Program for one-dimensional interpretation of VES and VES-EP data (onshore and offshore measurements).

ZONDIP1D

User manual

Zond geophysical software

Table of contents

Purpose and capabilities of the program

ZondIP1D program is intended for one-dimensional interpretation of profile data of different modifications of vertical electrical sounding. Convenient interface and wide possibilities of data representation allow to solve the set geological problem in the most effective way.

ZondIP1D program is a convenient tool for automatic and semi-automatic (interactive) interpretation of profile data and can be used on IBM PC-compatible personal computers with Windows operating system.

The vertical electrical sounding (VES) method is one of the oldest electrical surveying methods. The first applications of the method date back to the 1920s. XX century. The comparative simplicity and clarity of VES has led to its wide spread and development all over the world.

Today, electrical sounding remains one of the most used electrical prospecting methods. On the basis of VES other modern technologies are developed - for example, electrotomography, based on the same principles as for "classical" electrical sounding. One of the main requirements for the use of geophysical methods is the contrast in the physical properties of the object of study relative to the host medium. For electrical prospecting by resistivity methods, to which VES belongs - this means that the studied object (body, layer, stratum, etc.) should differ markedly in specific electrical resistance from the surrounding rocks [MSU, 2007].

Specific Electrical Resistance (SER), measured in ohmmeters (Ohmm), characterizes the ability of rocks to provide electrical resistance to the passage of current and is the most universal electromagnetic property. It varies in rocks and ores in a very wide range: from 10-3 to ^{1015} Ohmm. For the most common sedimentary, eruptive and metamorphic rocks, the resistivity depends on the mineral composition, physical, mechanical and water properties of rocks, the concentration of salts in groundwater and to a lesser extent on their chemical composition, as well as on some other factors (temperature, depth of occurrence, degree of metamorphism, etc.) [Hmelevsky, 1997].

The specific electrical resistivity of minerals depends on their intracrystalline bonds. dielectric minerals (quartz, mica, feldspars, etc.) with predominantly covalent bonds are characterized by very high resistivity (^{1012} - ^{1015} Ohmm). Semiconductor minerals (carbonates, sulfates, halides, etc.) have ionic bonds and have high resistances (^{104} - ^{108} ^{Ohmm}). Clay minerals (hydromica, montmorillonite, kaolinite, etc.) have ionic-covalent bonds and are distinguished by fairly low resistances.

Ore minerals (nuggets, some oxides) are electronically conductive and conduct current very well. The first two groups of minerals make up the "rigid" skeleton of most rocks. Clay minerals create a "plastic" skeleton capable of adsorbing bound water, while rocks with "hard" minerals can only be saturated with solutions and free water, that is, that which can be pumped out of the rock.

The electrical resistivity of free groundwater varies from fractions of Ohmm at high total salinity to 1000 Ohmm at low salinity. The chemical composition of salts dissolved in water does not play a significant role, so only the total salinity of groundwater can be judged by electrical survey data. The electrical resistivity of bound water adsorbed by rock solids is low and varies little (from 1 to 100 Ohmm). This is explained by their fairly constant salinity (3-1 g/l). The average mineralization of the world ocean waters is 36 g/l.

Since pore water (free and bound) has a much lower electrical resistivity than the mineral skeleton of most minerals, the resistivity of rocks is virtually independent of its mineral composition, but is determined by such rock parameters as porosity, fracturing, and water saturation. With their increase, the resistance of rocks decreases due to the increase of ions in groundwater. Therefore, the electrical conductivity of most rocks is ionic (electrolytic).

As the temperature rises by ^{400,} the resistivity decreases by about 2 times, which is explained by the increase in mobility of ions. During freezing the resistance of rocks increases by leaps and bounds, as free water becomes practically an insulator, and electrical conductivity is determined only by bound water, which freezes at very low temperatures (below -500 C). Increase in resistance when freezing of different rocks is different: it increases a few times for clay, up to 10 times for rocky rocks, up to 100 times for loam and sandy loam, and up to 1000 times or more for sand and coarse clastic rocks.

Despite the dependence of resistivity on many factors and a wide range of variation in different rocks, the basic patterns of resistivity are established quite clearly. Eruption and metamorphic rocks are characterized by high resistivity (from 500 to 10000 Ohmm). Among sedimentary rocks, rock salt, gypsum, limestone, sandstone and some others have high resistivity (100 - 1000 Ohmm). Clastic sedimentary rocks, as a rule, have a greater resistance, the larger the size of the grains that make up the rock, i.e. depend primarily on clay content. In the transition from clays to loams, loams, loams and sands, the resistivity changes from fractions and first units of ohmm to the first tens and hundreds of ohmm [Khmelevsky, 1997].

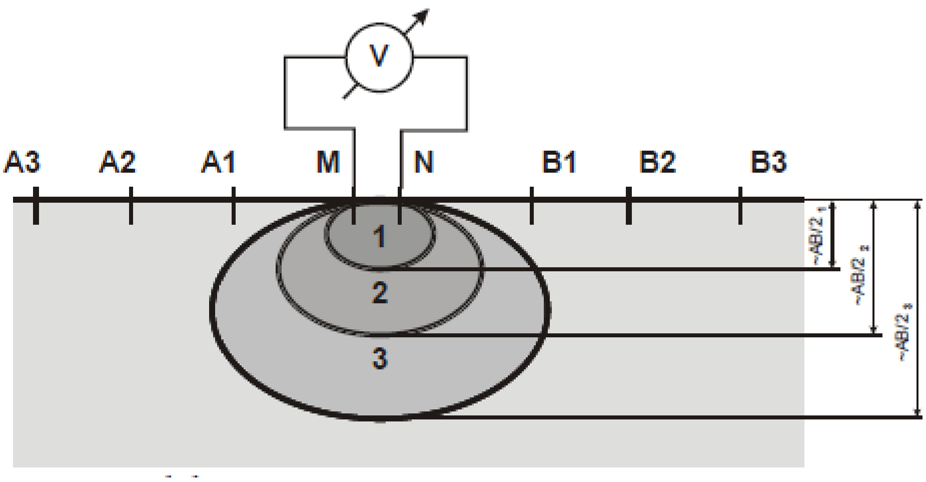
The ability of rocks to polarize, i.e. to accumulate a charge when current is passed through and then to discharge after this current is turned off is evaluated by the polarizability coefficient. The value is calculated in percentages as the ratio of the voltage that remains in the measuring line after a certain time (usually 0.5-1 s) after the current circuit is eroded to the voltage in the same line when the current passes through.

Polarization is a complex electrochemical process that occurs when a direct or low-frequency alternating (up to 10 Hz) current is passed through a rock. Ores with electronic conductivity (sulfides, sulfosols, some native metals, certain oxides, graphite, anthracite) have the greatest polarizability. The nature of these EP potentials is related to the so-called concentration and electrode polarization of ore minerals. Polarizability coefficients up to 2-6% are observed over watered loose sedimentary rocks, which contain clay particles. Their polarizability is caused by deformations of the outer covers of double electric layers, arising at the contact of solid and liquid phases. Most eruptive, metamorphic and sedimentary rocks saturated with mineral water are weakly polarizable [Khmelevsky, 1997].

The idea of the VES method is extremely simple. An electrical survey unit is assembled on the ground surface, which usually consists of two feeding and two receiving electrodes (Fig. 1).

There is an electric field in the ground and thus an electric current. The current in the supply line is measured with *an ammeter* included in the circuit. At the receiving electrodes M and N an electric potential difference occurs, which is measured with *a voltmeter*.

According to the results of measurements it is possible to judge the electrical properties of rocks at the depths of current penetration into the ground. The depth of "current penetration" depends mainly on the distance between the feeding electrodes.



Pic. 1 **Measurement scheme in the VES method.**

According to the results of the measurements performed, the apparent electrical resistance is calculated [MSU, 2007].

The program solves direct and inverse problems for arbitrary installations on the surface of a horizontally layered medium. The layer parameters are: layer thickness (in meters), electrical resistivity (in Ohm m) and polarizability (in percent).

When solving the direct problem, the point source potential is calculated using the following formula:

$$U(r) = \frac{\rho}{\pi}\left(\frac{1}{r} + \int_0^{\infty} R(m)\cdot J_0(mr)dm\right)$$

The apparent polarizability (in percent) is calculated using the Komarov-Siegel formula:

$$\eta_{\kappa} = \frac{\rho_K(\rho)}{\rho_K\left[\rho\cdot[1+\eta/100]\right]}\cdot 100\%$$

System requirements

The **ZondIPD** program can be installed on a computer with the Windows XP operating system and higher. Recommended system parameters: Processor P IV-2 GHz, 512 mb. memory, screen resolution 1024 X 768, color mode -True color (you should not change the screen resolution in data mode).

Installing and uninstalling the program

The **ZondIPD** program is delivered via the Internet. It comes with this manual. You can download the latest updates of the program at: www.zond-geo.com. To install the program, download the program from the website into the correct directory (for example, "Zond"). To install an update, simply write the new version of the program over the old one.

Before you start the program for the first time, you need to install the SenseLock security driver. To do this, open the SenseLock folder (the driver can be downloaded from the website) and run the InstWiz3.exe file. Once the driver is installed, insert the key. If everything is OK, you will see a message in the lower system panel that the key is detected.

To remove the program, erase the working directories of the program.

Symbols used in the program

Ro_a is the apparent resistance. $\rho_a = G \cdot \frac{\Delta U}{I}$ where G is the installation factor.

Eta_a - apparent polarizability. $\eta_a = \frac{\Delta U^{ВП}}{\Delta U^{ПР}} \cdot 100\%$, in percent.

Spacing: AB/2 for Schlumberger, Venus and symmetric installation; AM for two-electrode installation; AO for three-electrode installation; O'O for dipole-dipole installation.

Pseudo-depth is the approximate depth of the survey related to the setup factor (in this case the setup spread).

The probing point is the position of the center of the supply line for the Schlumberger, Venus and Symmetric unit; for other types of probing, the probing point is undefined (usually the position of the fixed electrode).

All geometric values of the program are defined in meters.

Creating and opening a data file

To start working with the **ZondIP1D** program it is necessary to create a data file of a certain format, containing information about the observing system and measured values of apparent resistivity and polarizability. Usually one file contains data for one profile of observations.

When using multi-electrode units, it should be borne in mind that the **ZondIP1D** program considers a probing curve as a data element. Therefore, the data should be presented as a set of probing curves.

Text data files organized in the **ZondIPD** program format have the extension "*. zlf", "*. zlp" and "*. ves". The program also supports induction sounding files with vertical magnetic dipole of *. vmd. The data file format is described in detail in the Data File Format section. The program also supports IPI2WIN program text files(dat, dtg) and some other formats (USF, es).

For the correct operation of the program the data file must not contain:

- curves with the number of gaps less than 3
- unconventional characters separating entries in a string (use TAB or SPACE characters)
- absurd values of measurement parameters

It is desirable that the total number of measurements contained in one file does not exceed 50,000.

When you open a project binary file with the extension "*. zlp" extension, not only the field data are loaded, but also the model corresponding to the previously performed interpretation.

Data file format

The program presents a universal data format, including information about the geometry of the installation, coordinates and relative heights of the sounding points.

Data format of Zond-IP data files (extension *. zlf).

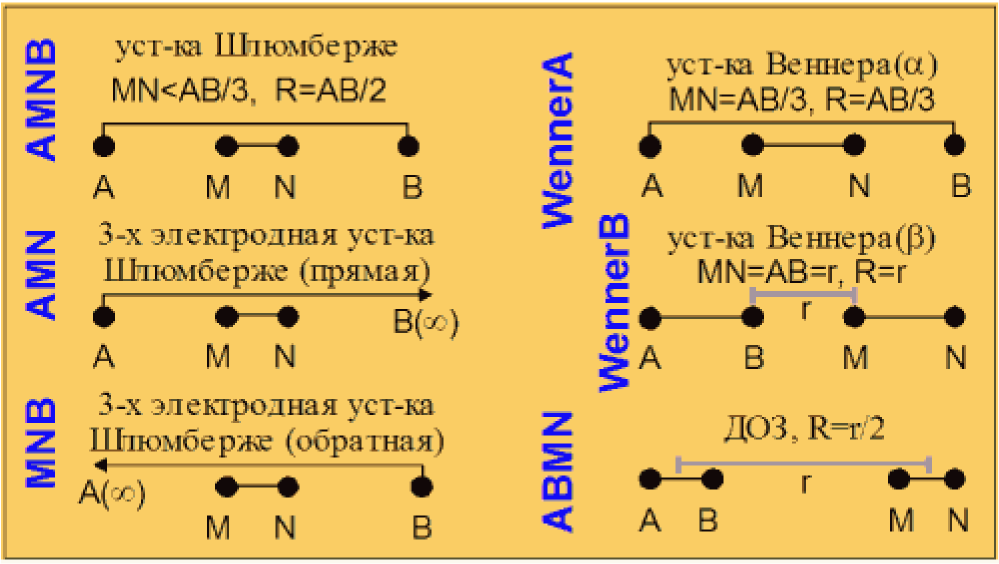
The first three lines contain information about the measurement parameters common to the whole profile.

The first line must contain the following entries separated by a space or tab character:

The first record is reserved by **ZondIP1D** program for frequency or for bottom measurements (value 8). The second record - (0-5) - type of installation: "0" - Schlumberger, Wenner (it is necessary to set the receiving line length for it), three-electrode installation; "1" - dipole-axis installation; "2" - two-electrode installation; "3" - three-electrode universal installation; "4" - four-electrode universal installation "5" - universal any installation (Figure 2). Sequence of entries in the row for the different units:

"0" - 0 0 1 (the third entry specifies the offset of the receiving line relative to the supply line along the Y axis (set if necessary, usually when measuring VP in frequency mode)).
"1" is 0 1 Len_dip, where Len_dip is the length of the feeding dipole.
“2” – 0 2
"3" - 0 3 delt_y dircosX dircosY, where delt_y is the offset of the measurement profile along the Y axis, dircosX and dircosY are the guide cosines of the receiving line.
"4" - 0 3 Bx By delt_y dircosX dircosY, where Bx and By are coordinates of the second supply electrode, delt_y is the offset of the measurement profile along the Y axis, dircosX and dircosY are the guide cosines of the receiving line.

Universal settings (3,4). The coordinates of the first supply electrode are zero. The differences are entered without taking into account the displacements relative to the first supply electrode. Measurements are taken along the X-axis, in the positive direction. The directional cosines determine the orientation of the receiving line relative to the profile. The offset of the measurement profile is made in the positive direction along the Y-axis. In general, there is no need for universal settings (in the latest versions of the program). They can be replaced by an arbitrary setting of the type (5).



Pic. 2 **VES arrays.**

If the bottom measurement system is selected, i.e. the electrodes are placed on the bottom of the reservoir, the first record must contain the number 8. In the next line the water resistance after the key "water" is written. The sounding depth is entered in the sounding description block, instead of the exceedance value, with a minus sign. Example of description of the bottom system:

8 0
Water 30

The second line - must contain the values of the spacing at which the measurements were made. For units 0, 3, 4 this is the distance from the first supply electrode to the center of the receiving line. For dipole-axis setup (1) it is the distance between the centers of the dipoles. For an arbitrary setup (5) the user chooses the span values himself. For two-electrode setup (2) - the distance between the receiving and supplying electrodes.

An arbitrary setting can also be used.

Description of an arbitrary installation:

arbitrary setting of type "5" implies setting the positions of feeding and receiving electrodes explicitly. When describing this setting, enter the X and Y coordinates of the electrodes, in the lines immediately after the line describing the spacing. The spacing can be set arbitrarily, i.e. the values are user-selected (they will be used only for displaying the curves). Coordinates of electrodes are set by lines beginning with a key, indicating to the program what coordinates and which electrode are contained in the given line. The following keys are available in the program: Ax, Ay, Bx, By, Mx, My, Nx,Ny. If an electrode is absent, its value is replaced by the symbol *. Y - coordinates are entered only if necessary. An example of setting the type "5":

```
0 5
1 2 3 4 5            //spacing - set by user
Ax -1 -2 -3 -4 -5    //X -coordinates of electrode A for each spacing
Bx -2 -3 -4 -5 -6 //X - coordinates of electrode B for each spacing
Mx 1 2 3 4 5         //X - coordinates of electrode M for each span
Nx 2 3 4 5 6         //X - coordinates of electrode N for each spacing
```

In this example, as we see, the Y coordinates of the electrodes are missing. They can be entered in the same way as the X coordinates.

The third line can contain the values of the receiving line spacing.

If the third line is missing, the receiving line spacing is considered infinitesimal. Since the program takes into account the length of the receiver lines when calculating the direct problem, it is recommended to enter this line. For the arbitrary setting (5), this line is not entered.

When using complex observation systems, with varying from point to point, in the second and third rows must be entered all the unique pairs (the distance of the receiving and feeding line), used on the profile, according to the above described rules.

This is followed by records containing information about each sounding point on the profile, combined into the blocks described below.

Block description of the sounding point:

The first line is an indicator of the beginning of the sounding point description block (must contain the notation "{").

The second line is the name of the sounding point.

The third line - additional probing parameters.

The first entry is the coordinate of the sounding point along the profile, the second entry is the elevation or depth (positive number for the terrestrial system, negative for the bottom; (in meters)).

The fourth and fifth lines contain the actual field measurements.

Each of the lines described must start with a key entry that tells the program what type of data to assign the following values to.

The values of the control keys that control the data type:

"Ro_a"-apparent resistances.

In the case of the bottom system (8), in order to avoid errors in the calculation of the apparent resistance, the ratio of the measured signal to the current (normalized signal) is entered in this line.

"Eta_a" - apparent polarizabilities.

If "_w" is added to the key, this line specifies the weights of the individual measurements.

The number and sequence of entries in the lines must correspond to the system of differences described in the second line of the file. If there is no measurement on any span, its value is replaced with an "*".

The sixth line is an indicator of the end of the sounding point description block (must contain the notation "}").

Next comes the description of the next probing point starting with "{" and so on.

The program also works with files with the *. ves extension. This format is convenient for preparing field data in any text editor. Below is an example of the *. ves format is shown below:

The first line contains the current frequency value (Hz), the second line contains the name of the probing point, the third line contains the coordinates, followed by a table of installation parameters and values of apparent resistances. The block containing information for the first point is followed by the block for the next one, starting with the title field, etc.

```
frequency 4.88
name: 1
X 10
Y 0
Z 0
ab/2 mn rk nk
1      1 100 2
2.5    1 101 2
5      1 90 2
8      1 80 2
10     1 70 2
25     1 80 2
50     1 90 2
80     1 101 2
```

ZondIP1D program supports induction sounding files with vertical magnetic dipole of *. vmd format. These files include information about installation geometry, coordinates and relative heights of sounding points.

The first line must contain the height of the installation.

The second line must contain the values of the distances (distances between the receiving dipole and the supply dipole) at which measurements were taken, sorted in ascending order.

The third line must contain the values of frequencies (Hz) at which measurements were made, sorted in descending order.

When using complex observation systems, with changing from point to point system of variances, in the second line it is necessary to enter all unique variances used on the profile, according to the rules described above.

This is followed by records containing information about each sounding point on the profile, combined into the blocks described below.

Block description of the sounding point.

The first line is an indicator of the beginning of the sounding point description block (must contain the notation "{").

The second line is the name of the sounding point.

The third line - additional probing parameters.

The first entry is the coordinate of the sounding point along the profile, *the second entry* is the elevation excess (a positive number; the minimum height is taken as 0, the others are calculated as the excesses over it (in meters)).

The fourth and subsequent lines contain the actual field measurements.

Each of the lines described must start with a key entry that tells the program what type of data to assign the following values to.

The values of the control keys that control the data type:

"Hz/HrN" are the ratios of the magnetic field components measured at the frequency written under the number **N** in the third line of the file.

The number and sequence of entries in the lines must strictly correspond to the observation system described in the second and third lines of the file. If there is no measurement at any frequency, its value is replaced by the symbol "*". If there are no measurements at any of the frequencies, the line with the index of this frequency is skipped.

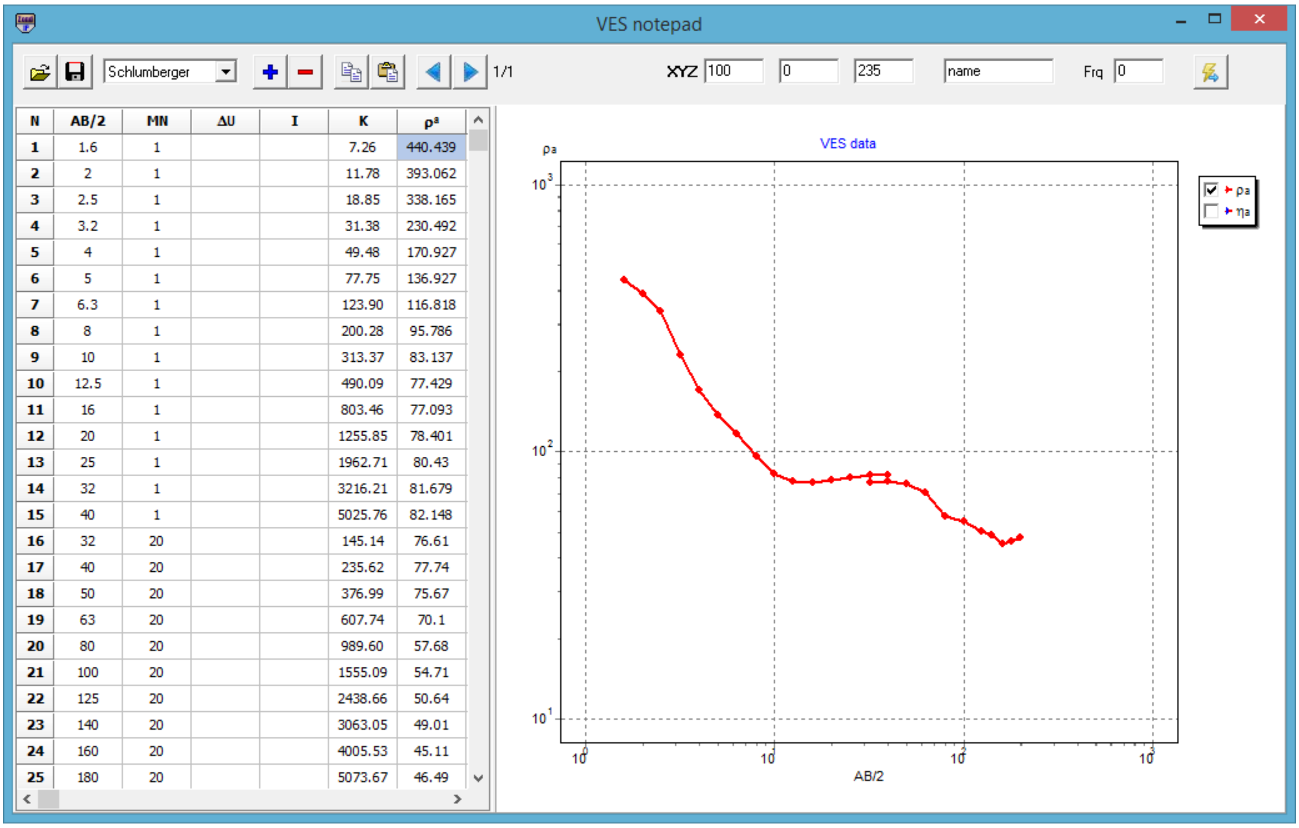
If "_w" is added to the key, this line specifies the weights of the individual measurements.

The last line is an indicator of the end of the sounding point description block (must contain the notation "}").

The stations must be recorded in the order in which they are located on the profile (in order of coordinate increase).

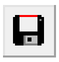
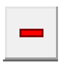
Field data entry dialog (VES notepad)

The dialog box is for entering new data and is available in the **File/VES notepad** main menu. The VES notepad option dialog box is shown in Fig. The **VES notepad** option dialog box is shown in Figure 3.



Pic. 3 **VES notepad dialog window.**

The main window menu contains the following buttons:

	Open a data file for one sounding or a database for several in *. txt.
	Save the file for one sounding or the database for several soundings in the *. txt.
	The field for selecting the type of installation.
	Add a probe point.
	Remove the probing point.
	Copy the data in the table.
	Paste data from the clipboard (copied, for example, from an MC Excel file).

	Move to the next point.
	input fields, where the X,Y,Z (elevation) coordinates of the sounding point are entered sequentially.
	In this field you should enter the name of the sounding point. Preferably not more than 5 characters.
	If the current frequency is relatively high (>20 Hz), then starting at some intervals, this can affect the results. For relatively low frequencies (<10 Hz), a value of zero can be used.
	Form the project and enter the interpretation mode.

Data can be copied directly into a table from an MS Excel application ([CTRL] + [C], [CTRL] + [V]).

In the ***XYZ*** field you must specify the coordinates of the sounding point, as well as the name of the point (the "name" field in Fig. 3).

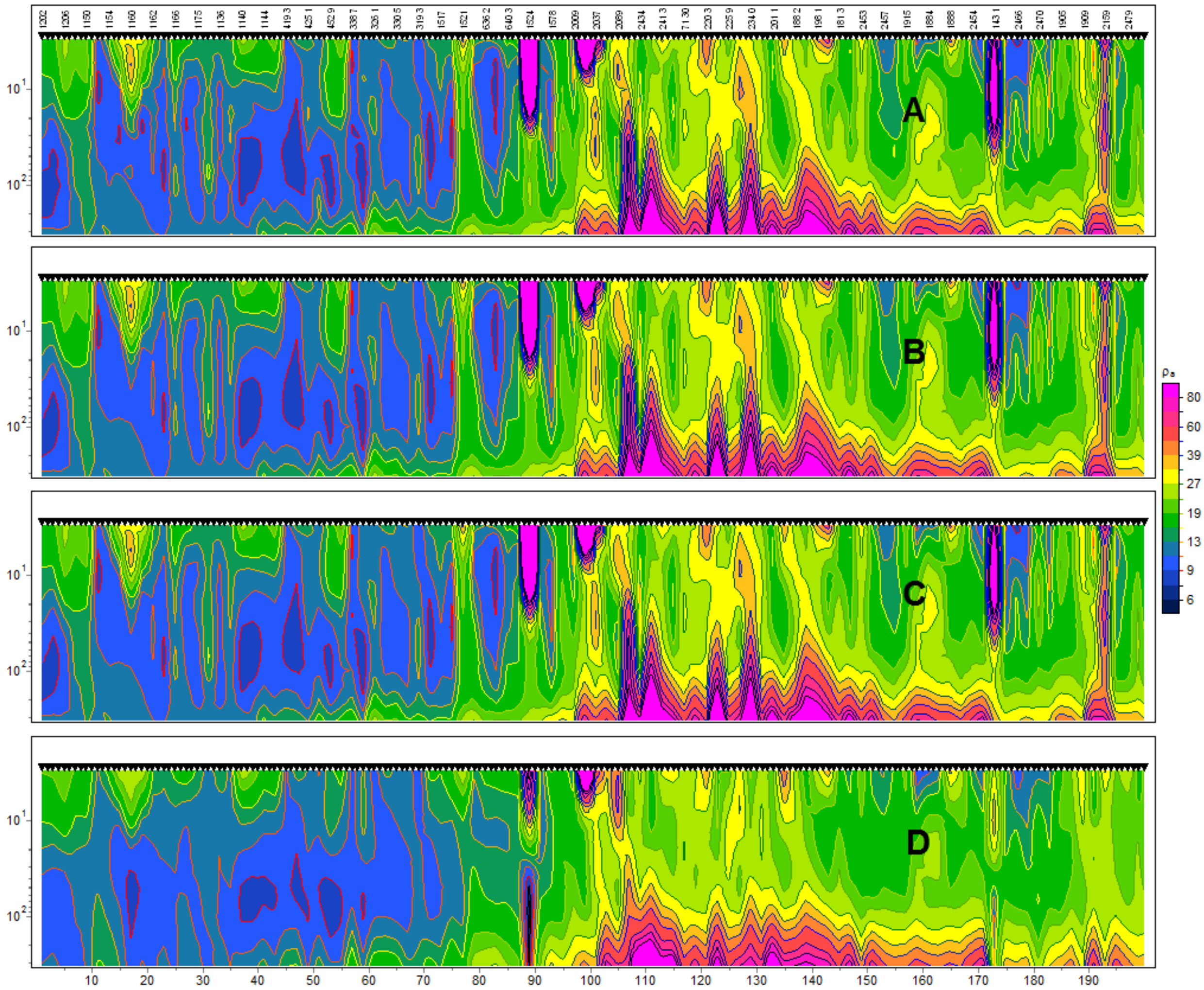
In the ***Frq*** field - set the current frequency. Allows the calculation to take into account the frequency, it is recommended to use in the case of current frequencies greater than 20 Hz.

