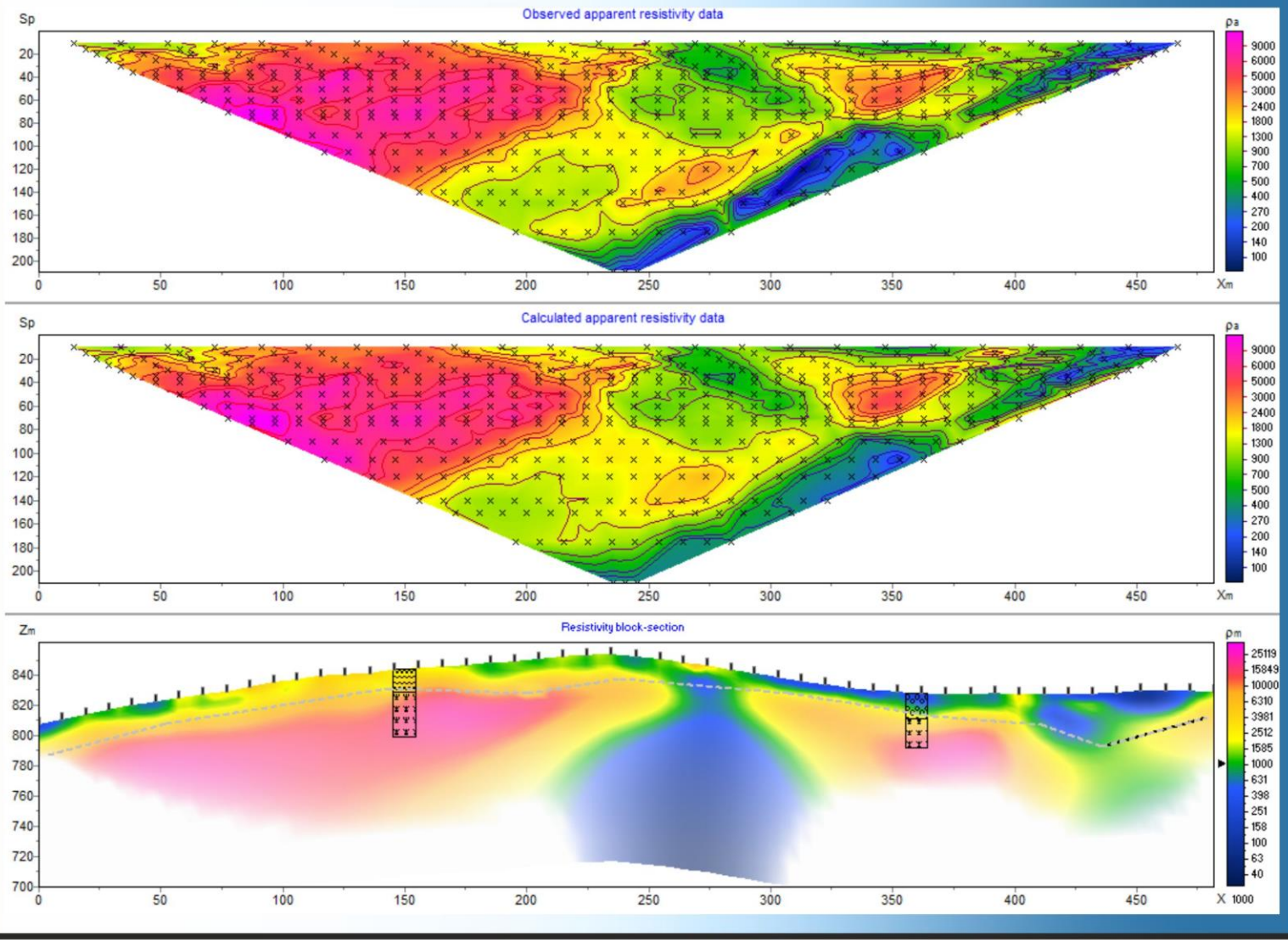
Zond Software

ZondRes2D



Introduction

ZondRes2D is a program for 2-dimensional (2D) and 2.5-dimensional (2.5D) interpretation of electrical resistivity tomography (ERT), induced polarization (IP) and mise-a-la-masse methods data for land, cross-hole and marine surveys.

ZondRes2D is an effective tool for automatic and interactive interpretation of ERT data with a user-friendly interface and a broad range of visualization options. One of the main features of the program is a joint inversion of various geophysical methods. The software can be used on any Windows operated PC.

ZondRes2D features a forward modeling routine that supports all common types of arrays used in resistivity surveying. Array configuration and the number of measurement points are determined by the user.

The program can be used to determine an optimal survey design at the survey planning stage. For this purpose, the software allows you to analyze the resolution of a particular survey array. The essence of the analysis is to let you study the influence of a particular portion of the model on the measurement results using the sensitivity function:

$$S = \sqrt{\mathrm{diag}(A^T A)}$$

The sensitivity study allows you to make an optimal choice of the type and parameters of the survey array for a specific geological problem.

ZondRes2D has a powerful set of data visualization tools, a data editor and tools for analysis of sensitivity and resolution of the applied survey method.

To display the measured and calculated data values, as well as the difference between them, the program uses two types of data representation – the graphic plot and the pseudo-section.

In the data editor, the user can view measurement parameters, set weights for individual measurements and edit the measured values.

ZondRes2D uses a simple data file format that allows combining several types of arrays on one survey line. The measured parameter can be apparent resistivity, resistance, or apparent polarization. The program allows you to import and display results of other methods, which contributes to a more comprehensive approach to data interpretation.

In addition to the resistivity and IP methods, the program performs joint inversion with data obtained with MT, TEM, refraction seismic, gravity and magnetic survey methods.

System requirements

The **ZondRes2D** program can be installed on a computer with Windows 98 or higher operating system. Recommended system parameters: Pentium IV 2 Hz CPU, 1 GB RAM, 1024 x 768 screen resolution, 24-bit (True color) color mode (the screen resolution should not be changed when working with data).

Because of the use of system registry, on operating systems higher than Windows XP the program needs to be run as an administrator (right-click on the program icon – ***Run as administrator***).

Installing the software

ZondRes2D is supplied via the Internet. You can download the latest program updates from our website: www.zond-geo.com.

To install the program, copy the program file to the desired location on your hard drive (e.g., create C:\Zond folder). To install an update, simply overwrite the old version of the file with the new one.

If the **ZondRes2D** dongle driver is not installed, you must install the SenseLock dongle driver before starting the program. To do this, open the SenseLock folder and run the InstWiz3.exe file. The latest version of the driver can be downloaded from the SenseLock website (http://www.senselock.com). After installing the driver, insert the dongle. If the driver was installed successfully, a message will appear in the notification area that the key has been found.

To uninstall the program, delete the folder containing the program.

Basic theory

Electrical resistivity

Electrical resistivity (SI unit: ohm-meter, Ohm·m) is the most universal electromagnetic property which characterizes the ability of material to resist electric current. In soils and rocks, it varies in a very wide range from 10^{-3} to 10^{15} Ohm·m. For the most common sedimentary, igneous and metamorphic rocks the resistivity depends on the mineral composition, water saturation, physical and mechanical properties, groundwater salinity, and to a lesser extent on their chemical

composition, as well as on some other factors (temperature, depth, grade of metamorphism, etc.) [9].

The electrical resistivity of minerals depends on their chemical bonds. Dielectric minerals (quartz, mica, feldspar, etc.) with predominantly covalent bonds are characterized by very high resistivity (10^{12}-10^{15} Ohm·m). Semiconductive minerals (carbonates, sulfates, halides, etc.) have ionic bonds and are characterized by high resistivity (10^{4}-10^{8} Ohm·m). Clay minerals (hydromica, montmorillonite, kaolinite, etc.) have ionic-covalent bonds and are characterized by relatively low resistivity. Metallic ore minerals (native metals, some oxides) are electronically conductive and conduct current very well.

The first two groups (dielectric and semiconductive minerals) make up the "rigid" structure of most rocks. Clay minerals make up the "plastic" structure, capable of adsorbing (bonding) retained water, whereas rocks with "rigid" minerals can only be saturated with solutions and bulk (or free) water, which means the water that can be pumped out of the rock.

The electrical resistivity of free groundwater ranges from fractions of Ohm·m for water with high mineral content (salinity) to 1000 Ohm·m in case of water with low salinity. The chemical composition of salts dissolved in water does not play a significant role; therefore, only the overall salinity of groundwater can be determined through electrical resistivity surveys. The electrical resistivity of bound water adsorbed by solid mineral particles is low and varies within a small range (from 1 to 100 Ohm·m) due to the relatively constant salinity of bound water (1-3 g/l). The average salinity of seawater is 36 g/l.

Since pore waters (free and bound) have a much lower electrical resistivity compared to most minerals that make up rocks, the resistivity of rocks is generally independent of their mineral composition and is determined by such parameters as porosity, fracture density, and water saturation. With an increase of these parameters, the resistivity of rocks decreases due to the increase of ions in groundwater. Therefore, the electrical conductivity of most rocks is ionic (electrolytic).

When the temperature is increased by 40°C, the resistivity drops approximately by half due to the increase in the mobility of the ions. Upon freezing, the resistivity of rocks increases dramatically, as free water practically becomes an insulator, and electrical conductivity is determined only by bound water which freezes at very low temperatures (below -500°C). The increase in resistivity is different for various types of rocks: it increases up to several times its value for clays, up to 10 times for hard rocks, up to 100 times for silts and sandy silts, and up to 1000 times and more for sands and coarse unconsolidated deposits.

Despite resistivity being dependent on many factors, and a wide range of resistivity values in different types of rocks, the basic ranges of resistivity are established quite clearly. Igneous and

metamorphic rocks are characterized by high resistivity (from 500 to 10000 Ohm·m). Among sedimentary rocks, high resistivity values (100-1000 Ohm·m) are typical for rock salt, gypsum, limestone, sandstone and some others. For unconsolidated sediments, as a rule, the larger the size of the particles, the higher the resistivity, i.e. it depends primarily on clay content. With the increasing clay content the resistivity changes by several orders of magnitude from fractions of ohm-meters in clays to hundreds of ohm-meters in sands [9].

Induced Polarization

Polarization is a complex electrochemical process that occurs when a direct or low-frequency (up to 10 Hz) current passes through rock or soil. Minerals with electronic conductivity (sulfides, sulfosalts, some native metals, certain oxides, graphite, anthracite) are characterized by the highest polarizability. The nature of the induced polarization (IP) potentials is associated with the so-called membrane and electrode polarization of ore minerals. Polarizability coefficients of up to 2-6% can be observed in the saturated unconsolidated sediments containing clay particles. Their polarizability is associated with deformations of the outer planes of electrical double layers that are formed at the contact area of the solid and liquid phases. Most igneous, metamorphic and sedimentary rocks saturated with saline water are characterized by weak polarizability.

To quantify the induced polarization effect, the ratio of polarization voltage at some time after the current is shut off V(t) to the initial voltage V_0 is used. This ratio is called polarizability of rocks (or polarizability coefficient) and is expressed in percentage. In early IP applications, the polarizability defined as normalized voltage value in the receiving circuit 0.2-0.5 s after the current shut off was used:

$$\eta^{0.5} = \frac{V(0.5s)}{V_0} \cdot 100\%$$

In theoretical models and when studying IP spectral characteristics, the limiting value of polarizability at t=0 s is widely used. This value is called stationary or asymptotic polarizability, but sometimes simply polarizability.

From the standpoint of noise reduction, it is more advantageous to use an integral measure of IP called chargeability (M), which is the averaged polarizability measured over a time window Δ□:

$$M = \frac{1}{\Delta t}\int_{t_1}^{t_2} \frac{V(t)}{V_0} dt$$

Forward and inverse problems of ERT

Calculating the geophysical response from a model with known geometry and physical properties is called solving the forward problem (or forward modeling). The inverse process, determining the geometry and physical properties of the model from the observed data, is called solving the inverse problem (or inverse modeling).

For solving the forward and inverse problems the program uses the mathematical apparatus of the finite element method which provides better results compared to mesh methods [1, 2]. When modeling the field created by a point source, the medium is divided into a network of cells with different resistivities. The behavior of the potential within a cell is approximated by a linear basis function:

$$N(x,z) = \frac{(a + \mathrm{b}x + \mathrm{c}z)}{2A}$$

The field of a point source within a two-dimensional medium has a three-dimensional structure. Using the Fourier transform, the solution of the problem can be taken to the spatial frequency domain:

$$\frac{\partial}{\partial x}\left(\sigma\frac{\partial\phi}{\partial x}\right)+\frac{\partial}{\partial z}\left(\sigma\frac{\partial\phi}{\partial z}\right)-\lambda^2\sigma\phi=-I\delta(x)\delta(z)$$

$$\frac{\partial\varphi}{\partial n}+\nu\cdot\varphi=0$$

,

where **φ** – spectral potential value, **λ** – spatial frequency, ***I*** – current value, **σ** – electrical conductivity of the medium, **δ** – Dirac delta function.

The subsequent solution for a set of spatial frequencies and the application of the inverse Fourier transform to the obtained spectral potential values yields the unknown values of the point source potential at the grid nodes [3].

$$U(x,y,z)=\frac{2}{\pi}\int_0^\infty\varphi(x,\lambda,z)\cos(\lambda\cdot y)d\lambda$$

For solving the inverse problem the regularized least-squares method is also used. Regularization results in a more stable solution and allows you to obtain a smoother distribution of the resistivity or polarizability in the medium [6]:

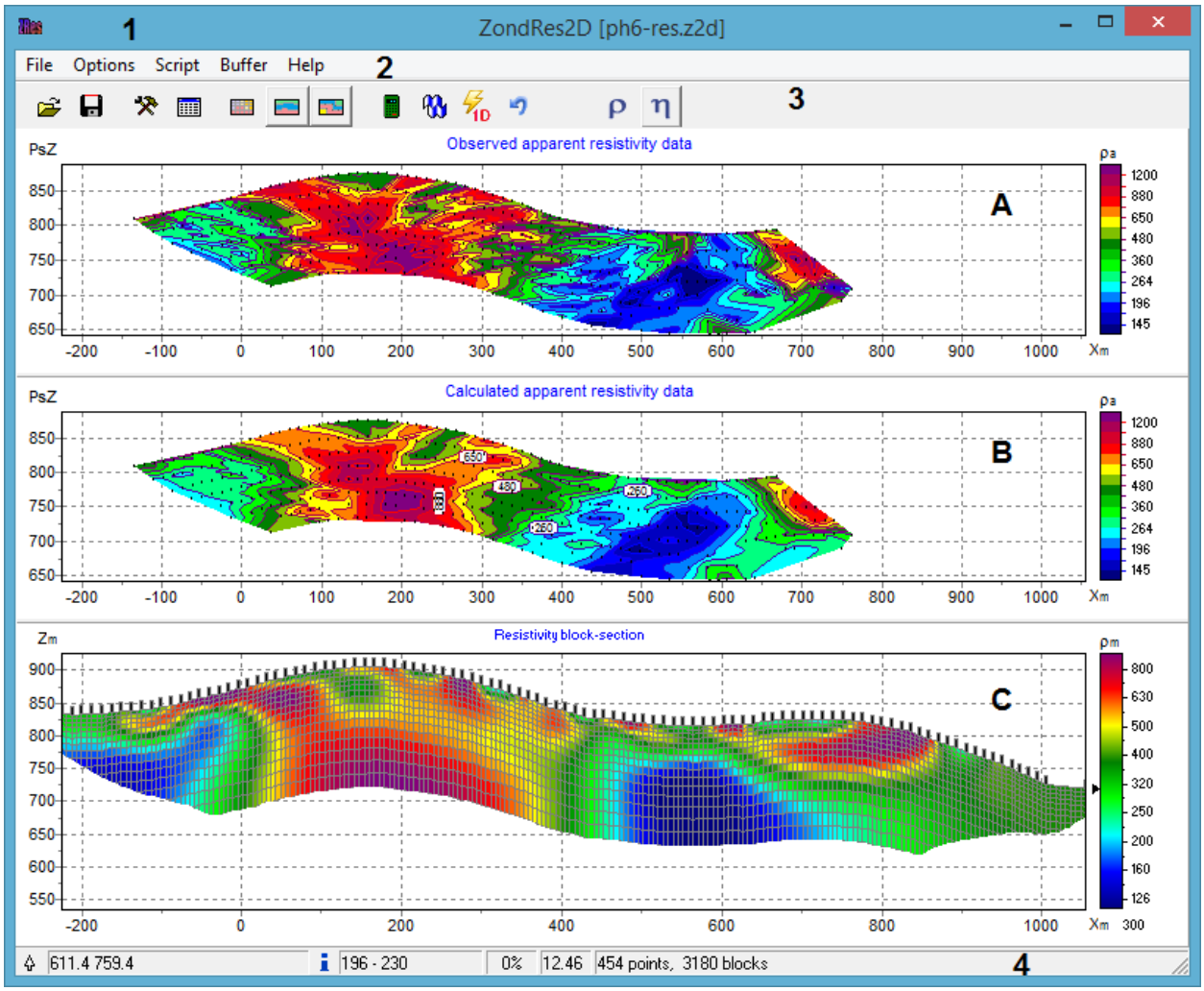
$$(A^T W^T \mathrm{WA} + \mu C^T \mathrm{RC})\Delta m = A^T W^T \Delta f - \mu C^T \mathrm{RCm}$$

,

where ***A*** – matrix of partial derivatives of measured values (Jacobian matrix), ***C*** – smoothing operator, ***W*** – matrix of relative measurement errors, ***m*** – vector of cross-section parameters, **μ** – regularizing parameter, **Δf** – vector of discrepancies between calculated and measured values, ***R*** – focusing operator.

Main window of the program

The main window of the program consists of *a header* (1), ***a series of menus*** (2), ***a toolbar*** (3), ***a status bar*** (4) and a workspace divided into three sections – *observed data* (A), *calculated data* (B) and *model section* (C):



Main window toolbar

The toolbar is used for quick access to the most frequently used functions. It contains the following buttons:

	Open data file (multiple files can be opened).
	Save data file.
	Open Program setup dialog box.
	Open data editor.
	Block (mesh) model mode.
	Arbitrary layered model mode.
	Polygonal model mode.
	Start forward modeling.
 	Start/stop inversion.
	Run one-dimensional inversion.
	Undo changing the model.
	Switch into resistivity mode.
	Switch into IP mode.

Description of the main menu commands

To bring up a tip about a particular menu command, right-click on that command in the menu. The table below provides the description of menu commands.

