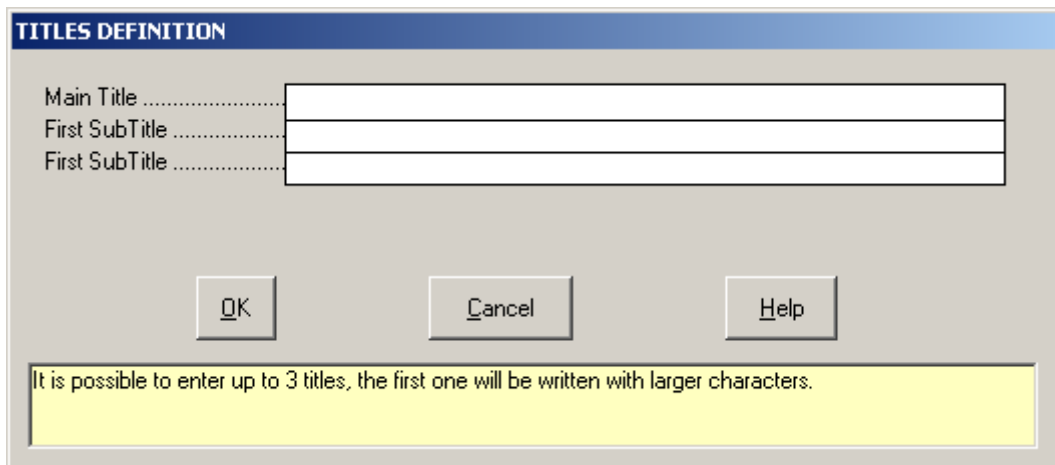
In the window with the caption "Resistivity values" one should enter:



- A: electrode A
- B: electrode B
- M: electrode M
- N: electrode N
- V/I [ohm]: ratio between V (volt) and I (ampere)

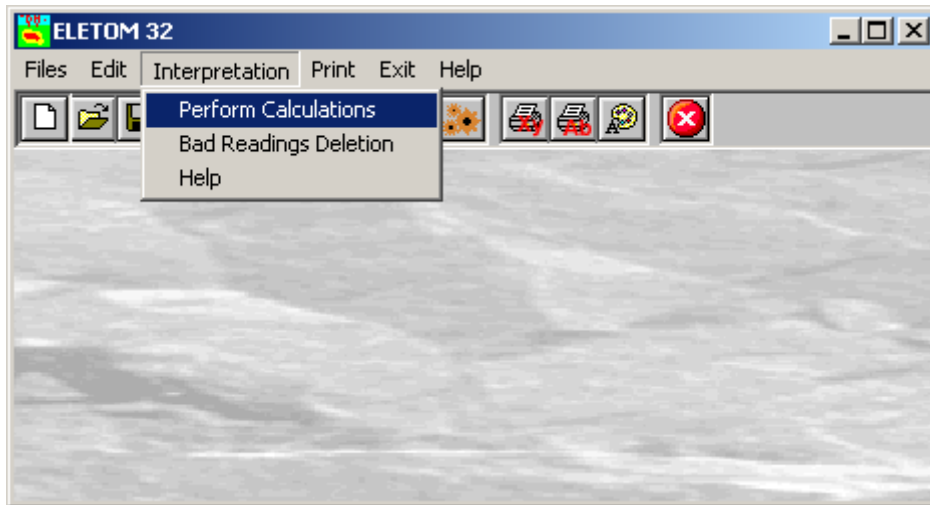
Titles Command

In order to annotate the drawing generated one can enter a main title and two sub-titles with this command.



Interpretation Menu

With this menu it is possible to start the procedure of elaboration of the data.