In the left area of the window there is a table containing the following columns: the first one or two columns (depending on the type of installation) are for setting the installation geometry. For a Schlumberger installation: [AB/2[- half of the supply line length, [MN] - receiver line length; for Schlumberger 3-electrode setup - A0, [MN] - receiver line length; for Wenner setup [AB/2] - half of the supply line length, for dipole setup [OO] - distance between the centers of the receiving and supply lines, [MN] - supply line length; for pole-pole setup AO. Then follow the columns that contain the measurement data directly: [ΔU] - voltage, [I], [A] - current, [K] - installation factor (calculated automatically), [ρa,Ohmm] - apparent resistance (can be both set and calculated automatically), [ηa] - apparent polarizability.

Use the button  to create the next probing points. After all the parameters are set, press the button  to go to the inversion.

Processing field data

Field data processing is performed by the **Data operations** menu section functionality and applies to the entire measurement profile at once.



Pic. 4 **Example of applying Data operations to profile data. Where A - original data, B - data after applying the *Denoise data* option, C - data after applying the *Smooth data* option, D - data after applying the *2D Smoothing* option.**

Saving interpretation results

The result of data profile interpretation is stored in a binary file of Zond-IP project files (*. zlp extension). This file stores the name of the file with field data, selected parameters and limits of parameter changes for each sounding point and much more. You can save the result of interpretation by pressing the toolbar button or its corresponding menu item. If the auto-save option is enabled, the interpretation result will be saved automatically after a specified period of time. You can also save the theoretical and observed curves in zlf format.

Possible options for saving data:

Project data files	Save the current project to a zlp file.
Calculated data files	Save the calculated values to a text data file.
XYZ model files	Save the environment model to a tabular file.
Section file	Save the current model in sec format (a graphical image with corner snapping). This file type can be used as a substrate.
Project with calculated	Save the current project to a file, with the observed data replaced by the calculated data.
Model in columns	Save the current model as a table of parameters in DAT format
Observed data files	save the observed data file, taking into account the editing performed (extension *. ves)

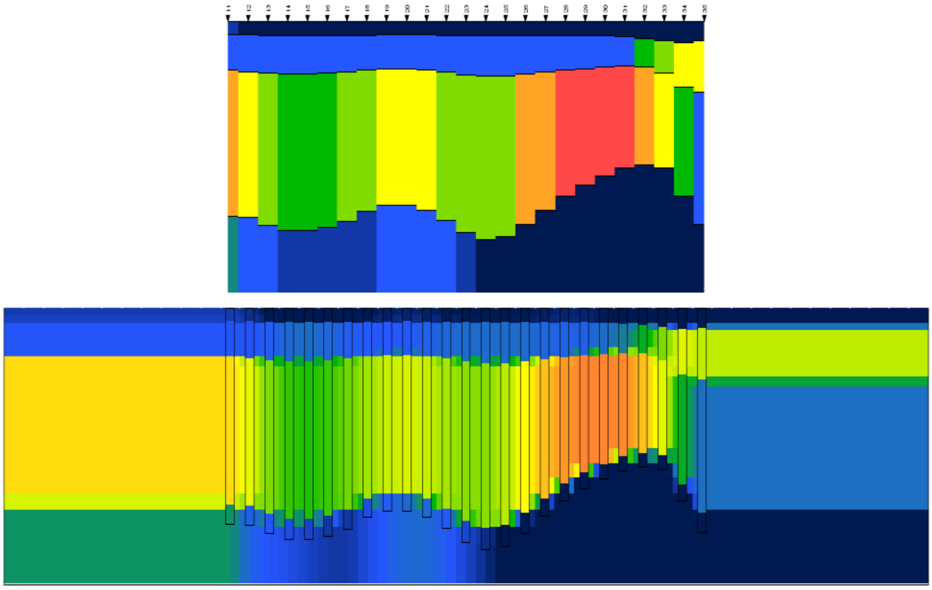
Exporting data

ZondIP1D program allows you to export data to MS Excel, Auto CAD format and create files for use in other **Zond** programs. This function is available in the **File/Export to** tab**.**

The following data export options are available:

Excel report
Create a file containing reporting information for each sounding point, measured and calculated data, sounding point coordinates, parameter models, etc.
Excel map
Create a file containing the name of the stations, their coordinates and the values of resistivity and the position of the bottom for each of the layers at each sounding point (useful for constructing area maps of the parameter distribution).
SEG-Y file
Export the model to a Seg-Y file.
CAD section
Export models to an AutoCad *.dxf file.
Pseudo BH data
Create a set of files corresponding to the sounding points in crt well data format.
MOD1D file
Save the model in a file with *.mod1d extension for further use in other **Zond** programs.
Joint_inv file
Create a data file for joint inversion that can be used by other one-dimensional **Zond** programs.

The ***Export/Pseudo BH data*** option creates a set of models in log and lithology data format. These data are convenient to use as, for example, a reference when interpreting other methods in **Zond** programs, or when interpreting data from neighboring profiles. In this case, the data are loaded as lithologic columns, the color scale of which corresponds to resistivity values.



Pic. 5 **Example of using models saved with the Export/Pseudo BH data option in another program.**

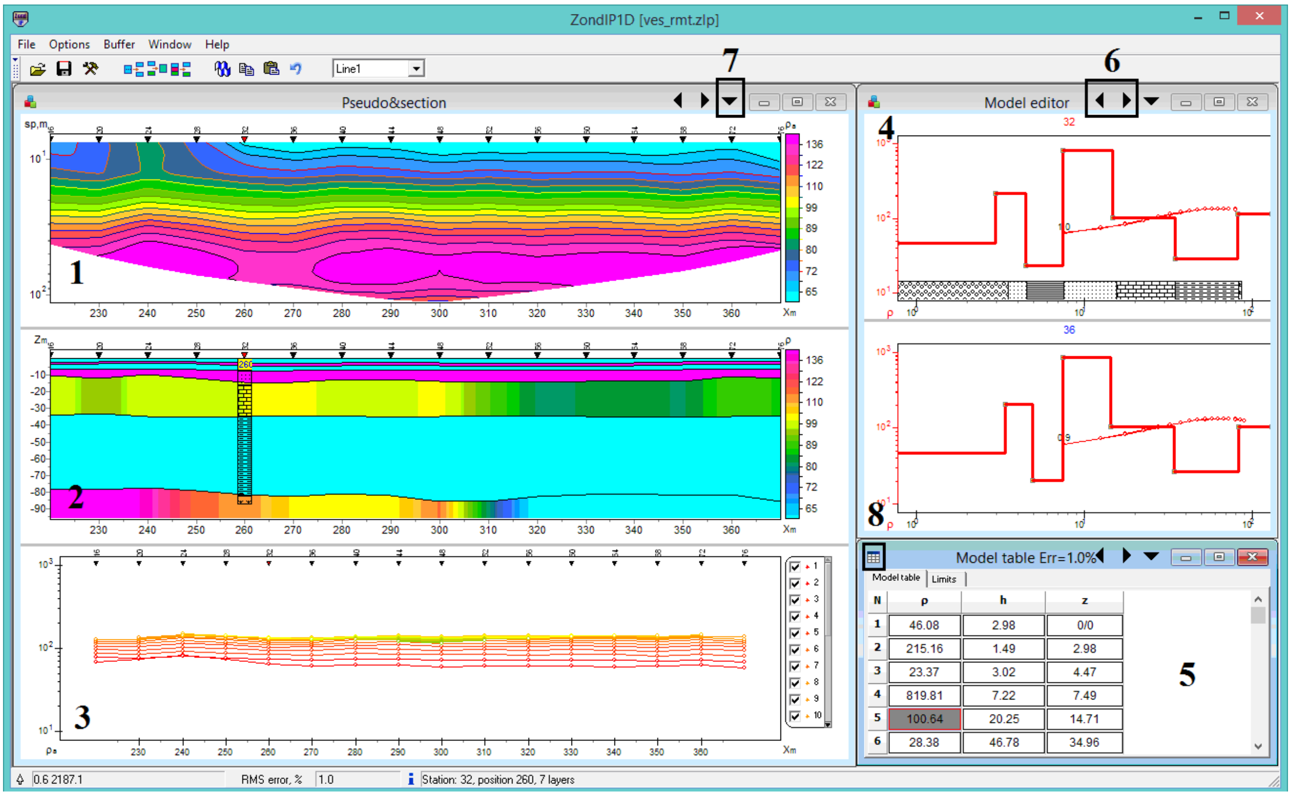
You can also use the **MOD1D file** export function to compare the results of different methods. If you use this file in another **Zond** program or for a different profile in an area survey, the model saved in this way will be loaded as a new model.

How to work with the program

Main program window

After creating a data file "*. zlf" or "*. ves", it should be loaded using the button or the corresponding menu item. When the file is successfully loaded, the function buttons for working with the data are activated on the toolbar of the main program window, and brief information about the first probing point appears in the right section of the status bar (Fig. 6). When moving the mouse cursor over the windows created during work with the program, the coordinates corresponding to the own axes of the given window are displayed in the left section of the status panel of the main program window. Clicking the first system button of the window designated in the plan as "setting objects" brings up a context menu allowing you to call a setting dialog box for each of the objects contained in the window. The order of editing the objects is described in detail in the corresponding sections (ModelEditor, Section, Parameter Table, Pseudo-Section, ProfilingPlots, TheoreticalandExperimental CurvePlots).

The second ◀ and third ▶ buttons allow you to move from one probing point to another.



Pic. 6 **The main window of the program. The following windows are marked with numbers: 1 - Pseudo-cut, 2 - Section, 3 - Profiling Plots, 4 - Model Editor, 5 - Parameter Table; 6 - Transition buttons, 7 - Object Setup button, 8 - Additional Options button.**

Program style library

Use the **Window** tab of the function menu of the main program window to select.

The library contains 4 variants of object configuration.

In the **Interpretation** style (Fig. 6) the user simultaneously operates with four windows: the first window contains a pseudo-section, a section and graphs of profiling (with the possibility to choose the depicted parameter by the user), separated by a floating slider; the second window contains graphs (from one to three) of model editing; the third graph contains a table of parameters, the fourth - graph of theoretical and experimental curves.

In the **Standard** style - there are no graphs of theoretical and experimental curves and graphs of profiling.

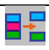
The **Profile** style does not contain a graph of theoretical and experimental curves.

The **User** style uses one window instead of the first two. The user chooses which parameter will display each of the objects.

The ***Tile vertical*** and ***Tile horizontal*** options of the **Window** tab allow you to change the location of the open working windows of the program. The ***Default*** option returns the default window settings for the selected viewing style.

Toolbar of the main program window

The toolbar is used to quickly call the most frequently used functions in the program. It contains the following function buttons (from left to right):

	Open a data or project file.
	Save the result of interpretation or calculated curves.
	Start the dialog box for setting the program parameters.
	Add Layer. The user-selected layer is split into two with equal (logarithmic scale) power.
	Combine the two layers into one with a power equal to the combined power of both.
	Delete the selected layer.
	If you right-click: displays a pop-up menu where you have to choose for which items the inversion will be performed: *current* - for the current point, *to end* - from the current point and to the end of the profile, *to start* - from the current point and to the beginning of the profile. Left-click: Start the inversion in the selected direction or for the current point.
	Write to the buffer the model on the current picket.
	The right mouse button brings up a pop-up menu with the following options: *To end* - copy the model from the buffer to the current and all subsequent stations (to the end of the profile); *To start* - copy the model from the buffer to the current and all previous stations (to the beginning of the profile); *unfixed only* - copy only unfixed parameters.
	Cancel the last action on the current station.
	Selecting a profile (when working with area data).

Functions menu of the main program window

If you need to call a tooltip about a particular menu function while working with the program, this is done by right-clicking on that option.

The names of the menu items and their purpose are listed below:

File
File/VES ***Notepad***
Open the dialog box for entering field data (details).
File/Open ***data***
Open a data file of the "*. zlf" or "*. zlp".
File/Save ***data***
Save interpretation result or theoretical curves.
File/Edit ***data***
Open and edit the data file used by the program in the Notepad editor.
File/Program ***setup***
Start the dialog box for setting the program parameters (more details).
File/Project ***information***
Add information about the current project (company name, location, project name, equipment, operator, additional information, etc.).
File/Load ***MOD1D/2D***
Load a model file of the *. mod1d or *. mod2d file previously saved in other **Zond** programs**.**
File/Export ***to***
Export data or model (more details).
File/Print ***preview***
Call the Print Model or Data dialog box (details).
File/Russian
The program menu is in Russian.
File/English
The program menu is in English.
File/Recent
Open one of the previously opened projects.
File/Exit
Exit the program.
Options
Options/Set ***lines/coordinates***
Set profile line (when working with area data) (more details).
Options/Plane ***data***
Display the isoline plan of the selected parameter. It is recommended for area data (more details).
Options/3D ***fence diagram***

Invoke the dialog to view the volumetric model. Works with area data (more details).
Options/Geological ***editor***
Call the polygonal section editor, which allows you to easily superimpose geological objects on the results of geophysical interpretation (more details).
Options/Boreholes
Options/Boreholes/Create/edit borehole data
Call the dialog of creating (or editing) columns of well information (more details).
Options/Boreholes/Load ***borehole data***
Load well data from a file of a special format. Such file should be previously saved in this or other **Zond** program. In this way it is easy to exchange borehole information between different projects or programs.
Options/Boreholes/Remove borehole data
Delete well data from the project.
Options/Data ***operations***
Options/Data operations/Denoise ***data***
Procedure of automatic correction of observed data. In this variant only sharp jumps on the curves are removed. It is applied to the entire profile. It is recommended to save the original data beforehand (more details).
Options/Data operations/Smooth ***data***
The procedure of automatic correction of the observed data. This option removes sharp spikes and makes a slight smoothing of the curves. It is applied to the whole profile. It is recommended to save the original data beforehand.
Options/Data operations/2D ***smoothing***
The procedure of two-dimensional automatic correction of the observed data. In contrast to the previous variants, the features of neighboring sounding curves are used in the analysis. It is possible to suppress P/C effects. The procedure is applied to the whole profile. It is recommended to save the original data beforehand.
Options/Invert&Apriori
Options/ Invert&Apriori/Set boundaries
Calls up an auxiliary dialog for introducing a set of a priori boundaries into the section. If accurate information about the geometry of the geoelectric horizons is available, it can be entered with the mouse. When using a priori boundaries, the following settings are recommended: smooth inversion, m0=median (more details).
Options/ Invert&Apriori/Load ***boundaries***
Build a model based on a priori bounds from a text file.

Options/ Invert&Apriori/Smooth ***model***
Smooth the current model horizontally and vertically.
Options/ Invert&Apriori/Global ***search***
Procedure for finding the global minimum of the solution. For successful execution of this procedure, the general and individual limits of parameter variation must be specified. The procedure does not guarantee the real global minimum of the solution and the success of its application depends on the correctly set limits of parameter variation.
Options/BiLog ***scale 1:1***
Sets the bi-logarithmic scale for graphs displaying individual probing curves.
Options/MT ***Data***
A menu subsection for joint interpretation of MT (AMT, RMT) and VES data (more details).
Buffer
Buffer/Model ***1,2...5***
Write to or load from the clipboard(s) a model.
Buffer/Open
Open in one window all models saved in the buffer for the current station (more details).
Window
Window/Tile ***vertical***
Vertical ordering of windows.
Window/Tile ***horizontal***
Horizontal arrangement of windows.
Window/Zond ***"Standard"***
Loading the "Standard" style.
Window/Zond ***"Profile"***
Loading the "Profile" style.
Window/Zond ***"User"***
Loading the "User" style.
Window/Zond ***"Interpretation"***
Loading style "Interpretation.
Window/Default
Set the default window settings for the current style.
Help
Help/About
About the program.
Help/Context

Download the help file.
Help/ ***Check for updates***
Check for updates at **www. zond-geo. com.**
Help/ ***Error! Clear setting***
Restart the program if an error occurs and clear all settings.
Help/Bing ***maps api_key***
If automatic downloading of the map from the Internet does not work, then the current api-key of the program has exceeded the allowable limit of requests. We recommend that you create your own api-key in your Bing account and enter it in the input field of this option.
Help/Show ***news***
Show **Zond Software** news announcements**.**
Help/Send ***message to us***
Send message to developers Message must be typed in transliteration.

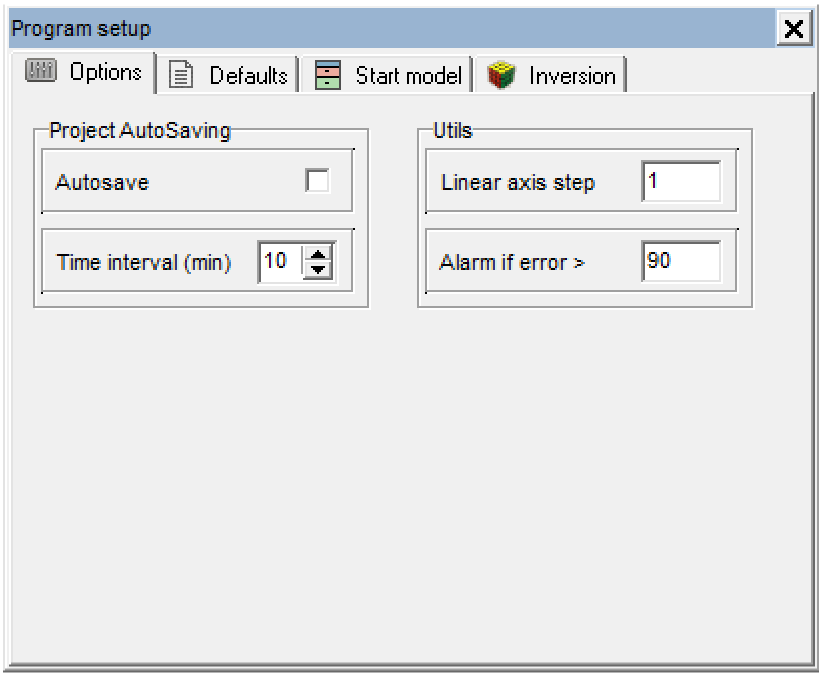
"Hotkeys

[Space]	Call the automatic model selection procedure for this item
[S]	Add Layer. The user-selected layer is split into two with equal (logarithmic scale) power.
[M]	Combine the two layers into one with a power equal to the combined power of both.
[D]	Delete the selected layer.
->	Move to the next station
<-	Move to the previous station
[Escape]	Interrupt the automatic selection process.

Program properties window

The window allows you to configure the options for automatically saving the project, the initial model, the default values and the inversion parameters; called by the toolbar  button or the corresponding menu item (**File/Program setup**).

Options tab



Pic. 7 **Program Setup window, Options tab**

Project AutoSaving - sets the automatic saving mode of the opened project.

The ***Autosave*** option enables the automatic saving of the opened project after a specified time interval.

The ***Time interval*** field sets the time interval after which the project is automatically saved in minutes (the data is saved to a file whose name is made up of the name of the opened file, with 'Temp' appended).

Utils area - configures additional parameters.

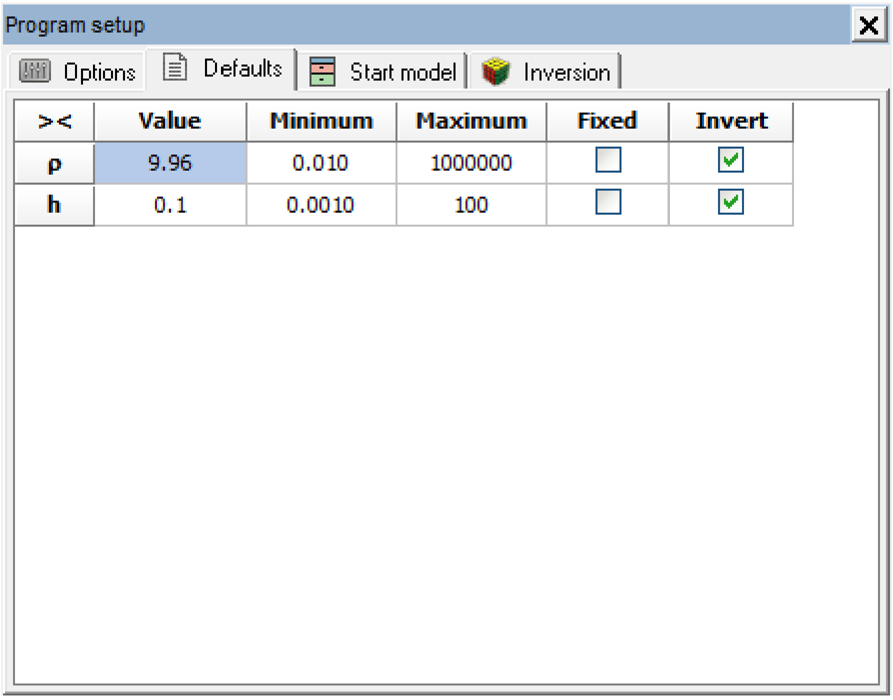
Linear axis step - sets the desired linear axis step. This option is used to change the scale of the image.

Field ***Alarm if error*** > - calls the dialog box to confirm the profile inversion, if the error at the current picket exceeds the value set in the field.

Defaults tab

This tab is used to specify a variety of parameters used when working with the model. The rows correspond to the type of layer parameters (properties and power), the columns to the options (fig. 8).

The first column [Value] is the default value (used when setting the initial model).



><	Value	Minimum	Maximum	Fixed	Invert
ρ	9.96	0.010	1000000	☐	☑
h	0.1	0.0010	100	☐	☑

Pic. 8 **Program Setup window, Defaults tab**

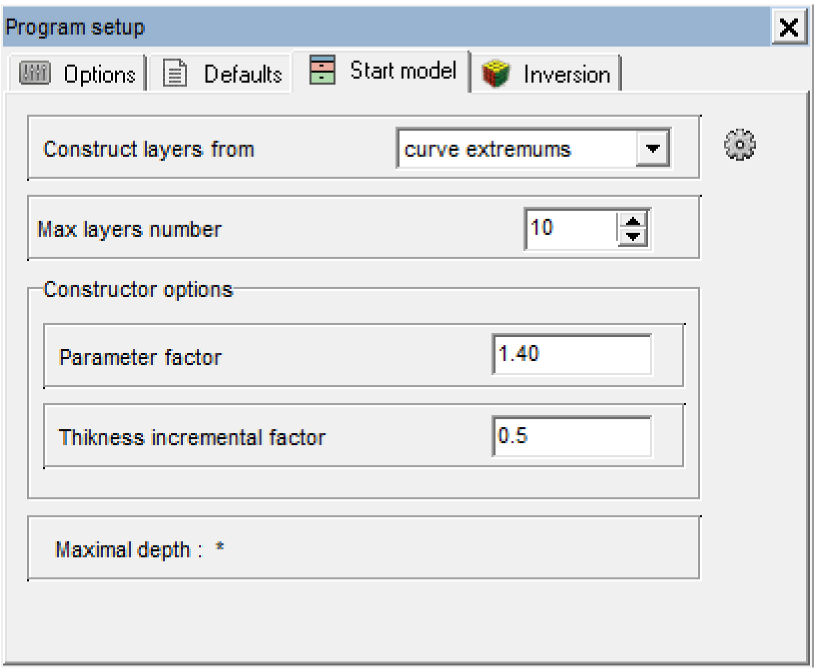
The second column [Minimum] is the value of the lower limit of the default parameter change (used when fixing parameters). It is used in *Global search*, if individual limits are not set.

The third column [Maximum] - the value of the upper limit of the default parameter change (used when fixing parameters). It is used in Global search, if individual limits are not set.

The fourth column [Fixed] - defines whether the parameter of the given type should be fixed after reading the field data file. The option has no effect when the project file is loaded.

The fifth column [Invert] determines whether the parameter of the given type should be corrected during automatic matching.

Start model tab



Pic. 9 **Program Setup window, Start model tab.**

The tab serves as a constructor of the initial model for all stations of the profile.

The ***Construct layers from*** field specifies the algorithm used to determine the positions of layer boundaries.

Value of *curve extremums* - boundary positions are determined by field curves (from extremum analysis). The maximum number of layers is specified in the Layers number field.

The *incremental factor* value - the default values are used when specifying layer powers. The power of each subsequent layer is equal to the power of the previous one multiplied by the *Thickness incremental factor*. The number of layers is specified in the ***layers number*** field.

The **Constructor options** area contains options for setting the initial parameters of the model.

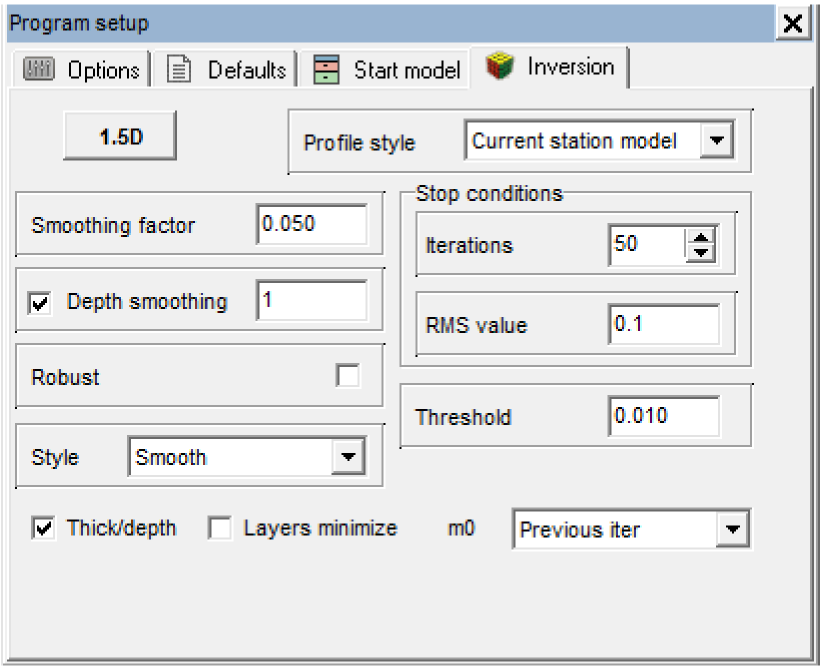
The ***Parameter factor*** field sets the coefficient by which the maxima and minima of the model parameters will be multiplied (limits 1 - 4). The coefficient increases the contrast of the initial model, which is especially important if the curves have no asymptotes.

The ***Thickness incremental factor*** field sets the factor by which the thickness of layers will be multiplied (limits 0.2 - 1) or the incremental factor for each successive layer in incremental factor mode (limits 1 - 4).

Maximal depth - shows the maximum depth, for the current partitioning parameters of the model builder.

After setting the start model parameters, you can apply them to the current project without closing the program parameter settings window by pressing the button .

Inversion tab



Pic. 10 **Program Setup window, Inversion tab.**

Profile style field - defines the style of automatic matching during profile interpretation.