File menu
File/ ***Open file***
Open a data or project file (multiple files can be opened).
File/ ***Import data from text/excel***
Import data from a multi-column text file or Excel spreadsheet. You should specify column

headres in the first row of the table.
File / ***Save file***
Open the Save As dialog box.
File / ***Edit file***
Open the data file opened in the program in the Notepad editor.
File / ***Create synthetic survey***
Open the module for creating a synthetic land measurement system for modeling. You can select the spread configuration and one of the conventional arrays. The created protocol can be exported to a text file. This module is useful for planning land surveys.
File / ***Create CBH survey***
Open the module for creating a synthetic cross-hole measurement system for modeling. You can select the spread configuration and one of the conventional arrays. The created protocol can be exported to a text file. This module is useful for planning cross-hole surveys.
File / ***Change electrode array***
Open the dialog box for changing the array type (this option is used in the forward modeling routine). The spread geometry cannot be changed. This option allows you to quickly estimate the model response for different types of arrays.
File / ***Print preview***
Open the dialog box for printing the workspace.
File / ***Recent***
Show the list of the most recently used files.
File / ***Russian***
Switch the program language to Russian.
File / ***English***
Switch the program language to English.
File / ***Exit***
Exit the program.
Options menu

Options / ***Quality control module***
Open the quality control and data editing module. The QC module is used to analyze and process the observed data. The module contains a set of options for automatic and semi-automatic analysis of data errors. The [Appl&exit] and [Del&exit] buttons should be used to exit the module.
Options / ***Project information***
Show information about the project that is currently opened. This information can be edited.
Options / ***Mesh constructor***
Open the dialog box for configuring the inversion model mesh. The constructor contains a set of options for automatic mesh creation, as well as advanced options for experienced users.
Options / ***Data editor***
Open the data editor. All individual measurements are shown in a table format. The editor can be used to exclude certain measurements from the inversion process, view individual resistance values, etc.
Options / ***Program setup***
Open the main program settings dialog box. Most of the settings are related to the inversion process.
Options / ***3D fence diagram***
Open the module for three-dimensional visualization of model sections based on their coordinates. Several model sections in MOD2D file format can be plotted in the module.
Options / ***Geological editor***
Open the module for plotting an interpretive (geological) cross section based on the interpretation of the model section. A polygonal object assignment interface is used.
Options / ***Inversion***
Options / Inversion / ***64 bit version***
Switch to a 64-bit version. Use this option only for very large data sets. Some commands may not work in the 64-bit version. If this option is not working, check the settings of your antivirus which may be blocking the library.
Options / Inversion / ***Set boundaries***

Open the toolbar for specifying known boundaries (contacts) to be taken into account when performing the inversion. Use this tool if the exact position of the boundaries is known (e.g., from borehole data). Try to set the boundaries as close as possible to the model mesh nodes. It is recommended to use the following combination: Occam inversion, Smoothing factor = 0.1 – 1.
Options / Inversion / ***Whole model inv***
Use this option if you are not certain about the correct choice of the maximum survey depth or if distortion appears at the edges of the inversion model. Typically, the distortion is a result of the influence of the structures outside the limits of the survey line. If the option is enabled, the cells outside the limits of the survey line (external cells) will be included in the inversion model, otherwise, a 1D modeling is used at the edges of the section. It is recommended to use this option during joint inversion with audio-frequency MT data.
Options / Inversion / ***Optimization***
Inversion optimization parameters.
Options / Inversion / Optimization / ***Display process***
Show the optimization progress during the inversion process in a separate window. The Line search option should be enabled.
Options / Inversion / Optimization / ***Line search***
Find the optimal damping factor at each iteration (if enabled, the inversion process will take approximately 3-4 times longer). In most cases, this option allows you to achieve the lowest RMS error in a reduced number of iterations, but it increases the time taken for each iteration and may result in the solution trapped in a local minimum. If the option is disabled, the damping factor is controlled by two parameters – Smoothing factor (initial value) and Factor (reducing factor).
Options / Inversion / Optimization / ***Lim based inv***
If the overall parameter limits or individual limits for some cells are set too narrow, the inversion will try to exceed the parameter values beyond the set limits. This can greatly affect the rate of convergence. In this case, activating this option will reduce the contribution of cells that exceed the set limits, at the same time making such an exceeding difficult by applying special parameter normalization.
Options / Inversion / Optimization / ***Factor***

If the Line search option is disabled, the Factor controls the degree of damping during the inversion. For the first iteration, the Smoothing factor value is used, and for each subsequent iteration, this value is divided by the user-specified Factor. The Smoothing factor can be determined automatically if the checkbox near the Smoothing factor input box in the Model tab of the Program setup dialog box is checked.
Options / Inversion / Optimization / ***Precondition***
The use of this option is somewhat controversial. On one hand, preconditioning of the inverse problem can speed up the convergence and provide a more stable model. On the other hand, it can completely wreck the final solution. It is not recommended to use this option in combination with focusing inversion.
Options / Inversion / ***Norms***
Options / Inversion / Norms / ***Automatic sres scale***
During cross-hole, and sometimes land surveys, negative apparent resistivity values may occur. In this case, it is not possible to apply logarithmic normalization to the data (although it is possible to apply it to absolute values). For such cases, the program uses a pseudo-logarithmic normalization that requires setting a linear-logarithmic coefficient. Its value should be chosen close to the noise (background) level. If you set it to -1, the coefficient will be determined automatically. This option is available when the SRes key is used in the input Z2D data file
Options / Inversion / Norms / ***Automatic IP pseudolog scale***
By default, the program uses a pseudo-logarithmic normalization for the IP data, which requires setting a linear-logarithmic coefficient (its value should be in the order of background IP values). If you set it to -1, the coefficient will be determined automatically. If you set it to *, the linear normalization of IP data will be used.
Options / ***Inversion***
Options / Inversion / ***Resolution***
A set of parameters to increase the resolution of the inversion. Utilizing these options increases the contribution of low-sensitive cells and decreases the contribution of high-sensitive cells (typically, the cells near the electrodes). This increases the resolution, i.e., the ability to detect smaller objects at depth. Use these options with care.
Options / Inversion / ***Method***

Options / Inversion / Method / ***Gauss-Newton***
Use the iterative Gauss-Newton method, a standard least squares method. The method has fast convergence and is recommended for most cases. All inversion settings are available for this method.
Options / Inversion / Method / ***Incomplete G-N***
Use the incomplete Gauss-Newton method. Instead of the full Jacobian matrix, it calculates J * V and J ^ t * v, which significantly reduces the computer memory required, but the trade-off is a slower convergence rate and incomplete set of available inversion options. This method is not recommended.
Options / Inversion / Method / ***NLPCG***
Use the non-linear conjugate gradient method. This method significantly reduces the computer memory required but shows a slower convergence rate. This method is not recommended.
Options / Inversion / ***Smoothness***
A set of options to control the smoothness and smoothing operator. These options can strongly influence the result of the inversion. They represent the second member of the target function C^C*(m-m0).
Options / Inversion / Smoothness / ***m0 - start model***
m0 (reference model) is a user-defined model or inversion result. The main goal of inversion in this case is to reduce the RMS error while keeping similarity to the reference model. The degree of similarity is controlled by the Smoothing factor set in the Model tab of the Program setup dialog box. This option works with the Occam and Focused inversion methods.
Options / Inversion / Smoothness / ***m0 - median model***
m0 (reference model) is the median of the model at the current iteration. The main goal of inversion in this case is to reduce the RMS error while keeping the model as smooth as possible. Smoothness is controlled by the Smoothing factor. This option works with the Occam and Focused inversion methods and is best suited for focused inversion and inversion with a priori boundaries.
Options / Inversion / Smoothness / ***m0 - previous model***
m0 (reference model) is the model obtained at the previous iteration. The main goal of inversion in this case is to minimize the RMS error while ensuring a stable convergence. The

convergence rate is controlled by the Smoothing factor. This option works with the Occam and Focused inversion methods. The focused inversion sometimes might not yield the desired result (a piecewise-constant model).
Options / Inversion / Smoothness / ***Natural smoothness***
The smoothing operator is constructed taking into account the real geometry of the model mesh. Typically, this results in a smoother inversion model.
Options / Inversion / Smoothness / ***Average window***
Apply 2D moving average (median) filter to the reference model.
Options / Inversion / Smoothness / ***Diagonal flatness filter***
Apply diagonal smoothing during the inversion. Use this option if there are inclined structures in the section.
Options / Inversion / ***Cross-gradient***
Options / Inversion / Cross-gradient / ***Pushing factor***
The main parameter of the joint inversion which controls the degree of similarity (minimum of cross-gradient) of the models of two different methods. A suitable value is chosen by trial and error in the range from 0 to 1000. If the value is set to zero, the input data sets are inverted independently. Larger values may result in higher RMS error for one or both of the methods. You can balance the misfits by assigning weights to the respective data sets.
Options / Inversion / Cross-gradient / ***Off-layers num***
Often the near-surface portion of the section is highly heterogeneous and variable in different ways for different geophysical methods. In such a case, several upper layers should be excluded from the cross-gradient operator. The near-surface portions of the models will then be inverted independently.
Options / Inversion / Cross-gradient / ***MinMax range***
Set the parameter range for the second method involved in the joint inversion.
Options / Inversion / Cross-gradient / ***CC criteria***
Use the common correlation criterion for joint inversion of two models. If this algorithm is activated, the Pushing factor should be set in the 0-2 range.
Options / Inversion / Cross-gradient / ***Seismic data***

Select seismic data for joint inversion. If this option is enabled, the Background V off option appears in the menu subsection. The Background V off option enables the algorithm that minimizes the cross-gradient value for the anomalous component of velocity (velocity reduced by the background value) instead of velocity itself.
Options / Inversion / Cross-gradient / ***Gravity data***
Select gravity data for joint inversion.
Options / Inversion / Cross-gradient / ***Magnetic data***
Select magnetic data for joint inversion.
Options / Inversion / Cross-gradient / ***IP data***
Select IP data for joint inversion. The joint inversion with IP data allows you to obtain the best correlation between anomalies in the model sections.
Options / Inversion / Cross-gradient / ***BG image***
Select an image as a base for image-guided inversion. It is recommended to use grayscale images.
Options / Inversion / Cross-gradient / ***Magnetic data***
Select magnetic data for joint inversion.
Options / Inversion / Cross-gradient / ***IP data***
Select IP data for joint inversion. The joint inversion with IP data allows you to obtain the best correlation between anomalies in the model sections.
Options / Inversion / Cross-gradient / ***BG image***
Select an image as a base for image-guided inversion. It is recommended to use grayscale images.
Options / Inversion / ***Invert only visible graphs***
With this option enabled, only the data displayed in the graphic plot will be inverted. In the legend box, you can disable the graphs you want to exclude from the inversion. The option is available only when the graphic plot mode is enabled.
Options / Inversion / ***Static shifts***
Automatic correction of spurious effects (galvanic shifts) on the potential (Suppress P-static shift) or current (Suppress C-static shift) electrodes during inversion. The program models the

galvanic shift as an addition to the logarithm of the value. You should set the maximum absolute value of the shift according to the real data distribution. It is recommended to enable the iterative determination and suppression of galvanic shifts after 3-4 inversion iterations without suppression.
Options / Inversion / Static shifts / ***Not suppress***
Inversion without suppressing P/C effects.
Options / Inversion / Static shifts / ***Suppress P static shift***
Inversion with suppression of the P-effect. The P-effect is caused by resistivity heterogeneities near potential electrodes.
Options / Inversion / Static shifts / ***Suppress C static shift***
Inversion with suppression of the C-effect. The C-effect is caused by resistivity heterogeneities near current electrodes.
Options / Inversion / ***Long line inversion***
This procedure splits the profile into several overlapping sections which are inverted independently. If the number of electrodes does not exceed 5000, this procedure is not recommended.
Options / Inversion / ***Underwater options***
A set of options for marine/aquatic surveys.
Options / Inversion / Underwater options / ***Resistivity***
Set the water resistivity.
Options / Inversion / Underwater options / ***Invert***
Determine the water resistivity automatically during inversion.
Options / Inversion / Underwater options / ***Sublayers number***
Set the number of subdivisions for the water column (5-10).
Options / ***Data***
Options / Data / ***Apparent resistivity***
Display apparent resistivity values.
Options / Data / ***Resistance (V/I)***

Show resistance (V/I) values.
Options / Data / ***Pseudo-section***
Display the calculated and observed data in the form of pseudosections.
Options / Data / ***Graphics-plot***
Display the calculated and observed data in the form of graphs.
Options / Data / ***Vertical axis***
Options / Data / Vertical axis / ***PsZ***
Plot pseudodepth on the vertical axis.
Options / Data / Vertical axis / ***Spacing***
Plot spacing on the vertical axis.
Options / Data / ***Calculated data***
Display the calculated data in the second (middle) section.
Options / Data / ***Data misfit***
Display the distribution of relative misfits (RMS errors) in the second (middle) section.
Options / Data / ***Data weights***
Display the distribution of measurement weights in the second (middle) section.
Options / Data / ***Iso-CC graphics***
With this option enabled, each graph in the graphic plot corresponds to a specific position of current electrodes.
Options / Data / ***Iso-psZ graphics***
With this option enabled, each graph in the graphic plot corresponds to a specific pseudodepth (geometric factor).
Options / Data / ***Iso-PP graphics***
With this option enabled, each graph in the grap hic plot corresponds to a specific position of potential electrodes.
Options / Data / ***Display***
Options / Data / Display / ***Display every N point***

Display every N data point in the pseudo-section. This option is used for large large data sets with more than 3000 data points.
Options / Data / Display / ***Forward array***
Display data measured with forward array.
Options / Data / Display / ***Backward array***
Display data measured with reverse array.
Options / Data / Display / ***Error gates***
Show measurement errors in the form of confidence intervals on the graphs.
Options / Data / ***U/AB/MN norming***
Apply the additional normalization of the resistance (V/I) by the lengths of potential and current electrode spacings. This option is available when the Data / Resistance (V/I) mode is activated. The option is useful when the size of the potential electrode spacing varies over a wide range.
Options / Data / ***G-Res plot***
Display a plot of resistance (V/I) values against the geometric factor of the array. This option is used for borehole measurements, where other ways of data representation are not possible.
Options / ***Model***
Options / Model / ***Block-sec***tion
Display the model section in the form of blocks (cells).
Options / Model / ***Smooth-section***
Display the model section in smooth interpolated palette mode.
Options / Model / ***Contour-section***
Display the model section in the form of contours.
Options / Model / ***Resistivity***
Display resistivity in the model section.
Options / Model / ***Sensitivity***
Display sensitivity in the model section.
Options / Model / ***Model quality***

Display the quality function in the model section (sensitivity normalized to the inversion misfit).
Options / Model / ***DOI index***
Run the special algorithm that estimates the depth of investigation (DOI). It is based on two cycles of inversion with two different reference models. The DOI parameter characterizes the influence of certain parts of the model on the data. Large DOI values indicate that the corresponding part of the model is not reliable. Large values in the topmost layer of the model indicate that the cell size is too small compared to the resolution of the array. The algorithm is available only when the Contour-section display mode is activated. The algorithm allows the estimation of the minimum size of the object at a particular depth that can be reliably modeled. For this, after calculating the DOI index, press the A key on the keyboard.
Options / Model / ***DOI as transparence***
Overlay the DOI values as a transparency function on the model section. The model section will appear more transparent in areas with larger DOI values. This option is available when the DOI index is calculated and the Smooth-section display mode is activated.
Options / Model / ***X:Z=1:1 scale***
Set proportional (1:1) scaling of the model section.
Options / Model / ***ModelEditor toolbar***
Open the toolbar for editing the mesh and the model.
Options / ***Topography***
Options / Topography / ***Topo coefficient***
Set the coefficient for topography distortion with depth. If the survey line contains topography, the coefficient specifies the rate with which the mesh flattens out with depth. If it is set to 0, each new layer of the mesh has the same geometry as the first one, i.e. the geometry of the last layer matches the topography. If set to 1, the last mesh layer is horizontal.
Options / Topography / ***Strike***
Set the direction of the geo-electrical strike. Typically, it is the same as the direction of the geological strike. This option is used when the survey was not carried out across the strike of geological structures. Set to 0 if the survey was carried out across the strike.
Options / Topography / ***Import topography***

Read in topography data from a text file. The file should contain two columns (X and Y).
Options / Topography / ***Remove topography***
Delete the line's topography data. This option is used for testing purposes.
Options / Topography / ***Restore topography***
Restore the previously deleted topography data.
Options / Topography / ***Edit topography***
Edit topography profile in a table mode. The topography data can be copied from an Excel spreadsheet.
Options / Topography / ***Smooth topo***
Smooth the topography profile by averaging the neighboring values.
Options / Topography / ***Set by mouse***
Draw the topography profile on the model section using the mouse. This mode is very similar to adding a priori boundaries.
Options / Topography / ***Splined intermediate***
If this option is enabled, spline interpolation is used to calculate the elevations of nodes between the topographic points, otherwise linear interpolation is used. The option is available when the project contains topography.
Options / Topography / ***X factor***
Multiply horizontal (X) coordinates of the line by a specified coefficient.
Options / Topography / ***Reverse Line***
Flip the line horizontally from left to right.
Options / Topography / ***Shift Line***
Shift the line horizontally by a specified value (in meters).
Options / ***Import/Export***
Options / Import/Export / ***Import model/data***
Open a data file with a graph (the file should contain two columns, X and Y) or a MOD2D file to display the graph/model in a separate window.

Options / Import/Export / ***Remove model/data***
Remove the imported graph/model from the project.
Options / Import/Export / ***Model parts***
Options / Import/Export / Model parts / ***Save selection***
Save a selected portion of the model to a text file (in the Block-section mode).
Options / Import/Export / Model parts / ***Load selection***
Load a portion of the model from a text file and insert it into the model (in the Block-section mode).
Options / Import/Export / Model parts / ***Extract 1d log***
Export a 1D model for a specified position to a text file.
Options / Import/Export / Model parts / ***Load 1d log***
Load a 1D model from a text file and insert it into the model.
Options / Import/Export / ***Background image***
A set of options for configuring model section background.
Options / Import/Export / Background image / ***Load image***
Load a background image in the following formats: BMP, PNG, SGY, SEC. SEC is an internal Zond format which contains a spatial reference for an image (coordinates of the image corners).
Options / Import/Export / Background image / ***Remove background***
Remove the background image from the model section.
Options / Import/Export / Background image / ***Change sizes***
Change the size and position of the background image.
Options / Import/Export / Background image / ***Create shaded map***
Use the shaded relief map based on the current model as a background.
Options / Import/Export / Background image / ***Save background***
Save the background image to a graphic file.
Options / Import/Export / ***Save synthetic with noise***
Export model data with user-defined noise level added.

Options / Import/Export / ***Export to Excel***
Options / Import/Export / Export to Excel / ***Model***
Save the current model to an Excel spreadsheet.
Options / Import/Export / Export to Excel / ***Data levels***
Save the observed data to an Excel spreadsheet.
Options / Import/Export / ***Import RAW data***
Options / Import/Export / Import RAW data / ***COMx64, POLARES***
Import data collected with the MARY / COMx64 equipment
Options/Import/Export/Import RAW data/ ***ELECTROTEST***
Import data collected with the ELECTROTEST equipment.
Options / Import/Export / Import RAW data / ***ERAMULTIMAX***
Import data collected with the ERA-MULTIMAX equipment.
Options / Import/Export / ***Load MOD1D/2D and ZondIP1D***
MOD1D and MOD2D are internal Zond formats. This option allows you to open models created in other Zond programs and projects. The imported model is embedded into the current model.
Options / Import/Export / ***Save MOD1D/2D file***
Save the model in the MOD1D or MOD2D format for future use in other Zond programs and projects.
Options / Import/Export / ***Export model to CAD (DXF)***
Export the model (contour section in vector representation) to a DXF CAD file.
Options / Import/Export / ***Export model to geosoft***
Export the current model to the Geosoft Inc. generic format.
Options / Import/Export / ***Export model to SEG-Y***
Export the current model to the SEG-Y format.
Options / Import/Export / ***Convert to VES (ZondIP1D)***
If the ERT data contain elements of 1D soundings, they can be selected and exported to a ZLF

(ZondIP1D format) file. The dialog box that appears prompts you to select the array and sounding parameters.
Options / Import/Export / ***Convert to 3D (ZondRes3D)***
Convert a set of 2D files (Z2D) into one Z3D file (ZondRes3D format). Z2D files should be text files. XY coordinates of the start and end of the lines are specified in the 3D fence diagram module.
Options / Import/Export / ***Direct drawing in Surfer***
Plot the current model in the Surfer program. This option is available in the Contour-section mode. It might not function properly if several versions of Surfer are installed or exchange libraries are not installed.
Options / ***Cutting***
A set of options for trimming the portions of the model section where the sensitivity is low. The settings are not applicable to cross-hole data.
Options / Cutting / ***Cutting angle***
Specify the trimming angle. As a rule, the shape of the resulting section should be close to the trapezoidal shape of the pseudo-section; the trimming angle should depend on the type of array used.
Options / Cutting / ***Group cut off cells***
Group cells in the trimmed areas into 1D layers for inversion.
Options / Cutting / ***No cutting***
Do not trim the model section.
Options / Cutting / ***Cut by angle***
Trim the edges of the model section at a specified angle.
Options / Cutting / ***Cut by sensitivity***
The trimming contour is determined based on the specified cutoff sensitivity level. The cutoff level is set in percentage of the maximum sensitivity.
Options / Cutting / ***Extend bottom***
If the section contains topography and the topography distortion coefficient Topo coefficient < 1, this option extends the last model layer to the bottom edge of the model section.