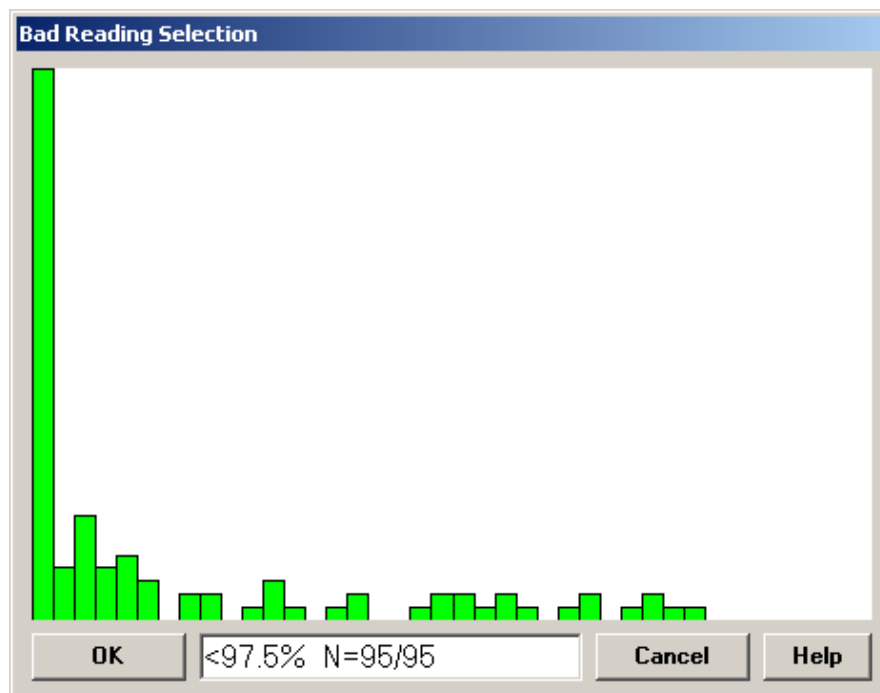


Perform Calculations Command

When one chooses this command the data are calculated with the pseudosection method and an iterative inversion algorithm that uses the finite elements method (FEM) in two dimensions. The pseudosection method is quite fast and yields mainly qualitative results from the survey. The second method is based on an iterative inversion algorithm that uses the finite elements method (FEM) in two dimensions to simulate the terrain's electrical response. Contrary to the pseudosections method, the inversion method used by ELETOM makes it possible, even if in only two dimensions, a resistivity section that is extremely precise. The calculation time requires more onerous computing times.

Bad Readings Deletion Command

This command is used to remove the values which, regardless a low noise level, show a large difference when comparing the effective readings and the inverted ones. Such values could be originated by an insulated or defective electrode, and could decrease the quality of the inversion. Clicking on the histogram it is possible to remove the bad readings, then repeating the inversion procedure.



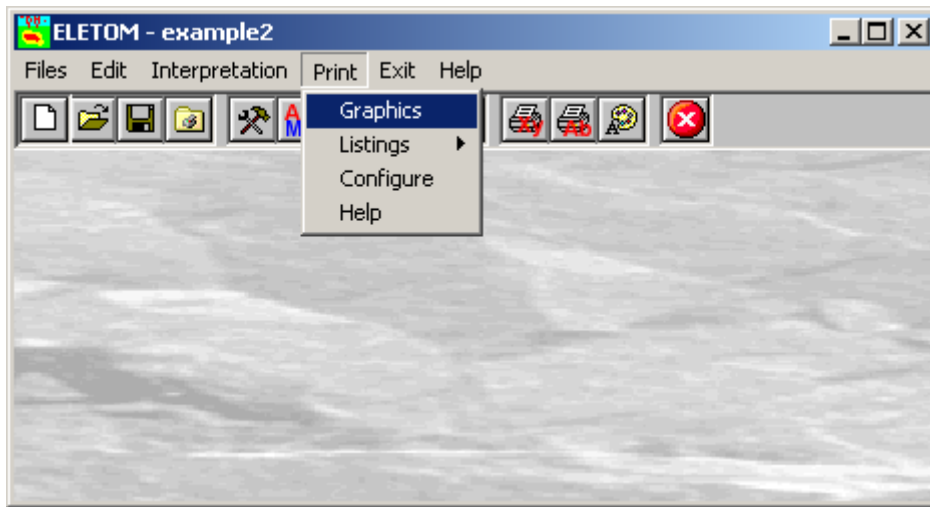
The error percentage shown in the window is calculated as

$$\left| \frac{R_c - R_m}{R_m} \right|$$

where R_c is the apparent resistivity of the inverted model, and R_m is the measured apparent resistivity.