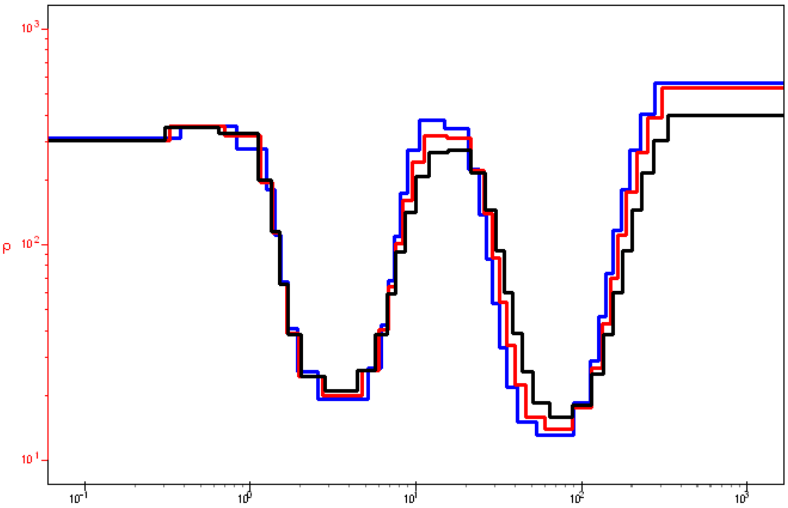
Current station model value - the current station model is used as the starting model for the selection.

Start station model value - the model of the point where the interpretation began is used as the starting model for the selection.

Previous station model value - the previous station model is used as the starting model for the selection.

The ***Smoothing factor*** field sets the initial value of the smoothing parameter. The value of this parameter depends on many factors: the number of Jacobian conditionality, the ratio of useful signal to noise in the measured values, the number of determined model parameters, and is chosen empirically. For data with a high level of noise or to get a smoother distribution, relatively high values of the smoothing parameter are chosen: 0.05 - 10; for high quality of measurements, values: 0.005 - 0.01 are used.

Below (Fig. 11) the models obtained for three values of the smoothing parameter (0.01 - blue, 0.1 - red, 1 - black) are shown.



Pic. 11 **Models for different values of Smoothing factor (0.01 - blue, 0.05 - red, 0.1 - black)**

The **Stop conditions** area contains the conditions of the end of automatic picking on the point.

Input field ***Iterations*** - sets the maximum number of iterations, after reaching which the automatic interpretation process stops.

The ***RMS value*** field specifies the minimum misalignment at which the automatic interpretation process stops.

Depth smoothing option - defines the degree of smoothing of the model with depth (if the option is enabled). The greater the value of the parameter (1 - 10), the stronger the averaging of the parameters of neighboring layers of the depth part of the model. It is used if *Smooth* or *Focused* is selected in the **Style** inversion type list when the model with depth is strongly oscillating.

Robust option - this option should be enabled if individual strong spikes associated with systematic measurement errors are present in the data.

The ***Style*** option is the type of procedure for restoring the parameters of the cut.

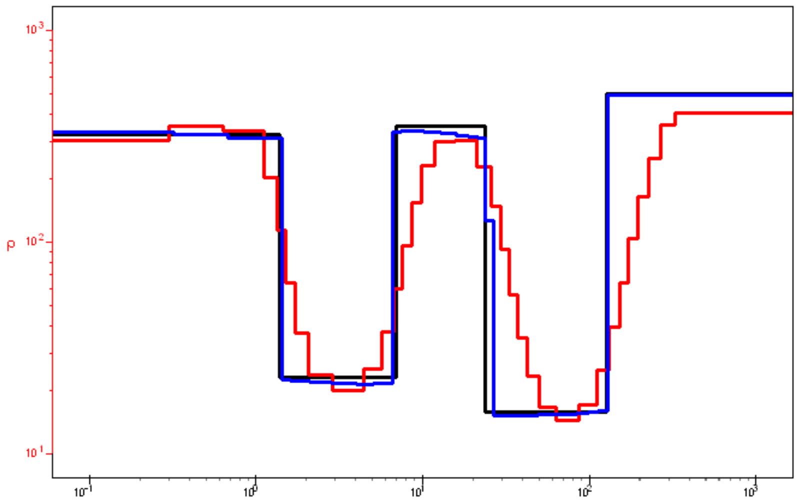
The value of *Standard* is a least-squares inversion with regularization by a dumping parameter. The algorithm allows you to obtain a model of the medium with sharp boundaries. Careless use of this modification of the inversion can sometimes lead to unstable results or an increase in the standard deviation. It is best to apply this method as a refinement, after inversion with the *Smooth* algorithm.

The *Smooth* value is a least-squares inversion using a smoothing operator and additional contrast minimization. As a result of applying this algorithm, the smoothest parameter distribution is obtained. It is recommended to use this type of inversion at the initial stages of interpretation. In this case, the number of layers in the model should significantly exceed the number of layers in the real model. It is desirable that the number of layers exceeds 10 and their capacities are fixed.

The *Focused* value is a least-squares inversion using a smoothing operator and additional contrast focusing. As a result of applying this algorithm, a piecewise-smooth distribution of parameters is obtained, i.e. a model consisting of layers with constant resistivity. It is recommended to use this type of inversion at initial stages of interpretation. The number of layers in the model should significantly exceed the number of real model layers. It is desirable that the number of layers exceeds 10 and their capacities are fixed.

Careless use of the focusing inversion parameters can lead to algorithm divergence and produce unstable models.

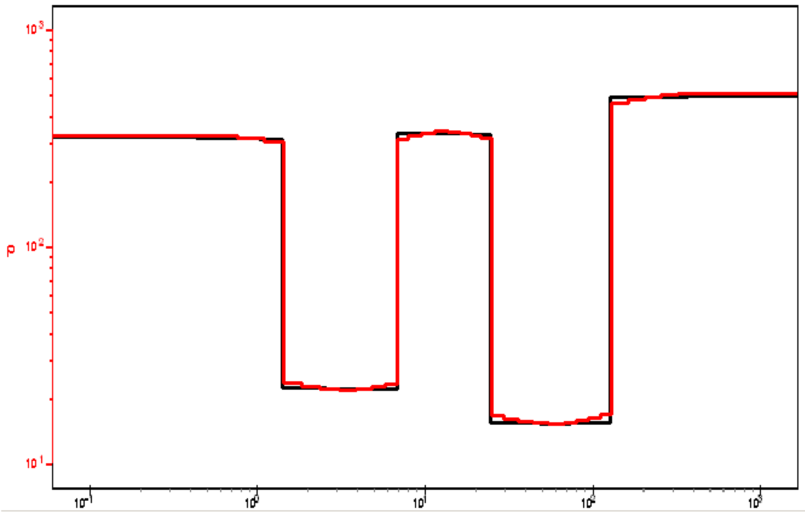
Figure 12 shows the models obtained using three inversion algorithms (standard - black, focusing - blue, smooth - red).



Pic. 12 **Models obtained using different inversion algorithms (Standard - black, Focused - blue, Smooth - red)**

The ***Threshold*** option sets the maximum contrast value of neighboring layers, after reaching which the parameters of these layers are not averaged between each other (i.e. it is considered that there is a boundary between the layers). The value of this parameter is chosen empirically (0.001-1). Choosing a very small value of the parameter can lead to algorithm divergence (in this case you should increase its value). Too large values of the parameter lead to a smooth distribution.

Figure 13 shows the models obtained for two values of the ***Threshold*** parameter (0.01 - black, 0.1 - red).



Pic. 13 **Models obtained for two values of the contrast parameter *Threshold* (0.01 - black, 0.1 - red)**

The ***Thick/depth*** option enables the power selection mode (if this option is active) or the depth selection mode (if not active). Depth selection is useful when the depths of the boundaries are known and fixed. In case of profile data inversion with depth selection, it is recommended to set "default" limits for depths beforehand.

The ***Layers minimize*** option turns on the mode of minimizing the number of layers. It is used in the combination of multilayer cut and focusing inversion. Layers minimize algorithm. A multilayer model (10-15 layers) and focusing inversion (smoothing factor~0.1) is selected. In the course of the inversion, layers close in parameter are combined and the process continues, but with a smaller number of layers.

Field ***m0*** - this option has a significant effect on the inversion results and determines the variant of smoothness that will be minimized in the inversion.

m0 defines the reference model, and *the smoothing factor* the degree of closeness to it.

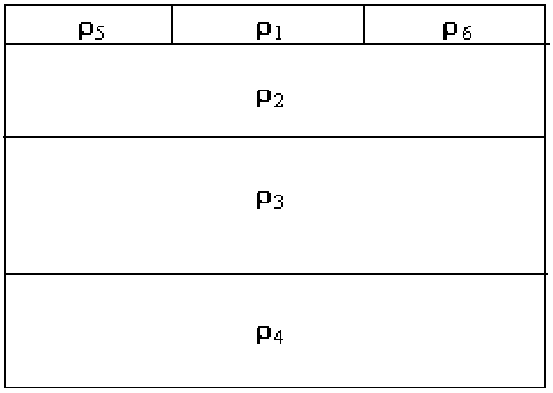
The *Median model* - m0=median *model* value is used to get the smoothest model of the environment in multilayer inversion. This option should also be used in focusing inversion or in the presence of introduced a priori boundaries. The degree of smoothness is determined by the smoothing factor field value.

The value *Start model* - *m0=start model* (reference model - model at the first iteration, initial model) is used when there is an a priori geoelectric model of the profile, if it is necessary that the deviation of the final model from the initial model is not too strong. It is often used in multilevel inversion, when after each model cycle the model is smoothed (for example, horizontally) and a new refinement cycle is made. The degree of closeness to the initial model is determined by the value of the smoothing factor field.

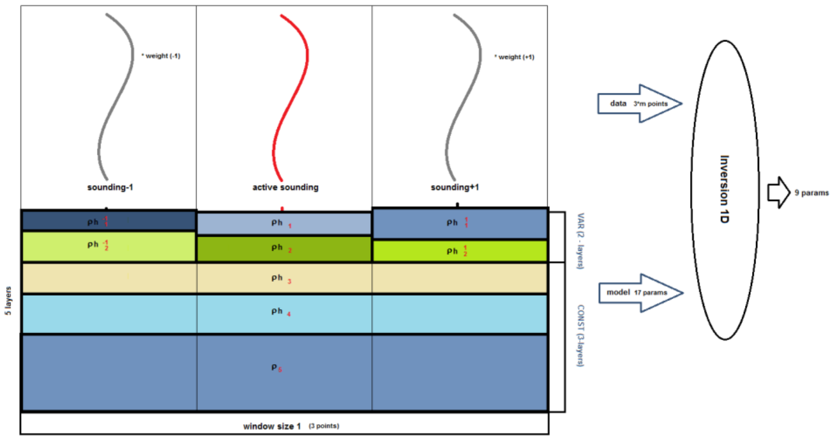
The value *Previous iterator* - *m0=previous model* (reference model - model at the previous iteration) is used to obtain models with the best unconstrained in multilayer inversion. In this variant smoothness is not minimized, but the operator contributes to a more stable convergence process. The convergence rate is determined by the value of the smoothing factor field.

The button **1.5D** opens a dialog box for configuring a special algorithm for profile data inversion.

When this algorithm works, the medium model is represented by a horizontal-layered or subhorizontal-layered section (with smoothly varying boundaries) in the lower part. The upper part of the section can vary greatly from point to point (Fig. 14). When solving the inverse problem, several adjacent soundings with a common bottom and a variable top are used. Selection is performed simultaneously for all curves in the window, and the central point is given the greater weight in the calculation of the inconsistency (Fig. 15).



Pic. 14 **Media model in 1.5D inversion**



Pic. 15 **1.5D inversion scheme**

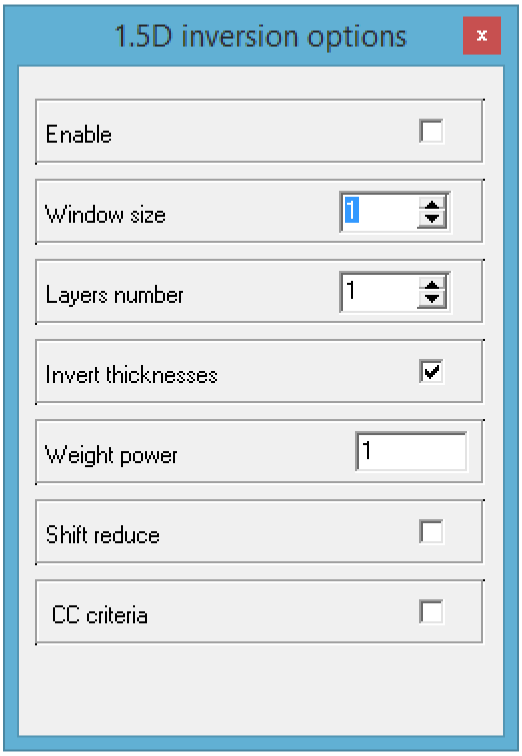
To combat the P-effect, each window curve is given an additional parameter - the offset. This parameter is minimized for all curves during selection, thus significantly reducing the effect of the P-effect.

The proposed algorithm differs from the standard inversion by additional parameters and the design of the smoothing operator. The parameters modeling the P-effect have less weight compared to the others. The main characteristics of the algorithm are given below.

Selection is performed simultaneously for several adjacent curves - "in the window", and the central point is given more weight in the calculation of the inconsistency.

The P-effect of each curve is picked up in the inversion process.

Each curve corresponds to its own model with a common bottom and a variable top.



Pic. 16 **The 1.5D inversion parameter window.**

The ***Enable*** option enables this algorithm.

The ***Window size*** option defines the size of the window for which the algorithm is applied. A value of 1 means 3 probes, 2 means 5 probes in the window.

The ***Layers number*** option sets the number of layers modeling the top (heterogeneous) part of the section.

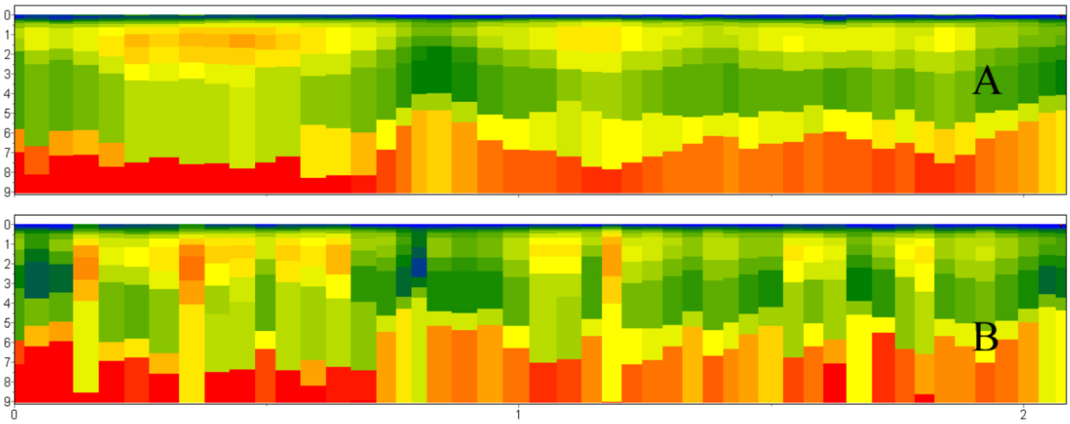
The ***Invert thicknesses*** option specifies whether the power of the first layers simulating the upper (heterogeneous) part of the section will be adjusted.

The ***Weight power*** option sets the coefficient of curves weight reduction depending on the distance from the central window curve (0 - all window curves have the same weight).

The ***Shift reduce*** option specifies whether the P effect will be taken into account in the inversion.

CC criteria option - uses the criterion of total correlation of neighboring stations.

Figure 17 shows the results of the algorithm (A) in comparison with the results obtained by the standard method (B).



Pic. 17 **Comparison of the results of the standard 1D inversion (B) and application of the 1.5D inversion algorithm (A).**

Interpretation of field data

ZondIP1D program allows to solve one-dimensional direct and inverse VES problems. The program makes automatic selection of parameters and powers of layers. It is possible to fix the parameters and set the limits of their variation, as well as to determine the significance (weights) of individual measurements.

The forward problem is solved using the linear filtering algorithm, while the inverse problem is solved using Newton's method.

Interpretation mode becomes available after reading the data from the file(s).

Automatic selection of model parameters

It is used for fast model selection of one or all points of the profile. The automatic selection is performed by minimizing the standard deviation between the calculated and the field curve by Newton's method. Click the button on the toolbar with the right mouse button. A pop-up menu will be displayed, in which it is necessary to choose for which points the inversion will be performed: current - for the current point, to end - from the current point and to the end, to start - from the current point and to the beginning of the profile. After selecting the option, click the button with the left mouse button.

The status bar displays information about the current relative divergence between the calculated curve and the field curve. The selection is pumped when the specified relative divergence between the calculated curve and the field curve is reached or when the specified number of iterations is reached.

The automatic selection does not provide a single solution, since the calculated curves may be the same for several models. Therefore, in automatic selection, when setting the starting model, it is necessary to take into account the a priori information about the geoelectric section. If there is reliable a priori information about the studied geoelectric section, it is reasonable to fix the known parameters or to limit the area of their change in the starting model.

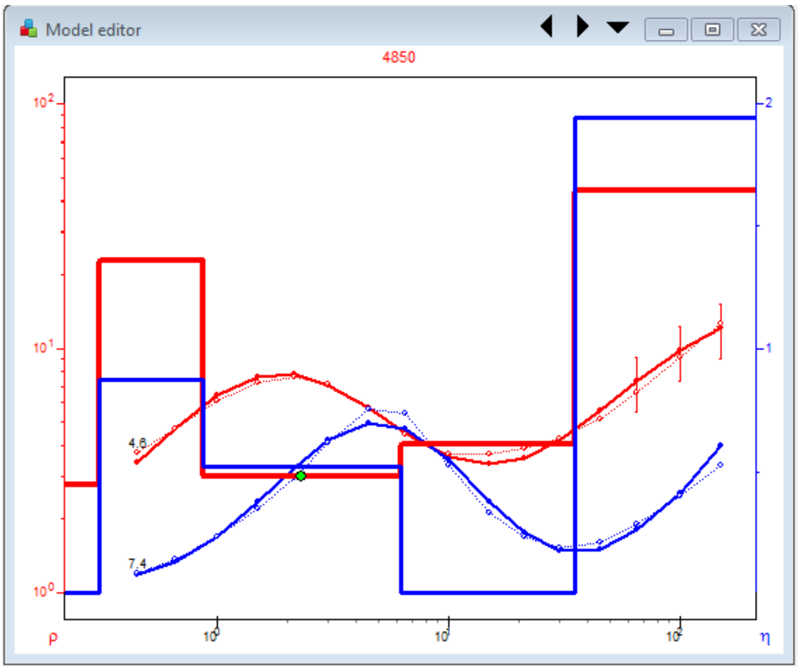
Program objects

Interactive interpretation is performed in objects: ModelEditor, Section, Parameter Table, visualization in objects: Pseudo-cut, Profilinggraphs, Theoreticaland experimentalcurvesgraph, which appear automatically after data loading.

Model editor (Model editor)

Designed to visualize the field and calculated sounding curves, as well as visualize and edit the curve parameters of the calculated model.

Figure 18 shows the model editor window. The red and blue curves with shaded circles are the experimental curves of apparent resistivity (left red axis) and polarizability (right blue axis) as a function of the separation in meters (lower horizontal axis). The model is displayed with a solid red line.



Pic. 18 **Model editor window**

The graphical parameters of the observed, calculated and model plots can be set in the graphsettings dialog box (right-click+[SHIFT] on the plot). Axis parameters can be set in the axis settings (right-click+[SHIFT] on the axis).

The window may contain one, two or three similar graphs, allowing you to edit the model at three neighboring stations.

Parameters of the model are edited with mouse. To change the model parameters, place the cursor on the model curve (the cursor shape should change to ⇳⇔) and click the left mouse button, then drag the selected section of the model curve with the mouse button pressed. The green circle on the model curve shows the activated layer.

Changing the position of the vertical sections of the curves corresponds to changing the geometry of the model (i.e. the capacities [right button] and the depths of the upper edges of the layers [left button]).

A change in the position of the horizontal sections of the model curves corresponds to a change in the parameters of the model layers.

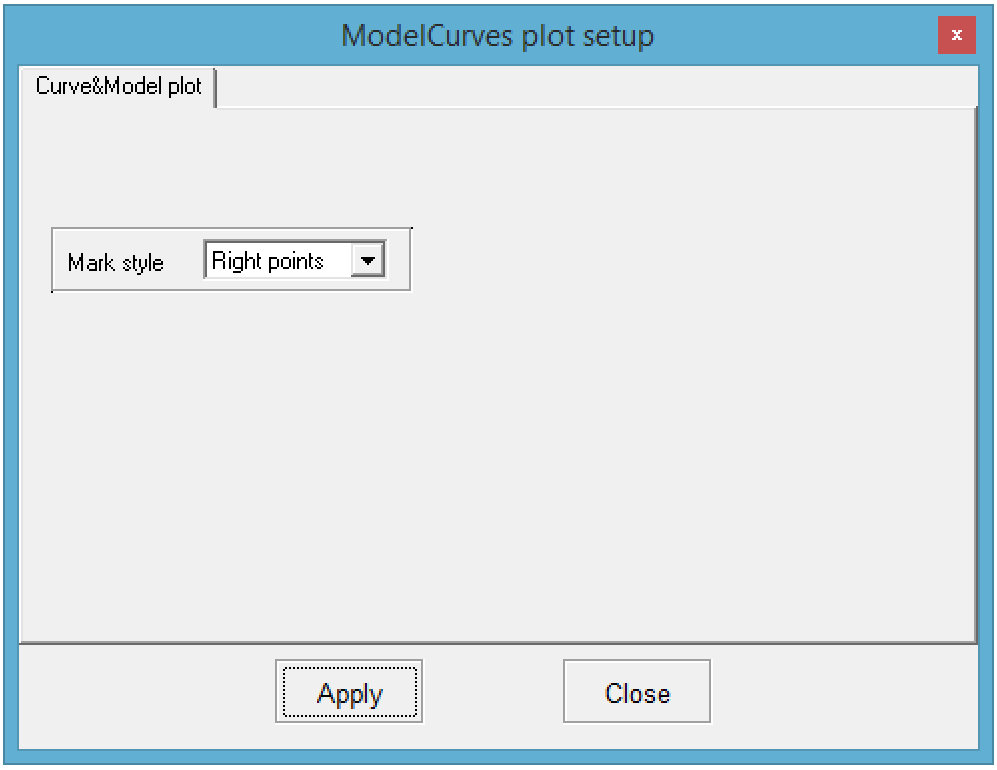
Right-clicking on the points of the observed curves brings up a context menu that allows you to set weights to the observed data:

Good point	Set the weight to 1 selected point.
Bad point	Set the weight to 0.5 for the selected point.
Very bad point	Set the weight 0 of the selected point.
Good points >>	Set the weight to 1 selected point and all points to the right of it.
Bad points >>	Set the weight to 0.5 for the selected point and all points to the right of it.
Very bad points >>	Set the weight 0 to the selected point and all points to the right of it.
Good points <<	Set the weight to 1 selected point and all points to the left of it.
Bad points <<	Set the weight to 0.5 for the selected point and all points to the left of it.
Very bad points <<	Set the weight to 0.5 for the selected point and all points to the left of it.
Delete point	Delete point.
Delete point>>	Delete the point and all points to the right of it.
Delete point<<	Delete the point and all points to the left of it.

Right-clicking in the object area brings up a context menu with the following options:

Print preview	Call the graph print dialog box.
Display legend	Show or remove the legend to the charts.
Setup	Call the Object Settings dialog box .
Left axis Resistivity	Show the apparent resistance for the left axis.
Left axis polarizability	Show apparent polarizability for the left axis
Right axis Resistivity	Show the apparent resistance for the right axis.
Right axis polarizability	Show apparent polarizability for the right axis.

The ***Setup*** option brings up *a dialog box for setting the curve parameters* (fig. 19).



Pic. 19 **Curve parameter settings dialog box.**

The ***Mark Style*** field determines how to draw captions to the charts.

The *Left points* value is on the left side of the graphs.

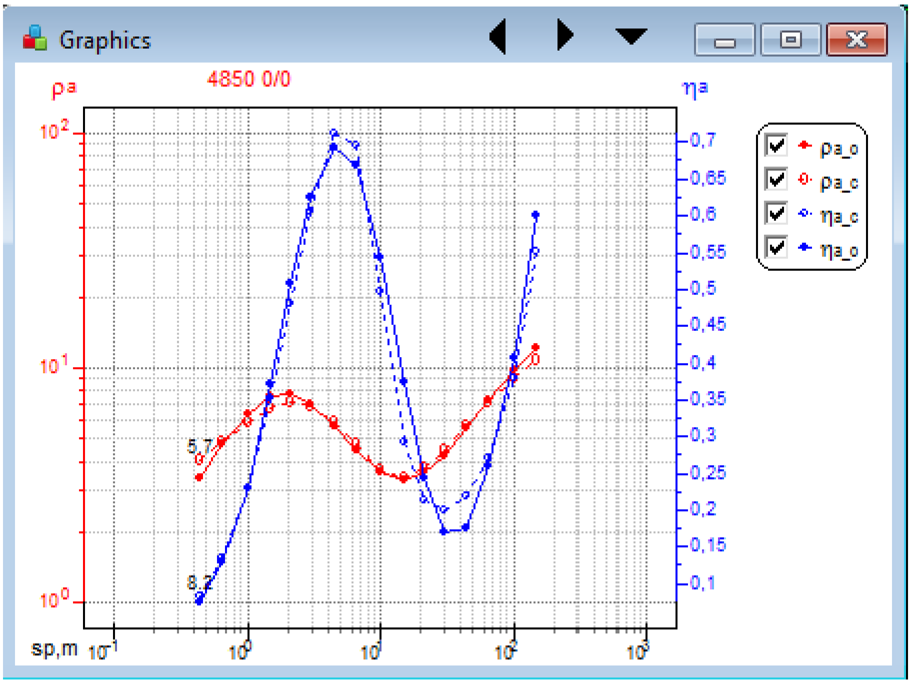
The value of *All points* is from point to point.

The *Right points* value is to the right of the graphs.

Value *None* - do not draw captions to the charts.

Graph of theoretical and experimental curves (Data Editor)

It is displayed when the **Interpretation** style is selected (**Window** tab). The window may contain one, two, or three similar blocks - a graph. By default, red curves usually correspond to the left axis, blue to the right one. If one type of measurement is used in the program, the right axis is missing.

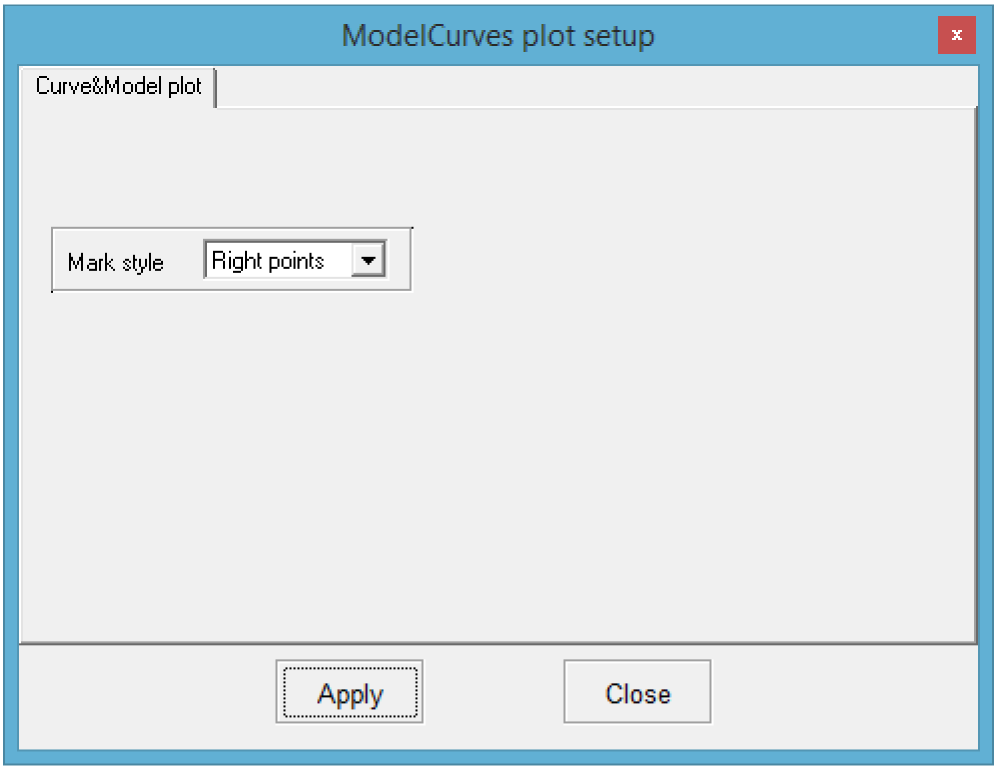


Pic. 20 **Graph of theoretical and experimental curves.**

Double-clicking in the object area brings up a context menu with the following options:

Print preview	Call the graph print dialog box.
Display weights	Show or hide weights of points on graphs
Display legend	Show or remove the legend to the charts.
Setup	Call the Object Settings dialog box.