Options / Cutting / ***Pseudosection bounded***
Trim the model section at the minimum and maximum horizontal coordinates of the pseudosection.
Options / Cutting / ***Write points out of cut***
If this option is enabled, the whole model is exported to an XYZ text file, otherwise, only visible area is exported.
Options / ***Borehole***
Options / Borehole / ***Create/Edit borehole data***
Add and edit borehole data (columns and logs).
Options / Borehole / ***Load borehole data***
Open a file containing borehole columns and/or logging data, or a MOD1D file (one-dimensional sounding results).
Options / Borehole / ***Remove boreholes***
Remove the borehole data from the project.
Options / Borehole / ***Set column width***
Set the width of the borehole columns in the model section (in percentage of the survey line length).
Options / ***Extra***
Options / Extra / ***Model smooth/raster***
This tool allows you to smooth the model or a portion of it, or group cells into blocks. The tool can be used for smoothing the strongly heterogeneous top of the model or for smoothing the Blocks inversion results.
Options / Extra / ***IP units***
Set IP measurement units in percentage or mV/V.
Options / Extra / ***Potential& sensitivity in model***
Display potential and sensitivity in the model section. Particular data points are selected in the Data editor table.
Options / Extra / Potential& sensitivity in model / ***Potential isolines***

Enable the mode for displaying potential distribution isolines for a specific position of current electrodes. The Data editor table appears where you can select a particular data point.
Options / Extra / Potential& sensitivity in model / ***Sensitivity isolines***
Enable the mode for displaying sensitivity isolines for a particular measurement. The Data editor table appears where you can select the measurement.
Options / Extra / Potential& sensitivity in model / ***Sensitivity contours***
Enable the mode for displaying filled sensitivity contours for a particular measurement. The Data editor table appears where you can select the measurement.
Options / Extra / ***Display Rho&IP together***
Show polarizability isolines on top of resistivity model or vice versa.
Options / Extra / ***Extra isoline color settings***
Open the dialog box for configuring the isolines of the second parameter displayed when the previous option is activated.
Options / Extra / ***Resistivity&IP summary plot***
Open a new window with resistivity and polarizability model sections.
Options / Extra / ***Resistivity&MF summary plot***
Open a new window with resistivity and metal factor model sections.
Options / Extra / ***Monitoring summary***
Open a new window with monitoring (time-lapse survey) summary sections. Two sections are displayed: 1) a section of parameter's average values; 2) a section of variation of parameter's values in percentage. You can print or save the resulting sections in the BMP/PNG format. You can also open the sections in the Surfer program.
Options / Extra / ***Model&Data histograms***
Open a chart showing the distribution of apparent (observed) and true (modeled) values of the model parameter.
Options / Extra / ***Before loading***
Options in this sub-menu should be activated before opening a data file.
Options / Extra / Before loading / ***Open in modeling mode***

Open a data file in the modeling mode (the measured data will not be displayed).
Options / Extra / Before loading / ***Include extended nodes***
Add additional nodes along the edges of the model. This option might be useful for cross-hole surveys.
Options / Extra / Before loading / ***Round position to ****
This option is used to reduce the number of unique electrode positions on the survey line. It is typically used when the current and potential electrode cables are separated. Exercise caution when setting this parameter (it is usually set equal to the minimum spacing), especially when the electrode positions are specified in UTM coordinates.
Options / Extra / Before loading / ***Remove duplicates***
If the data file contains repeated measurements, they will be deleted.
Options / Extra / Before loading / ***Average duplicates***
If the data file contains repeated measurements, they will be averaged.
Options / Extra / ***Array converting***
A set of options for converting three-electrode arrays to four-electrode arrays and handling of "infinity".
Options / Extra / Array converting / ***Convert AMN&MNB → AMNB***
Recalculate the data collected with forward and reverse three-electrode arrays into four-electrode array data (Schlumberger array, if possible).
Options / Extra / Array converting / ***Convert AMN&MNB → AMNA***
Recalculate the data collected with reciprocal three-electrode arrays into anomaly-sensitive symmetric AMNA array data (if possible). This array belongs to the class of dedicated anomaly detecting arrays and is insensitive to the influence of layered half-space. It is very sensitive to localized objects, especially near the surface, is alternating, and does not allow calculation of apparent resistivity. The results of inversion of such data sets should be regarded as complementary, for the purpose of obtaining a better resolution of localized anomalous objects. The inversion should be performed based on initial models obtained with conventional arrays.
Options / Extra / Array converting / ***Remote electrodes to infinite***
If data are collected using "infinity" electrodes, and the coordinates of these electrodes are

specified in the data file, after applying this option the electrodes will be considered to be at true infinity.
Options / Extra / Array converting / ***Edit remote electrodes position***
Edit position (coordinates) of "infinity" electrodes.
Options / Extra / ***Remove data with large misfit***
Delete data points with misfits exceeding the specified cutoff level. Pre-filtering inversion should be run with the Robust reweighting scheme option enabled.
Options / Extra / ***Get synthetic for big misfits***
Replace data points with misfits exceeding the specified cutoff level with synthetic (calculated) data. Pre-filtering inversion should be run with the Robust reweighting scheme option enabled. This procedure is not recommended.
Options / Extra / ***Display electrode RMS***
Display RMS error for each electrode. The Visible checkbox in the Marks tab of the Graphic editor dialog box should be activated.
Options / Extra / ***Misfit in percentage***
Show RMS error in percentage. The RMS error is typically given in percentage, however, for some types of surveys (cross-hole, IP) it is more practical to use absolute misfit values.
Options / Extra / ***EM coupling removal***
If the data were acquired using high-frequency AC equipment (typically 10-20 kHz for capacitively-coupled systems), the data can be converted to zero frequency using this option.
Options / ***Graphics***
Options / Graphics / ***Observed graphics***
Open the dialog box for configuring the observed data graphs.
Options / Graphics / ***Calculated graphics***
Open the dialog box for configuring the calculated data graphs.
Options / Graphics / ***Data X (4-electrodes)***
A set of options for determining the data point X position for a four-electrode array. Auto – automatic determination; **M-O-N** – the data point is at the midpoint between potential electrodes M and N; AB-O-MN – the data point is at the midpoint between current (AB) and

potential (MN) dipoles.
Options / Graphics / ***Data X (3-electrodes)***
A set of options for determining the data point X position for a three-electrode array. (A+M)/2 – the data point is at the midpoint between the current electrode A and the nearest potential electrode M; (M+N)/2 – the data point is at the midpoint between potential electrodes M and N.
Options / Graphics / ***Perpendicular array***
A special approach of data representation in the case when measurements are carried out along the strike of geological structures. Such can be the case when electrical soundings are performed at right angle to the direction of the main survey line.
Options / Graphics / ***Smooth contours***
Smooth the model contours in the Contour-section display mode.
Options / Graphics / ***Smoothness***
Set the degree of smoothness of the model contours. The greater the smoothing parameter, the smoother the contours.
Options / Graphics / ***Alternative smooth***
Alternative way of plotting the contour section. It creates a more blocky structure and renders slower than the default version.
Options / Graphics / ***Copy pseudo colors to model***
Copy the color scale of the pseudosection to the model section. вопрос
Options / Graphics / ***High quality isolines***
Render contours in high quality. It is recommended to use this option only prior to printing/exporting the results because it slows down the program performance.
Options / Graphics / ***Bitmap output settings***
Open the dialog box for setting up export image parameters.
Options / Graphics / ***Axis behind***
Plot axes grid lines in front of or behind the model section image.
Options / ***MT data***
Options / MT data / ***Load MT data***

Load MT data. Both M2D (**ZondMT2D**) and MDF (**ZondMT1D**) text formats can be imported to perform the joint inversion. Stations coordinates (distances in km) should be in the same coordinate system as the current model.
Options / MT data / ***Invert MT data***
Include MT data in joint inversion with ERT data.
Options / MT data / ***Components***
Choose the components of the MT field to participate in the inversion.
Options / MT data / Components / ***TM inversion***
Select TM mode for joint inversion.
Options / MT data / Components / ***TE inversion***
Select TE mode for joint inversion.
Options / MT data / Components / ***Tipper inversion***
Select tipper for joint inversion.
Options / MT data / Components / ***DET inversion***
Select effective impedance for the joint inversion
Options / MT data / ***Show MT plot***
Open a window with MT data.
Options / MT data / ***With static shift***
Automatic correction of galvanic shifts in MT data during inversion. The program models the galvanic shift as an addition to the logarithm of the value. You should set the maximum absolute value of the shift according to the real data distribution. It is recommended to enable the iterative selection and suppression of galvanic shifts after 3-4 inversion iterations without suppression.
Options / MT data/ ***Set weight of MT***
Set the weight of MT data. Assigning weights allows you to control joint inversion misfits.
Options / ***Seismic data***
Options / Seismic data / ***Load SRT data***
A simple text file format is used to import seismic data, which contains three columns with the

following headers: sx – source coordinates (m), rx – receiver coordinates (m), fb – first arrival picks (ms). The coordinates should be in the same coordinate system as the current model.
Options / Seismic data / ***Show SRT plot***
Open a window with seismic data.
Options / Seismic data / ***Invert SRT data***
Include seismic data data in joint inversion in layered model mode.
Options / Seismic data / ***Set weight of SRT***
Set the weight of seismic data. Assigning weights allows you to control joint inversion misfits.
Options / GraviMagnetic
Options / GraviMagnetic / ***Load new data***
Read in gravimagnetic data from a multi-column text file. The first row should contain column headers. The data must be in the same coordinate system as the current model.
Options / GraviMagnetic / ***Add new data***
Add gravimagnetic data to the project.
Options / GraviMagnetic / ***Save data***
Save the measured and calculated data to a text file.
Options / GraviMagnetic / ***Field setting***
Set up gravity and magnetic field parameters.
Options / GraviMagnetic / ***Substract median grav***
Subtract the median value from the measured gravity data to obtain the anomalous field.
Options / GraviMagnetic / ***Substract median mag***
Subtract the median value from the measured magnetic data to obtain the anomalous field.
Options / GraviMagnetic / ***Inversion***
Invert gravity and magnetic data in polygonal mode.
Options / GraviMagnetic / ***Invert gravity***
Enable inversion of gravity data in layered model mode.
Options / GraviMagnetic / ***Invert magnetic***

Enable inversion of magnetic data in layered model mode.
Options / GraviMagnetic / ***Set weight of gravity***
Set the weight of gravity data. Assigning weights allows you to control joint inversion misfits.
Options / GraviMagnetic / ***Set weight of magneti***c
Set the weight of magnetic data. Assigning weights allows you to control joint inversion misfits.
Options / GraviMagnetic / ***Display GM window***
Open a window with gravimagnetic data and the model (in cross-gradient mode).
Script menu
Scripts are used for automatic batch processing of multiple files. First, a script is recorded while a file is being opened and processed, then the recording stops and the script (processing sequence) can be applied to other files.
Script ***/ Start recording script***
Start recording the script. You should enable this option before opening a new file.
Script / ***Stop recording script***
Finish recording the script.
Script / ***Run packet processing***
Select a list of files for batch processing with the recorded script.
Script / ***Save script file***
Save the current script to a text file.
Script / ***Load script file***
Load a script from a file.
Script / ***Preset***
Script / Preset / ***Stable 1***
A preset script – a step-by-step strategy to get consistent results even for very low quality data. The first iteration uses 8x8 inversion cells, each subsequent iteration uses the model from the previous iteration and reduces the cell size.

Script / Preset / ***PC-suppress***
A preset script – a special algorithm for suppressing P/C effects in the observed data. It should be applied only if the data is highly distorted.
Buffer menu
The buffer allows you to store up to five models obtained using different inversion parameters. The models can be opened in a separate window for comparison.
Buffer / ***Model 1-5***
Save/load the model into/from the buffer.
Buffer / ***Open***
Open a window displaying all buffer models. This option is useful for comparing inversion results obtained with different settings.
Help menu
Help **/ *About***
About the program.
Help / ***Manual***
Open the program manual.
Help / ***Erorr!!! Clear settings***
Reset all settings and return to the default settings after restarting the program.
Help / ***Check for updates***
Check for updates.
Help / ***Bing maps api_key***
If a Bing Maps image is not downloaded automatically from the Internet, you should enter an actual Bing API key.
Help / ***Show news***
Show news.
Help / ***Send message to us***
Send a message to the developer. Only messages written in the Latin alphabet can be sent (use

transliteration if sending messages in Russian).

When you switch to the layered model mode (the  button on the main window toolbar), the **Layered** sub-menu becomes available with the following options:

Layered sub-menu
Layered / ***Model constructor***
Open the layered model constructor.
Layered / ***Save to mesh***
Embed the layered model into a mesh model.
Layered / ***Load from mesh***
Use the average value of all cells comprising a layer as the layer's parameter value.
Layered / ***Invert Rho&IP***
Joint inversion of resistivity and IP data using a common geometry of layers.
Layered / ***Invert boundaries***
Determine the geometry of layers during the inversion (sometimes you only need to determine parameter values, e.g., when the boundaries are known and fixed).
Layered / ***Draw labels***
Display parameter values in the nodes. The choice of the parameter to be displayed is made in the model constructor.
Layered / ***Transparent***
Do not fill layers. This allows you to see the results of the inversion in the mesh mode in the background and set the optimal initial model.
Layered / ***Edit mode***
Enable editing mode of the layered model. Editing is done using the mouse. The boundaries can be dragged in the vertical direction; right-clicking on the layer's label allows editing the parameter value.
Layered / ***Save layers***
Save the layered model to a text file.

Layered / ***Load layers***
Load a layered model from a text file.

Additionally, when switching to the layered model mode, options for working with the time-domain electromagnetics (TDEM) and frequency domain electromagnetics (FDEM) data will appear in the **Options** tab.

Options menu
Options / TDEM data
Options / TDEM data / ***Load TDEM data***
Import data in TDF (**ZondTEM1D**) or USF (*universal sounding format*) text formats. The coordinates should be in the same coordinate system as the current model. Joint inversion is available only for layered model mode.
Options / TDEM data / ***Show TDEM data***
Open a window displaying the TDEM/FDEM data.
Options / TDEM data / ***Invert TDEM data***
Include the TDEM/FDEM data in joint inversion in layered model mode.
Options / TDEM data ***/ Set weight of TDEM***
Set the weight of TDEM/FDEM data. Assigning weights allows you to control joint inversion misfits.

When you switch to the polygonal modeling mode (the  button on the main window toolbar), the **Modeling** sub-menu becomes available with the following options:

Modeling sub-menu
Modeling ***/ Get values from mesh***
Assign parameter values to polygons automatically. The value will be equal to the average value of all model cells within the polygon.
Modeling / ***Set values to mesh***
Embed the polygonal model into a mesh model.
Modeling / ***Save polygons***

Save polygns to a text file.
Modeling ***/ Load polygons***
Load polygons from a text file.
Modeling / ***Remove all polygons***
Remove all polygons.
Modeling / ***Display color scale***
Show the color scale bar next to the model section.
Modeling / ***Colors from color scale***
Assign colors to polygons according to the color scale.

When switching to the polygonal modeling mode, options for working with the self potential (SP) method will appear in the **Options** tab.

Options
Options / ***Self potential***
Options / Self potential / ***Load SP data***
A simple text file is used to import SP data, which contains four columns: 1) data point X (m); 2) M electrode X (m); 3) N electrode X (m); 4) measured value (mV). The X coordinates should be in the same coordinate system as the current model. The data point X is at the midpoint between electrodes M and N for the gradient method, or at the rover electrode X for the potential (fixed-base) method.
Options / Self potential / ***Remove SP data***
Delete SP data from the project.
Options / Self potential / ***Get SP from ERT***
Extract SP data from the resistivity data. Quite often files with ERT data already contain SP values.
Options / Self potential / ***Export to ZondSP2D***
Export SP data to the ZondSP2D program text file format. In addition to the data, the current resistivity model is exported to the MOD2D file format.

Options / Self potential / ***Redox factor vs Z***
Set the depth distribution of the potential-determining factor.
Options / Self potential / ***Use polygons resistivity***
If this option is enabled, the polygon resistivities are used for SP calculations, otherwise, the resistivities of the mesh model are used.
Options / Self potential / ***Invert redox factor***
Determine the depth distribution function of SP sources during inversion. The distribution depends on the Z coordinate only. A conductivity type (electronic or ionic) should be set for each polygon. The conductivity type is set in the Body parameters dialog box which can be opened by double-clicking on the polygon. If the checkbox in the dialog box is activated, the electronic conductivity is selected, otherwise, it will be ionic conductivity.
Options / Self potential / ***Invert volume sources***
Determine the volumetric distribution of SP sources within a specific, user-defined polygon.
Options / Self potential / ***Remove volume sources***
Remove the volumetric distribution of SP sources.
Options / Self potential / Display SP window
Open a window displaying the SP data.

Status bar

The status bar is located at the bottom of the main program window and is divided into several sections containing various information:

- Coordinates of the mouse pointer and the active cell.
- Parameter value of the active cell.
- Model editor operating mode.
- Process indicator.
- RMS error.
- Additional information, e.g., a number of data points and model cells, or the inversion status.

Hotkeys

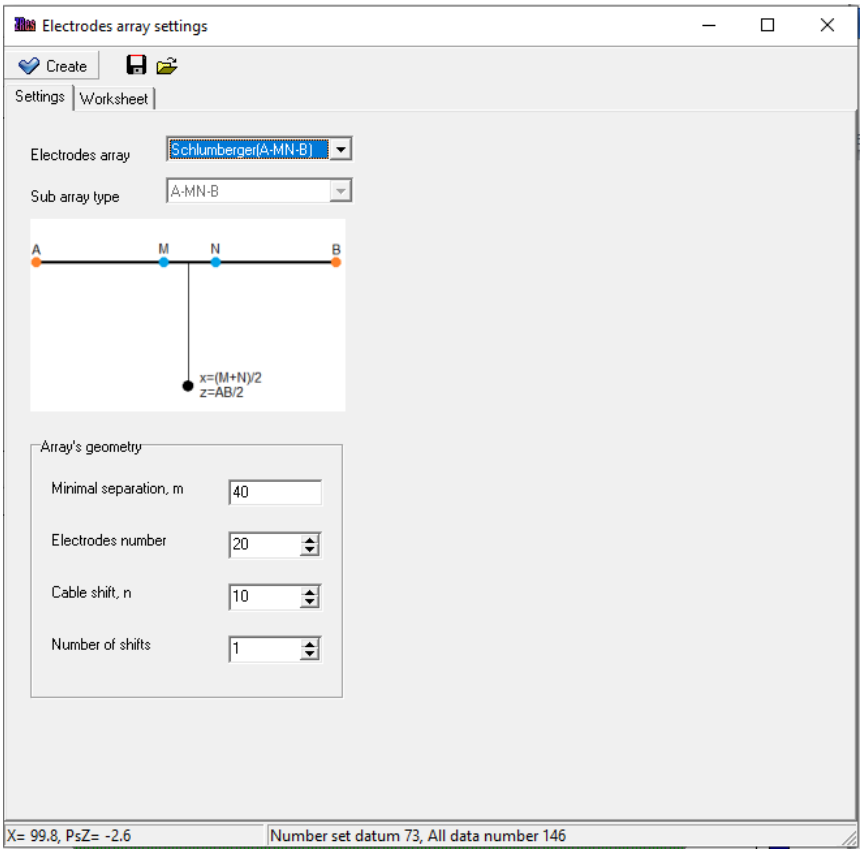
Arrow keys or mouse pointer in the model editing mode
Select active model cell.
[Delete] / cursor in the model editor
Reset the cell parameter to its initial value.
[Insert] / cursor in the model editor
Assign the current value to the active cell.
[F] / cursor in the model editor
Fix the value of the active cell.
[X] / cursor in the model editor
Select cells with similar parameter values (the “Magic Wand” tool).
[V] / cursor in the model editor
Remove selection.
[Ctrl] + [C] (press and hold down)
Drag a selection in the model section using the left mouse button.
[Up] / Down arrow keys after left-clicking on the color bar
Change the current value.
[Space]
Solve the forward problem.

Creating a synthetic measurement system

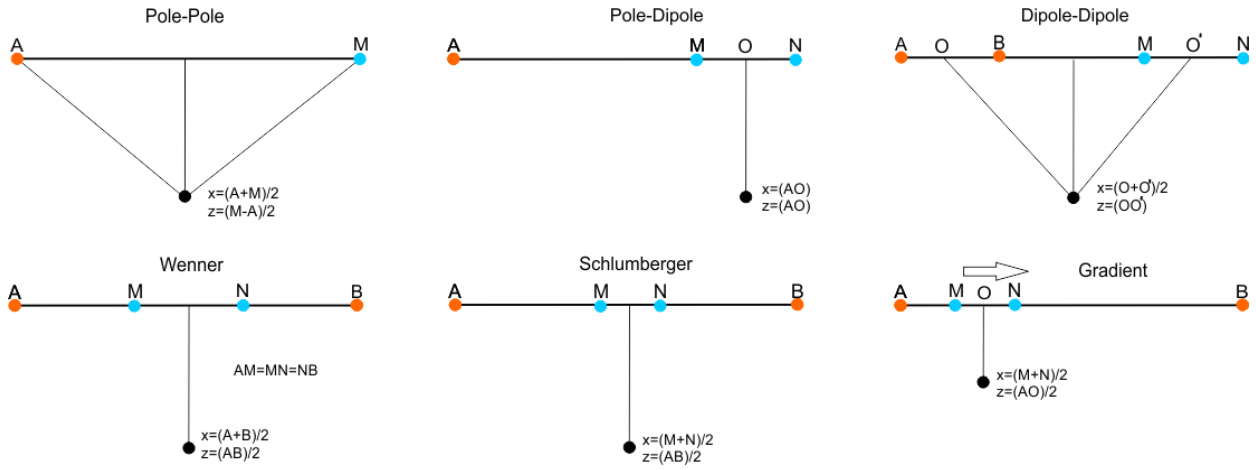
The system of observations (measurements) can be set in two ways: you can create it directly in the program or read it in from an existing data file. Note that creating a measurement system in the **ZondRes2D** program allows you to perform modeling only for arrays with constant spacings between electrodes. To perform modeling for measurement systems with arbitrary electrode separations refer to the **Importing a measurement system from a data file** section.