Print Menu

Through this menu it is possible to print the results of the interpretation.

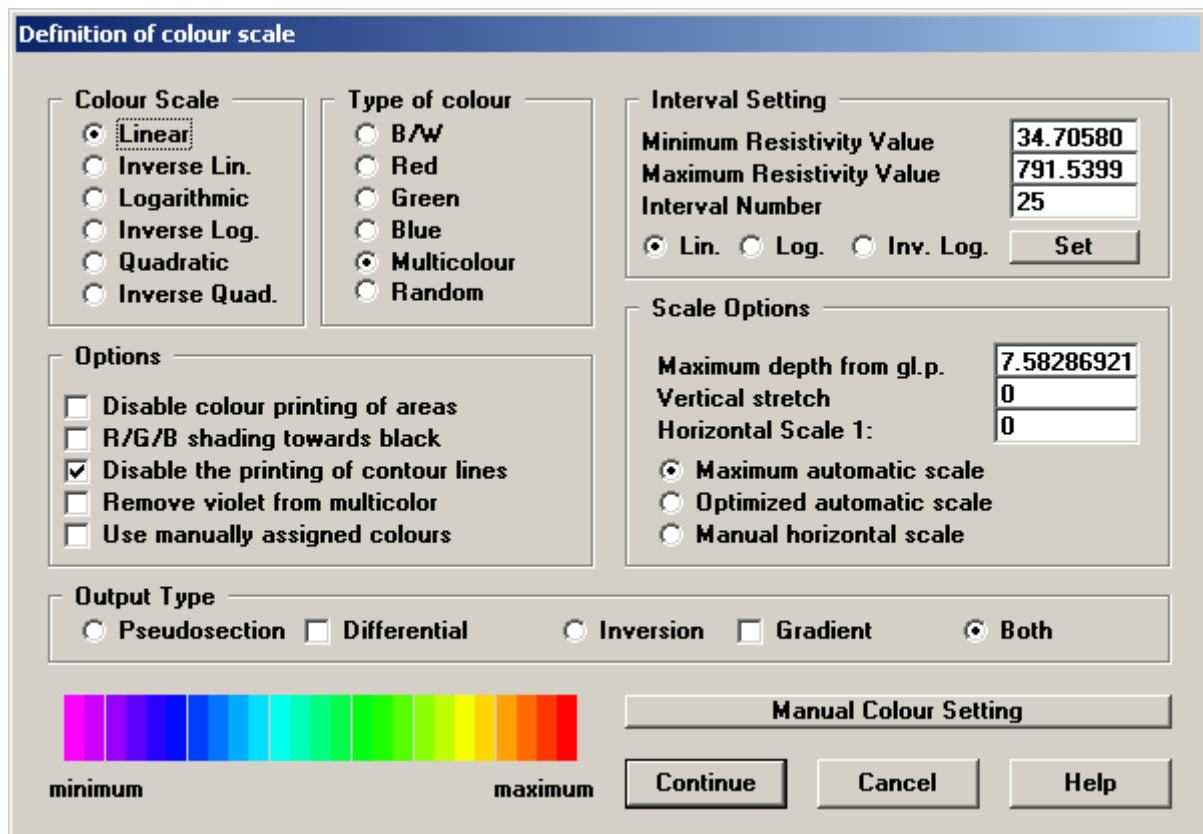


Graphics Command

On selecting this command the program shows, in the print preview window, the graphical results of the interpretation. Before visualizing and printing the results one ought to define the general parameters for the representation.

The “Definition of colour scale” dialogue box

Many different options are available for colors and scale of the graphical outputs. Once assigned the graphical parameters, click the <Continue> button to open the print preview window, or <Cancel> to abort.



Colour Scale

To visualize different resistivity values, different scale types can be used: linear, inverted linear, logarithmic, inverse logarithmic, quadratic, inverse quadratic. This section involves the colour distribution but does not affect the interval values.

Type of colour

Each of these can be visualized by using one of the colors present in this field: Black and White, Red, Green, Blue, Multicolor, and Random. With the Black and white option you will obtain a gray-scale output; With the Red, Green or Blue option the different values of resistivity will be represented varying from the selected color to the white; the Multicolor options allows to create outputs using a full scale of colors, while the Random option assigns a color randomly chosen to each resistivity level.