The ***Setup*** option brings up a dialog box for setting curve parameters (fig. 21)



Pic. 21 **Curve parameter settings dialog box.**

The ***Mark Style*** field determines how to draw captions to the charts.

The *Left points* value is on the left side of the graphs.

The value of *All points* is from point to point.

The *Right points* value is to the right of the graphs.

Value *None* - do not draw captions to the charts.

An additional context menu is invoked by pressing the **Options** button 

The ***Change orientation*** option allows you to change the order of the sounding graphs for neighboring points: from top to bottom or from left to right.

The ***MultiCurves Plot Setup*** option (see Fig. 21) calls the curves settings dialog box described above.

The ***Set MultiCurves Plot number*** option allows you to set the number of neighboring sounding points for which curves are displayed simultaneously (from 1 to 3).

Right-clicking on the points of the observed curves brings up a context menu that allows you to set weights to the observed data:

Good point	Set the weight to 1 selected point.
Bad point	Set the weight to 0.5 for the selected point.
Very bad point	Set the weight 0 of the selected point.
Good points >>	Set the weight to 1 selected point and all points to the right of it.
Bad points >>	Set the weight to 0.5 for the selected point and all points to the right of

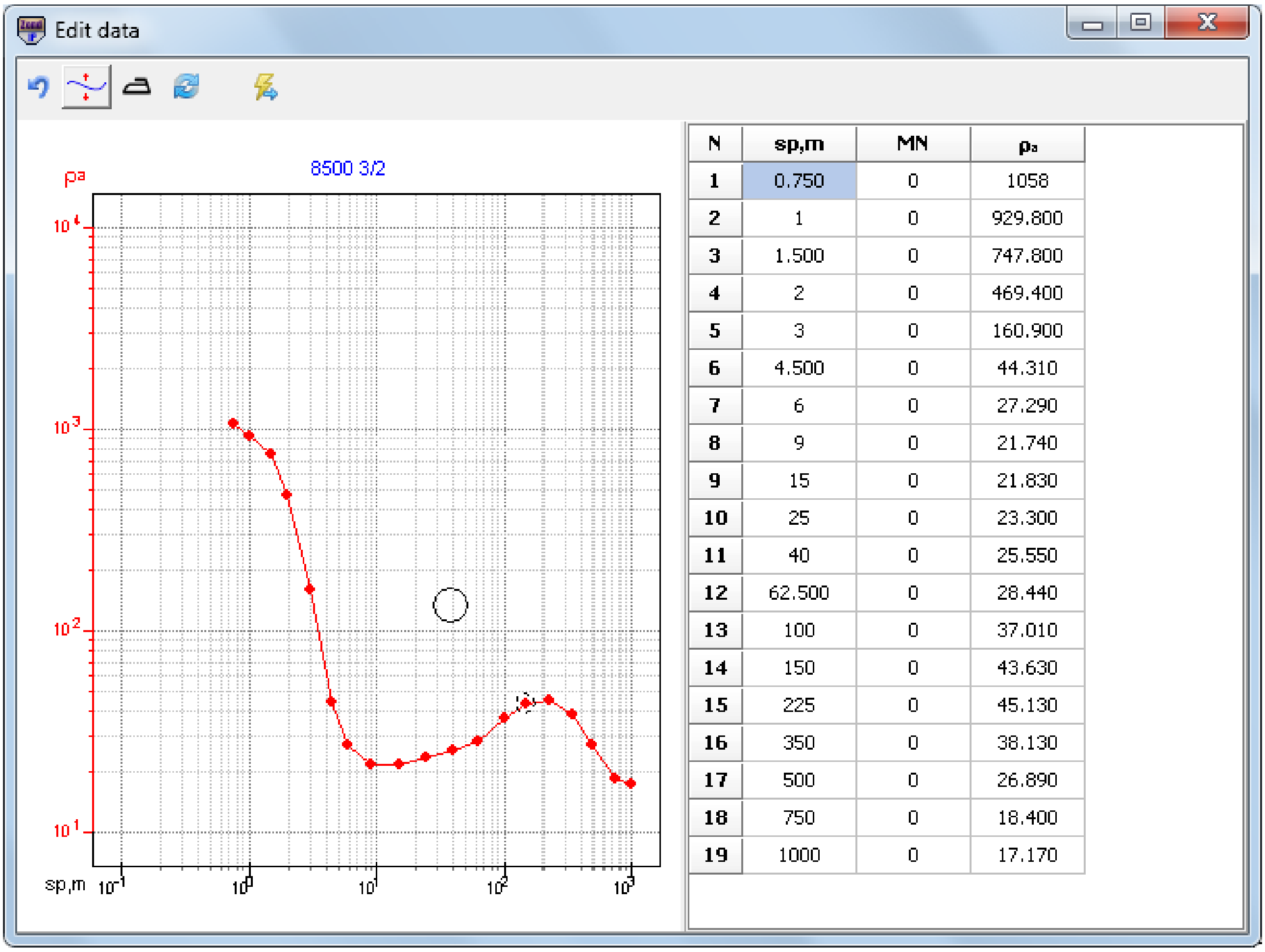
	it.
Very bad points >>	Set the weight 0 to the selected point and all points to the right of it.
Good points <<	Set the weight to 1 selected point and all points to the left of it.
Bad points <<	Set the weight to 0.5 for the selected point and all points to the left of it.
Very bad points <<	Set the weight to 0.5 for the selected point and all points to the left of it.
Delete point	Delete point.
Delete point>>	Delete the point and all points to the right of it.
Delete point<<	Delete the point and all points to the left of it.
Edit data	Edit Curves.

To increase or decrease the point weights, use the right and left mouse buttons with the [ALT] key pressed.

To delete points, use the mouse wheel with the [ALT] key pressed. The size of the deletion area can be adjusted with the wheel.

Setting the point weights plays a significant role in automatic model matching. Points with weight 0 are not considered in automatic matching. The point weights are defined as follows: = 1-dispersion/(measured value). Measurement weights can be set in the data file and saved in the project file.

The **Edit data** option is intended for manual editing of the sounding curves. After selecting this option, the **Edit data** dialog box appears (Fig. 22).



N	sp,m	MN	ρa
1	0.750	0	1058
2	1	0	929.800
3	1.500	0	747.800
4	2	0	469.400
5	3	0	160.900
6	4.500	0	44.310
7	6	0	27.290
8	9	0	21.740
9	15	0	21.830
10	25	0	23.300
11	40	0	25.550
12	62.500	0	28.440
13	100	0	37.010
14	150	0	43.630
15	225	0	45.130
16	350	0	38.130
17	500	0	26.890
18	750	0	18.400
19	1000	0	17.170

Pic. 22 **The Edit data window for probing curves.**

The window toolbar contains the following buttons:

	Return to the original curve.
	Select the mode to move the entire curve or a segment of it.
	Smooth the curve.
	Redraw the curve in the other windows.
	Exit edit mode by saving the changes.

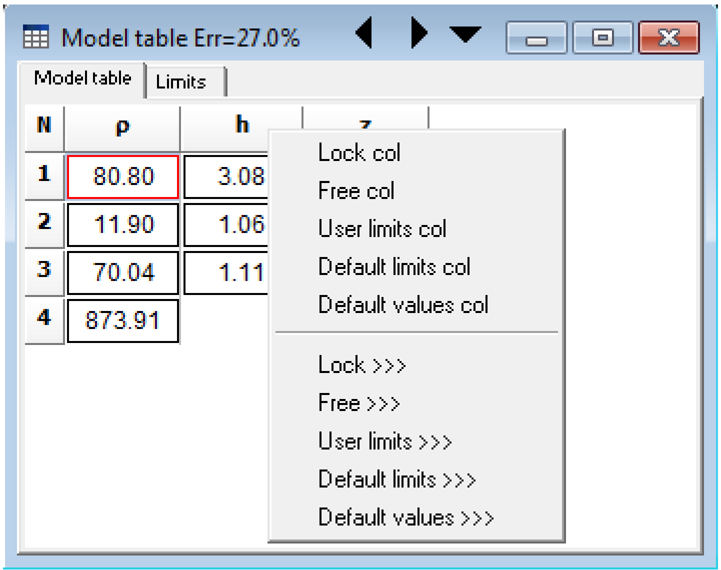
The window consists of two areas. On the left the edited curve is displayed graphically. On the right side the table, the gaps and the values of the edited parameter are shown. To delete a point on the curve use the right mouse button. At that, in the table the deleted point will be displayed in gray color. To restore the point, use the left mouse button. To delete several points, use the mouse wheel with the [ALT] key pressed. The size of the deletion area can be adjusted by rotating the wheel. When editing is finished, go to inversion by clicking the button on the window toolbar.

Parameter table

The tabular edit model parameters window is used to change the model parameters using the keyboard. The table contains 3 or 4 columns (depending on the presence of polarizability). Each row of the table contains parameters of one layer.

The first column contains the values of resistivity of layers, the second column (if available) - the values of polarizability, the third - the power, and the last - the depth to the upper edge of the layers, taking into account the height of the probing point. In the case when the variation limits are set for the parameter, the output field is shaded in a certain color (light gray by default). If the layer parameter is fixed, the output field also has its own color, dark gray by default.

The context menu is called by right-clicking on the table cells (fig. 23). If you click on the first row (the table header), the operation selected in the menu will be applied to the given parameter of all layers (cell in this case is replaced with col). If you click on the first column, the operation selected from the menu will be applied to all the parameters of this layer.



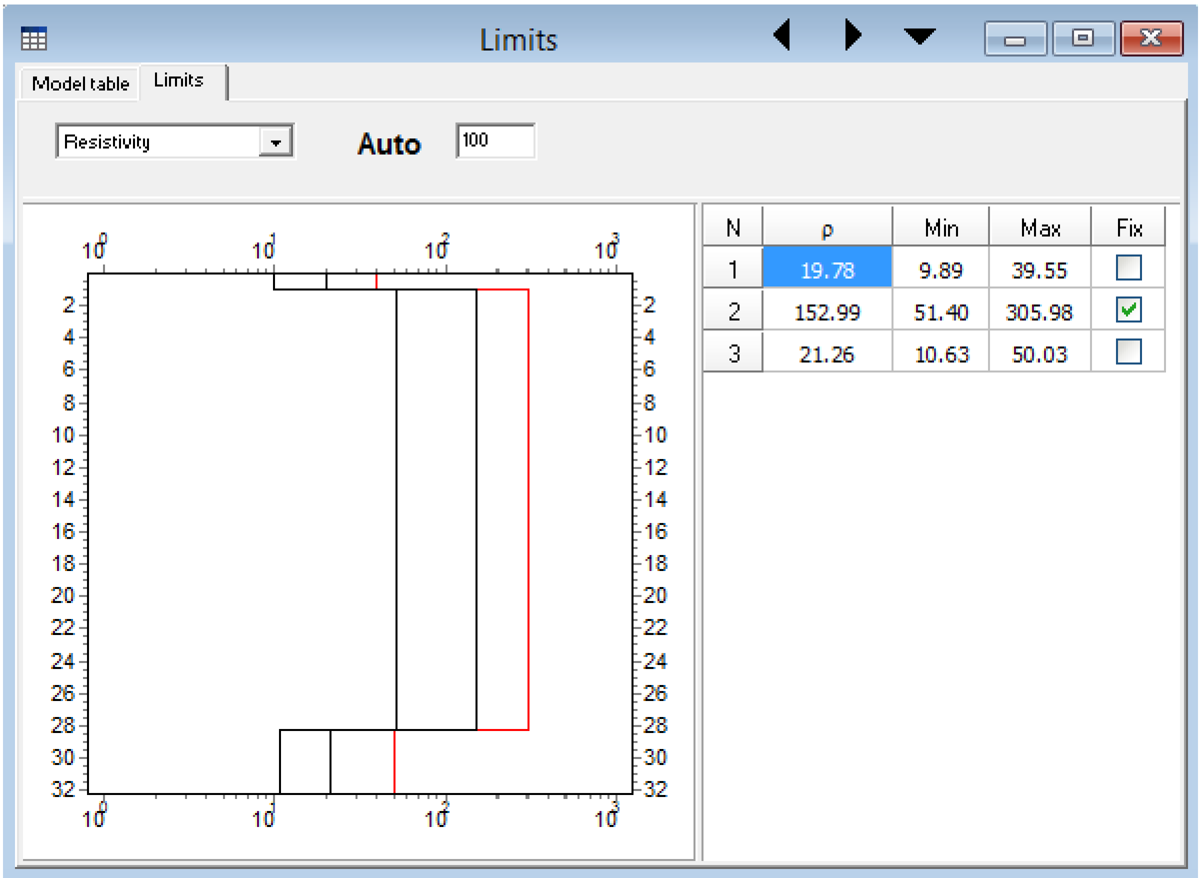
Pic. 23 **Model table window**

Lock col/cell/row	Fix parameter.
Free col/cell/row	Remove the fixation of the parameter.
User limits col/cell/row	Set the custom ranges of parameter change.
Default limits col/cell/row	Set the parameter change ranges (set by "default").
Default values col/cell/row	Set the parameter to the "default" value.
Lock >>>	Fix the parameter in the current model and in the models of all

	subsequent points.
Free >>>	Remove the parameter fixation in the current model and in the models of all subsequent points.
User limits >>>	Set the user parameter change ranges in the current model and in the models of all subsequent points.
Default limits >>>	Set the parameter change ranges (default) in the current model and in the models of all subsequent points.
Default values >>>	Set the parameter to the "default" value in the current model and in the models of all subsequent points.

Limites tab

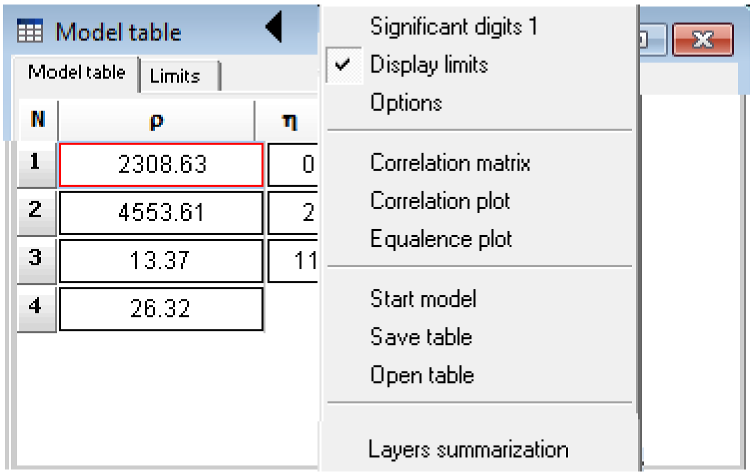
This tab is used to set the parameter change limits (Fig. 24). The buttons on the toolbar allow you to select the type of parameter for which the change limits will be set. The [**Auto**] button assigns limits for all parameters of this type automatically, according to their values and the specified percentage of deviation. The graph shows the model of the selected parameter type (black), the lower (red) and the upper (blue) boundary of the parameter change.



Pic. 24 **Model table window, Limites tab.**

The lower and upper limits of the model parameters can be edited with the mouse. The parameter limits can be edited in the table to the right of the graph.

An additional context menu, called by clicking on the icon in the upper right corner of the window (fig. 25).



Pic. 25 **Model table window, additional settings.**

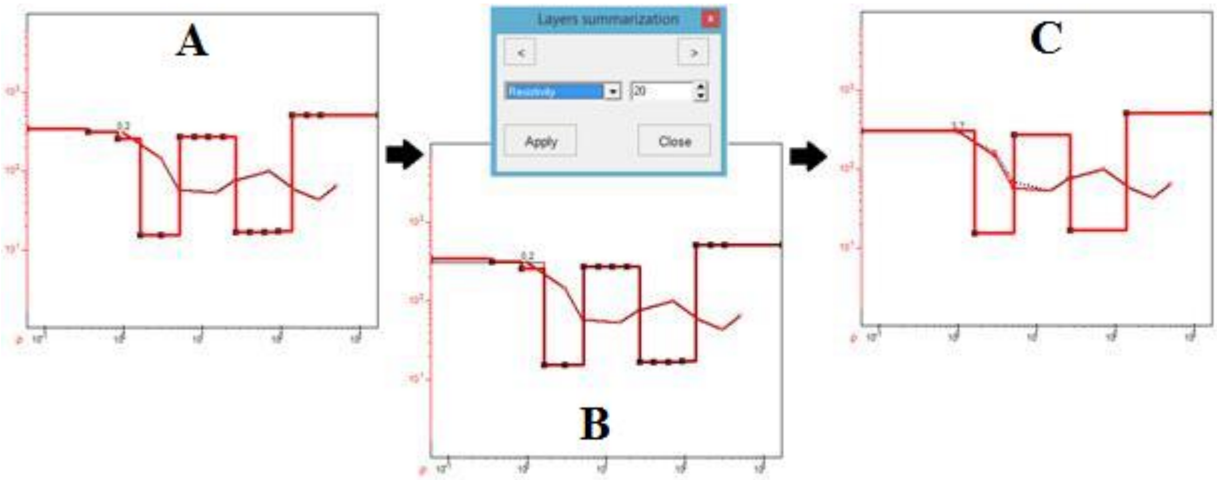
Significant digits	Set the accuracy with which the parameters will be displayed.
Display limits	Show or hide the parameter change limits.
Options	Opens the Table Graphics Settings dialog box.
Correlation matrix	Show the correlation matrix and confidence limits of the model parameters.
Correlation plot	Call the correlation map window for a pair of parameters.
Equivalence plot	Call the window for building a cloud of equivalent models.
Start model	Go back to the starting model.
Save table	Save the current model to a file (MDL extension).
Open table	Load the model from a file (MDL extension).
Layers summarization	Calls the Layer Merge dialog box.

The **Layers summarization** dialog is used to switch from multilayer models resulting from type inversion (Smooth or Focused) to low-layer models, which are more understandable from a geological point of view. At the beginning of interpretation, it is convenient to use a layered model consisting of 14-20.

The results of such inversion (*Smooth, Focused*), most often should be approached as a starting approximation for further, meaningful interpretation. They give an understanding of the approximate geoelectric structure of the section. Then proceed to the low-layer model, using the **Layer summarization** dialog**.** The new model is displayed on top of the old black line.

The input field specifies the contrast parameter, at which point the two layers are merged into one.

Then fix some parameters and make a manual selection or inversion.



Pic. 26 **An example of the inversion results after applying the Layers summarization function. Where A is an example of multilayer model, B is an example of Layers summarization dialog (red line - multilayer model, black line - merged model), C is an example of inversion results after applying Layers summarization function.**

Analysis of equivalence. The theorem of uniqueness of the inverse problem in the case of error-free continuous measurements is proved in electrical exploration. In practice, measurements are made in a finite interval with a certain discretization, moreover, they contain errors. The presence of errors and incompleteness of data transforms the theoretical singularity of the solution into the practical singularity of the solution, that is, the equivalence of different solutions of the inverse problem. Two geoelectric transects are called equivalent if the relative discrepancy in the data for these transects does not exceed the accuracy of field measurements or the fitting discrepancy. Practically, the equivalence principle means that some parameters of a transect cannot be determined during interpretation if some other parameters of the transect are unknown. The equivalence principle makes it very difficult to interpret the data. The solution to this problem is to fix some parameters (based on a priori information).

The analysis of the equivalence principle action is based on two approaches - information-statistical for all model parameters by constructing a correlation matrix of relationships and direct calculation of equivalence areas for a pair of section parameters with their visualization.

Another variant of equivalence analysis is to construct a cloud of equivalent models, that is, a family of model curves giving very close theoretical curves.

Usually, a statistical evaluation of the equivalence of all parameters is carried out first, and then the study of individual pairs of parameters with high correlation coefficients.

The correlation matrix is called by the option: **Correlation matrix** (Fig. 27). On the main diagonal of the correlation matrix are ones (two parameters with the same value, e.g. ρ1 and ρ1 always have a correlation coefficient equal to 1, and this parameter does not need to be analyzed). If the correlation coefficient is much less than unity modulo, the parameters of the section, for which it is calculated, affect the data differently and are determined with a small error. Thus, it becomes possible to determine them separately.

Parameters for which the correlation coefficient modulo is close to 1 are jointly indeterminate. To increase the accuracy of the solution in this case, one of the equivalent parameters should be fixed, if it is possible to obtain independent information about it. In the case of strongly correlated parameters of neighboring layers, either fix one of the correlated parameters, or combine these two layers into one, i.e. simplify the model.

Correlation matrix

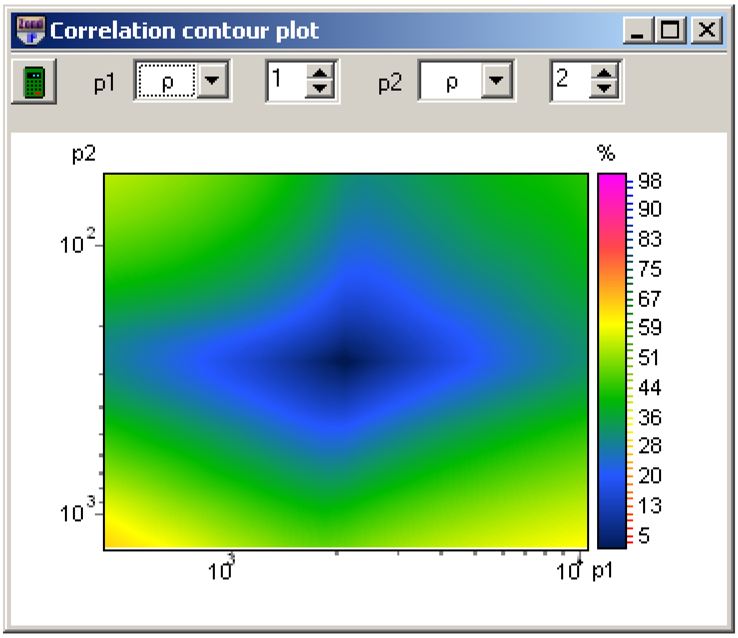
	ρ1	ρ2	ρ3	ρ4	ρ5	h1	h2	h3	h4
ρ1	1.00	0.15	-0.014	0.0039	-0.000	-0.61	0.14	0.0087	0.0024
ρ2	0.15	1.00	-0.063	0.011	-0.002	-0.63	0.94	0.031	0.0066
ρ3	-0.014	-0.063	1.00	-0.086	0.010	0.050	0.057	-0.92	-0.060
ρ4	0.0039	0.011	-0.086	1.00	-0.063	-0.012	-0.025	-0.089	0.94
ρ5	-0.000	-0.002	0.010	-0.063	1.00	0.0026	0.0055	0.015	0.077
h1	-0.61	-0.63	0.050	-0.012	0.0026	1.00	-0.57	-0.029	-0.007
h2	0.14	0.94	0.057	-0.025	0.0055	-0.57	1.00	-0.072	-0.016
h3	0.0087	0.031	-0.92	-0.089	0.015	-0.029	-0.072	1.00	-0.075
h4	0.0024	0.0066	-0.060	0.94	0.077	-0.007	-0.016	-0.075	1.00
par	324.26	22.91	351.17	15.82	503.38	1.39	5.50	16.85	104.55
min	308.90	20.28	307.90	13.91	419.39	1.34	4.84	14.76	91.52
max	340.38	25.89	400.52	18.00	604.19	1.44	6.26	19.25	119.43

Pic. 27 **Correlation matrix window.**

The last two lines of the table show the confidence intervals for each of the parameters. The confidence intervals are a criterion for the reliability of parameter determination and are related to the total sensitivity of the parameters of the section. If the confidence interval is wide, the parameter value is considered to be unreliable. When estimating the width of the confidence interval, the absolute values of the resistivity and power of the layers should be taken into account.

Double-clicking a correlation matrix cell brings up **a correlation** plot for the selected parameter pair (fig. 28).

The map of correlation dependence of a pair of parameters represents the plan of isolines of inconsistency between the theoretical data for the current model and the theoretical data for the modified model. Assuming that the current parameter values are the center of the equivalence area, several more solutions of the direct problem for the parameters varying around this point are calculated and the value of the maximum error of the difference between the data and the central one is determined. To construct a correlation map between two parameters, a set of values is assigned to the value of each parameter in some range, the unconformity with the data for the current model is calculated, and an isolinear map is constructed. If the parameter has a logarithmic distribution, all the above actions are performed with the logarithms of the parameters. Isolines of maximum error values are plotted on the equivalence map, showing the configuration of the region and the limits of the equivalence principle. Isometric regions of equivalence indicate no correlation of parameter estimates, strongly elongated regions indicate correlation relationships of parameter estimates. The analysis of the correlation relationship contributes to the successful identification of the equivalence of the two parameters.



Pic. 28 **Correlation contour plot window.**

The color scale establishes the relationship between the value of the inconsistency and a particular color.

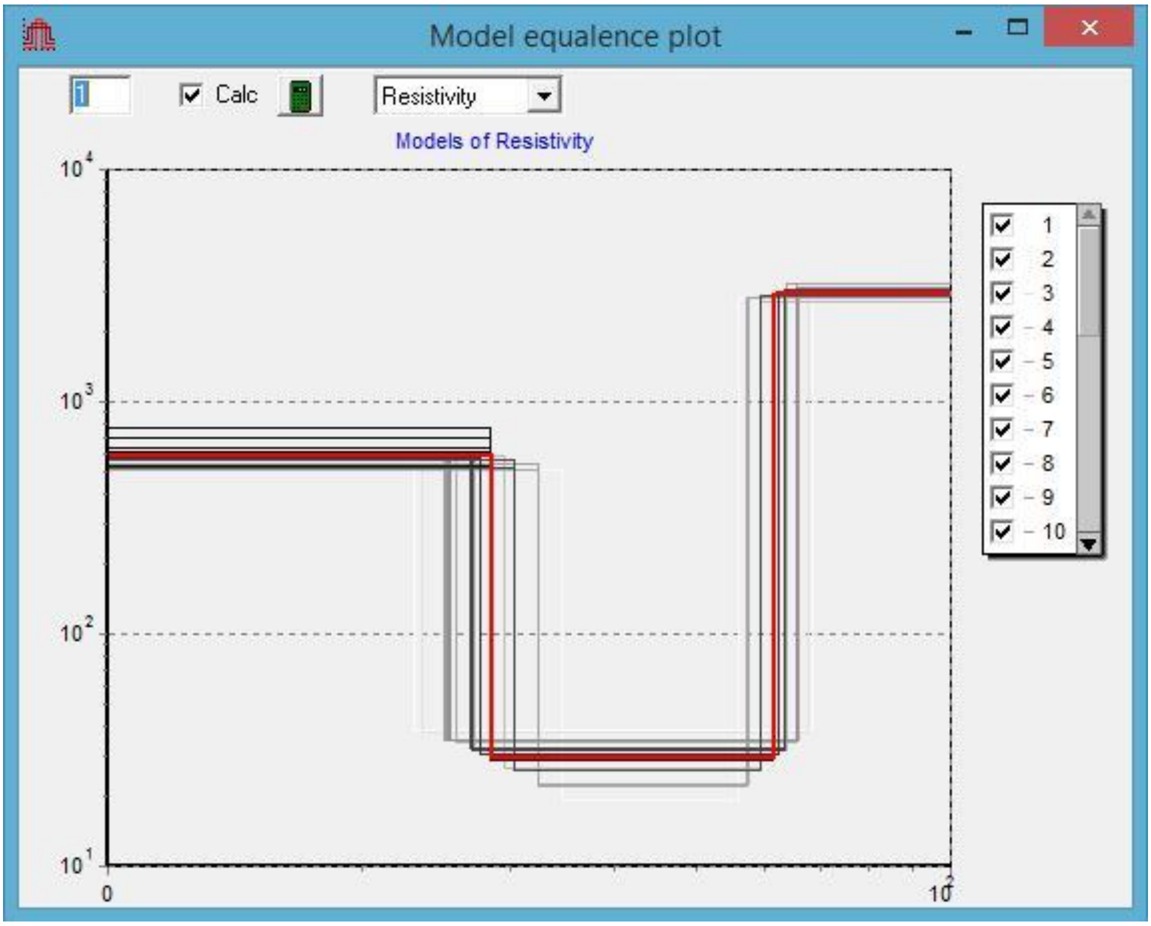
The ***p1*** field sets the type of the first parameter for which the correlation analysis is performed.