Use the **File** / **Create synthetic survey** menu command to open the **Electrodes array settings** dialog box containing settings for configuring the parameters of the ERT measurement system.

The **Settings** tab allows creating a measurement system using one of the conventional array types.



The type of array is selected from the **Electrodes array** drop-down list. The following options are available: Pole-Pole (A-M), Pole-Dipole (A-MN), Dipole-Dipole (AB-MN), Wenner (A-M-N-B), Schlumberger (A-MN-B), and Gradient (A-MN-B). A sketch corresponding to the selected array type is displayed in the central portion of the tab:



The **Sub array type** drop-down list allows you to select the array sub-type (the order of the electrodes). For example, for a three-electrode (Pole-Dipole) array the options are the **Forward (A-MN)** array, the **Reverse (MN-A)** array, and a combination of both (**A-MN & MN-A**).

The bottom portion of the **Settings** tab contains the **Array's geometry** settings**,** which define the basic geometrical parameters of the measurement system:

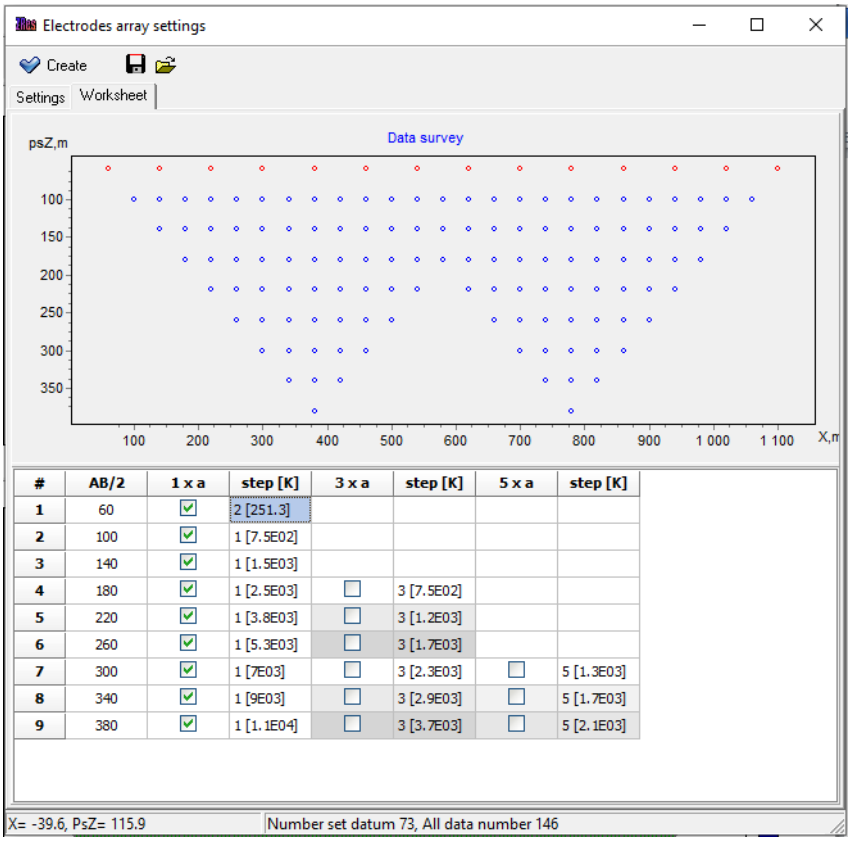
Minimal separation, m – smallest spacing between electrodes in meters;

Electrodes number – number of electrodes in one spread;

Cables shift, n – step size of cable roll-along, **n** is the number of electrode spacings;

Number of shifts – number of roll-along shifts along the line. If 0 is selected, a single spread is used.

The **Worksheet** tab allows you to configure the geometry of the measurement system in more detail.



The **Data survey** window in the upper portion of the tab contains a graphical representation of the created measurement system. It shows schematically the location of the data points in the form of small circles. The horizontal axis shows the distance along the line in meters, and the vertical axis shows the array spacing in meters. When pointing the mouse cursor at a data point you can see the electrode numbers corresponding to this data point. For example, **C1C4P2P3** for the **Schlumberger(A-MN-B)** array means that for this data point electrodes 1 and 4 are used as current (**C**) electrodes and electrodes 2 and 3 are used as potential (**P**) electrodes.

The bottom portion of the **Worksheet** tab contains a table. The first column of the table contains all possible measurement spacings in meters **(AO, OO, AB/2**, or **AM/2** depending on the selected array type). For the **Pole-pole (A-M)** array you can include or exclude particular measurement spacings using checkboxes in the "**M**" column. For other array types, checkboxes in the "**n x a**" columns are used**,** where **n** is an odd number, e.g. "**1 x a**" or "**5 x a**". The "**n x a**" number represents the measurement spacing (length of the specific electrode configuration), where "**a**" is the **Minimal separation, m**, i.e. the smallest electrode spacing in the spread. The **step[K]** columns define the frequency (density) of measurements for a particular spacing. The **step** value can be set manually for each spacing and determines the frequency (density) of measurements. **[K]** is a geometric factor and is calculated automatically. For example, if the **step** value for a specific spacing is set to 3, the measurements using this spacing will be performed at every third possible point. The **Data survey** plot will be updated correspondingly.

The measurement protocol can be saved in the Syscal sequence (TXT), ABEM sequence (XML), MAE sequence (SEM), Multimax sequence (TXT) and AGI sequence (CMD) formats using the  button in the upper left corner of the dialog box.

After configuring the measurement system and pressing the 

button, the **Mesh constructor** dialog box appears. In the ***Start resistivity*** (and ***Start polarizability***) input box you can set the resistivity (and polarizability) of host rocks. After pressing ***Apply***, the tool buttons for working with data become active on the main window toolbar, and brief information about the data and the model (the number of data points and model cells) appears in the right section of the status bar.

To change the measurement system without changing the model, use the **File** / **Change electrodes array** menu command, which reopens the **Electrodes array settings** dialog box.

Creating a cross-hole measurement system in ZondRes2D

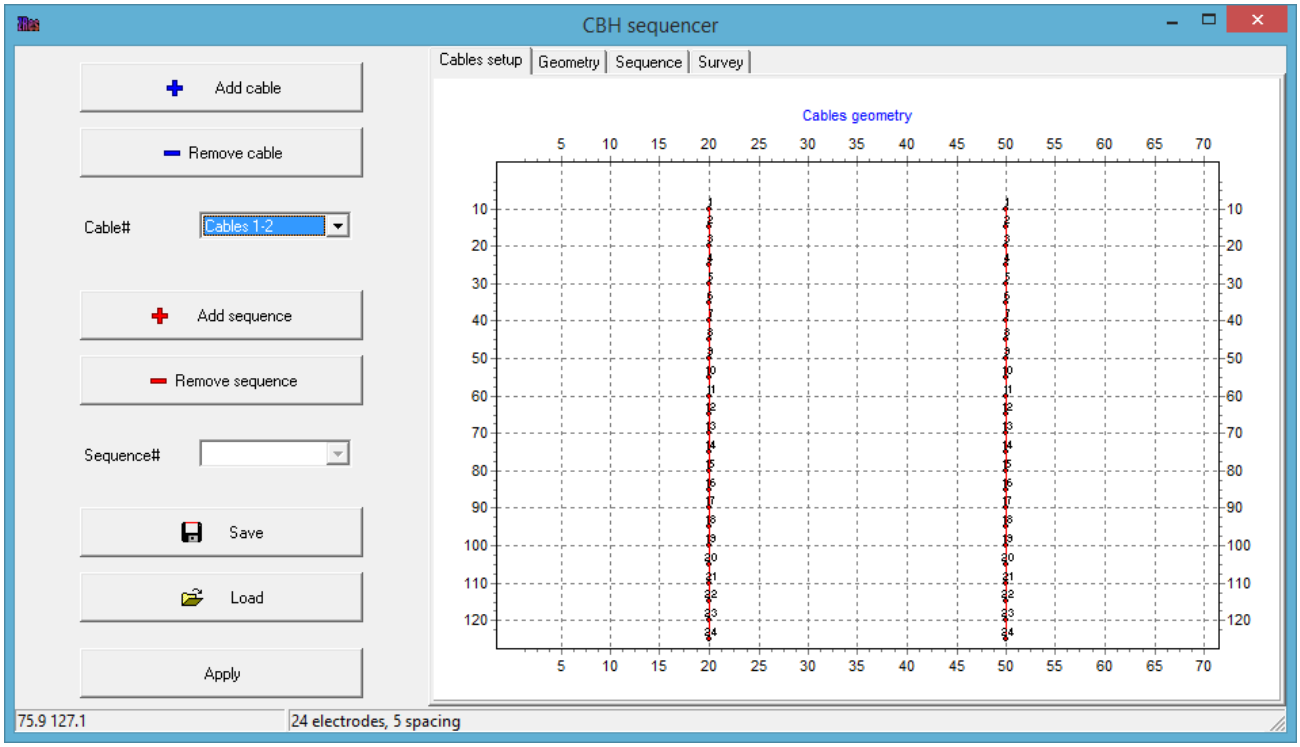
Use the **File / Create CBH survey** menu command to open the **CBH sequencer** dialog box containing settings for configuring the parameters of the cross-hole measurement system.

In the **Cables setup** tab, you can set the array geometry on the XZ coordinate plane using the mouse.

Add a spread (cable) by clicking the 

button, setting the number of electrodes and the spacing between them in meters. Left-clicking on the XZ coordinate plane will specify the starting point of the spread; right-clicking will complete the process of creating the spread. Delete the spread by clicking the 

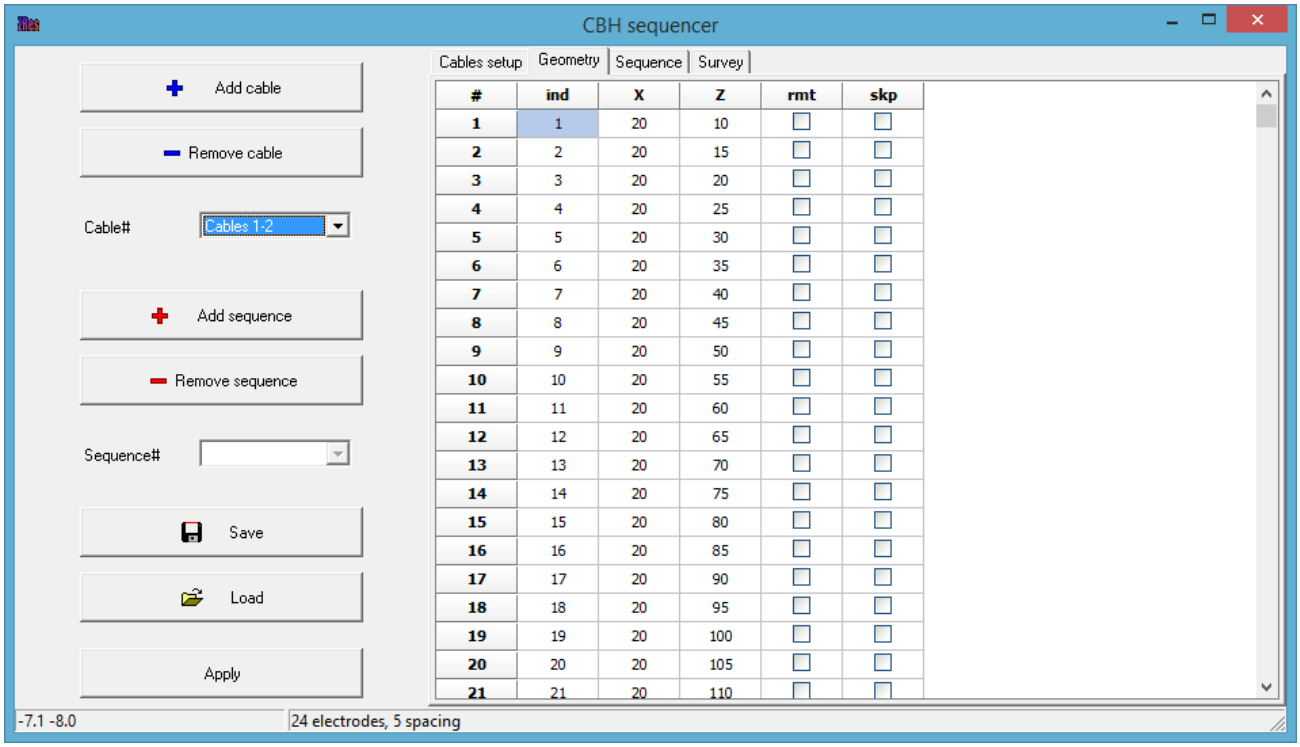
button. From the **Cable#** drop-down list, you can select the spread (or a pair of spreads) to work with.



In the **Geometry** tab, you can edit the geometry of the spread in the table.

The *Ind* column allows you to set an integer index number for each electrode.

An activated checkbox in the *rmt* column indicates that the electrode is considered "at infinity". An activated checkbox in the *skp* column excludes the corresponding electrode from the created protocol.



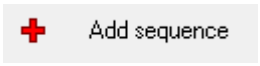
#	ind	X	Z	rmt	skp
1	1	20	10	☐	☐
2	2	20	15	☐	☐
3	3	20	20	☐	☐
4	4	20	25	☐	☐
5	5	20	30	☐	☐
6	6	20	35	☐	☐
7	7	20	40	☐	☐
8	8	20	45	☐	☐
9	9	20	50	☐	☐
10	10	20	55	☐	☐
11	11	20	60	☐	☐
12	12	20	65	☐	☐
13	13	20	70	☐	☐
14	14	20	75	☐	☐
15	15	20	80	☐	☐
16	16	20	85	☐	☐
17	17	20	90	☐	☐
18	18	20	95	☐	☐
19	19	20	100	☐	☐
20	20	20	105	☐	☐
21	21	20	110	☐	☐

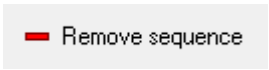
The **Sequence** tab is used for creating a measurement protocol. The **Array** drop-down list allows you to select the array type. The **Geometry** drop-down list allows you to select the orientation of dipoles.

In the **Filter** group box the maximum geometric factor (**MaxK**), minimum and maximum dipole lengths (**MinDIP** and **MaxDIP**), and unstable values check (**Unstable check**) are set. Unstable values are the data points for which the geometric factor values are very sensitive to errors in the geometry.

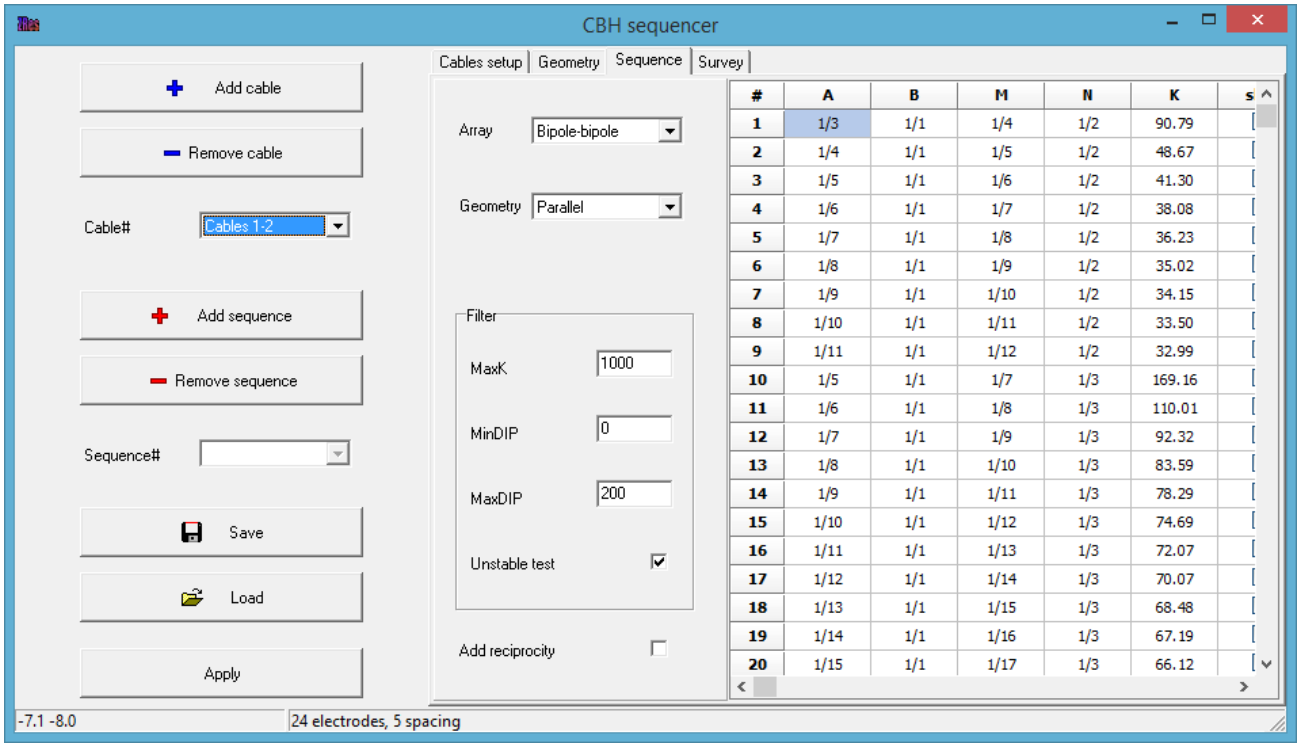
Add reciprocity – add reciprocal measurements for a two- or four-electrode array.

Clicking on a row in the table will open the **Array** window, which shows the position of the electrodes corresponding to the selected row. An activated checkbox in the *skip* column excludes a specific data point.

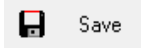
Create a protocol for the selected spread and array type using the 

button. Delete the current protocol using the 

button. For each spread (or a pair of spreads) you can create several protocols with different settings. The active (current) setting is selected from the **Sequence#** drop-down list.

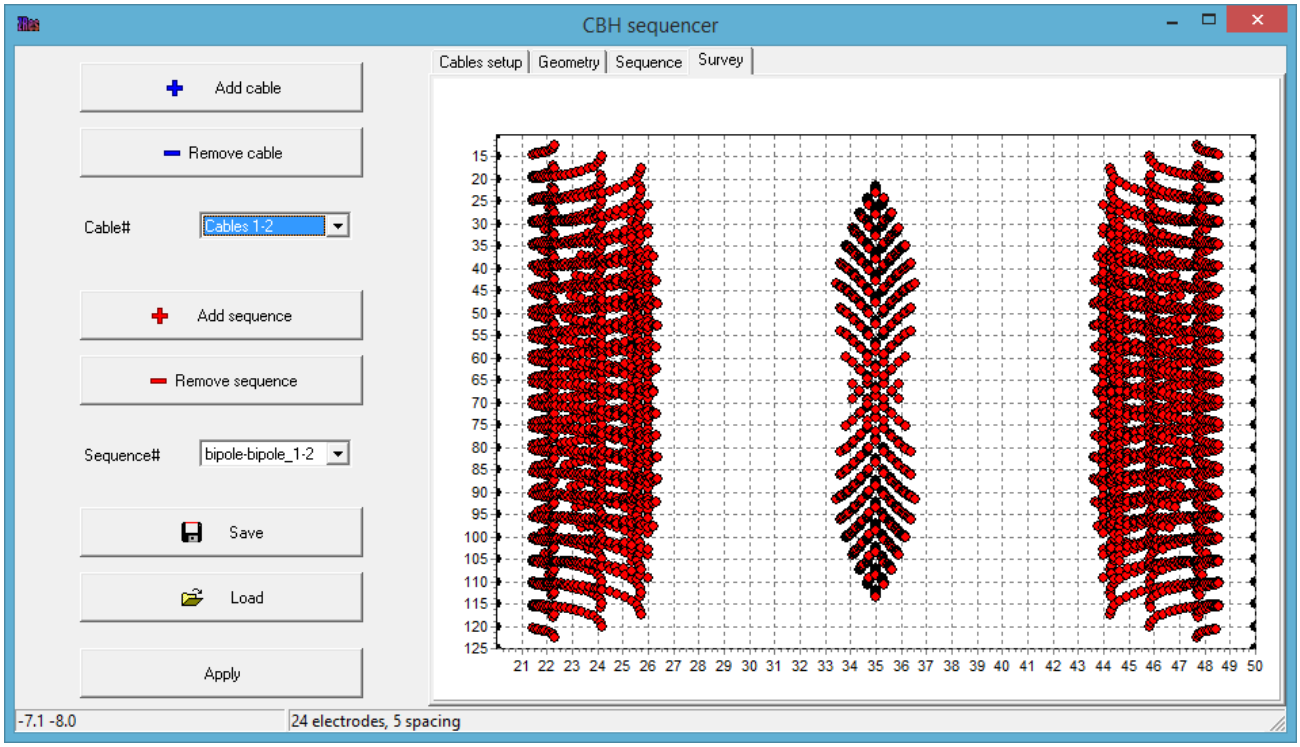


In the **Survey** tab the measurement scheme is visualized.