Options

Disable color printing of area: with this option activated, the program will only draw the iso-resistivity lines.

R/G/B shading towards black: with this option the selected color shades towards black instead of to white.

Disable the printing of contour lines: with this option activated, the program will no longer draw the iso-resistivity curves.

Remove violet from multicolor: this option allows to start the multicolor scale (if selected) from the blue instead of from the violet.

Use manually assigned colors: you can choose to define manually both the limits of each range and the color to be used. To manually define those parameters click the <Manual Color Settings> button.

Interval Setting

This section involves the interval values but does not affect the colour distribution. Different interval types can be used: linear, logarithmic and inverse logarithmic. In this frame it is possible to select the range of values and the required number of intervals.

Scale options

Many options are available to personalize the graphical outputs.

Maximum depth from the ground level: this option allows to remove from the drawing the most deep part of the inverted section; this is useful when few measures with an high noise level have been taken.

Vertical stretch: allows to apply two different scales along the x- and y-axis.

Horizontal scale: it is the scale along the x-axis used when the Manual Horizontal Scale option is selected.

Maximum automatic scale: when this option has been selected the program creates a drawing as big as allowed by the sheet size.

Optimized automatic scale: The program chooses the optimal scale between 1:1, 2, 2.5, 5, 7.5 (or a multiple) in function of the sheet size.

Manual Horizontal Scale: The program uses the user-defined scale.

Output Type

In this section it is possible to choose which kind of output would be printed.

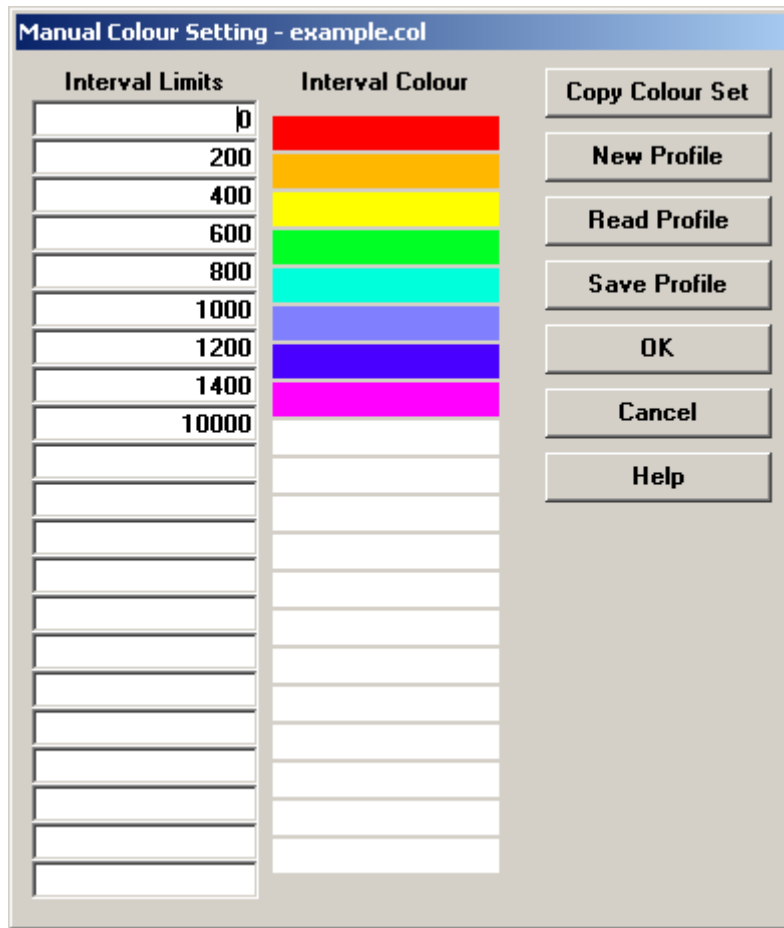
- Pseudosection: as an alternative to the standard inverted pseudosection it is available a differential pseudosection, calculated as the difference between the measured and the inverted apparent resistivity values; this kind of output is a good tool to verify the quality of the interpretation and to detect defective readings.
- Inversion: as an alternative to the standard inverted section, it is possible to produce a section showing the gradient distribution of the calculated resistivity; this kind of output is a good tool for a more clean identification of the shape of the underground objects, as the resistivity gradient increases in correspondence of the contact between different materials.