Field ***p2*** - sets the type of the second parameter, for which the analysis of the correlation dependence is performed.

The following water fields, set the indices of the layers of the first and second parameters, for which Field ***p1*** - sets the type of the first parameter, for which the analysis of the correlation dependence is performed.

- build a correlation map.

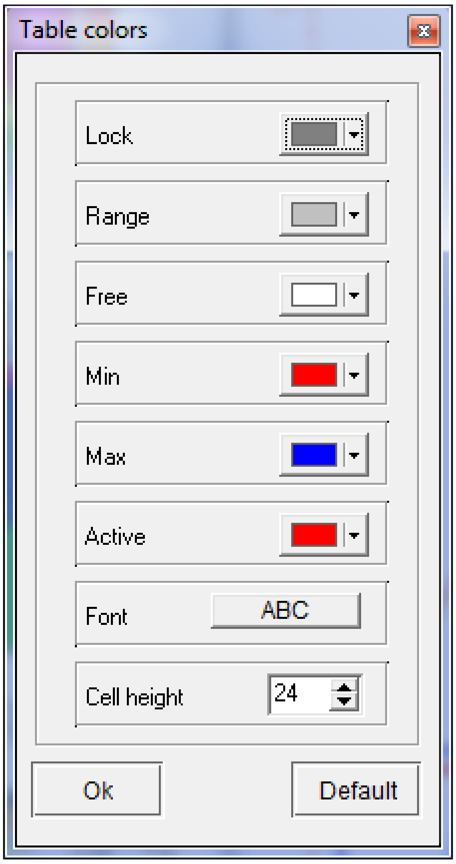
The window for building the cloud of equivalent models is called by the option: **Equivalence plot**. It implements a fairly resource-intensive algorithm for finding equivalent models, within a given error, by brute-force method (Fig. 29).



Pic. 29 **Equalence plot window.**

It is necessary to set the minimum level of error, at reaching which the model will be considered equivalent to the current model (input field). If the ***Calc*** option is not set, the equivalent models will be calculated for the value of model fitting error. Usually this level is chosen slightly higher than the current fitting error. Next, select the parameter for which the calculation will be performed (*Resistivity*). And then run the search procedure. The result of the algorithm is a set of model curves. It is possible to disable one or another curve on the legend on the right side of the image. Use scrolling to move from one graph to another.

The **Options** item opens the dialog box for setting the graphical parameters of the table.



Pic. 30 **The Options dialog box for setting the graphical parameters of the table.**

The ***Lock*** option sets the color of the cell whose parameter is locked.

The ***Range*** option sets the color of the cell for which the limits are set.

The ***Free*** option sets the color of the cell for which no limits are set.

Min option - sets the color of the minimum limit of parameter changes.

Max option - sets the color of the maximum limit of parameter changes.

The ***Active*** option sets the color of the active cell frame.

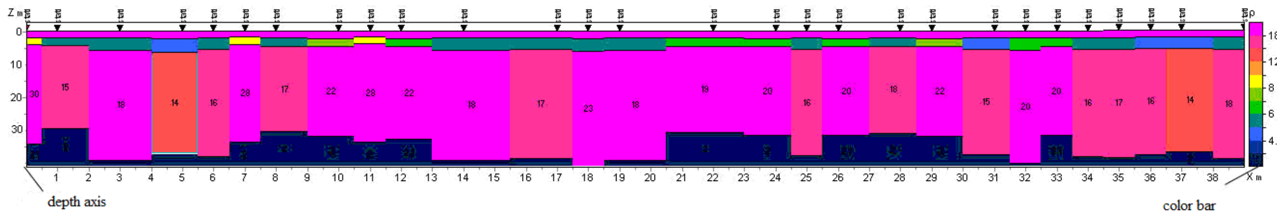
The ***Font*** option sets the font of the cell.

The ***Cell height*** option sets the thickness of the cell.

Section

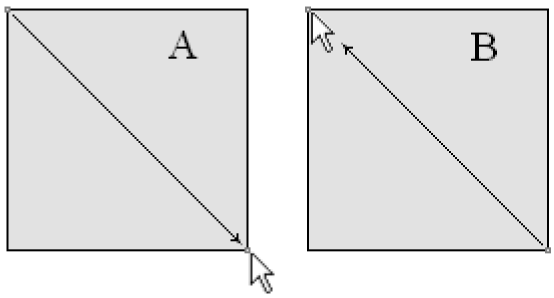
This object is used to display the change in the geoelectric section along the profile. The construction is performed in the axes: coordinate along the profile, depth. The color scale sets the relation between the depicted value and the color (Fig. 31).

When the cursor is located within the geoelectric section, the slider selects the layer over which it is located. When the cursor approaches the boundary between the layers, its shape changes and you can edit its position. To do this, drag the selected boundary to the desired position with the left mouse button pressed. When the right button is pressed, the underlying boundaries will be shifted along with the selected one. Double-clicking on a layer brings up a dialog box for setting its parameters.



Pic. 31 **Geoelectric section window**

Zooming in or moving an individual area is done with the button depressed (the tool is a "rubber rectangle"). To select the area to be zoomed in, the mouse pointer moves down and to the right, with the left button pressed(A). To return to the original scale, the same actions are performed, but the mouse moves up and to the left (B).

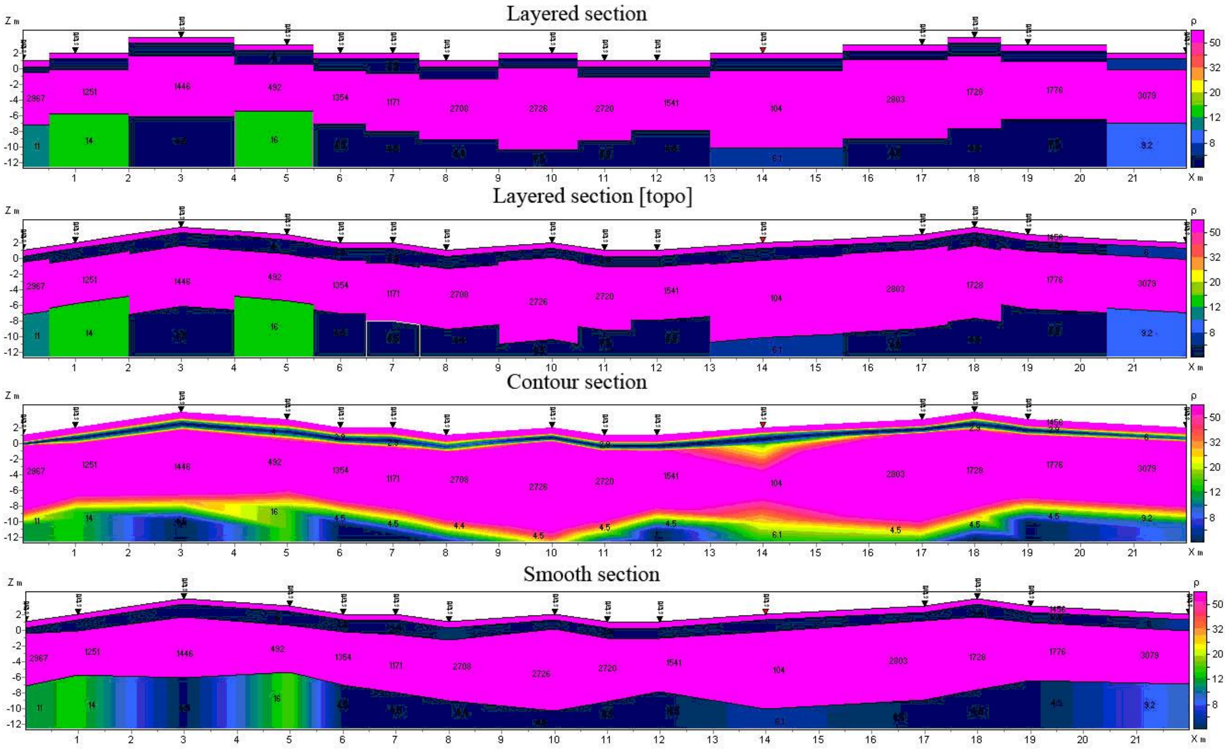


Double-clicking in the object area brings up a context menu with the following options:

Log data scale	Use the logarithmic scale for the color scale.
Display labels	Show labels (parameter values) on layers.
Display ColorBar	Show the color scale.
Setup	Call the Object Setup dialog box.
Print preview	Print out the cut.
Refresh section	Redraw the cut.
Save picture	Save the section in a graphical file.
Output settings	Call the dialog box for setting the scale of exported graphics.
Layered section	Display the section as layers.
Layered section [topo]	Display the section in the form of layers, taking into account the topography data.
Contour section	Display the section as isolines.
Smooth section	Display the section as a smoothed section.
Add background	Add a backing image.
Remove background	Remove the substrate image.

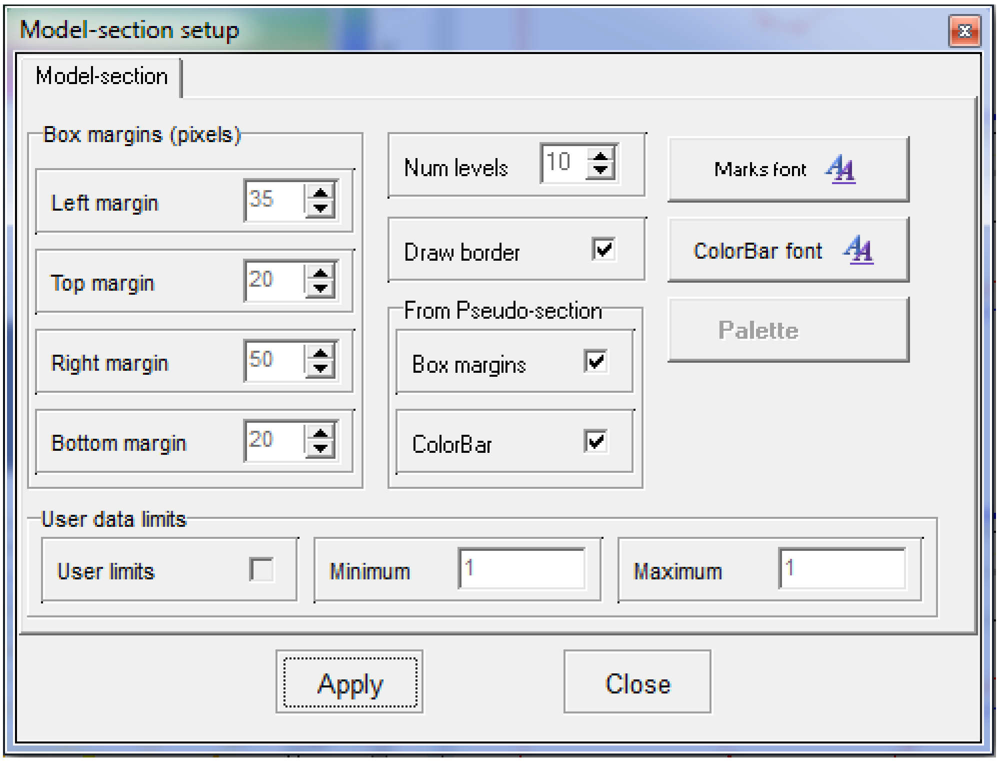
Set bottom	Set the maximum value of the vertical axis manually.
Model interpolation	Interpolation of all models between two given sounding points.
Bad data interpolation	Model interpolation (with a fitting error above a given level) between two given sounding points.
Column percent	Set the width of the model columns.
Resistivity display	Show section of resistivity
Polarizability display	Show polarizability section

Fig. 32 shows four options for representing the geoelectric section.



Pic. 32 **Variants of the geoelectric section representation.**

The ***Setup*** option brings up a dialog box for setting the cut parameters (fig. 33).



Pic. 33 **Geoelectric section parameter settings window.**

Box margins area

The ***Left margin*** field sets the margin (in pixels) of the image from the left edge of the window.

Right margin field - sets the margin (in pixels) of the image from the right edge of the window.

The ***Top margin*** field sets the margin (in pixels) of the image from the top edge of the window.

Bottom margin field - sets the margin (in pixels) of the image from the bottom edge of the window.

The ***Num levels*** field defines the number of colors. Sections are set in uniform linear or logarithmic steps, depending on the type of data.

The *Draw border* function tells the program whether or not to draw a border between layers of different colors.

The **[Palette]** button calls the color settings dialog for the cut layers (more details).

The **[ColorBar font]** button calls the color bar font settings dialog box.

The **[Marks font]** button calls the dialog box to set the font for the layer labels.

User data limits area

The ***User limits*** option tells the program whether to use the minimum and maximum data values or to use the [Minimum] and [Maximum] values when setting the color scale.

The ***Minimum*** field sets the minimum value when setting the color scale.

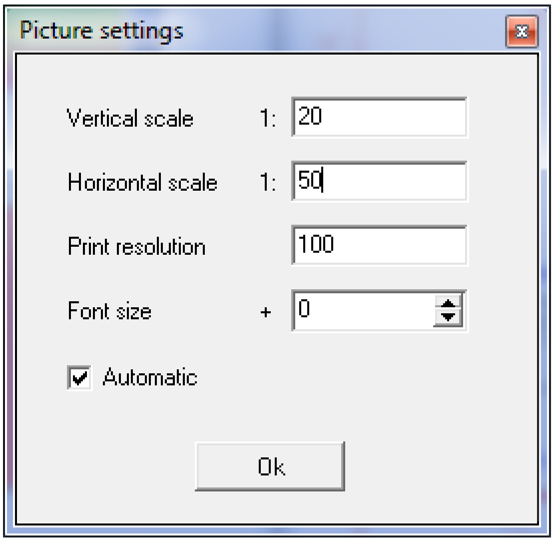
The ***Maximum*** field sets the maximum value when setting the color scale.

Area **From Pseudo-section**.

Box margins option - tells the program to use the values of the Box margins area fields that correspond to the pseudo-cut.

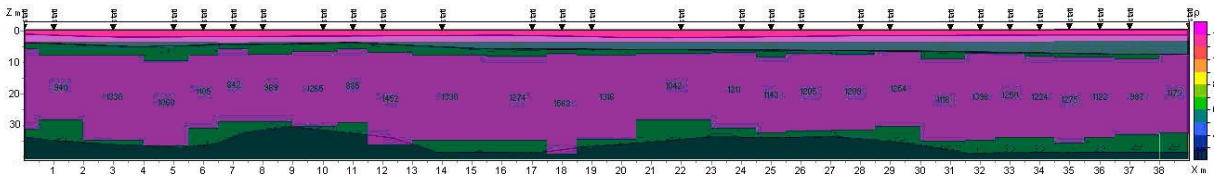
The ***ColorBar*** option tells the program to use the color bar corresponding to the pseudo-section.

The **Output settings** dialog, with the ***Automatic*** option off, allows you to adjust *the Vertical* scale, *Horizontal* scale, *Print* resolution in dpi and *Font size* (fig. 34).

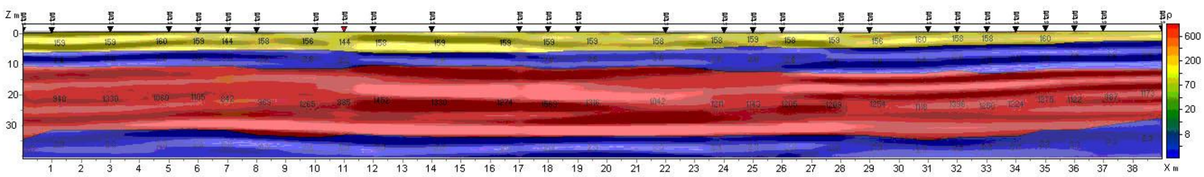


Pic. 34 **The window for setting the parameters of the exported image**

The ***Add background*** option is intended for inserting a substrate model into the area. This function is useful if you have a priori information (geological section along the profile), data from other methods or for comparing the results of the inversion at different stages. The underlying file should be in bmp format. After choosing this option, a window will appear in which you should set the position of the exported file. The substrate will be reflected from above on the model, and the substrate layer is transparent (Fig. 35, 36).



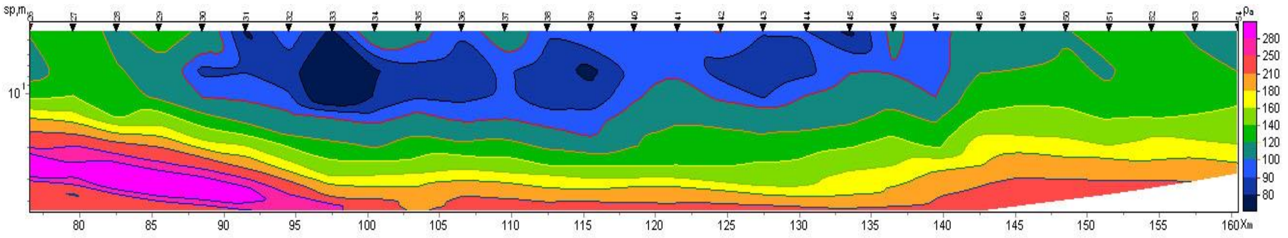
Pic. 35 **Example of a substrate of a geological section.**



Pic. 36 **Example of a seismic section substrate.**

Pseudosection

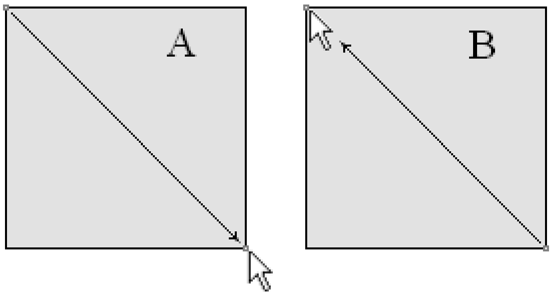
This object is used to display the changes in the observed values along the profile, in the form of isolines (Fig. 37).



Pic. 37 **Example of a pseudo-split of apparent resistance.**

Drawing is performed in the axes: coordinate by profile, spacing. The color scale establishes the relationship between the depicted value and the color.

Zooming in or moving an individual area is done with the button depressed (the tool is a "rubber rectangle"). To select the area to be zoomed in, the mouse pointer moves down and to the right, with the left button (A) pressed. To return to the original scale, the same actions are performed, but the mouse moves up and to the left (B).

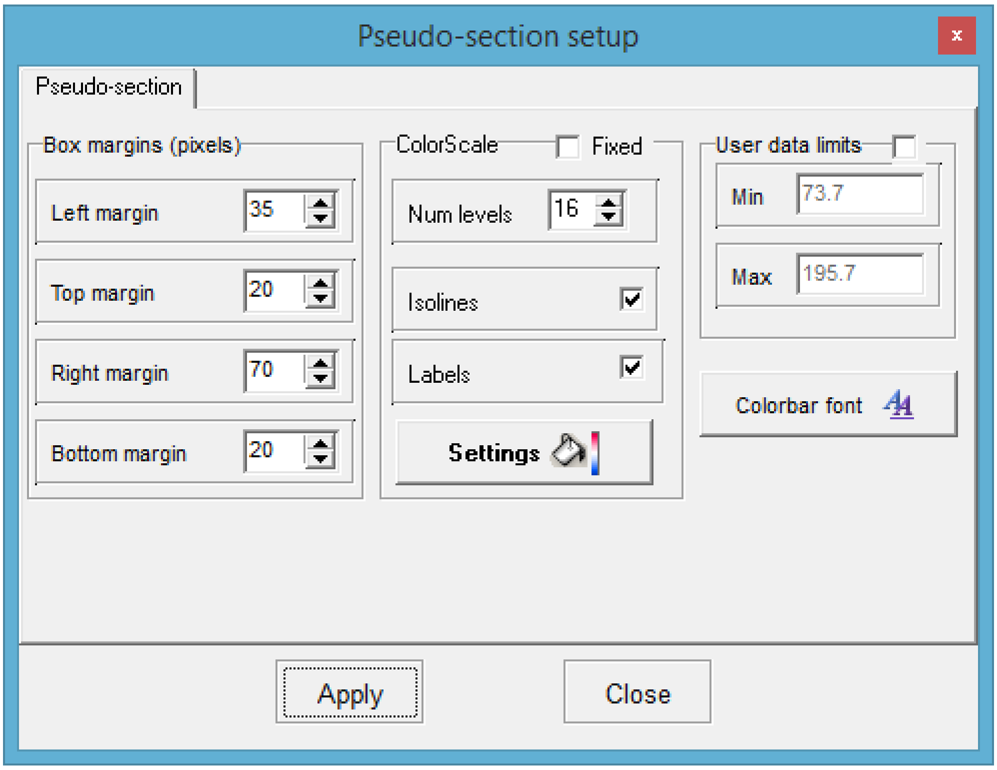


Right-clicking in the axes area brings up a context menu with the following options:

Setup	Call the Object Settings dialog box.
Print preview	Print preview dialog box (more details).
Log data scale	Use the logarithmic scale for the color scale.
Smooth image	Enables continuous gradient fill mode instead of contours.
Display grid	Show measurement point marks.
Display ColorBar	Show the color scale.
Save XYZ file	Save the pseudo-cut in Surfer format.
Save picture	Save the pseudo-cut to a graphical file.
Apparent resistivity display	Display the pseudo-section of the apparent resistance.
Apparent polarizability display	Display the pseudo-cut of the apparent

	polarizability.

The ***Setup*** option is used to call the pseudocut parameters setup window (fig. 38).



Pic. 38 **The window for setting the pseudo-cut parameters.**

Box margins area:

The ***Left margin*** field sets the margin (in pixels) of the image from the left edge of the window.

Right margin field - sets the margin (in pixels) of the image from the right edge of the window.

The ***Top margin*** field sets the margin (in pixels) of the image from the top edge of the window.

Bottom margin field - sets the margin (in pixels) of the image from the bottom edge of the window.

ColorScale area:

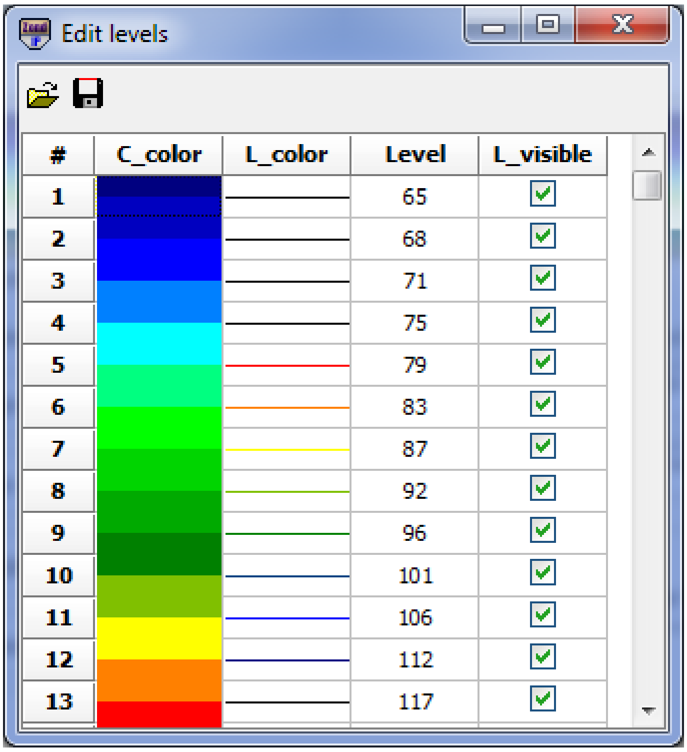
The ***Fixed*** option fixes the current color scale with all the settings.

Num levels field - defines the number of isoline sections. The isoline sections are set in uniform linear or logarithmic steps, depending on the type of data.

Isolines option - tells the program whether or not to draw isolines.

The ***Labels*** option tells the program whether or not to draw captions to isolines.

The [**Settings**] button sets the color palette (fig. 39)



Pic. 39 **Color Palette Settings dialog box.**

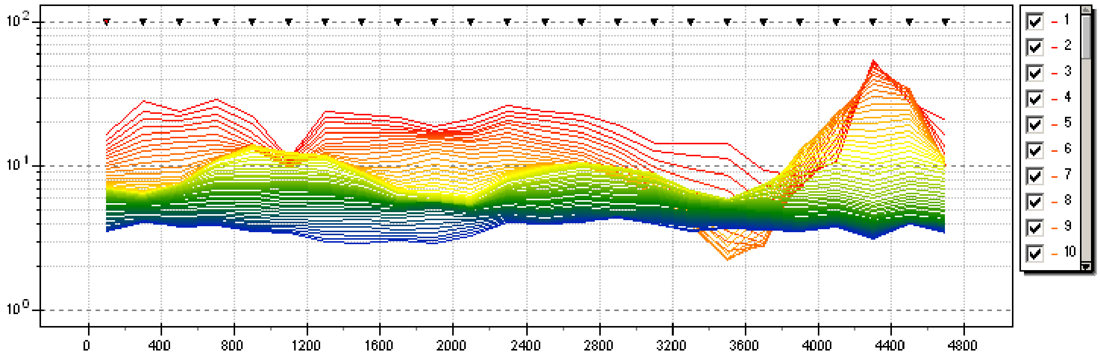
The [**Colorbar font]** button calls the color bar font settings dialog box.

The ***User data limits*** option tells the program whether to use the minimum and maximum data values or to use the values of the Minimum and Maximum fields when specifying the isoline sections.

The ***Minimum*** field - sets the minimum value when defining isoline sections.

The ***Maximum*** field - sets the maximum value when defining the isoline sections.

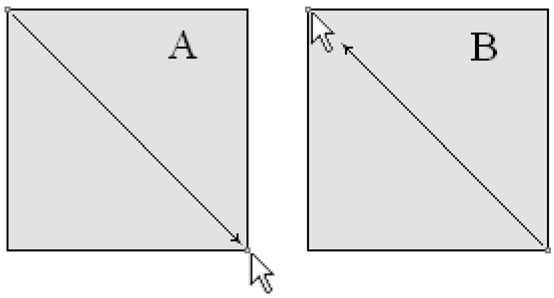
Profiling graphs (Profile)



Pic. 40 **The window of profiling graphs.**

This object is used to display profiling plots (theoretical and experimental) at different spacing. The color of the curve corresponds to the specific spacing. Available for **Profile** and **Interpretation** styles**.**

Zooming in or moving an individual area is done with the button depressed (the tool is a "rubber rectangle"). To select the area to be zoomed in, the mouse pointer moves down and to the right, with the left button pressed(A). To return to the original scale, the same actions are performed, but the mouse moves up and to the left (B).



Moving the mouse with the right button pressed allows you to move the graphs vertically.

Labels on the experimental curves denote the numbers of the spacing for which they are constructed. If you click the left mouse button on the curve, the curves for the other spacing disappear, and appear again after releasing the button. To display only one curve, left-click with the [SHIFT] key on the curve list. Use scrolling to jump to adjacent graphs. Press [SHIFT] + left mouse button on the curve list again to display all graphs.