Using the 

button, the measurement protocol can be saved in the *Sequence project*, *ZondRes2D*, *MAE sequence* and *IRIS sequence* formats.

A measurement protocol in the **Sequence project** format can be opened using the Load button.



Importing a measurement system from a data file

An alternative to creating a measurement system is to import it from a data file.

Before opening a data file, it is necessary to switch to the modeling mode (**Options** / **Extra** / **Before loading** / **Open in modeling mode**). After activating the **Open in modeling mode** option, you can open the data file using the **File** / **Open file** command. This option allows you to perform forward modeling with the same measurement system that is used for field measurements.

The **File** / **Change Electrodes array** option will be disabled if the data file is opened without activating the **Open in modeling mode** option first.

Opening data files

Supported Formats

To start working with **ZondRes2D** in the interpretation mode, you need to open a data file of a certain format containing electrode coordinates, topography and measurement data.

In addition to its internal Z2D file format, the program supports the most widely used data formats: RES2DINV (Geotomo Software, M.H.Loke, Malaysia), SENSINV2D (Geotomographie,

T.Fleschner, Germany), ABEM (Guideline Geo AB, Sweden), AGI (Advanced Geosciences, Inc., USA), Syscal (Iris Instruments, France), ProfileR (Lancaster Environment Center, A.Binley), and others. The program also allows reading in data from an arbitrary tabular text file (the **File / Import data from text/excel** option). When selecting the **File** / **Open file** menu command you will be prompted to select one of the following file formats:

Zond data file [*.z2d]
Open data file or project file in Zond format.
ProfileR data file [*.in]
Open the ProfileR program data file.
ABEM/AGI/SYSCAL text file
Open data file in ABEM/AGI/SYSCAL format.
Res2dInv [*.dat], ARES
Open the Res2dInv program data file.
Sens2dInv [*.gem, *.imp, *.sen]
Open the Sens2dInv program data file.
Zond1d file conversion [*.zlf]
Import data from a ZondIP1D file.
Program configuration [*.cfg]
Open file with program configuration settings.
Omega-48 data files
Open data file in Prosys format.
Skala-48/64 csv file
Open data file in Skala-48/64 format.
SmartTEM-24 file [*.ip]
Open data file in Smart TEM-24 format.
GDD instruments [*.gdd]
Open data file in GDD format.
MAE instruments [*.tsv]
Open data file in MAE instruments format.
GeoSoft data [*.dat]
Open the GeoSoft program data file
Zonge data
Open data file in Zonge instruments format.
LGM device
Open data file in LGM device format.

Note. *To be opened correctly, the data file should not contain the following:*

- non-conventional delimiters separating entries in a string (use Tab or Space characters);
- absurd values of measured parameters (e.g., negative values of apparent resistivity).

Preferably, the total number of measurements in one data file should not exceed 100000, and the number of unique electrode positions should not exceed 2000.

Typically, one file contains data for one survey line. The text data files formatted in the **ZondRes2D** program format have the Z2D extension. The following paragraphs describe this format in detail.

Z2D file format

Z2D files can have a different structure – they are either data files (the structure of such a file is described below), or project files created by the program when saving a project. The project file is binary and can only be read with the **ZondRes2D** program. It contains all the project information, i.e. observed data, calculated data, models, stored a priori information, data from other methods, etc.

Z2D is a universal data file format that contains information about the coordinates of current and potential electrodes, topography, and measured values (resistance, apparent resistivity, or apparent polarizability).

A Z2D file can be created using any text editor, e.g. Notepad. To be able to open the file in the **ZondRes2D** program, you should change its extension from *.txt to *.z2d.

A data file can be conventionally divided into two parts: 1) observed data; 2) topographic data (which is optional).

Part I of the data file: observed data

For conventional ERT surveys, the program accepts files with the Z2D extension which have the following structure.

The first line contains headers that specify the type of data in the respective columns.

The following headers for electrode coordinates are used:

C1 C2 C1z C2z C1y C2y – for current electrodes;

P1 P2 P1z P2z P1y P2y – for potential electrodes.

Y and Z electrode coordinates should be present when necessary, for example, when using non-collinear arrays, submerged sources or for marine surveys. An example file containing Z coordinates: *sample_with_z_source*. The distance along the Y-axis should not exceed 1/3 of the maximum measurement spacing (distance between the current and potential electrodes). An example file containing Y coordinates: *sample_with_y*. When entering Z coordinates of electrodes, remember that a positive value means the distance below the surface of measurements. Negative Z coordinates are only used for marine surveys (where the surface of measurements is the bottom).

A * symbol in front of an electrode header (e.g., *C1) means that this is a remote electrode which coordinates are used for calculations, but it is not involved in the generation of the mesh or images.

The following headers for the measured data are used:

Ro_a – apparent resistivity;

Res – absolute value of resistance (V/I);

SRes – resistance (V/I) (e.g., for measurements with submerged source, with the account of the sign);

Eta_a – apparent polarizability.

Note. *It is recommended to ALWAYS use the* ***Res*** *or* ***SRes*** *values in data files when topography is present to avoid geometric errors.*

Apparent polarizability measured in the time domain and calculated by the following formula can be used as **Eta_a**:

$$\text{Eta\_a}\ (\%) = (V(t) / V_0) \cdot 100\%,$$

where $\mathbf{V_0}$ – initial voltage measured during the on-time, **V(t)** – polarization voltage measured during the off-time. If chargeability is used as an input parameter, the measured chargeability values should be divided by 10 beforehand.

The error, or measurement weight, specified in the column with the **Weight** header, determines the quality of measurements. The measurement weight values should be set in the range from 0 to 1. If there is no information about the measurement errors (i.e. no column with the **Weight** header in the data file), the program will automatically assign weight 1 to each measurement.

The second and subsequent lines in the data file contain the actual data corresponding to each measurement, in the same order as the headers in the first line.

In the case when different array types are used on the same survey line, the * symbol is written instead of the coordinate of the missing (remote) electrode.

If necessary, additional horizontal nodes (data points) can be entered to extend the model area beyond the ends of the spread, e.g., when steep topography is present outside the spread limits. Coordinate (X) of each node is entered in a new line after the *** symbol.

After reading the file, if necessary, resistance values are converted to apparent resistivity using the following formula:

$$\rho = K\frac{|V|}{I},$$

where **K** – geometric factor, **V** – measured voltage, **I** – measured current.

For monitoring (time-lapse) surveys the program accepts files with *.z2d extension which have the following structure: the first four columns contain the electrode coordinates; the following columns with headers res1, res2, etc. contain the **Res** (V/I) values for the first, second, etc. measurement cycles.

Below is a sample of a three-electrode time-lapse survey data file:

C1	*C2*	*P1*	*P2*	*res1*	*res2*	*res3*	*res4*	*res5*
0	*	*2*	*4*	*5.57*	*5.57*	*5.57*	*5.60*	*5.63*
0	*	*4*	*6*	*1.55*	*1.55*	*1.56*	*1.59*	*1.62*
0	*	*6*	*8*	*0.84*	*0.84*	*0.85*	*0.88*	*0.91*
0	*	*8*	*10*	*0.48*	*0.48*	*0.49*	*0.51*	*0.52*
0	*	*10*	*12*	*0.44*	*0.44*	*0.45*	*0.49*	*0.46*
0	*	*12*	*14*	*0.25*	*0.26*	*0.26*	*0.29*	*0.24*

For IP surveys the program accepts files with *.z2d extension which have the following structure.

The first line after the word **time_#chann** contains the list of delay times for the time-domain or frequencies for the frequency-domain method of measurements. Then follows a table, with each line corresponding to a data point, and columns containing electrode coordinates (**C1**, **C2**, **P1**, **P2**), **Res** values (V/I), and IP values at corresponding delay times (column headers – **ipi1**, **ipi2**, etc.) or channels (frequencies) (column headers for absolute value – **mod1**, **mod2**, etc., column headers for phase shift – **pha1**, **pha2**, **etc**.). The **Res** column will be used in the inversion to obtain the resistivity model; geometric factors will be calculated from the electrode coordinates. The values in the **mod1**, **mod2**, etc. columns can be used in the inversion of the IP data (determination of the Cole-Cole model parameters).

A sample of a time-domain data file:

time_#chann		*0.05*	*0.065*	*0.085*	*0.115*	*0.155*			
C1	*C2*	*P1*	*P2*	*res*	*ipi1*	*ipi2*	*ipi3*	*ipi4*	*ipi5*
0	*	*5*	*10*	*0.001*	*0.279*	*0.260*	*0.242*	*0.218*	*0.194*
5	*	*10*	*15*	*0.001*	*0.306*	*0.286*	*0.266*	*0.241*	*0.215*
10	*	*15*	*20*	*0.002*	*0.338*	*0.317*	*0.295*	*0.268*	*0.239*

A sample of a frequency-domain data file:

time_#chann		*0.0001592*		*0.000427*		*0.001145*				
C1	*C2*	*P1*	*P2*	*res*	*mod1*	*mod2*	*mod3*	*pha1*	*pha2*	*pha3*
0	*	*5*	*10*	*0.001*	*0.001*	*0.001*	*0.001*	*-0.004*	*-0.006*	*-0.010*

5	*	*10*	*15*	*0.001*	*0.001*	*0.001*	*0.001*	*-0.005*	*-0.007*	*-0.012*
10	*	*15*	*20*	*0.002*	*0.002*	*0.002*	*0.002*	*-0.005*	*-0.009*	*-0.014*

To assign horizontal coordinates to the survey line, the **geocoord** key should be entered in a new line, followed by coordinates specifying the start and end of the line in the following order: X1 Y1 X2 Y2. For example:

geocoord 441000 7230500 442000 7230500

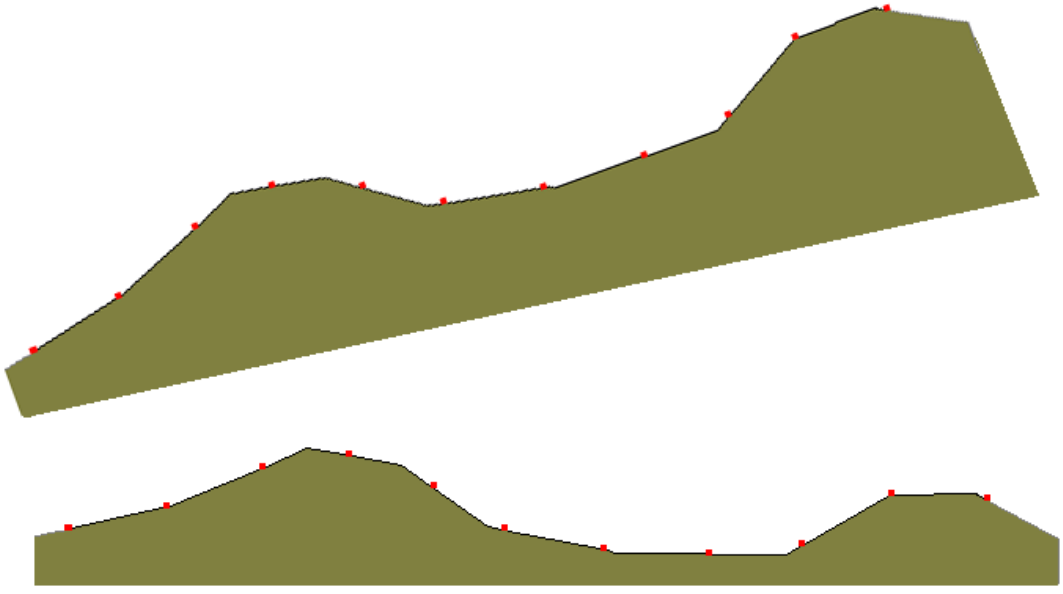
Part II of the data file: topography

If topography data are available, a new line containing the **topo** key is started, followed by a table of coordinates and elevations. When inverting data containing topography, it is recommended to use **Res** (resistance, V/I) as an input parameter.

The X coordinates of the survey line can be specified in two ways – as distances along the line and as horizontal projections. The following **topo** key ending symbols corresponding to different methods of assigning the X-coordinate of a topographic profile can be used:

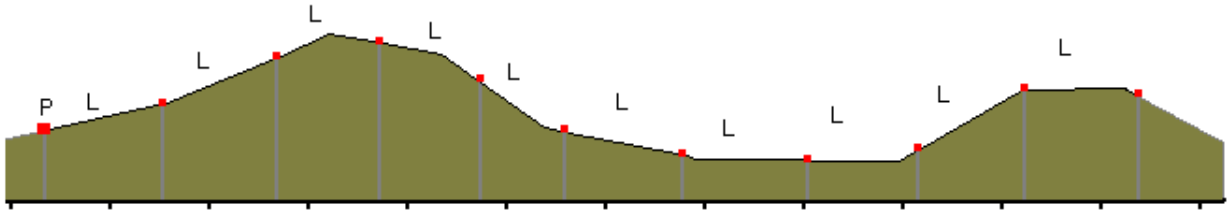
topo (no ending symbol) – electrode and topography coordinates are given in horizontal projections.

topo~ – reduction to a horizontal plane. The topography profile is approximated by a straight line using the least-squares method, then rotated until the line is horizontal (see figure below). This method should be applied when the survey is carried out along a slope with known elevations.



topo# – mixed method when the electrode X coordinates are given in line distances, topography coordinates – in horizontal projections. The horizontal coordinate of an electrode in this case is the distance (L) along the spread (ground length) (see figure below). In the program, the electrode

coordinates are recalculated from distances to horizontal projections. It is necessary to reference one of the electrodes (P) to a specific topographic point. For this, following the **topo#** key, there should be a line containing the X coordinate of the topography profile point and coordinate of the corresponding electrode (line distance), separated by space. Then follows the table of elevations.

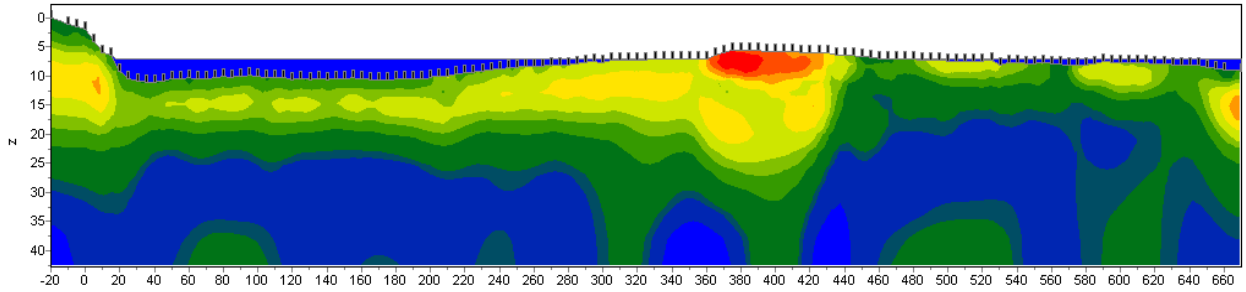


topo% – this key ending is used when **Ro_a** (apparent resistivity) values calculated using line distances are given as input data. The program will recalculate the input data into V/I values, then calculate the apparent resistivity using real coordinates (horizontal projections).

topo^ – this key ending is used when electrode and topography coordinates are given as distances along the survey line.

The above key endings can be combined, e.g. **topo~#**.

topow – this ending is used for interpretation of marine surveys (bottom or surface arrays). In this case, the topography is represented by the bottom profile or the bottom transitioning to the ground surface profile if a combination of marine and ground surveys is carried out (see figure below). In the same line, after the key, it is necessary to specify the elevation of the water surface, the water resistivity and the number of additional water layer partitions (3-10), separated by space (e.g. **topow** 0 100 5). The last two parameters can be changed in the **Options** / **Inversion** / **Underwater options** menu subsection. Then follows the table of elevations. The elevations are relative to the water surface elevation, so that negative values indicate bottom topography, positive values (if any) – ground surface topography. It is possible to create combined systems, in which electrodes are located on the bottom and on the water surface. For this purpose, it is necessary to specify the vertical coordinates of electrodes (**C1z C2z P1z P2z**) relative to the bottom elevations.



topo* – for marine surveys this ending simplifies topography data entry in the case when the spread is on the water surface (towed array).

It is recommended to review additional options described in the **Working with marine data** section.

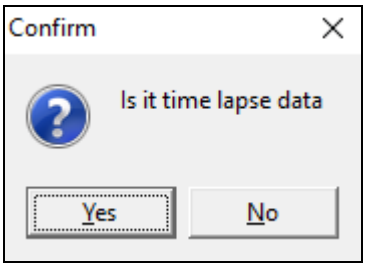
Opening several files

The program allows you to open several data files in *Zond* format (Z2D).

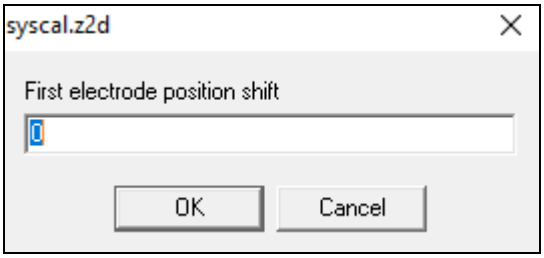
This is used in the following situations:

- when different portions of the survey line are in different files;
- to combine several measurement cycles into one monitoring (time-lapse) file;
- when the data along the survey line were collected using several different arrays.

A dialog box appears when the files are opened:



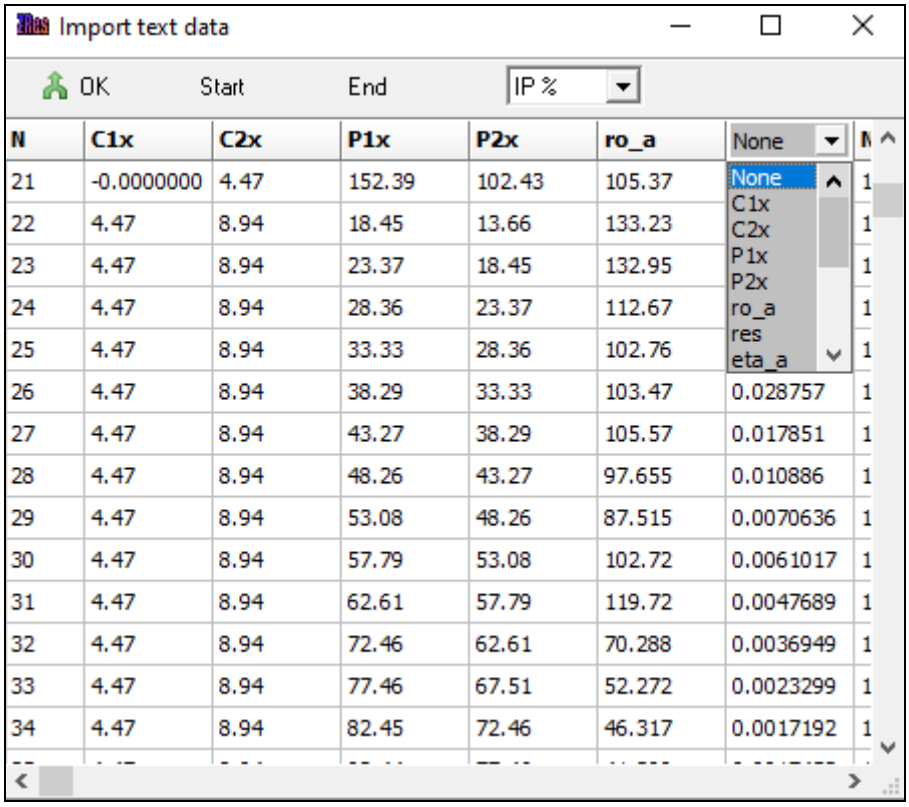
If you choose **Yes**, the program will merge the files into a single time-lapse file. If you choose **No**, a dialog box will appear prompting you to enter the offset for the first electrode:



Import data from a text file or Excel spreadsheet

Importing data from a text file or Excel spreadsheet is performed using the **File / Import data from text/excel** menu command. After selecting the file, the data import dialog box appears (see figure below). Use the **Start** and **End** buttons at the top of the dialog box to specify the range of data to be imported (the **Start** button specifies the first row, the **End** button specifies the last row).

Cells in the uppermost row of the dialog box table contain a drop-down list of headers (electrode coordinates, resistivity or IP data). For each column to be imported, it is necessary to specify the corresponding header. To import the data into the project, press the OK button.