The <Manual Color Settings> button opens the "Manual Color Settings" dialog box, where you can define ranges and colors of the resistivity intervals.

The <Manual Colour Setting> Button

The <Manual Colour Setting> button opens the "Manual Colour Setting" dialogue box, where you can define the resistivity intervals and the colour one wishes to assign to each interval. These settings can be saved as personalised colour profiles. To change a colour, double click the coloured field and choose the new colour to be assigned to the interval from the Colours Dialogue Box.

Each time a colour set has been defined and confirmed, it is saved as a default manual colour profile. It will then be used each time the *Use manual colour settings* option is selected, if a different colour profile has not been loaded or saved; in this case, the default colour profile is overwritten, and the selected (saved or loaded) colour profile will be permanently associated to the current project, regardless of the default manual colour configuration.



- <New Profile> button: if the current interval number is lower than 21, the current automatic settings are copied and can then be modified, otherwise all the fields are cleared.
- <Read Profile> button: the program reads a previously saved colour profile from the hard disk; this colour profile will be permanently associated to the current project.
- <Save Profile> button: this allows one to save the current colour profile on the hard disk; this colour profile will be permanently associated to the current project.
- <Copy Colour Set> button: if the current interval number is lower than 21 the current settings are copied and can then be modified, otherwise the button is disabled.

The Graphical Output Preview Window

The Toolbar is located in the upper side of the preview window. The Toolbar allows fast access to the commands with the mouse. Click the corresponding icon, to select one of the commands that belong to the Toolbar. The Toolbar displays the following commands:



- **Zoom +** : clicking the first icon activates the ZOOM function (it can only be used with the mouse) that permits the enlargement of part of the drawing. The function remains active until the Zoom - icon is selected. To enlarge a part of the drawing:
 - ✎ click the icon, and then select the area to be enlarged by clicking the upper-left corner and dragging the mouse pointer to the lower-right corner. At this point, release the mouse button. Note: due to the low resolution of the screen, the texts could appear in a slightly different scale at different zoom levels. This does not affect the quality of the final printouts.
- **Zoom -** : click this icon to return to the original scale of the preview.
- **Arrows**: click the arrow icons, in "Zoom +" mode, to pan the on-screen preview.
- **Print**: click the print icon to send the drawing to the default printer.
- **Resized print**: click this icon to send the drawing to the default printer. The image will be resized to fit the current paper size.

- **DXF**: one can export the graphic output as a DXF file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the DXF file, which has, by default, the same name as the current project.
- **EMF/W**: this icon allows the graphic output to be exported to a version of the Enhanced Windows Metafile which is compatible with Microsoft Word. Clicking this icon gives access to a dialogue window that enables one to assign a different name to the EMF file, which has, by default, the same name as the current project.
- **EMF/D**: this icon allows the graphic output to be exported to a version of the Enhanced Windows Metafile which is compatible with Corel Draw. Clicking this icon gives access to a dialogue window that enables one to assign a different name to the EMF file, which has, by default, the same name as the current project.
- **BMP**: one can export the graphic output as a BMP file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the BMP file, which has, by default, the same name as the current project.
- **GIF**: one can export the graphic output as a GIF file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the GIF file, which has, by default, the same name as the current project.
- **JPG**: one can export the graphic output as a JPG file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the JPG file, which has, by default, the same name as the current project.
- **Exit**: click this icon to close the preview window and return to the main menu.

Listings Command

By selecting this command the program opens a sub-menu where one can select the preview mode, and whether to print or export the listing.

Listings Preview Command

With this command it is possible to preview the listings pertaining to the calculation result. The listing can not be changed here.

To scroll the text use the scroll bars on the window or use the UP and DOWN arrow keys, or the PAGE UP e PAGE DOWN. keys. To exit press ESC or double-click any point of the listing.