To increase or decrease the point weights, use the right and left mouse buttons with the [ALT] key pressed.

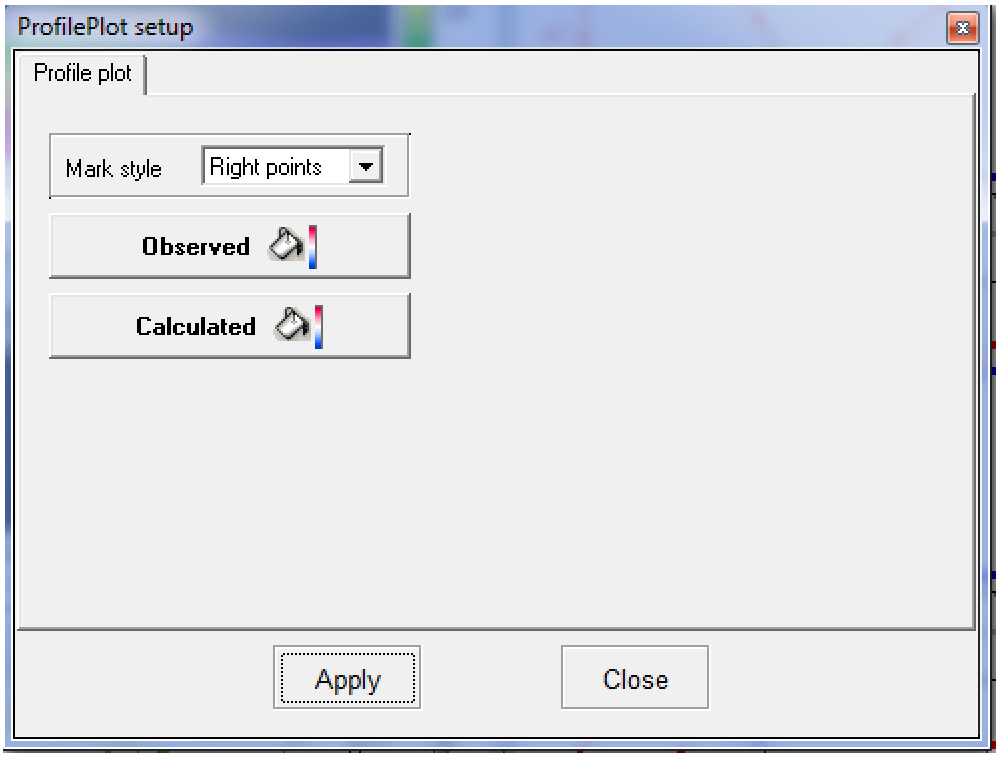
To delete points, use the mouse wheel with the [ALT] key pressed. The size of the deletion area can be adjusted with the wheel.

Right-clicking in the object area brings up a context menu with the following options:

Setup	Call the Object Settings dialog box.
Print preview	Print out a chart plan.
Display calculated	Show theoretical curves.
Delete invisible	Delete hidden graphs.
Apparent resistivity display	Display graphs of apparent resistance.
Apparent polarizability display	Display plots of apparent polarizability.

Right-clicking directly on the graphs brings up a pop-up menu that allows you to set weights to the graph points or delete individual graphs.

The ***Setup*** option is used to adjust the parameters of the graphs (Fig. 41).



Pic. 41 **The window for setting the parameters of the profiling graphs.**

The ***Mark style*** option determines how to draw captions to the charts.

The *Left points* value is on the left side of the graphs.

The value of *All points* is from point to point.

The *Right points* value is to the right of the graphs.

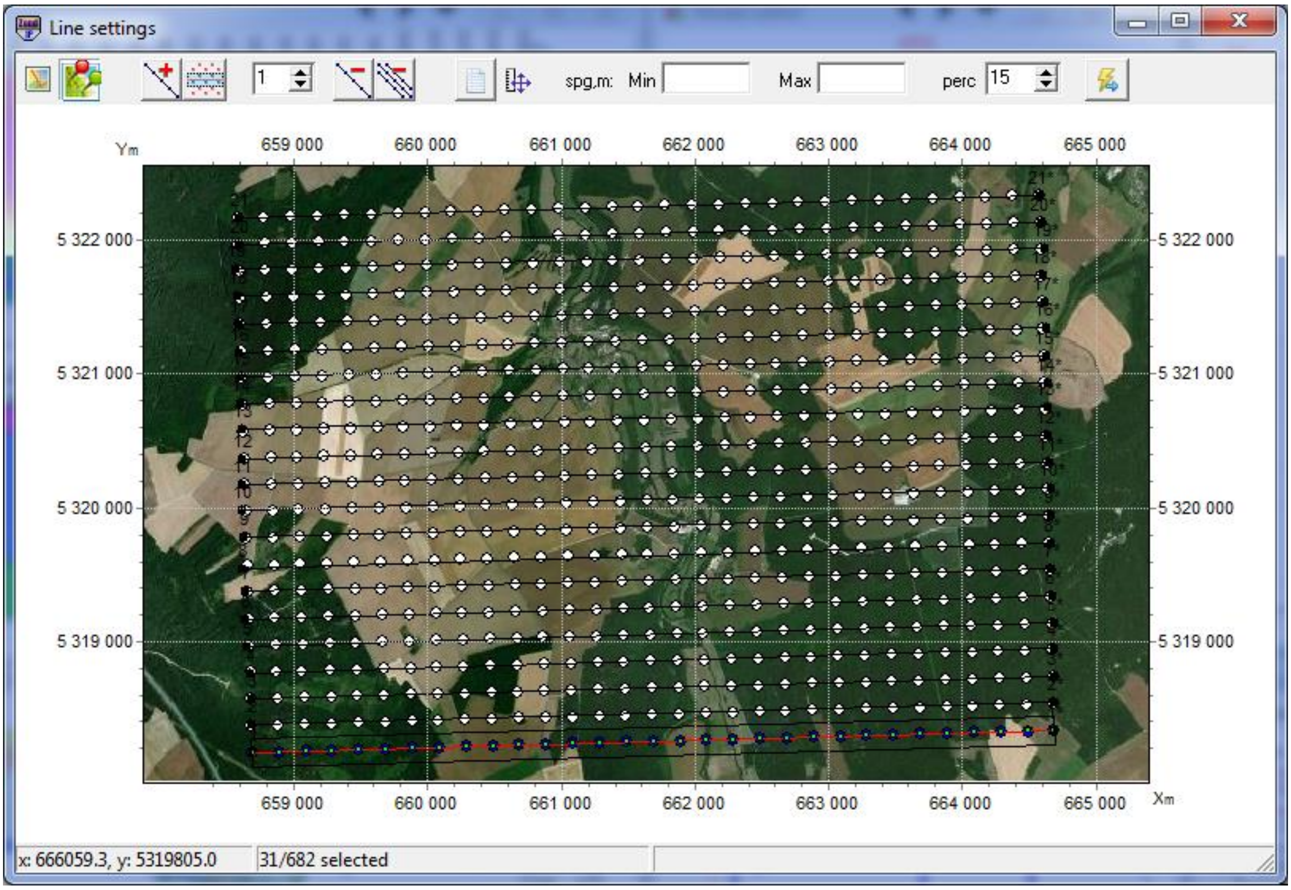
The **[Observed]** and **[Calculated]** buttons bring up a dialog box for setting the graphical parameters (details) for the observed and calculated curves.

Axis parameters can be set in *the axis editor* (right-click+SHIFT on the axis) (details).

Working with area data and 3D visualization

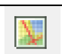
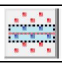
Line settings coordinates option

To set several profile lines, select **Options/Set lines/coordinates** in the main menu of the program. After that, the **Line settings** dialog box will appear (Fig. 42), where the position of the sounding points on the area is displayed.



Pic. 42 **Line settings dialog box for working with area data with topographic isolines.**

The main window panel contains the following functions:

	Load a raster map file (*Load map*) or draw relief isolines (*Draw topography*).
	Loads a site map from the Internet. In this case, the coordinates of the stations must be set in UTM coordinates. In case of loading problems, enter the current key in the field *Bing maps api_key*.
	Add a profile line. The left mouse button sets the profile line points, the right button sets the last point. Right-clicking the mouse brings up an additional menu. By clicking the right mouse button the context menu appears, allowing to set the coordinates of the line, to hold the cursor near the points, to invert the line and to draw lines automatically.
	Include probing points in the profile automatically. Those points that fall within a

	rectangular area around a given line.
1	Set the number of the active profile.
	Delete the current profile.
	Delete all profiles.
	Open and edit the coordinate table (**Stations locations** dialog).
	Select the image scale as equiaxial or with the maximum filling of the window area.
spg,m: Min Max	Set the minimum and maximum spacing respectively. Measurements outside the selected range will not be loaded into the program (when new data is loaded).
perc	Set the size of the area of automatic selection of probing points in the profile.
	Switch to the data inversion mode for the selected profile system.

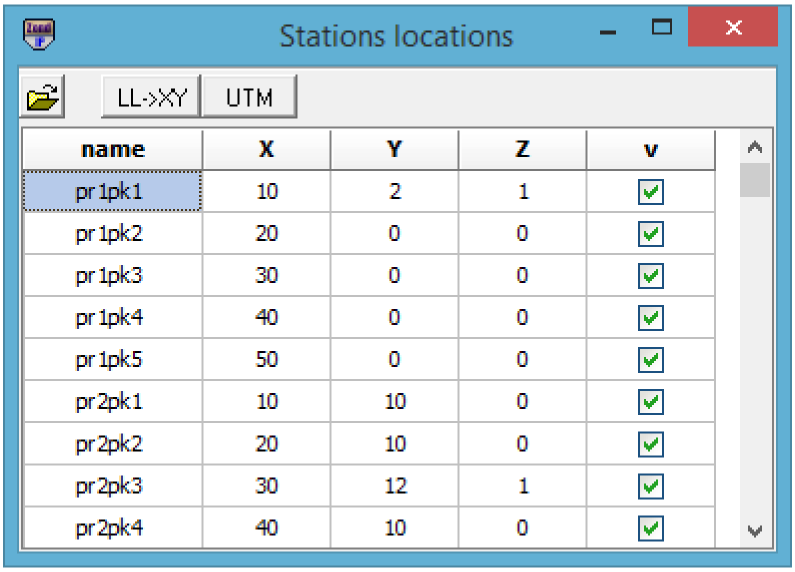
The program allows you to set several lines of profile lines. After setting a profile and selecting points along it, all the points included in the profile will be displayed in blue. It is also possible to exclude/include a point in a profile by clicking the left mouse button. If the profile line does not pass directly through the points, the position of the projection of the probing point on the profile will be displayed in green.

To view and edit the coordinates of points, use the **Stations locations** dialog box (fig. 43), where the user can set the coordinates of points manually, copy from Excel or load a text file with coordinates, using the button .

The buttons on the toolbar allow you to perform the following coordinate transformations:

LL->XY - to recalculate geographic coordinates into rectangular coordinates

UTM - converting geographic coordinates to the UTM system



Pic. 43 **Stations locations window. Setting coordinates.**

The sounding points table contains the following columns: [Name] - source file name, [X], [Y], [Z] - point coordinates and elevation, [v] - include or exclude point.

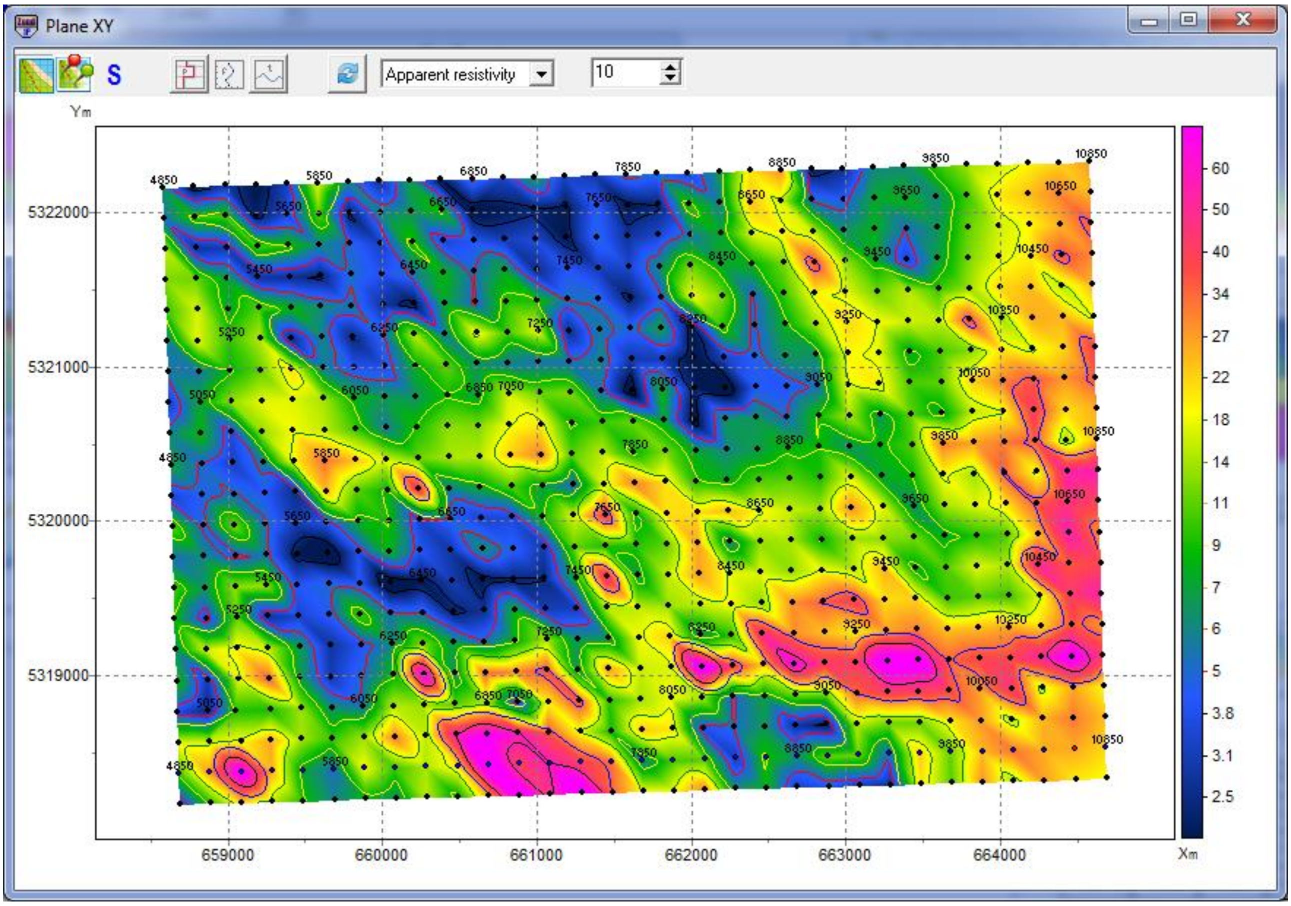
To view and edit the coordinates of the sounding points, right-click on the point of interest. In the window that appears you will see the coordinates that you can edit.

After setting the profile, it is necessary to press the button to switch to the data interpretation mode, after which the main program window will appear. For switching between profiles use the window 

on the toolbar of the main program window.

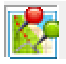
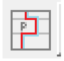
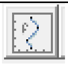
Plane data option

The **Plane data** option **(Options/ Plane data)** allows plotting the distribution of the selected parameter (resistivity, apparent resistivity, heights, etc.) as a function of depth or time over the area. Fig. 44 shows an example of plotting the resistivity distribution over an area.



Pic. 44 **PlaneXY window. Plan of resistivity isolines.**

The window toolbar contains the following buttons:

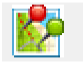
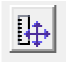
	Load a raster map file as a substrate.
	Loads a site map from the Internet. In this case, the coordinates of the stations must be set in UTM coordinates. In case of loading problems, enter the current key in the field *Bing maps api_key*.
	Export the area pattern of the parameter distribution to Golden Software Surfer and run the application.
	Draw isolines for model parameters (resistivity, conductivity or power).
	Draw isolines for measured parameters (apparent resistivity or polarizability).
	Draw isolines for elevation values.
	Update the current plan.
	The parameter that will be displayed on the area slice. This can be the model parameter for the selected depth or the measured values.
	Sets the depth for which the area model slice will be built. Depending on the mode, the values are taken from the set absolute depth (height) or from the *depth from* surface. If the mode of data building is set, sets the number of the data level to be displayed.
	Slice construction mode: *absolute depth* - select parameter values for a certain absolute depth (height), *depth from surface* - select parameter values for a certain depth from the surface, *layer index* - layer number.

When building the model parameters, the toolbar contains two windows, allowing you to select one of the parameters and set the depth at which it will be displayed. In the right window it is necessary to set the way of depth counting: *Depth from topo* - depth values are counted from the surface, *Absolute depth* - absolute depth values are used, *Layer index* - the plan of isolines is built for the given layer.

When constructing isolines of measured parameters, the number of the layer corresponds to the grid of variances of the initial data.

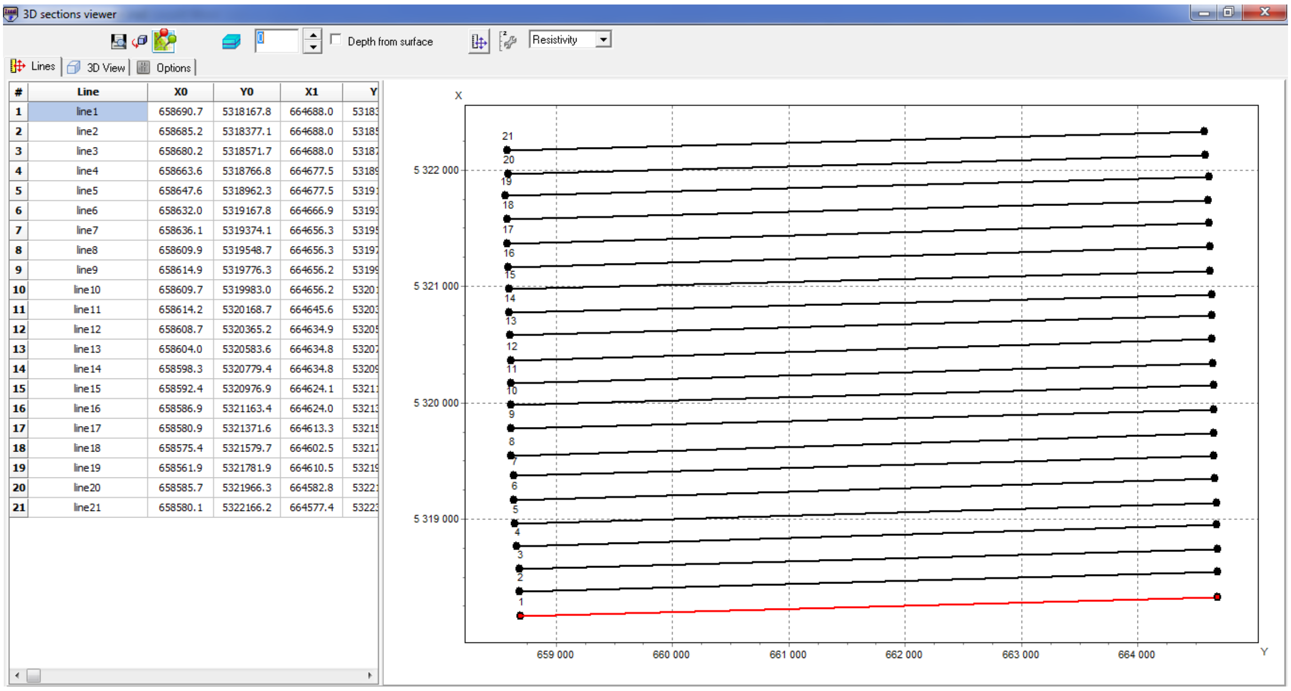
3D fence diagram option (Options/ 3D fence diagram)

This option is intended for three-dimensional visualization of interpretation results by profiles. After selecting this option the **3D sections viewer** window appears. The toolbar of the window contains the following buttons:

	Print preview.
	Rotate the 3D model.
	Loads a site map from the Internet. In this case, the coordinates of the stations must be set in UTM coordinates. In case of loading problems, enter the current key in the field *Bing maps api_key*.
	Build a set of horizontal slices in the program surfer.
	Show horizontal plan. The depth of the plan from the surface is set in kilometers in the window on the right.
0	Sets the depth of cut of the profile set.
☐ Depth from surface	Mode of building a slice. If the option is enabled, the slice is built for a given depth from the surface, otherwise for a given absolute elevation (flat slice).
	Clicking this button sets the same scale for all axes. At the same time, a window appears on the right allowing you to set the ratio of scales for each axis.
	Adjusting the vertical Z-axis.
Resistivity	Indicates which parameter model will be built.

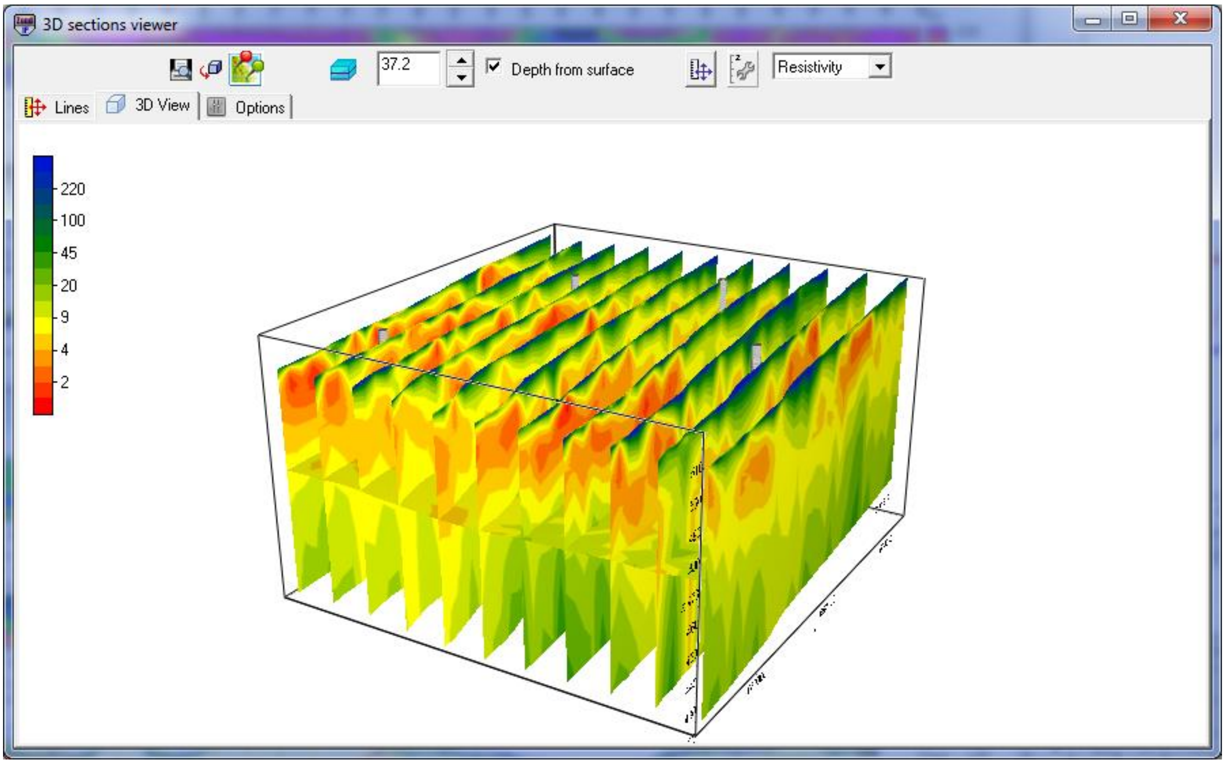
The **3D section viewer** has three tabs:

The **Lines** tab (Fig. 45) is used to edit the coordinates of the beginning and the end of the profiles, as well as to set the profiles that will be displayed in the 3D model. On the left side of the window there is a table with profile names, beginning and end coordinates. To display the profile on the 3D model it is necessary to check the last column of the table ([V] - visible). The plan of profiles is displayed on the right. Active profile is displayed in red color. It is possible to edit properties of axes, the editor is called by the right mouse button with the [SHIFT] key pressed. More information about setting the axes parameters in the Axeseditor section.



Pic. 45 **3D section viewer, Lines tab.**

The **3D view** tab (Fig. 46) is for viewing the 3D model.

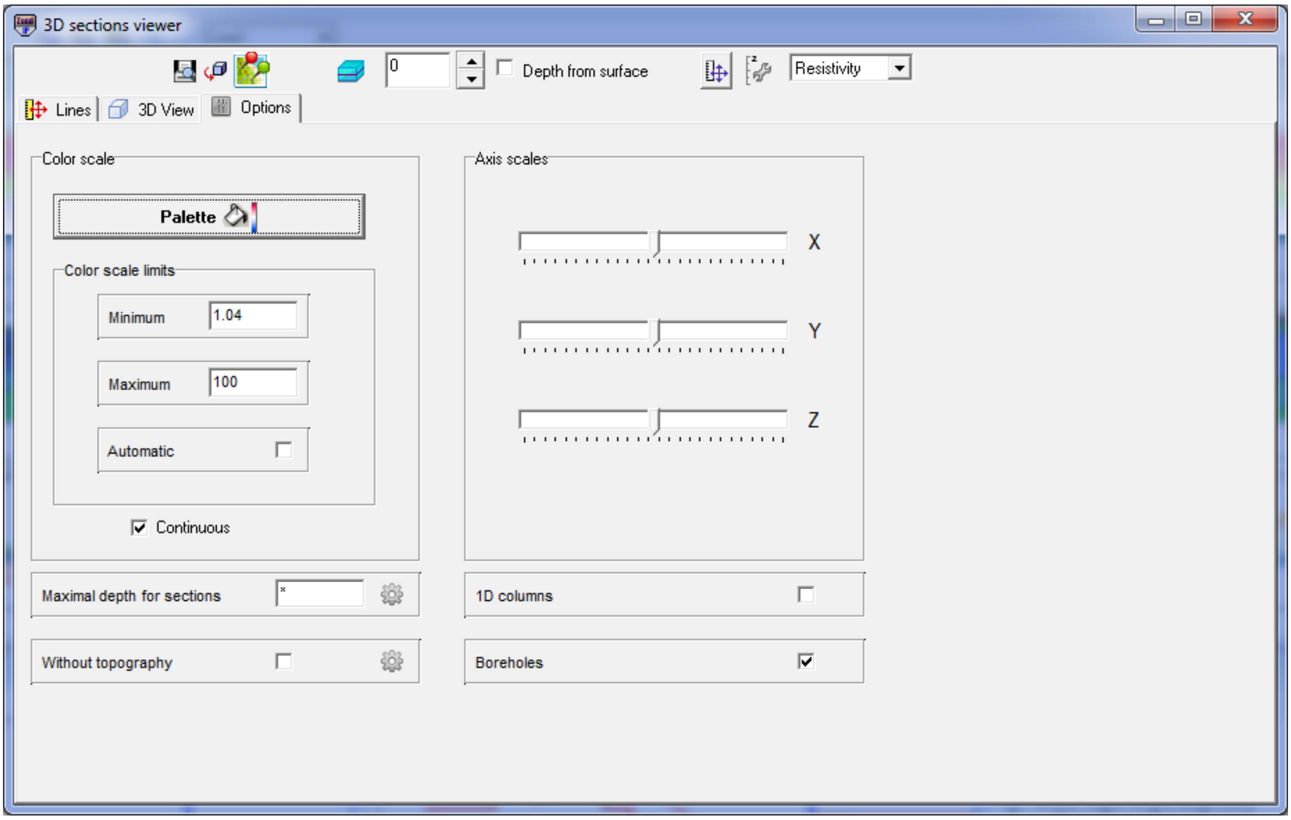


Pic. 46 **3D section viewer window, 3D view tab.**

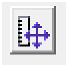
The **Options** tab (Fig. 47) is used for adjusting the image parameters.