Preparing the data for inversion

The processing of the data before running the inversion includes assessing the quality of field measurements, removing poor-quality data points, assigning weights to measurements depending on their quality, combining several spreads into one line, data filtering and topography input.

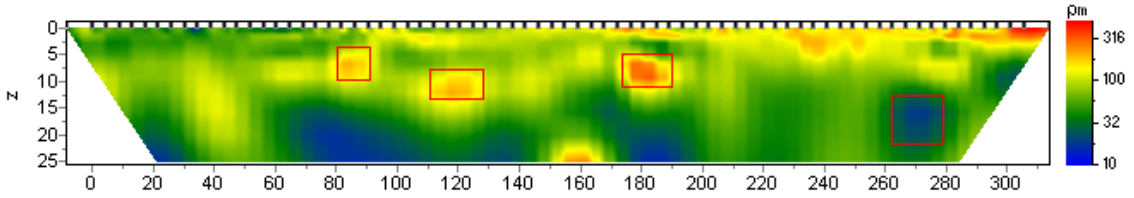
Estimation of quality of ERT data can be implemented at two levels – analysis of standard statistics obtained during the fieldwork and analysis of the distribution of the measured data (voltage and current values). The first level of estimation is typically carried out in the programs controlling the measuring equipment. The second level in the form of quality control procedures and data processing tools is implemented in the **ZondRes2D** program.

ERT operates with much larger data amounts compared to conventional resistivity profiling and sounding. Therefore, it is considered acceptable to use data with some percentage of erroneous readings for the inversion. The important aspect is not the percentage of bad data points, but the distribution of these points in the section. Three situations can be distinguished:

- uncorrelated noise (individual outliers);

- overall high noise level;
- correlated noise in some areas of the section.

In the first case (uncorrelated noise, outliers), the presence of poor-quality measurements may not affect the interpretation results but sometimes leads to a significant distortion of the resulting model. It has been found in a number of study cases that a small percentage of bad data points can lead to the formation of false anomalies in the form of systems of "compensated dipoles" (see figure below), appearing as alternating local anomalies of relatively low and high resistivity in the section. The use of adaptive robust inversion schemes does not help in this situation; removal of bad points from the data set is necessary.



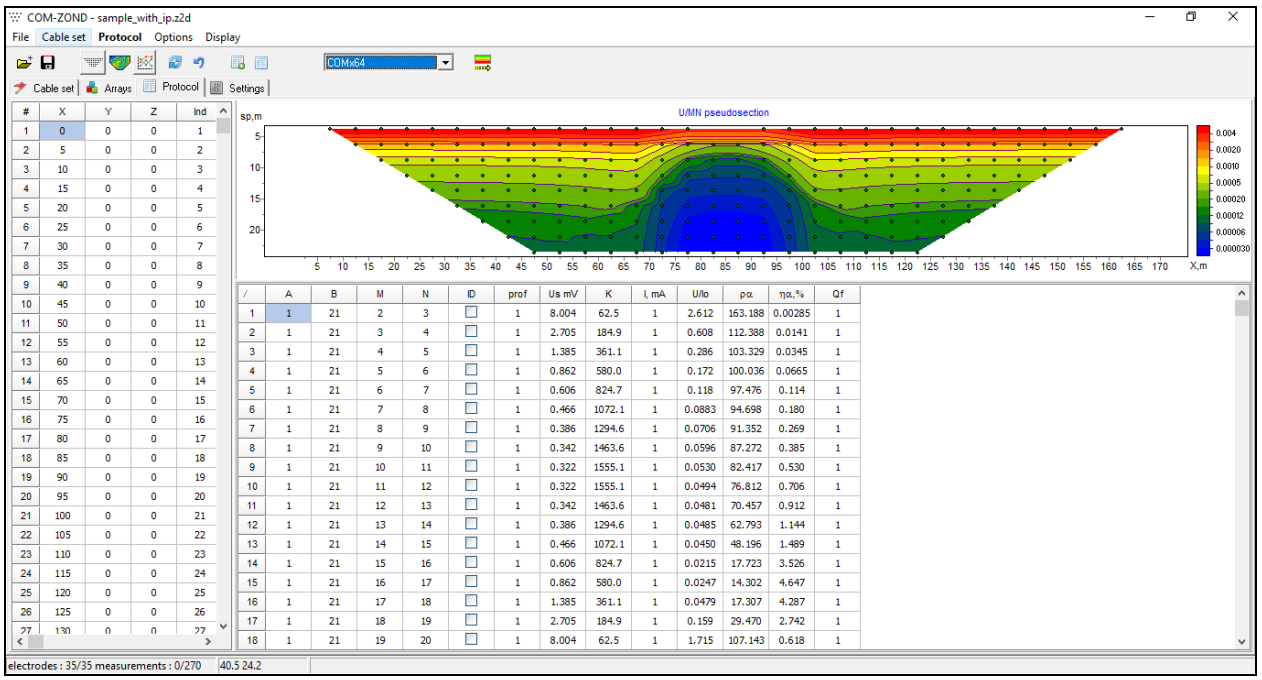
The second case (overall high noise level) requires data filtering. Filtering significantly reduces the noise level but may lead to a decrease in resolution.

The third case (correlated noise in some areas of the section) presents a more challenging problem because it is crucial to understand the cause for the poor data quality in the specific area to choose the appropriate processing tools. For example, bad readings may be caused by poor grounding of some electrodes, systematic external noise, low signal level, etc. Thus, it is necessary first to identify the cause of the poor data quality, and then find a way to eliminate it. For solving this problem, the program provides a variety of data analysis tools.

When assessing data quality and preparing the data for inversion you should keep in mind that a large number of erroneous readings, noisy or missing data in some areas of the section greatly affect the reliability of inversion results.

In **ZondRes2D** the data preparation is carried out using a special module for quality control and data editing – QC module (**Options** / **Quality control module**), the data editor (**Options** / **Data editor**) and graphic plots (**Options** / **Data** / **Graphics-plot**).

The **ZondProtocol** program of the **Zond** package provides an alternative for ERT data analysis and editing. The program performs the following tasks: creation of measurement protocols (automatic and interactive), visualization and analysis of measurement results, separate visualization of different arrays, conversion of data to another type of array, import and export, data editing. Due to the ease of use and full compatibility with the **ZondRes2D** program, **ZondProtocol** allows you to perform quality control directly in the field (the figure below shows the program's interface).



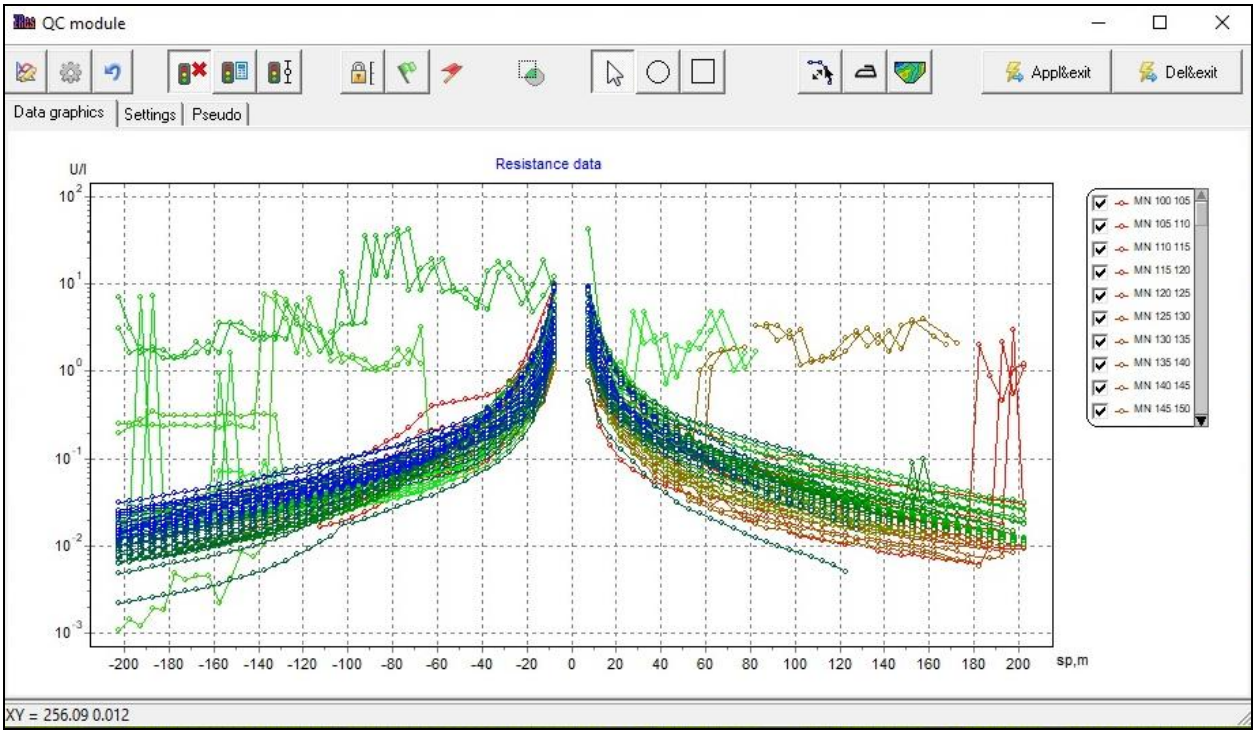
Quality control module

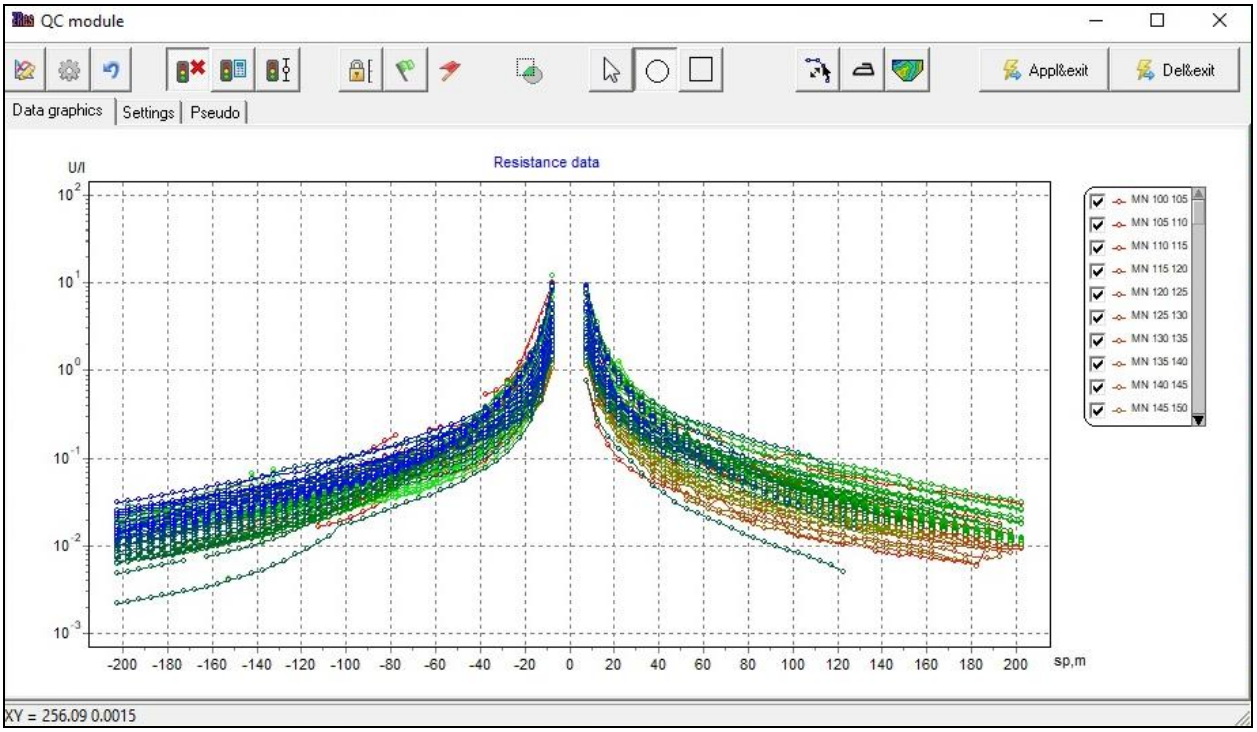
For quality control and data editing, a special module is implemented in **ZondRes2D**, which can be opened with the **Options / Quality control module** command. The module allows you to perform the following operations with data:

- visualize measurement results in the form of graphs for various parameters and in the form of pseudosections for various arrays;
- if possible, perform data quality evaluation (determine measurement weights) based on the principle of reciprocity (for some array types);
- remove bad data points from the graphs.

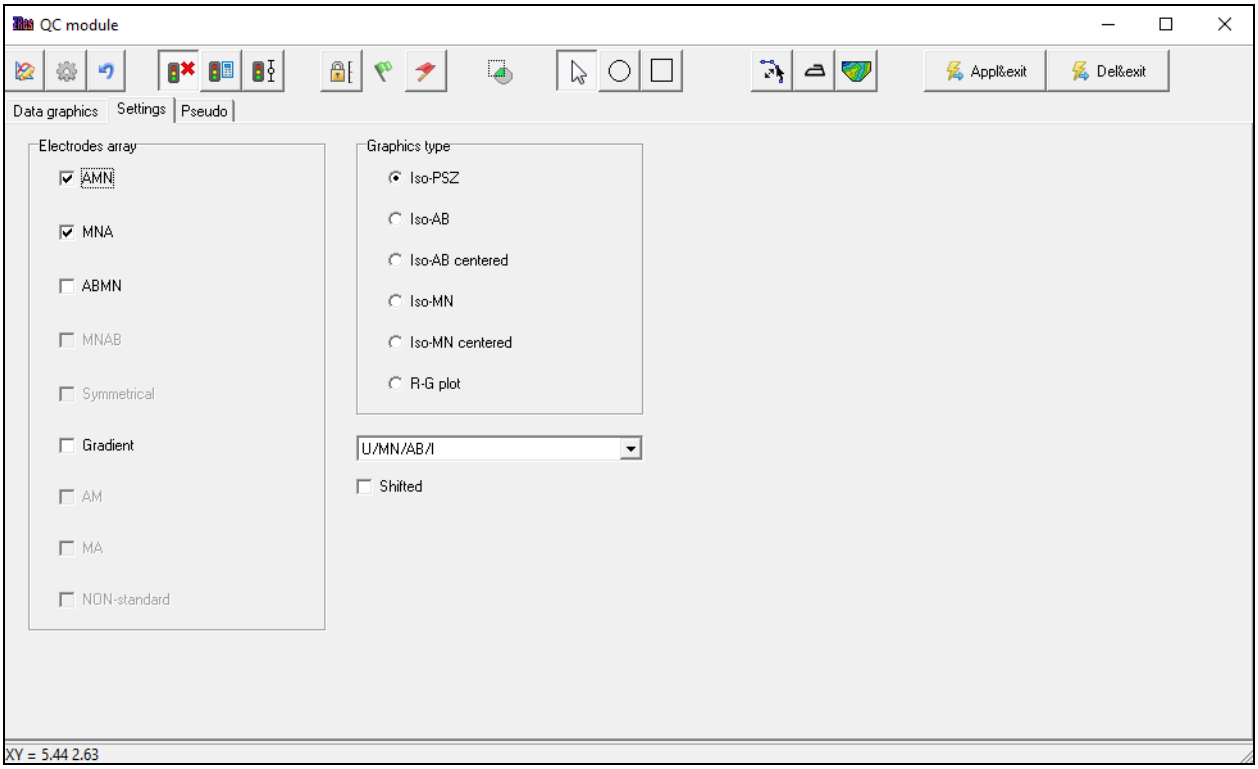
The ability to sort data by the type of array, measured signal, various geometric parameters and measurement weights, combined with manual and automatic bad points removal options, allows you to carry out the entire quality control sequence to prepare the data for inversion.

The **QC module** window consists of a toolbar and three tabs: the **Data graphics** tab displaying graphs of selected parameters (see figure below), the **Settings** tab for selecting the parameters to display, and the **Pseudo** tab displaying pseudosections.

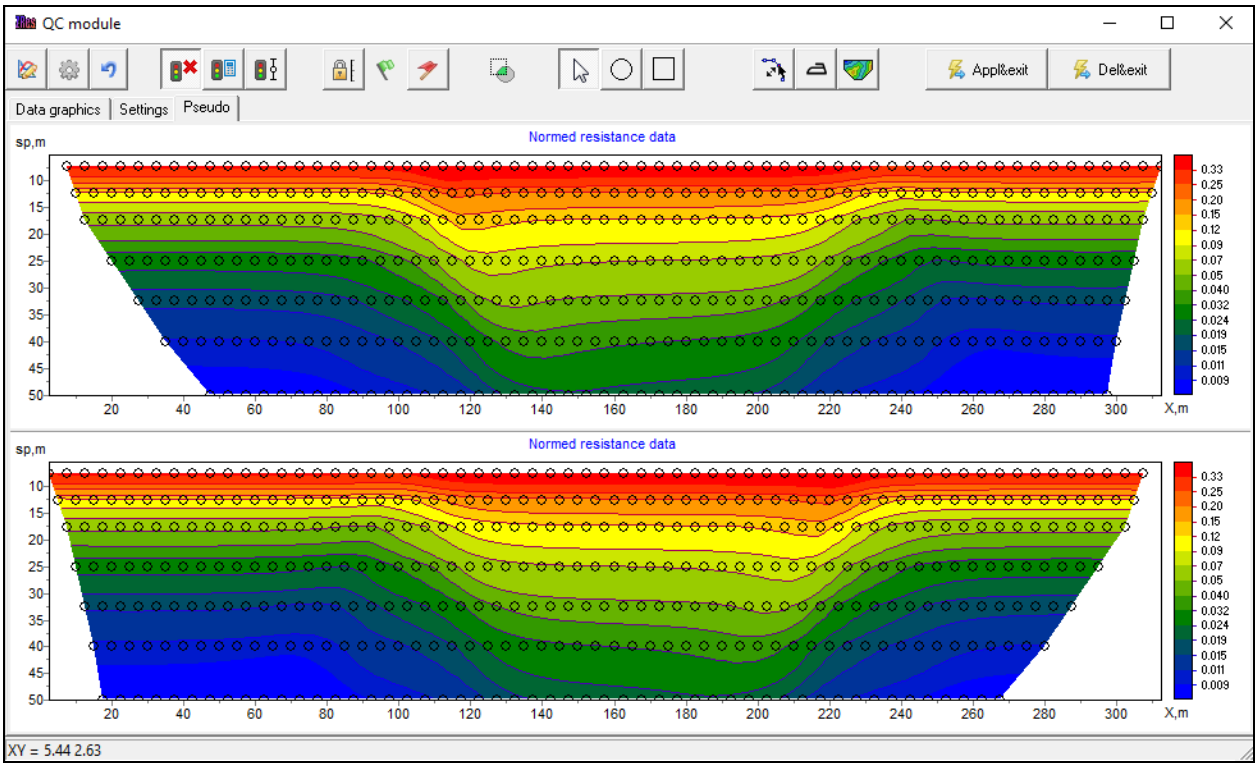




The **Settings** tab allows selecting the type of array for which the graphs will be displayed (the **Electrode array** group box), specify the data grouping parameter – by spacing, by pseudodepth, by the position of current electrodes, potential electrodes, etc. (the **Graphics type** group box), and to select the displayed value – resistance (V/MN/AB/I or V/I), apparent resistivity or apparent polarizability. The ***Shifted*** option, when activated, shifts the graphs by the average value of the displayed parameter.



The **Pseudo** tab displays a pseudosection.



The graphs of selected parameters are displayed in the **Data graphics** tab. In this tab, you can work with the data interactively using the toolbar. Measurements are represented by points (small circles) on the graphs. When selecting a measurement by left-clicking on a point, the position of the electrodes corresponding to the measurement is graphically displayed in the lower portion of

the graphic area (red dots represent current electrodes, blue dots – potential electrodes). When pointing the mouse cursor at a point on the graph, the parameter value corresponding to the point is displayed.

Data quality can be controlled in three ways: 1) based on measurement weights (provided by the measuring equipment or calculated based on the principle of reciprocity), 2) visually by assessing the smoothness of the plots, and 3) based on the comparison of raw data and the solution of forward problem for the inversion model (i.e., observed data and calculated data).

The display of measurement weights is controlled by the toolbar buttons: when the  button is pressed, the weights are not displayed; the  button displays the weights determined based on the principle of reciprocity (if possible for the current acquisition system); the  button displays the weights specified in the data file. The green color of the graph points indicate measurements of good quality, yellow – average, red – bad).