The file containing the listing is saved on disc in the same directory where the data files resides. It has the same name with the difference of the ".TAB" extension. The file is in the normal ASCII format and its pagination is such that all commands have the # sign as prefix.

Listings Direct PrintCommand

With this command the listings pertaining to the calculation result are printed directly.

Listings Export to DOC Format

With this command the listings pertaining to the calculation result are exported in the format used by Microsoft Word 97. The file will have the same name but with the ".DOC" extension and will be saved in the same directory where the data files reside.

Listings Export to TXT format

With this command the listings pertaining to the calculation result are exported in the "Text only" format. The file will have the same name but with the ".TXT" extension and will be saved in the same directory where the data files reside.

Listings Export to SLK format

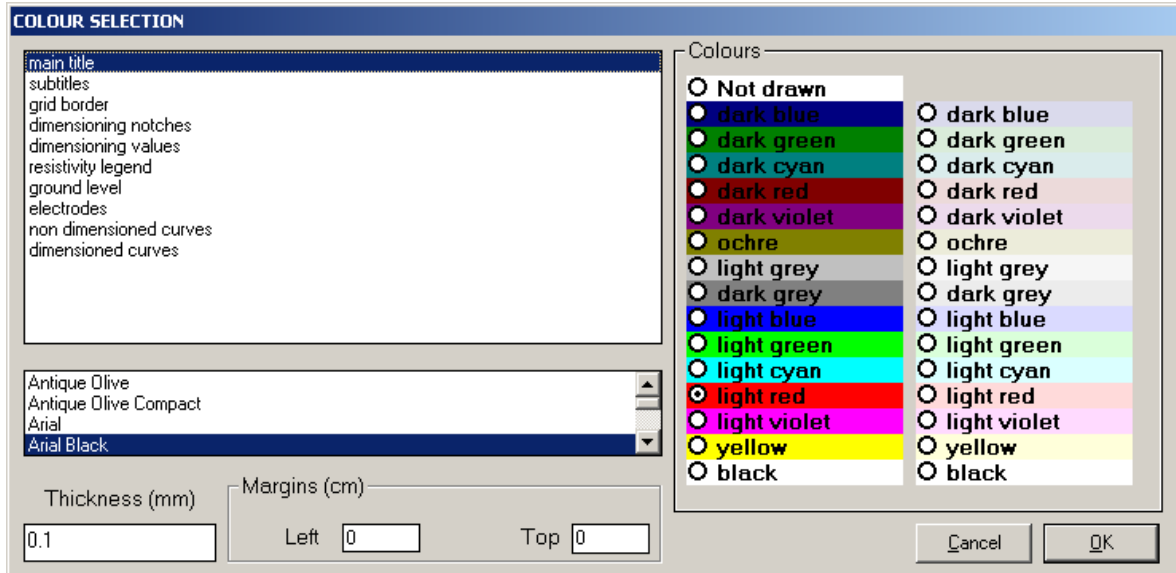
With this command the listings pertaining to the calculation result are exported in the format used by Microsoft Excel. The file will have the same name but with the ".SLK" extension and will be saved in the same directory where the data files reside.

Configure Command

With this command, one can define the graphical properties of the different objects in the graphical output. One can associate a colour, a line thickness and a character font to each object. Notice that not all the objects have both a line thickness and a character font: for example, a *title* only requires the colour and the character font to be defined.

- Click an object to select it (use the scroll bar to see the entire list)
- Click one of the option buttons, in the "colours" frame, to select the colour that has to be associated to the selected object. The *Not drawn* option hides the object of the graphical output.

- If the object contains texts, one can choose a character font from the list (use the scroll bar to see the entire list)
- If the object contains lines, one can assign a thickness to the lines (in mm). If the value is zero, the program uses the thinnest line on the output device.
- It is possible, as an option, to assign a left and a top margin (in cm) to the whole graphical output.



NOTE: although the available set is limited to fifteen colours, they can be manually customised; double-click the coloured bar to open the Colour Dialogue Box and choose a different RGB value associated to the selected colour.

Exit menu

This command allows one to exit the program. If the file in use contains data that have not yet been saved, the program will present the window used to save files, as described previously.