The **Color scale** area allows you to adjust the fill parameters. The [**Palette]** button opens the Fill settings dialog box (see details). The **Color scale limits area** allows you to set the minimum and maximum for the color scale manually or select the automatic mode for determining the limits by checking the corresponding checkbox.

Continuous option, if the option is enabled, the cut will be built using a continuous color palette, otherwise by a set of contours.



Pic. 47 **3D section viewer, Options tab.**

The **Axis scales** area is intended for setting the axes scales. The scales are set only when the button on the window toolbar is pressed. 

The ***Maximal depth for sections*** field sets the depth limit below which cuts will be cut.

The option ***Without topography*** allows you to build sections without taking into account the topographic elevations of the soundings. This is useful if you need to build a model with a logarithmic vertical axis.

1D Columns option - allows to show 1D models as pseudo-columns. The colors of the layers are selected according to the color scale.

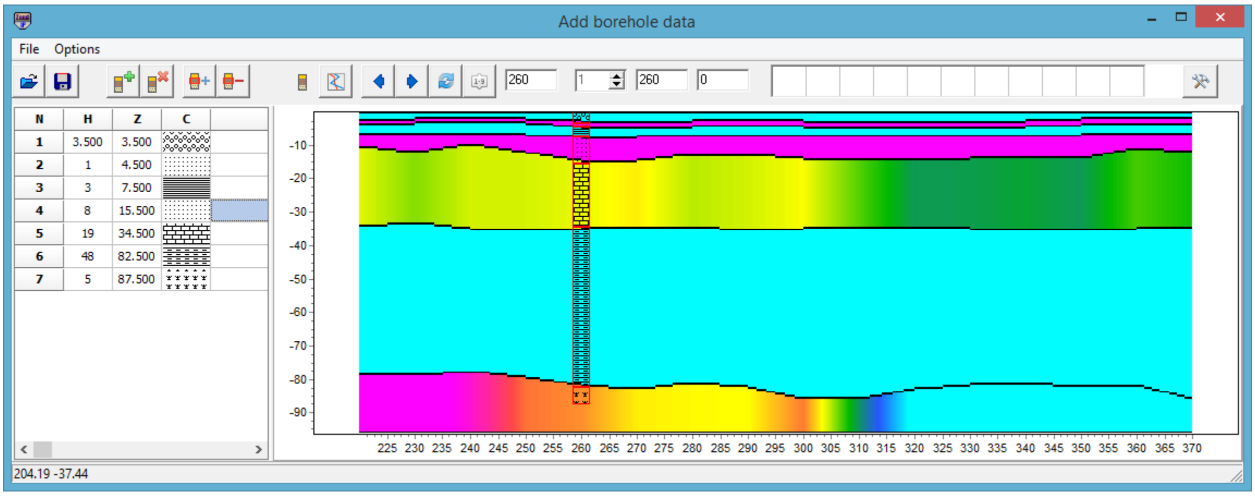
Boreholes option - allows to show well data in three-dimensional image. If there are many wells in the project, it may take considerable time to display them.

Introduction of a priori information

The availability of a priori information (data on wells) allows to increase significantly the reliability of the obtained geoelectric sections. **ZondIP1D** program has a built-in module that allows to display a priori data in graphical form on the sections.

Creating and adding a lithology and log file

To create or edit borehole data, select **Options/Boreholes/Create/edit borehole data** in the main program menu. The **Add borehole data** module dialog box will appear (Fig. 48).



Pic. 48 **The Add borehole data lithology file creation dialog box**

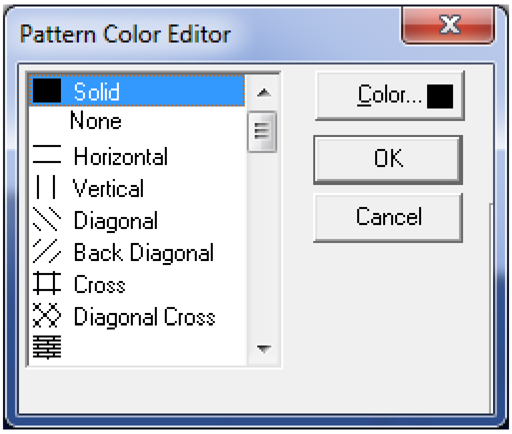
The toolbar of the dialog box contains the following buttons:

	Open the lithology file.
	Save lithology file.
	Create a new well.
	Remove the well.
	Add a layer in the well.
	Remove the layer in the well.
	Sets the type of borehole data: lithologic column. Otherwise log data (graph).
	Load log data.
	Go to the previous well.
	Go to the next well.

	Update the data drawing window.
	Sort the wells by coordinate.
	Sets the horizontal coordinate (along the profile) and the profile number if the project consists of several lines.
	A caption to the well (no more than 5 characters) and its angle of inclination in the XZ plane.
	Optional extras.

The module contains two main windows. On the left side there is ***the Data window***, which contains a table with the following columns: **[N]** - sequence number of the layer, **[H]** - layer thickness in meters, **[Z]** - bottom depth of the layer in meters, **[C]** - fill type. In the right window the data on the wells are displayed in a graphic form.

To create new well data it is necessary to click the button on the toolbar. After that a new table will appear in **the Data window.** Use the button to set the required number of layers. Then it is necessary to edit the table by setting values of thickness or depth of the bottom of each of the layers, and also by choosing the type of filling according to the lithology. The **Pattern Color Editor** fill settings dialog box is invoked by double-clicking the left mouse button in the **[C]** column of the data window (Fig. 49). The program offers a wide choice of lithological fills. In the ***Color*** option you can select the color of the fill.



Pic. 49 **Fill editor window**

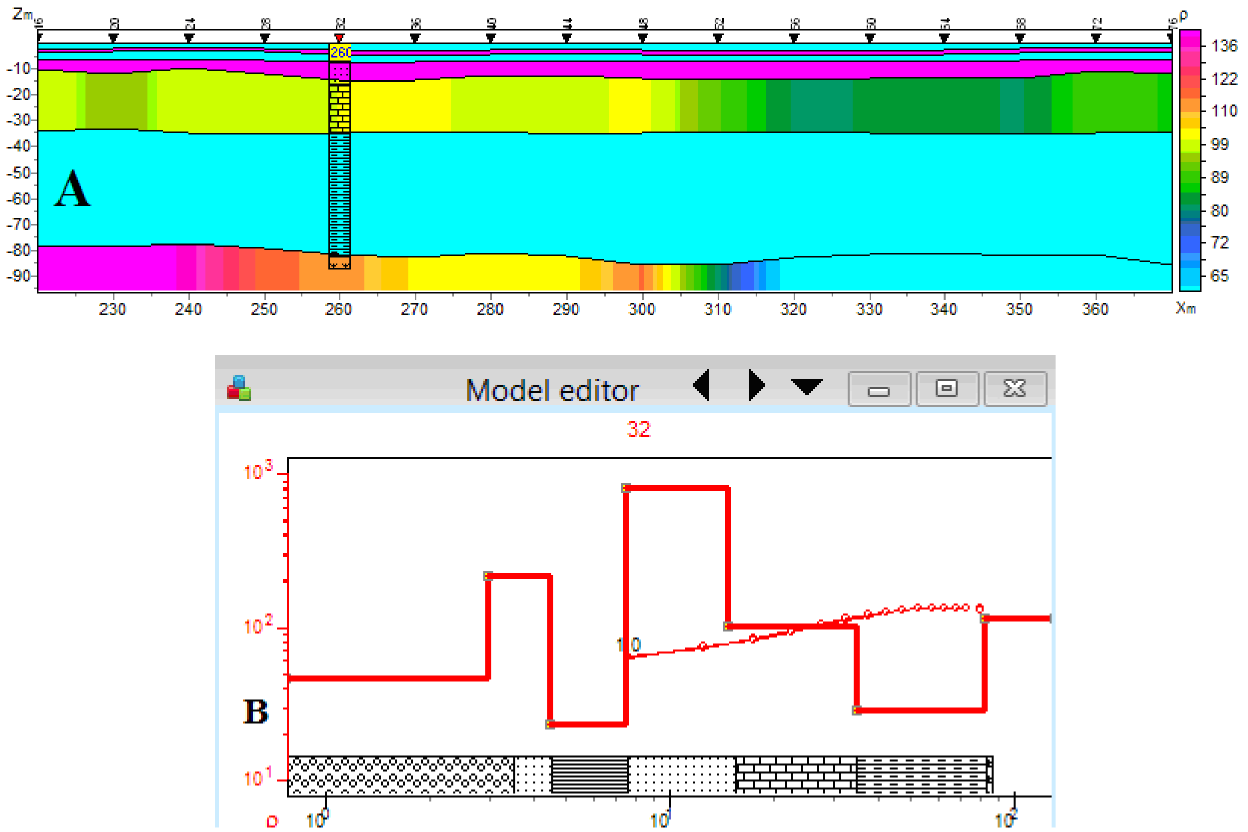
After finishing the well data input, you should press the button and the well will appear in the graphical window. After that, it is necessary to set the horizontal and vertical coordinates of the well on the toolbar in kilometers, and then the well will be represented according to its coordinates. The active well is displayed in red color in the graphical window.

For convenient work with a large number of wells, the program has the possibility to create a palette. To create a palette, select the necessary fill in the Fill column ***of the Data window*, and** then right-click in the Fill area on the main program panel. In this way, you can create a set of fills, which you can then save. To do this, click the button and choose ***Save default palette*.** The saved set of fills can be used when creating a new lithology and logging data file (- ***Load default palette***).

The *Set percent* function available by pressing the button is designed to change the scale of well data display in graphical form.

After saving the data file, several files will be created: ***. crt** - module project, which can be loaded in the program **ZondIP1D** and ***. txt** - files for each well, the names correspond to horizontal and vertical coordinates. For more details about the format of the lithology file, see Appendix1.

To add borehole data from a file, use the **Options/Boreholes/Load borehole data** command**. The** borehole data will be displayed both in the geoelectric section and in the model editor area (Fig. 50).

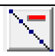


Pic. 50 **Displaying lithology data in the geoelectric section (A) and model editor (B).**

Logging data can also be added. To do this, click the button in the **Add borehole data** window and select the appropriate *. crt. The logging data can be added to an existing lithologic column.

Introducing geological boundaries (Set boundaries dialog)

The **Set boundaries** dialog is available in the **Options/Invert& Apriori/Set boundaries** menu **and** allows taking a priori geological information into account during inversion. After selecting this tab a menu appears containing the following buttons:

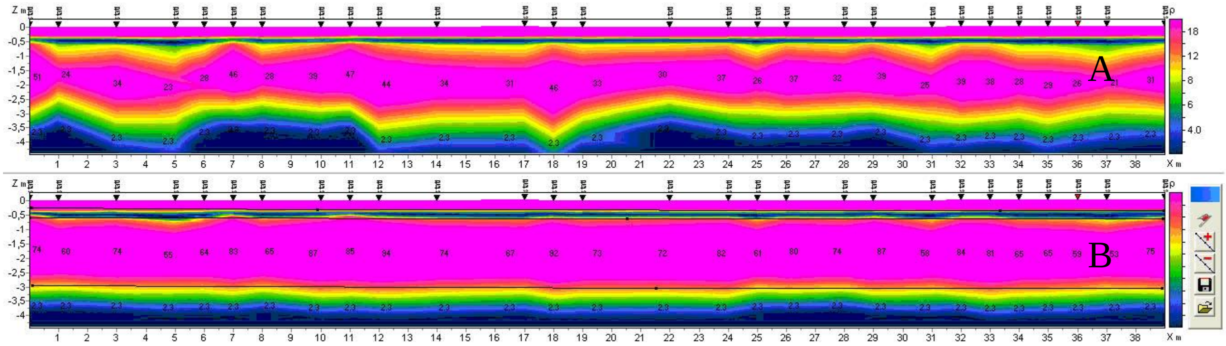
	Enable/Disable editing boundaries mode.	Enable/Disable boundary editing mode.
	Add new boundary.	Add a new border.
	Delete boundary.	Remove all boundaries.
	Save boundaries to file.	Save the boundaries to a file.
	Load boundaries from file.	Load boundaries from a file.

The introduction of a priori geological boundaries into the inverse problem is the most important technique to improve the quality of interpretation. This, on the one hand, increases the stability of the problem, on the other hand, reduces the area of equivalence and allows to obtain a more seasoned structure. In those areas of the model, where the parameters are low-sensitive, this is practically the only way to obtain an acceptable result.

Before setting the boundaries, it is recommended to perform inversion by selecting in the Inversion tab of the program properties window the type of inversion (***Style***) - ***Smooth*** and enabling depth matching (unchecking the ***Thick/depth*** box). For more information about the settings of the inversion parameters, see ProgramProperties window.

The boundaries should be drawn on the obtained geoelectric section taking into account the borehole data, or based on the a priori ideas about the structure of the studied area. The boundaries are added using the left mouse button when the mode is on. To close the boundary, use the right mouse button. It is not necessary to use a lot of nodes when marking the boundaries. It is desirable that the boundaries should be as smooth as possible.

After drawing the boundaries, run the inversion again, which will be performed taking into account the set boundaries (fig. 51).



Pic. 51 **Example of a geoelectric section based on the results of the inversion without (A) and with (B) geological boundaries.**

Options for joint interpretation of VES and MT (AMT, RMT) data

This functionality is intended for joint interpretation of VES data and such modifications of frequency sounding as MT, AMT, RMT, CSAMT, CSRMT. All these modifications differ in the range of frequencies studied and, accordingly, in the depth. Only the abbreviation of the deepest of all methods, MT sounding, will be used below.

The functionality is available in the main menu of the **Options/MT Data** program. The following options are available in the menu:

LoadMT data	Download MT data in TXT format (file format is described below).
Load MDF data	Load data file in MDF format created by **ZondMT1D** or **ZondMT2D.**
Remove MT data	Delete MT probing data.
Joint inversion	Joint inversion of VES and MT sounding data.
DisplayMT plot	Show window of MT curve graphs.

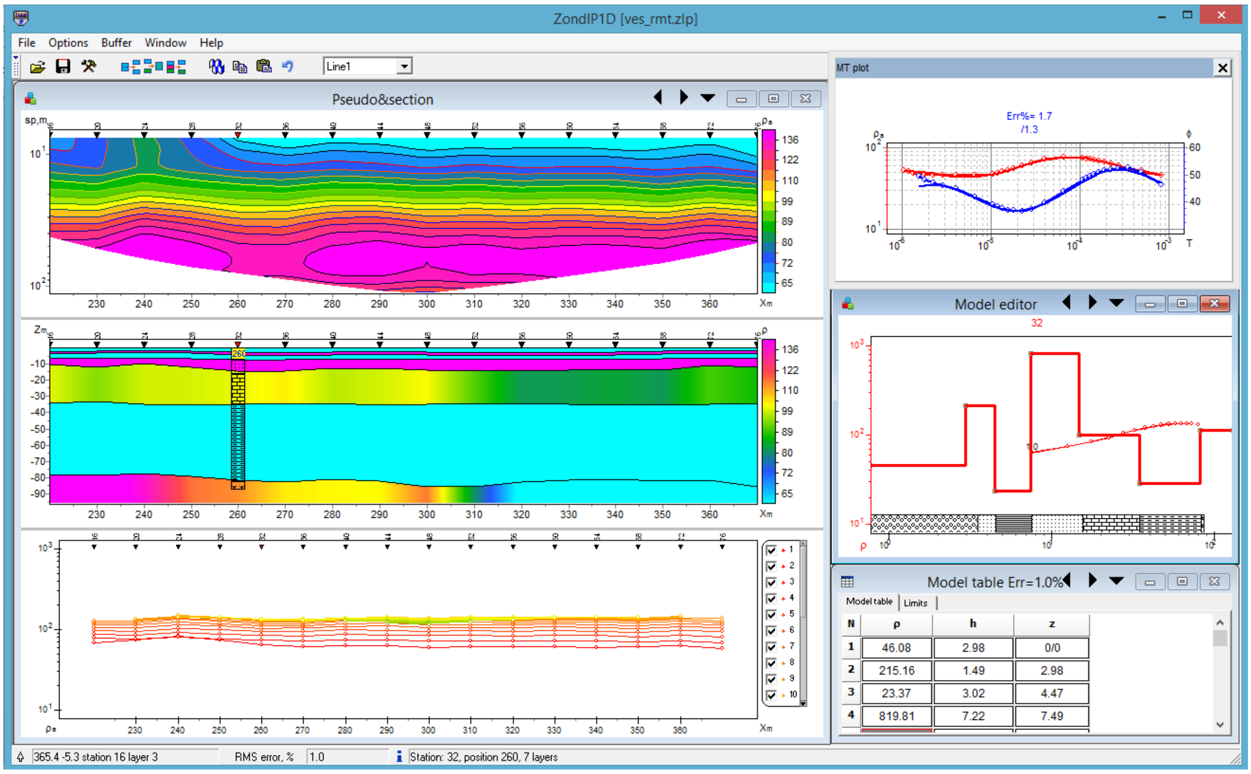
The text file of MT data must contain the following values written in strings:

per - set of periods

app. res - the corresponding values of apparent resistances

pha - the corresponding values of impedance phases.

The MT Curves MT Plot window displays plots of apparent impedance and impedance phase (Fig. 52). The curves with circles show the measured parameters, the solid lines show the parameters selected as a result of joint inversion. It is possible to edit the properties of the axes, the editor is called by the right mouse button with the [SHIFT] key pressed. More information about setting the axes parameters in the Axeseditor section.



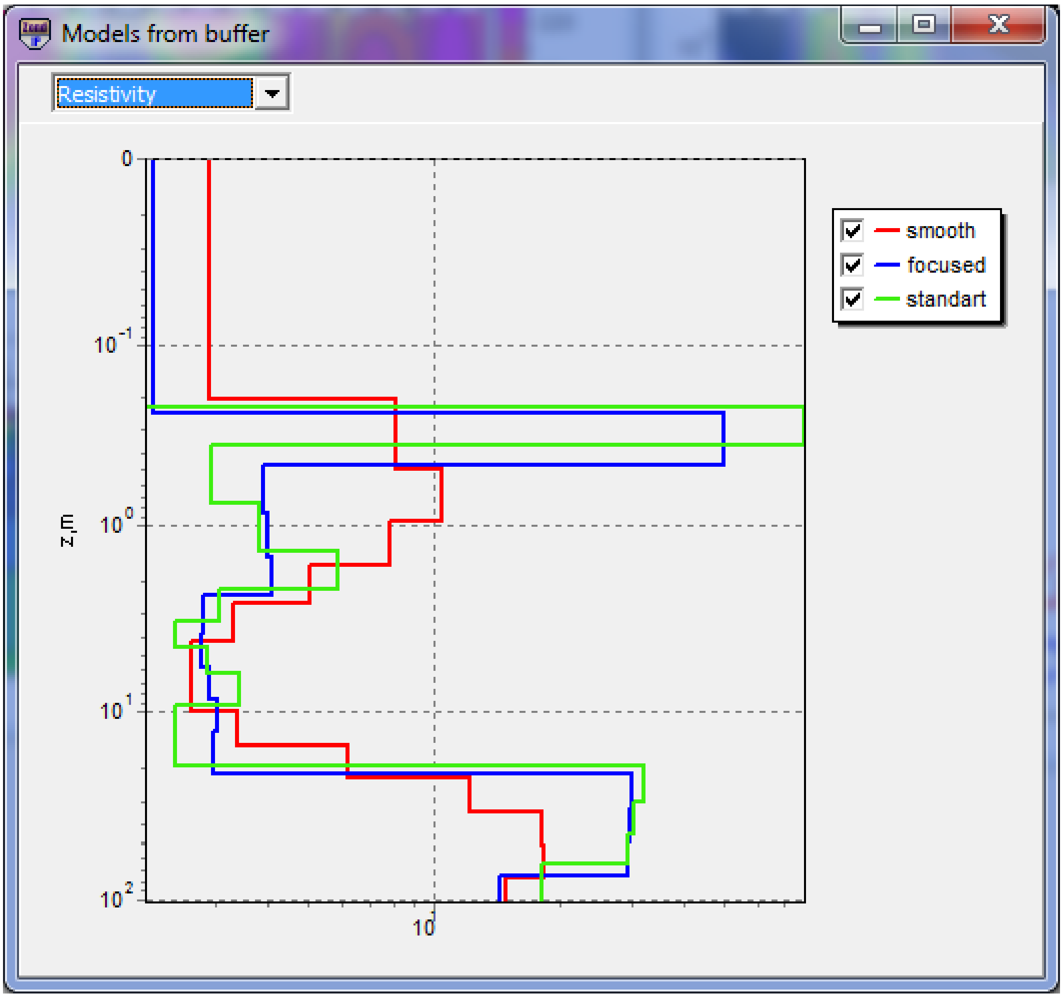
Pic. 52 **Display of MT curve plots. Results of joint inversion of VES and CSRMT data**

Buffer menu for comparing inversion results

The **Buffer** menu of the main program window allows you to compare the results of inversion of the data obtained using different parameters. After calculating the first model, go to the **Buffer** menu and select **Model 1**. In the dialog box that appears, you can specify a model name that reflects, for example, the parameters used in the inversion. Thus, from 1 to 5 models can be saved.

After saving the model, a checkmark will appear next to the name. If you click on a previously saved model, a dialog box appears that allows you to either load the selected model as the current model (**From Buffer** button) or save the current model as the selected model (**To Buffer** button).

The **Open** option in the **Buffer** menu opens all the saved models for the current sounding point in one window (Fig. 53).

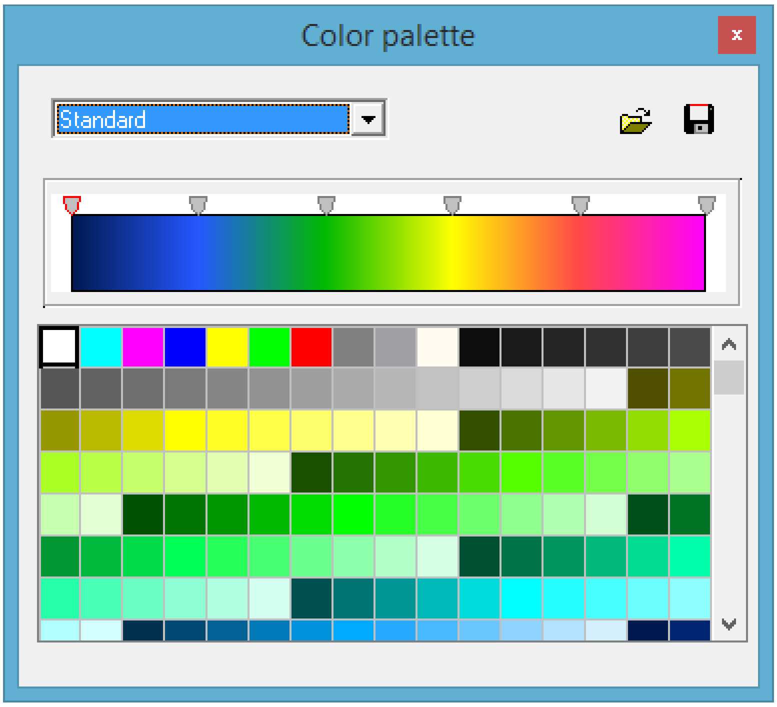


Pic. 53 **Two models for one sounding point opened in one window.**

Configuring graphical objects

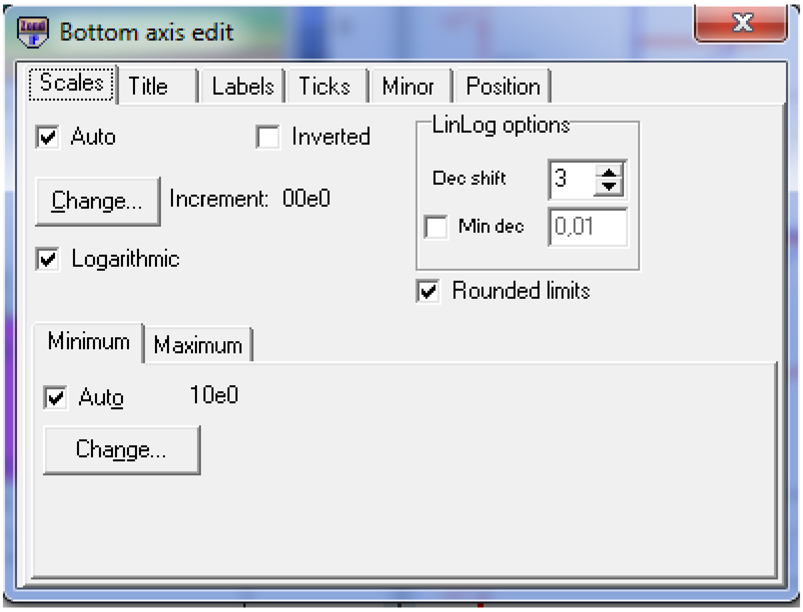
Palette setup dialog

The dialog is designed to configure the palette of the program object and is invoked by the **[Palette]** button (Figure 54). The dialog allows you to choose one of the default palettes (direct and reverse rainbow, grayscale, etc.) or create a custom scale. To add a slider to the scale, use the right mouse button with the [CTRL] key pressed. To delete a slider, use the [DELETE] key. You can also save a custom palette using the , or load an existing one using the .



Pic. 54 **The palette settings dialog box.**

Axis editor



The **Axis editor** is used to change axes configuration. It can be opened by right-clicking on the axis of interest while holding down the Shift key. A context menu will appear with three items: **Options**, **Default** and **Fix range**. The first option opens the **Axis editor** dialog box, the second sets the axis configuration parameters to default values.

The **Scales** tab of the dialog box contains axis scaling options.

If the ***Auto*** checkbox is checked, the program will determine the minimum and maximum of the axis automatically. Otherwise, the values set by the user in the **Minimum** and **Maximum** sub tabs are used.

The ***Inverted*** checkbox determines the axis direction.

The **Increment Change** button opens the dialog box for specifying the label interval.

The ***Logarithmic*** checkbox sets the axis scale to logarithmic or linear. If negative or zero values have to be represented, you should additionally use the options in the **LinLog options** group box.

The **LinLog options** group box contains options for configuring the linear-logarithmic axis. The linear-logarithmic scale allows to represent negative or zero values on a logarithmic scale.

Dec Shift – sets the offset in decades from the maximum axis limit (by absolute value) to zero. The decade closest to zero will have a linear scale, the rest of the axis will be logarithmic.

Min dec – sets the value of the decade closest to zero when the checkbox is checked.

If the ***Rounded limits*** checkbox is activated, the program will round the axis minimum and maximum values.

The **Minimum** and **Maximum** sub tabs contain a set of options for setting the axis limits.

If the ***Auto*** checkbox is activated, the axis limit is determined automatically. Otherwise, the axis limit can be set by pressing the **Change** button.

The **Title** tab contains axis title configuration options.

The **Style** sub tab contains the following options:

Title – defines the text for the axis title.

Angle – determines the orientation angle of the axis title.

Size – specifies the title offset. If set to 0, the offset is determined automatically.

The ***Visible*** checkbox shows or hides the axis title.

The **Text** sub tab contains the following options:

The **Font** button opens the font settings dialog box.

The **Outline** button opens the text outline settings dialog box.

The **Labels** tab contains axis labels configuration options.

The **Style** sub tab contains the following options:

The ***Visible*** checkbox shows or hides the axis labels.

Offset – specifies the label offset. If set to 0, the offset is determined automatically.

Angle – determines the orientation angle of the axis labels.

Min. Separation % – specifies the minimum separation (in percentage) between labels.

The **Text** sub tab contains the following options:

The **Font** button opens the font settings dialog box.

The **Outline** button opens the text outline settings dialog box.

The **Ticks** tab contains axis tick marks configuration options.

The **Axis** button opens the axis line settings dialog box.

The **Grid** button opens the major grid lines settings dialog box.

The **Ticks** button opens the dialog box for configuring the major external tick marks. The **Len** option specifies their length.

The **Inner** button opens the dialog box for configuring the major internal tick marks. The **Len** option specifies their length.