The display of plots calculated for the inversion model is controlled by the  button. The calculated data are plotted with dashed lines. If there is a strong disagreement between the measured and calculated data, there might be several reasons for this:

1. The measured data is susceptible to noise interference. In this case, the noisy data should be deleted.
2. The resistivity variation range is set too narrow. In this case, you can change the model parameter constraints (or do not use any) and repeat the inversion.
3. Three-dimensional effects of geology. In this case, a three-dimensional interpretation is recommended, if the survey grid allows.

To remove bad data points, the point selection mode should be activated by pressing the  button. Points can be selected by right-clicking either on each of them individually using the  tool, or on the area of the plot using the  and  tools. Selected points on the plot are indicated with crosses. The selected points can be hidden/displayed using the  button.

When the  button is pressed, the measurements can be edited by pressing the left mouse button and dragging the point up or down to the desired value.

The  button opens the window for setting graph properties such as line thickness, color, etc. The  button is used to fix the axes scale so that it does not change automatically when working with the data.

The  button gives access to a set of commands for working with the plotted data. The following options are available: select/deselect all points; smooth the selected points; set weights for selected points; change the sign (positive or negative) of selected points (sometimes required for cross-hole surveys); option to always connect all the points in a graph; display the confidence interval for each point (if the data file contains these data).

The  button performs smoothing of the data. Subsequently pressing the button increases the degree of smoothing. Typically, smoothing is not recommended because it distorts the data. This option should only be used in extreme cases when the data is heavily influenced by noise.

The  button is used to update the pseudosection plot.

The last executed command can be undone with the  button.

Changes to the data made in the QC module will not be applied if the module is closed using the X button in the top right corner. To apply changes and close the window, press the 

button. The data points that have been selected will be excluded from the inversion data set; however, these data points will remain in the original data set and will be available for further editing. The 

button is used to delete the selected data points from the data set completely. Right-click on this button allows selecting the following options:

Remove Rho&IP **–** when apparent resistivity data are edited, with this option enabled, resistivity and polarizability values of selected data points will be removed.

Remove all IP chan **–** removes values of all polarizability channels for the selected points.

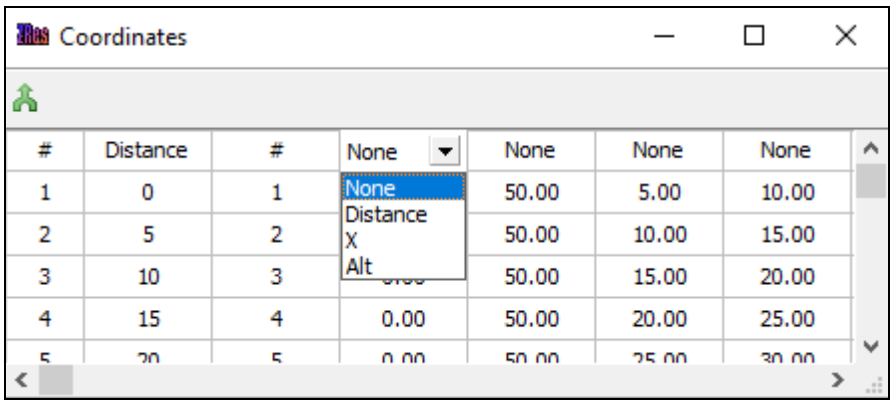
Additionally, the **Options / Extra / Remove data with big misfit** menu command allows you to quickly remove data points with higher misfits (after inversion).

Topography input and editing

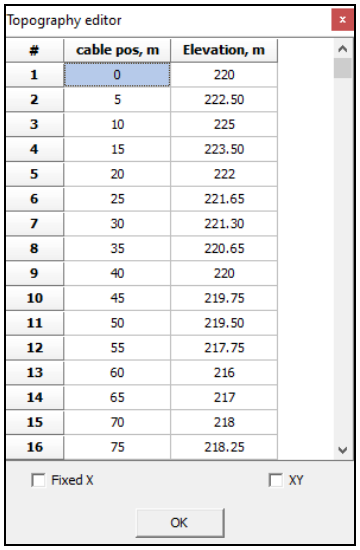
There are several ways to input topography data in **ZondRes2D**:

- via a Z2D data file (for more information, see the **Supported formats** section);
- by importing the data from a text file using the **Options / Topography / Import topography** menu command.

The **Options / Topography / Import topography** command prompts to select a text file containing the topography data. After selecting the file, the data import window appears (see figure below). The cells in the uppermost row of the table contain a drop-down list of headers: **Distance** (distance along the line), **X** (projected horizontal coordinate) and **Alt** (elevation). For each column of the imported table, it is necessary to specify the corresponding header. To import the data into the project, press the  button.



The **Options** / **Topography** / **Edit topography** command opens the topography editor which also can be used to input the topography data.



The Ctrl+X combination clears the *Elevation* column. You can then copy the data from a spreadsheet (e.g., Excel) and paste it into this column using the Ctrl+V combination.

If the ***Fixed X*** checkbox is checked, the X coordinates (**Cable pos**) will not change after the input of elevations, if unchecked, the X coordinates will be recalculated to projected coordinates.

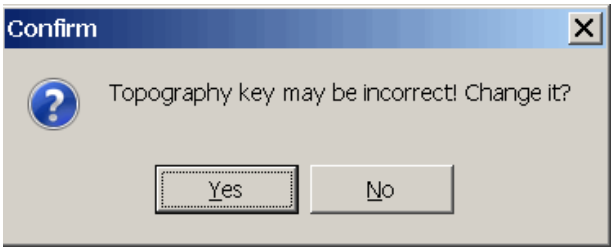
The ***XY*** option allows adding XY coordinates (e.g., in the UTM projection) to the table.

The program will analyze the topography data and warn if the input elevation data are irrational, which typically happens when incorrect keys (column headers) have been selected. Below are the

two cases when the program will detect incorrect input (wrong key/header) and suggest correcting it.

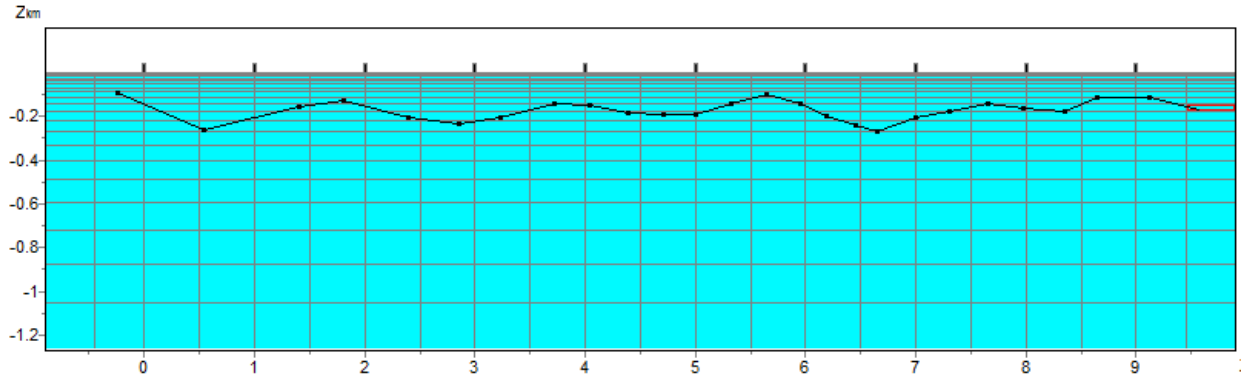
1. When the distances (lengths) between the electrodes along the line specified in the data file are smaller than the real (projected) horizontal distances.
2. When the elevation difference between two neighboring electrodes is greater than the distance between the electrodes.

A dialog box will appear suggesting to correct the input. Selecting **No** will leave the input unchanged, selecting **Yes** will convert the input data to the correct distance-elevation coordinate system.



Using the **Options** / **Topography** / **Remove topography** and **Restore topography** commands allows to you remove and restore the topography data, respectively.

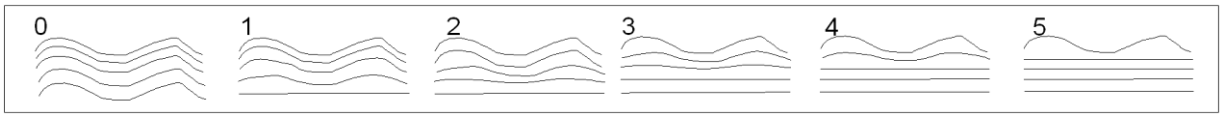
The manual topography input mode (**Options / Topography /Set by mouse**) can be useful for test purposes (generally, it is too coarse). The process of adding the topography profile is similar to adding a priori boundaries.



The coefficient of topography distortion with depth (values from 0 to 5) can be set using the **Option / Topography / Topo coefficient** menu command. If set to 0, the profile of each subsequent mesh layer will have the same geometry as the first one. If set to 1, the topography flattens with depth, and the last mesh layer will be flat (see figure below). The distorted depth is calculated using the following formula:

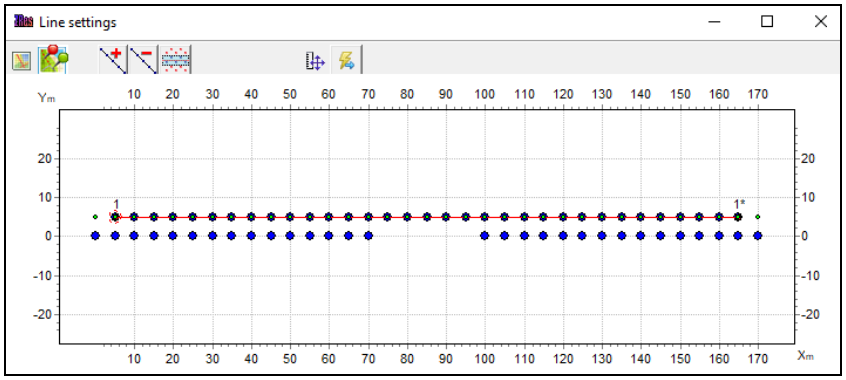
$$z(x,z)=\mathrm{Topo}(x)+z\cdot\left(1+\frac{\max(\mathrm{Topo})-\mathrm{Topo}(x)}{\max(z)}\cdot\mathrm{Tcoeff}\right),$$

where **Topo** is elevation, and *z* is depth below the ground surface.



Editing electrode coordinates in the plan view

During ERT surveys, the electrodes are not always located directly on the survey line. This can happen due to difficult terrain, or in the case when it is necessary to physically separate the current line from the potential line. To account for non-collinear electrode spreads, **ZondRes2D** has a special module that is started automatically when a data file containing Y coordinates is opened. The module allows you to assign a single line for the measurement profile, exclude unnecessary electrodes from the interpretation and project the measurement results on this line.



In the module window, the electrode locations are plotted on the XY plane according to their X (along the survey line) and Y (perpendicular to the line) coordinates specified in the data file. The position of any electrode can be edited in the dialog box opened by right-clicking on the electrode point. The  button on the toolbar allows you to import a raster image as a base map. The  button allows you to import a Bing map as a base layer (in this case coordinates in the input data file should be in the UTM coordinate system).

After the actual XY positions of electrodes are plotted it is necessary to draw the profile projection line. The  button enables/disables the line drawing mode. Points are added to the line by left-clicking on the XY plane; right-click on the plane to finalize the line plotting. Right-click

on the  button brings up a context menu, which allows you to set the coordinates of the line, snap the line points to the electrode locations, invert the line, and plot the line connecting the electrode points automatically. The line can be deleted by pressing the  button.

The next step is to select the electrode points that will be projected on the profile projection line. Select (or deselect) a single electrode by left-clicking on the point; the color of the selected point turns blue. To select all the points, press the  button on the toolbar. When the electrode is selected, its position on the profile is indicated by a green dot.

The scale of the plot can be changed by pressing the  button.

After completing the above operations, proceed to the mesh configuration settings and interpretation by pressing the  button.

Another application of the plan view module is to arrange data sets of several survey lines collected on a grid. If the XY electrode coordinates of all lines are specified in the data, you will be able to derive the data of any particular line.

Working with marine data

When interpreting marine (aquatic) survey data, it is preferred to know the water resistivity; if the water resistivity is unknown, it is necessary to enable the option to determine the water resistivity during the inversion (**Options / Inversion / Underwater options / Invert**).

The water resistivity can be set using the **Options/Inversion/Underwater options /Resistivity** command.

Using the **Options / Inversion / Underwater options / Sub layers number** option you can set the number of water layer partitions**.** The number of partitions is set based on the thickness of the water column.

Working with borehole data

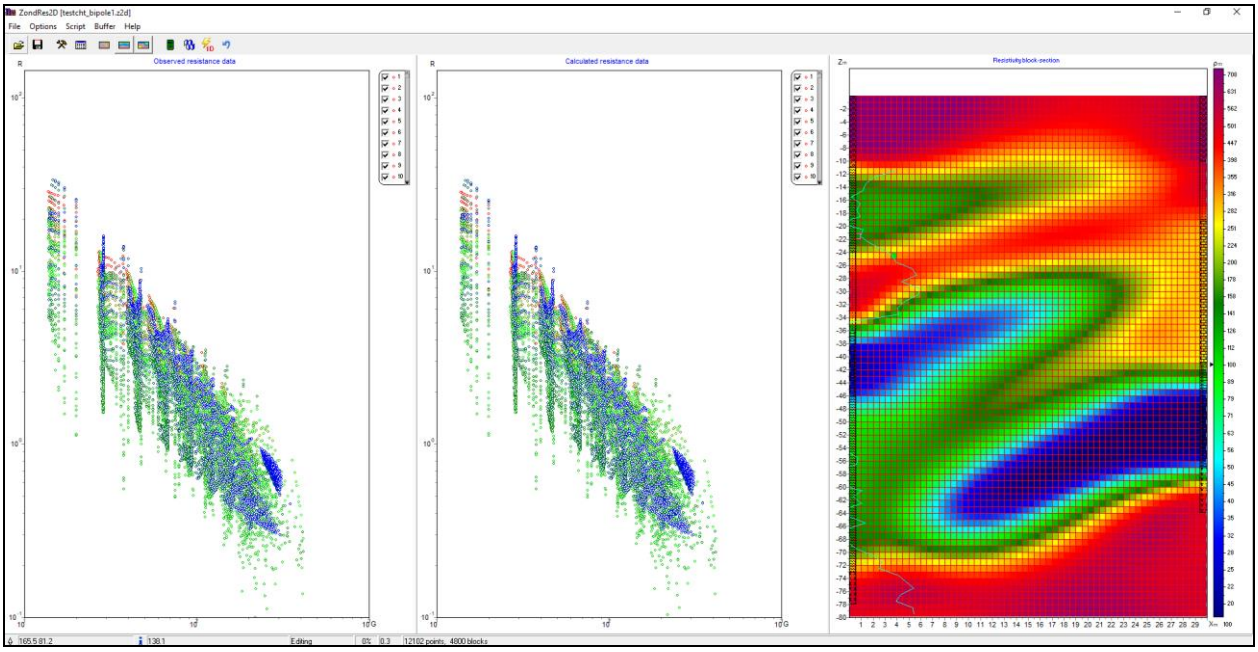
ZondRes2D allows the interpretation of borehole resistivity data. This includes cross-hole measurements, borehole-to-surface measurements and their combinations. Both vertical and inclined boreholes are supported.

Similar to land observations, horizontal and vertical coordinates of electrodes located in boreholes should be specified in the data file.

Modeling and inversion of cross-hole and borehole-to-surface resistivity data are carried out in the same way as for land ERT surveys. The difference is in the way the apparent resistivity is

presented – unlike conventional representation of land ERT data, for cross-hole surveys, the apparent resistivity is plotted as a function of geometric factor. The observed and calculated data can be viewed in a separate window accessible with the **Options / Data / G-Res plot** command.

During cross-hole surveys, measured apparent resistivity values may be negative. In this case, it is not possible to apply the logarithmic normalization to the data (although it is possible to apply it to absolute values). For such cases, the program uses a pseudo-logarithmic normalization that requires specifying a linear-logarithmic coefficient (its value should be close to the noise level). If you set it to -1, the coefficient will be determined automatically. The **Options / Inversion / Norms / Automatic sres scale** option becomes available when the **Sres** key is used in the data file.



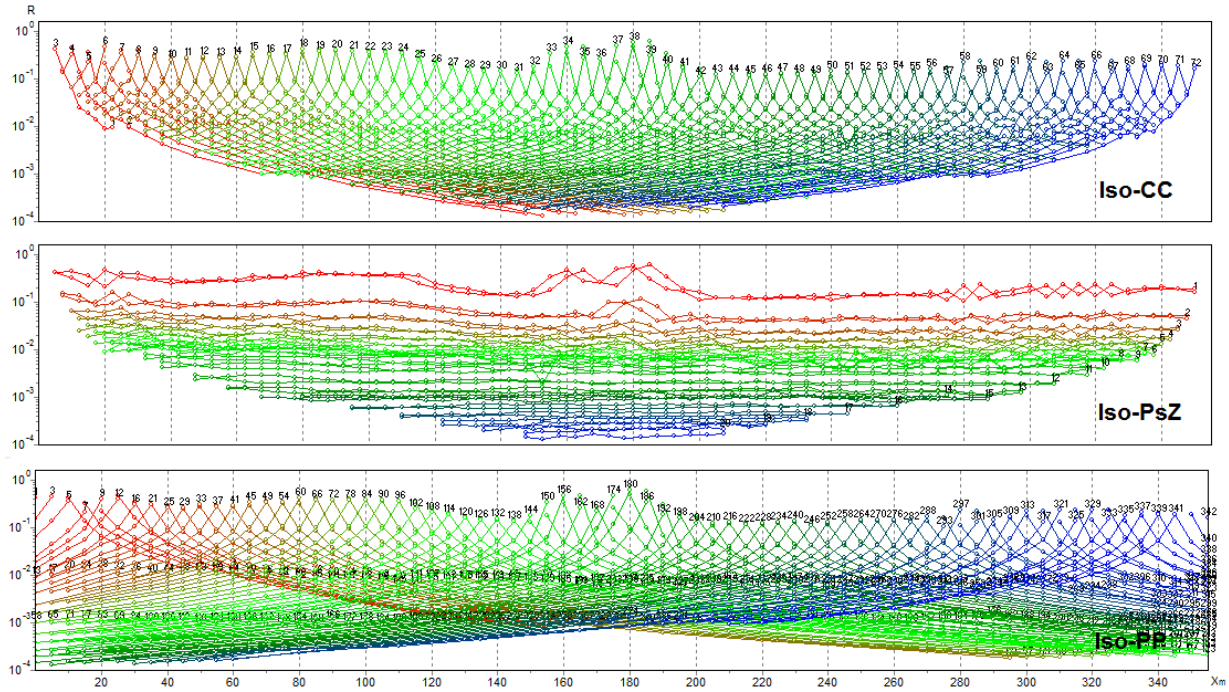
Data visualization

In **ZondRes2D** the data can be visualized in the form of graphs (**Options** / **Data** / **Graphics-plot)** and in the form of pseudosection (**Options** / **Data** / **Pseudo-section**). For ERT surveys, the data can be displayed in resistance values (**Option** / **Data** / **Resistance (V/I)**) or in apparent resistivity values (**Option / Data / Apparent resistivity**).

Graphic plot

The graphic plot (**Options** / **Data** / **Graphics-plot**) displays the data values along the survey line in the form of graphs.

In the **Options** / **Data** / **Graphics-plot** menu subsection you can choose to plot graphs for a specific geometric factor or pseudodepth (**Options** / **Data** / **Graphics-plot** / **Iso-PsZ graphics**), for a specific position of current electrodes (**Options** / **Data** / **Graphics-plot** / **Iso-CC graphics**), or potential electrodes (**Options** / **Data** / **Graphics-plot** / **Iso-PP graphics**).



Zooming in on a portion of the graphic area is done by pressing and dragging the mouse. To zoom in, press the left mouse button and drag the mouse up/down and to the right while holding down the button. To zoom out and return to the original scale, press the left mouse button and drag the mouse up/down and to the left. Moving (scrolling) the graphic area is done by moving the mouse with the right button pressed.

Pressing the left mouse button with the cursor pointed at a graph point clears the other graphs and displays the electrode positions for the selected point until the mouse button is released.

Graph properties such as line thickness, color, etc. can be changed in the **Graphics Setup** window accessible with the **Options** / **Graphics** / **Observed graphics** or **Calculated graphics** commands.

To scroll through several graphs, select several adjacent graphs in the legend and spin the mouse wheel, with the mouse pointer hovering over the legend. The indexes of the active graphs will change.

If the **Data editor** window is opened, right-click on a graph point will highlight the corresponding measurement in the table.