If the ***At labels only*** checkbox is activated, the tick marks will be plotted only where labels are present.

The **Minor** tab contains minor tick marks configuration options.

The **Ticks** button opens the dialog box for configuring the minor external tick marks.

The **Grid** button opens the minor grid lines settings dialog box.

Length – sets the length of minor tick marks.

Count – sets the number of minor tick marks between the major tick marks.

The **Position** tab contains options for specifying the dimensions and position of the axis.

Position % – sets the axis offset on the graph relative to the default position (as a percentage of the total graph size).

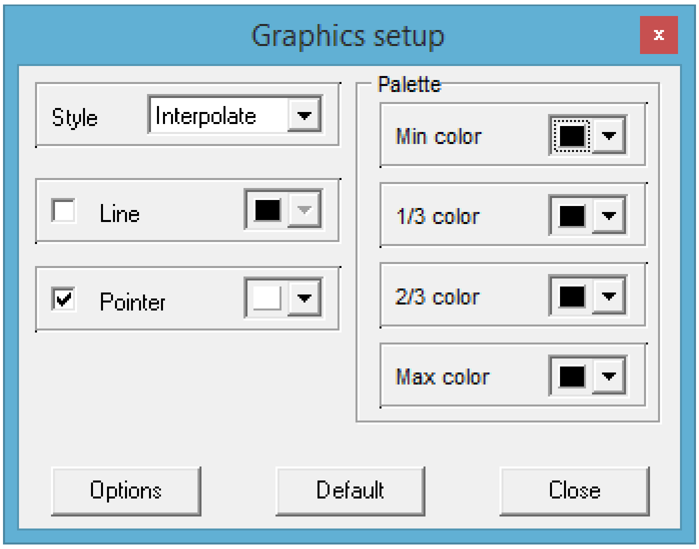
Start % – sets the offset of the start of the axis on the graph relative to the default position (as a percentage of the total graph size).

End % – sets the offset of the end of the axis on the graph relative to the default position (as a percentage of the total graph size).

The ***Other side*** checkbox plots the axis on the opposite side of the graph. For example, if the default axis position is at the bottom, checking this option will position the axis at the top.

Graphics setup

To open the dialog box for changing properties of a graphic plot (a set of graphs).



Style – defines the method which is used to assign colors to a set of graphs.

Interpolate – a color palette is created by interpolating the set of colors specified in the **Palette** group box (*Min color*, *1/3 color*, *2/3 color* and *Max color*).

Constant – the same color is assigned to all graphs.

Random – random colors are assigned to graphs.

Line – specifies the connecting line color. If the checkbox is checked, the color set in the **Line** box is assigned to lines, otherwise, colors from the **Palette** group box are used.

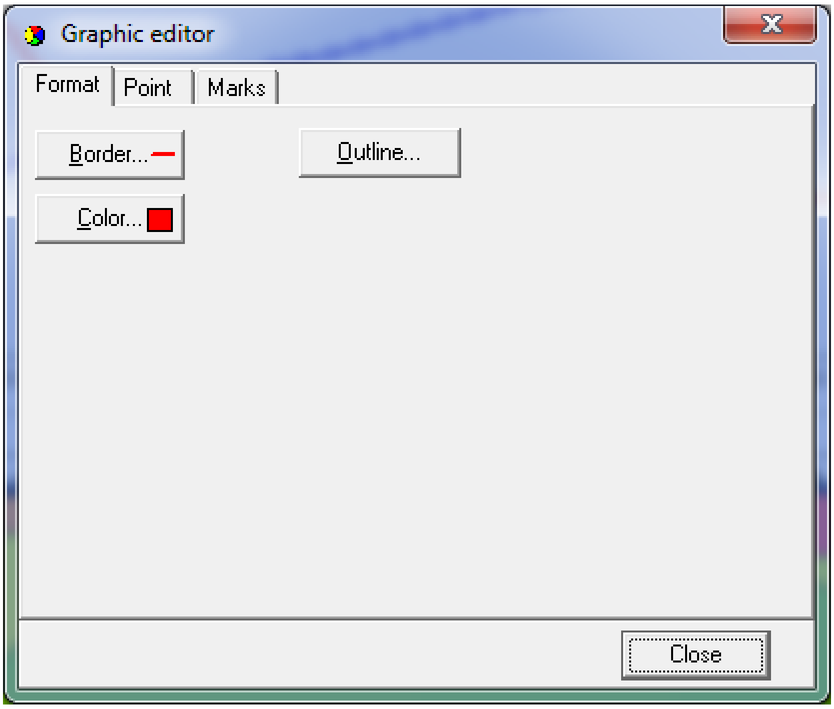
Pointer – specifies the graph point fill color. If the checkbox is checked, the points are filled with the color set in the ***Pointer*** box, otherwise, colors from the **Palette** group box are used.

The **Default** button restores the default dialog box settings.

The **Options** button opens the **Graphic editor** dialog box.

Graph editor

The **Graphic editor** is intended to customize the appearance of graphs. For graphic plots, it is accessible through the **Graphics setup** dialog box (the **Options** button). In the case of individual graphs, it can be opened by right-clicking on the graph while holding down the Shift key.



The **Format** tab contains the settings for the lines connecting graph points.

The **Border** button opens the dialog box for configuring the connecting lines.

The **Color** button opens the line color settings dialog box.

The **Outline** button opens the stroke outline settings dialog box.

The **Point** tab contains settings for the graph points.

The ***Visible*** checkbox shows or hides the graph points.

The ***Error gates*** checkbox shows or hides confidence intervals.

The ***Style*** option sets the shape of the pointer.

Width – sets the point width in pixels.

Height – sets the point height in pixels.

The **Pattern** button opens the point fill settings dialog box.

The **Border** button opens the point outline settings dialog box.

The **Marks** tab contains settings for the graph point labels.

The **Style** sub tab contains the following settings:

The ***Visible*** checkbox shows or hides the point labels.

Draw every – allows plotting every second, third, etc. label depending on the selected value.

Angle – determines the orientation angle of the point labels.

If the ***Clipped*** checkbox is activated, the labels that go outside the graph area are not plotted.

The **Arrows** sub tab contains settings customizing the appearance of arrows that go from the labels to graph points.

The **Border** button opens the arrow line settings dialog box.

Length – specifies the length of the arrows.

Distance – sets the distance between the arrowhead and the graph point.

The **Format** sub tab contains the label graphic settings.

The **Color** button opens the dialog box for specifying the label's background color.

The **Frame** button opens the frame line settings dialog box.

Round Frame – plots the rounded corner frame.

Transparent – sets the transparency of the label's background.

The **Text** sub tab contains the following settings:

The **Font** button opens the dialog box for the label font settings.

The **Outline** button opens the dialog box for the label text outline settings.

Print preview dialog

Use **File/Print preview** function of the program main menu to run print preview dialog. Double click on any program object also runs it. Two options are available when running via main menu:

Option *Station* serves to print sounding curves and current station model. Model parameters are displayed as table which contains number, resistivity value and depth of top layer boundary.

The **Print preview** window is opened with the **File / Print preview** menu command.

You can move objects on the sheet by dragging them with the mouse.

The toolbar of the **Print Preview** window contains the following items:

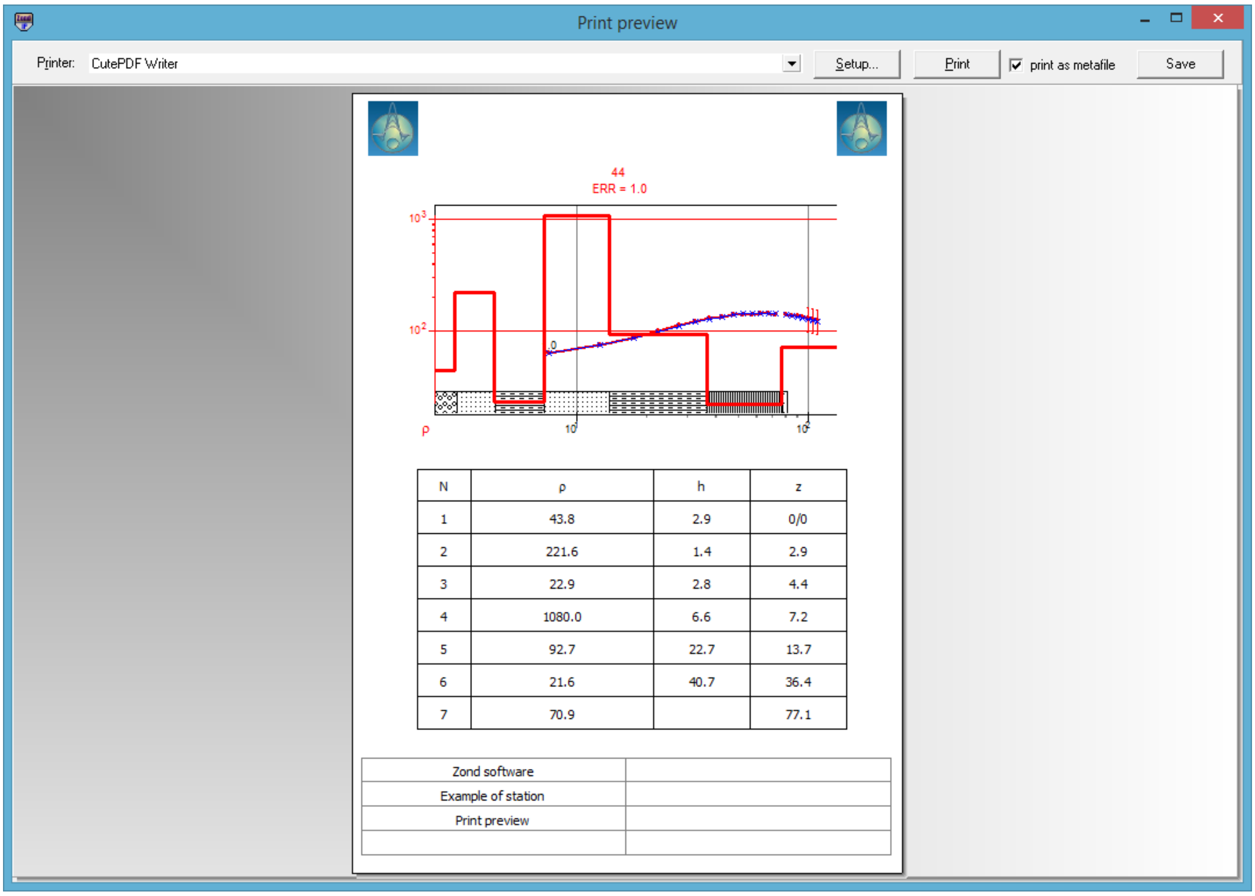
– select a printer.

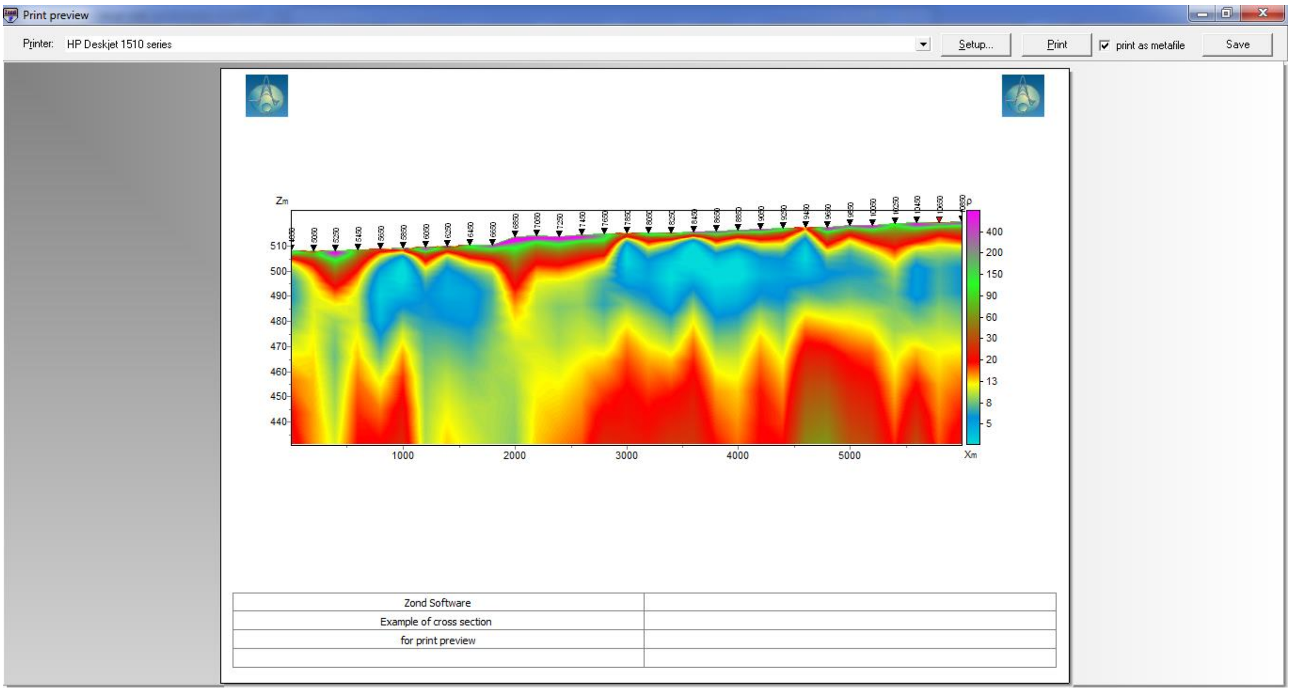
– open a standard print setup dialog box.

– print the drawing.

– if activated, the drawing is saved as a vector image.

– save as a bitmap image.





The square area in the top left corner of the sheet can be used for seals, stamps or company logos. Right-click on the area to upload a BMP image. The area can be resized with the mouse.

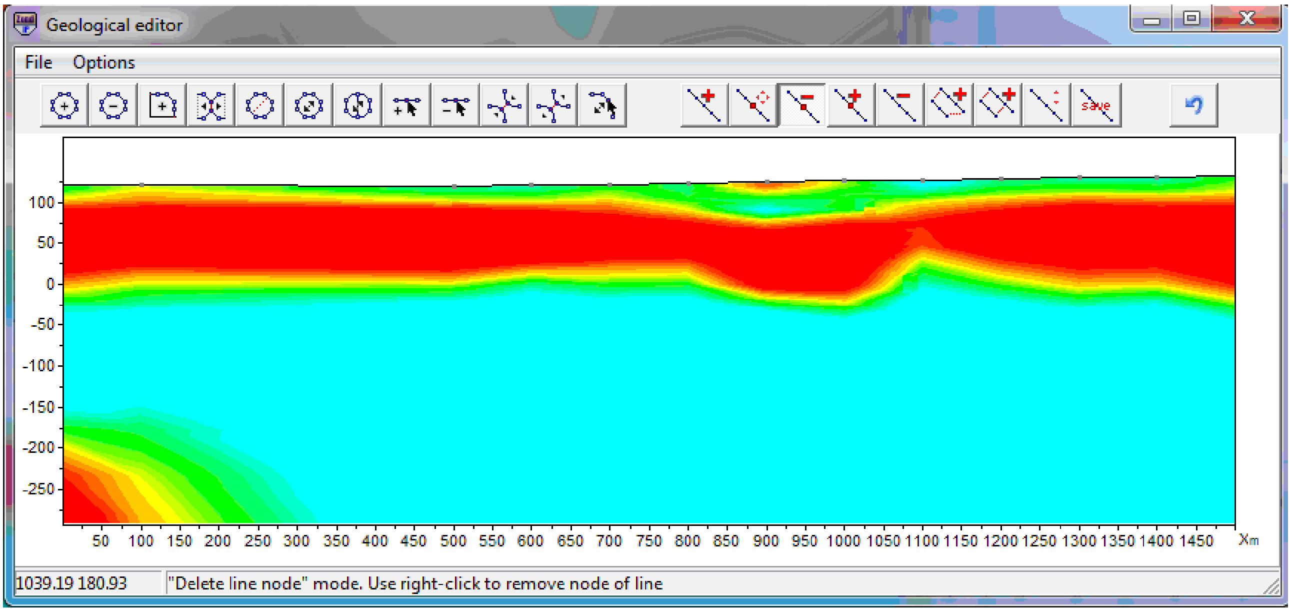
At the bottom of the sheet is an editable table. Right-click on the table for text input. The contents of the table can be saved and loaded using the and buttons.

Geological editor module

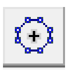
To build geological models (perform geological interpretation), a special **Geological editor** module. The editor allows to interactively create a geological model based on the current project model, borehole data, background images and models obtained in other **Zond** programs, as well as to print the resulting section at a user-defined scale, save and export the interpretation results.

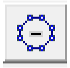
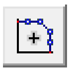
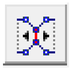
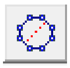
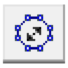
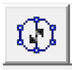
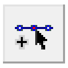
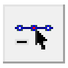
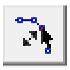
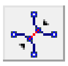
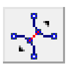
In the editor the current inversion model is used as a background over which the geological model is plotted. When creating the geological model, local objects (polygons) and layers are plotted and then filled with patterns and/or colors corresponding to the geology. The module also allows displaying borehole data to simplify the model building process.

The main purpose of the module is a rapid creation of geological sections based on inversion results and their further export for reporting. Before running the module, the display mode and graphic settings of the inversion model have to be chosen. In most cases, contour representation of the model is the best choice.



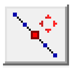
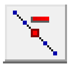
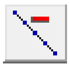
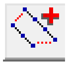
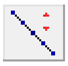
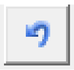
The toolbar of the **Geological editor** window contains a set of buttons for creating and editing polygons:

Tool	***Option***
	Create a polygon. Left-clicking on the model adds a new node to the polygon. Right-click specifies the location of the last node and finalizes the creation of the polygon.

	Remove a polygon. Right-click on the polygon to remove it.
	Create a polygon coupled to an existing polygon or external boundary of the model. The first and the last nodes of the new polygon should be located either on the boundary of an existing polygon or on the external boundary of the model. The program will select the common boundary automatically or will prompt the user to select it if several options are possible.
	Disconnect coupled polygons to allow editing of individual polygons (moving, node editing, etc.). Left-click on the polygon to be separated (its boundary will change color). Right-click finalizes the uncoupling.
	Divide a polygon by a straight line (create two polygons from one polygon). Click the left mouse button to indicate the first point of the line, then the right mouse button to indicate the second point. Both points should be on a boundary of the polygon to be divided.
	Move a polygon. Left-click on the polygon to capture it, right-click to release the polygon in a new location.
	Move a coupled polygon.
	Add a node. Right-click on the boundary to add a node.
	Remove a node. Right-click on the node you want to remove.
	Move a node. Left-click on the node to capture it, right-click to release the node in a new location. If the operation is not possible (there are intersecting boundaries), the node is returned to its original position. Nodes located on a model boundary can only be moved along the boundary.
	Couple two nodes belonging to different polygons. Left-click on the node to capture it, right-click to release it with the mouse pointer hovering over the node of the other polygon. The two polygons become coupled.
	Disconnect coupled nodes. Left-click on the common node of coupled polygons, right-click to release the uncoupled node in a new location.

The dialog box for changing graphic settings of a polygon is opened by double-clicking on the polygon.

The following toolbar buttons are used for creating and editing lines:

Tool	*Option*
	Add a line.
	Move a node.
	Remove a node.
	Add a node.
	Remove a line.
	Create a polygon from two lines.
	Move a line.
	Save a line.
	Undo last action.

The **File** menu of the **Geological editor** window contains the following commands:

File / Load polygons – load polygons from a file.

File / Save polygons – save polygons of the current model into a file.

File / Show background – show the background image.

File / Remove background – hide the background image.

File / Print preview – open the **Zond Print Preview** dialog box.

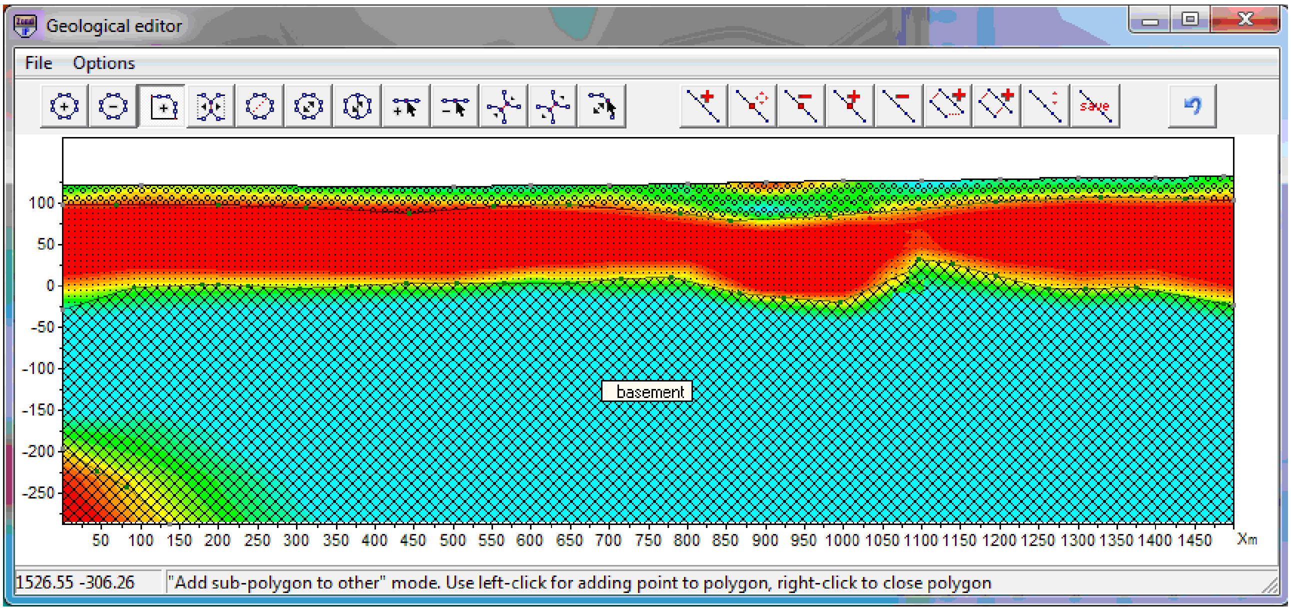
Get from modeling – load polygons from the polygonal inversion model.

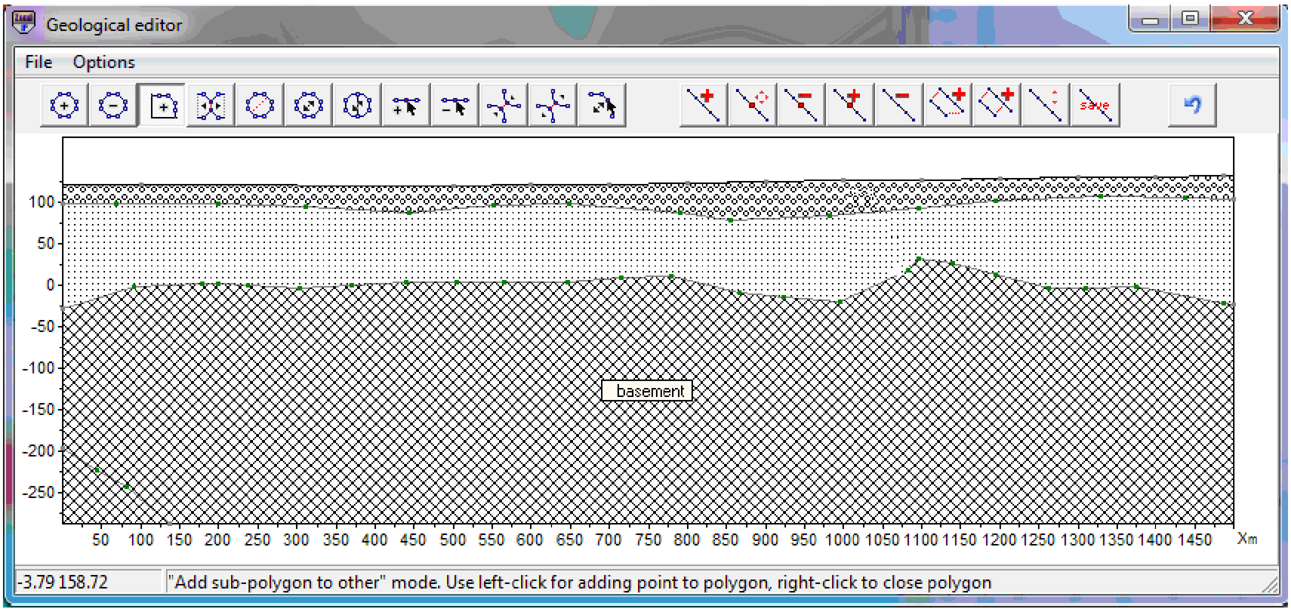
Options / Model setup – open the dialog box for specifying the model area size.

Options / Load borehole data – load borehole data from a file.

Options / Remove borehole data – remove borehole data from the module.

Options / Remove all polygons – remove all polygons.


Geological editor
File Options
100
50
0
-50
-100
-150
-200
-250
50 100 150 200 250 300 350 400 450 500 550 600 650 700 750 800 850 900 950 1000 1050 1100 1150 1200 1250 1300 1350 1400 1450
Xm
basement
1526.55 -306.26
"Add sub-polygon to other" mode. Use left-click for adding point to polygon, right-click to close polygon


Geological editor
File Options
100
50
0
-50
-100
-150
-200
-250
50 100 150 200 250 300 350 400 450 500 550 600 650 700 750 800 850 900 950 1000 1050 1100 1150 1200 1250 1300 1350 1400 1450
Xm
basement
-3.79 158.72
"Add sub-polygon to other" mode. Use left-click for adding point to polygon, right-click to close polygon

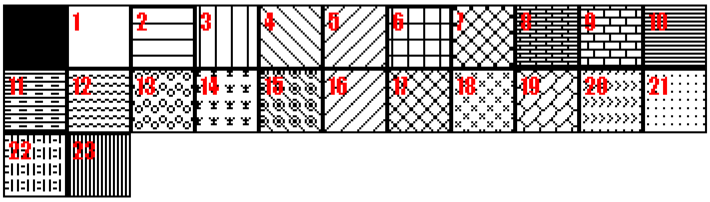
Appendix 1. Lithology data file format

Lithologic columns are hold in certain file formats. First type of files has txt extension.

The following file structure is used to create lithology data file:

First column contains depth (from ground surface) of layer boundary. Second column contains zeros. Third column defines layer color for visualization, forth – type of pattern

Lift of first 23 patterns which can be used for lithologic column creation is given below.



Lithologic data sample-file is given below:

0 1 0 13 Top of layer 1
4 1 0 13 Bottom of layer 1
4 1 0 19 Top of layer 2
11 1 0 19 Bottom of layer 2
11 1 0 27 Top of layer 3
16 1 0 27 Bottom of layer 3

Second type of files has *.crt extension; these are control files which specify type of data and way of visualization. Structure of CRT file for lithology data visualization for any number of wells is described below.

2280.txt	First line – logging or lithology data file
скв2280	Second line – Well name (is displayed on well)
18 2 2 1 0 1 0 0	Third line contains control parameters.

Data record 18 – well coordinate on profile.

2 – image width (in percents to profile length, usually 1 - 20).

2 – type of data visualization 0 - 3.

0 - logging data (as graph)*; carot1.crt*

1 - logging data (interpolated colour column), section colour scale is used for visualization; carot2.crt

2 - lithologic column; strati.crt

3 - logging data (colour column), colours for data visualization correspond to model colour scale, column colours are selected in compliance with model colour scale;;

1 - Logging data normalization parameter 0 - 2.

0,1 – the same minimum and maximum is used for all data;

1,2 - subtract average value from every well log;

0 - Logging method index (if different logging methods are displayed indices of all methods should be specified) 0 – n-1, where n – number of methods.

1 – Plot colour.

0 – Data scale is logarithmic 0 or linear 1.

0 – Vertical well shift relative to ground surface.

3246.txt Description of the following well on profile

скв3246

102 2 2 1 0 1 0 0