If measurement weights are specified in the input data file, you can display the confidence intervals on the graphs using the **Options** / **Data** / **Display** / **Error gates** command. The

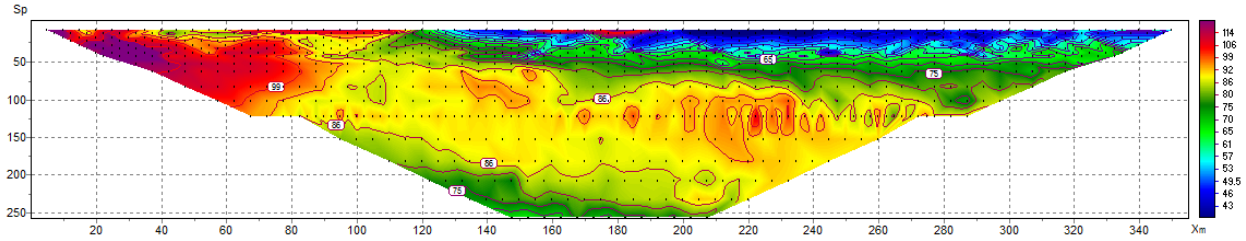
confidence intervals (weights) can be adjusted on the graphic plot by pressing the left or right mouse button while holding down the Alt key.

Pseudosection

The pseudosection is a distribution of data along the survey line in the form of a contour section.

Coordinates of the data points along the survey line are plotted on the horizontal axis of the pseudosection. On the vertical axis, pseudodepths or electrode spacings are plotted.

The color scale is displayed to the right of the pseudosection. The color scale represents the assignment of colors to the plotted data values.



Right-clicking on the axis area brings up a context menu with the following options:

Log data scale
Use the logarithmic scale for the color scale.
Smooth mode
Use a smooth interpolation palette.
Display grid point
Show/hide measurement points.
Display ColorBar
Show/hide the color scale.
Setup
Open the Contour-section setup dialog box.
Print preview
Print the pseudosection.
Save picture
Export the pseudosection to a BMP, EMF or PNG file.
Save XYZ file
Export the pseudosection to a DAT text file.
Draw in surfer
Plot the pseudosection in Surfer.

The **Axis Editor** is opened by right-clicking on the axis while holding down the Shift key (when hovering over the axis, the mouse pointer turns into a hand pointer). A context menu containing three items (**Options**, **Default**, and **Fix range**) will appear (for more information, see the **Axis editor** section).

The dialog box for changing data point graphical properties is opened by right-clicking on a point while holding down the Shift key.

The method of plotting the pseudosection is different for different types of arrays:

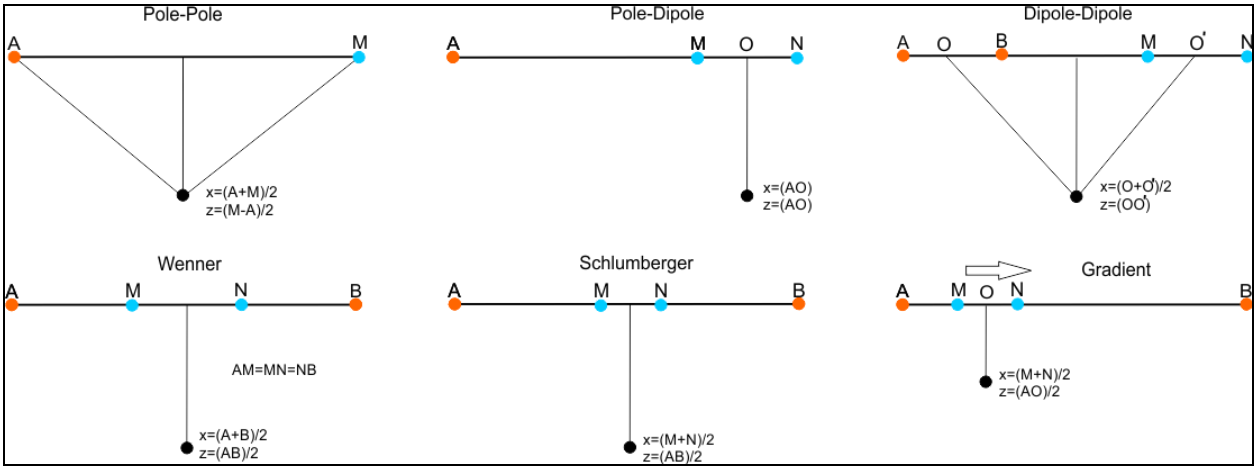
- For a *Wenner* array, the X coordinate corresponds to the center of the array; the Z coordinate corresponds to ¼ the distance between the outermost electrode to the center of the array.
- For a *Pole-Dipole* array, the Z coordinate corresponds to the spacing (distance from the current electrode to the midpoint between the potential electrodes). For the X coordinate, the program provides two options:

Options / Graphics / Data X (3-electrodes)
A set of options for determining the data point X position for a three-electrode array. **(A+M)/2** – the data point is at the midpoint between the current electrode A and the nearest potential electrode M; **(M+N)/2** – the data point is at the midpoint between potential electrodes M and N.

- For a *Dipole-Dipole* array, the Z coordinate corresponds to ½ the distance between the centers of the current and potential dipoles. For the X coordinate, the program provides three options:

Options / Graphics / Data X (4-electrodes)
A set of options for determining the data point X position for a four-electrode array. **Auto** – automatic determination; **M-O-N** – the data point is at the midpoint between potential electrodes M and N; **AB-O-MN** – the data point is at the midpoint between current (AB) and potential (MN) dipoles.

- For a *Gradient* array, the X coordinate corresponds to the midpoint between the potential electrodes; the Z coordinate corresponds to ½ the distance from the nearest current electrode to the midpoint between the potential electrodes.
- For a *Pole-Pole* array, the X coordinate corresponds to the center of the array; the Z coordinate corresponds to ½ the distance between the electrodes.
- For a *Schlumberger* array, the X coordinate corresponds to the center of the array; the Z coordinate corresponds to ½ the distance between the outermost (current) electrodes.



If the data set contains reciprocal measurement systems, the data of forward and reverse arrays can be displayed separately using the **Options** / **Data** / **Display** / **Forward array** or **Backward array** commands. For large data sets, the ***Display every N point*** option can be used to prevent overloading the pseudo-section. By default, this option is enabled when the data file contains more than 3,000 data points.

In the middle section of the program workspace which displays the calculated data, you can display measurement weights (**Options** / **Data** / **Data Weights**) and relative misfits (RMS errors) for each measurement (**Options** / **Data** / **Data Misfit**).

Data Editor

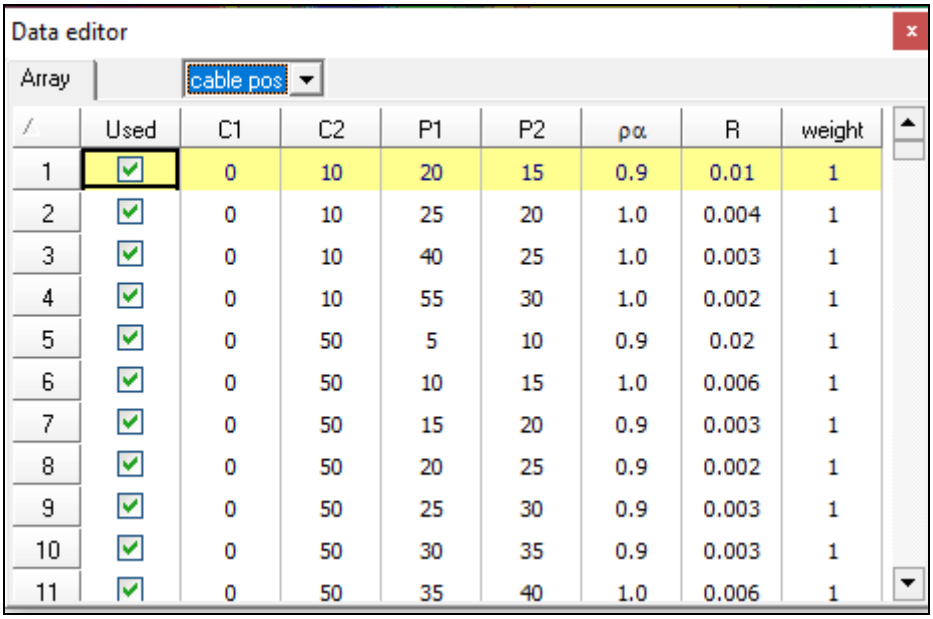
The data editor, which can be opened with the  button on the toolbar, is used for viewing and editing the observation parameters and measured data. The data editor window contains a table consisting of nine columns:

ID
Measurement index.
Used
Indicates whether the measurement is used in the inversion.
C1
Coordinate of the first current electrode.
C2
Coordinate of the second current electrode.
P1
Coordinate of the first potential electrode.

P2
Coordinate of the second potential electrode.
ρα
Apparent resistivity value.
R
Resistance value.
Weight
Measurement weight.

If one of the electrodes is absent, its coordinate is replaced by the * symbol.

The data in the last three columns, if necessary, can be edited using keyboard input. No absurd values should be entered for apparent resistivity or resistance. The measurement weights are set in the range from 0 to 1. By clicking on a column header, the table can be sorted by the values of that column.



/	Used	C1	C2	P1	P2	ρα	R	weight
1	☑	0	10	20	15	0.9	0.01	1
2	☑	0	10	25	20	1.0	0.004	1
3	☑	0	10	40	25	1.0	0.003	1
4	☑	0	10	55	30	1.0	0.002	1
5	☑	0	50	5	10	0.9	0.02	1
6	☑	0	50	10	15	1.0	0.006	1
7	☑	0	50	15	20	0.9	0.003	1
8	☑	0	50	20	25	0.9	0.002	1
9	☑	0	50	25	30	0.9	0.003	1
10	☑	0	50	30	35	0.9	0.003	1
11	☑	0	50	35	40	1.0	0.006	1

The electrode coordinates can be displayed as distances along the survey line (**Cable pos**) or as real coordinates (horizontal projections) **X**.

When you select a data point in the table, the corresponding electrodes and position of the data point are indicated on the pseudosection or graphic plot.

The data editor is also displayed when using the options from the **Options** / **Extra** / **Potential&sensitivity in model** menu subsection.

Modeling

Modeling is an important process preceding fieldwork. Using a priori information about the target object, its physical properties and parameters of the host medium, the geophysicist can define a theoretical model corresponding to the expected geological setting. Through the modeling routine, the expected signal level and resolution of various measurement systems can be assessed, allowing to optimize the fieldwork technique. This approach is considered professional.

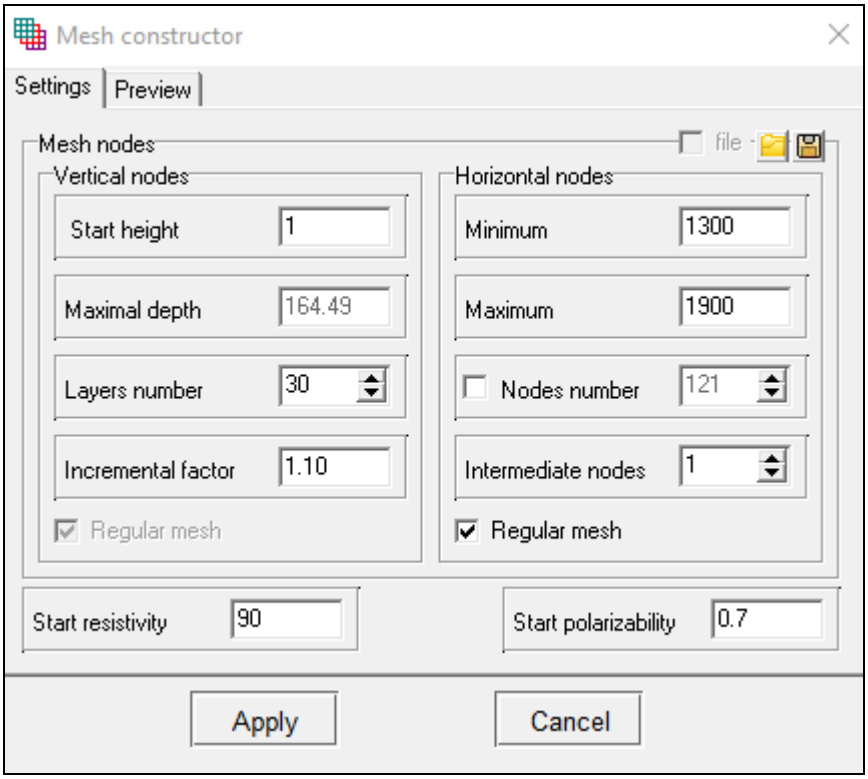
ZondRes2D allows you to model the data of ERT and IP surveys, with the account for the topography of the survey line. The input data for the modeling are the geometry of the measurement system and the resistivity model.

Creating a mesh model

After creating a new measurement system for modeling or opening a data file, a dialog box for setting up the initial model mesh appears.

To create a new model or change the parameters of an existing model, you can use the **Options / Mesh constructor** command.

In the **Mesh Constructor** dialog box, you are prompted to select the mesh parameters and the resistivity and polarizability of the model.



The **Vertical nodes** group box contains options that allow you to set the parameters of the model mesh in the vertical direction. Initially, the program automatically selects these parameters in accordance with the following rules:

- The depth to the deepest layer equals half the maximum pseudodepth of the measurement system. The pseudodepth is determined by the electrode spacings.
- The number of layers equals twice the number of unique geometric factors for a given measurement system, but does not exceed 40;
- The thickness of each consequent layer is ~ 1.1 times the thickness of the previous layer.

You can edit the default parameters of the initial model.

Start height **–** sets the thickness of the first layer. The value should correspond approximately to the horizontal size of the cell and meet the required resolution of the survey.

Maximal depth **–** indicates the depth of the deepest layer. Note that the maximum depth should not be overly deep because the influence of geoelectric parameters decreases with depth. This parameter is calculated automatically.

Layers number **–** sets the number of model layers. According to the physical context, it should not exceed significantly the number of unique geometric factors (spacings) for a given measurement system. Typically, 12-20 layers are enough for a comprehensive model. A large number of layers will significantly reduce the speed of calculations.

Incremental factor **–** sets the ratio between the thicknesses of adjacent layers. The values are typically chosen in the range from 1 to 2.

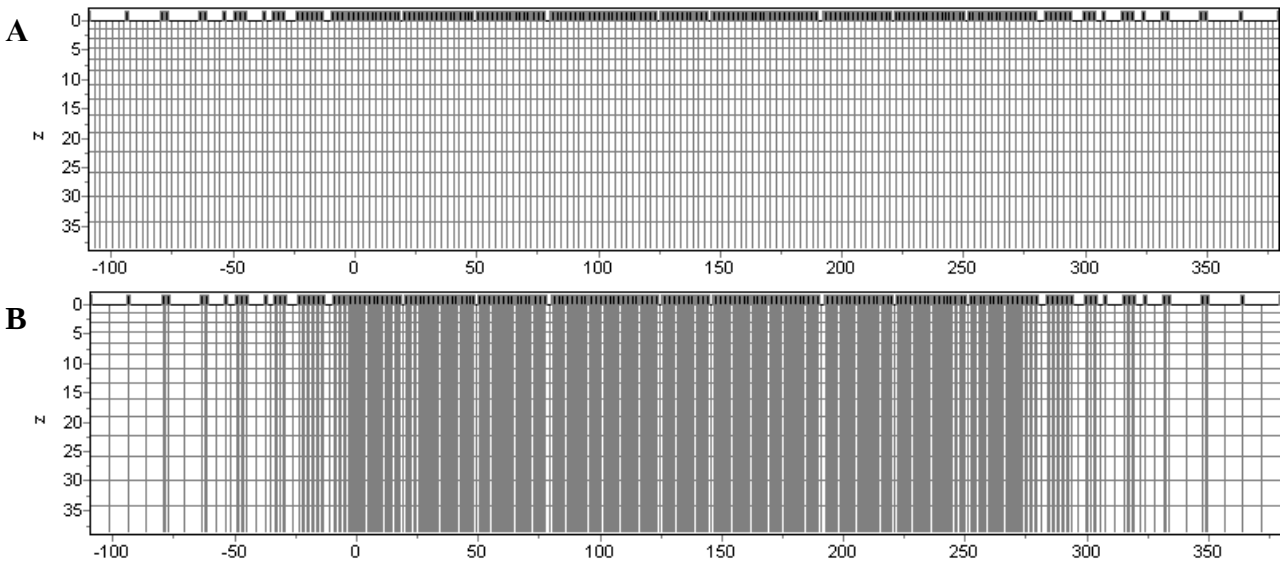
The **Horizontal nodes** group box contains options for defining the model mesh parameters in the horizontal direction.

Minimum **–** specifies the minimum horizontal coordinate of the survey line.

Maximum **–** specifies the maximum horizontal coordinate of the survey line.

Intermediate nodes **–** sets the number (0-4) of additional nodes between unique electrode positions on the survey line. The presence of additional nodes increases the accuracy of the forward problem solution, especially when using the total potential calculation.

Nodes number **–** if this option is enabled, the mesh is constructed with a constant (horizontal) step, from the value in the ***Minimum*** field to the value in the ***Maximum*** field (example A in the figure below). The number of nodes is specified in the ***Nodes number*** field. It is recommended to use this option in the case of irregular (non-tomographic) measurement systems, e.g., for 2D interpretation of VES data.



With the ***Nodes number*** checkbox activated, the number of cells is reduced significantly, but the algorithm neglects the measurements in the case when the current and potential electrodes are in the same cell due to the nonlinearity of the potential function.

Regular mesh **–** activates the mesh constructing algorithm in which additional nodes are created based on the condition of the uniform subdivision. The option should be enabled in the case when the spacing between adjacent electrodes varies significantly (it will increase the accuracy of the forward and inverse problem solution). By right-clicking on the area with the ***Regular mesh*** label you can specify the step of cell subdivision in the horizontal direction (the ***Nodes number*** option should be disabled).

When working with cross-hole data, the ***Regular mesh*** option is available for vertical direction as well allowing the uniform subdivision in both directions. The ***In elec*** option, available when working with cross-hole data, provides matching of cell nodes and borehole electrodes.

Start resistivity **–** sets the resistivity of the initial model.

Start polarizability **–** sets the polarizability of the initial model.

The mesh parameters can be saved and loaded using the  and  buttons located in the upper right corner of the dialog box.

The **Preview** tab allows viewing the plot of the configured mesh.

After configuring the mesh, press **Apply** to proceed to the working mode.

Editing the model

ZondRes2D provides the following three modeling modes:

- **Block (mesh)** mode (the  button on the main window toolbar) is intended for direct editing of the model cells (the width and thickness of a row or column of cells, as well as cell properties). When working in block mode, it is also possible to edit the mesh geometry initially created in the **Mesh constructor**.
- **Arbitrary layered** mode (the  button on the toolbar) is intended for creating and editing layered models;
- **Polygonal** mode (the  button on the toolbar) is intended for creating and editing models consisting of a set of connected or disconnected objects.

In each of the model modes, various auxiliary tools facilitating the modeling process are available, such as the import of borehole logs, base images, models from other programs of the **Zond** package.

Block (mesh) model mode

This mode is used to interactively change parameters of individual model cells or groups of cells. Working with the model cells is similar to editing a raster image in graphical editors. The active cell in the model is highlighted with a rectangle – cursor. When you move the cursor in the model area, the status bar at the bottom of the main program window displays the coordinates and parameters of the cell in which the cursor is located. A selected or fixed cell is indicated with a fill of white or black dots, respectively.

There are several sets of tools for working with the block model available in the program: the context menu (brought up by right-clicking in the model area), the floating toolbar (opened with the **Options** / **Model** / **Model Editor toolbar** command) and the color scale. The functions of the context menu and the floating toolbar generally duplicate each other.

Context menu item	***Toolbar item***	***Action***
		Zoom in by pressing the left mouse button and dragging the mouse up /down and to the right. To zoom out, drag the mouse in the opposite direction.
Display cell setup		Open the Cell Setup dialog box.
Cell to cursor value		Use the parameter of the active cell as the current value.

Edit mode		Enable editing mode.
Selection / Free form selection		Select an arbitrary group of cells.
Selection / Rectangular selection		Select a rectangular group of cells.
		Select an elliptical group of cells.
Selection / Magic wand		Select a group of cells based on the similarity of the cells' parameter values. The active cell and the adjacent cells with similar parameter values are selected. The degree of similarity is set in the Model setup dialog box (right-click on the color scale or the title of the model section and select Setup). This action can also be performed by pressing the X key.
Selection / Remove selection		Remove selection.
Mesh options / Add column/row		Add a new vertical or horizontal mesh boundary. The new boundary appears when you click on the desired location.
Mesh options / Remove column/row		Remove a mesh boundary by clicking on the boundary.
Mesh options / Resize column/row		Change the thickness of a row or column using the mouse.
Clear model		Clear the model completely. Remove all selections and reset the model to the initial settings specified in the Mesh constructor.
Clear parameters		Reset the model to the initial settings specified in the Mesh constructor. The selections made and the parameter limits specified by the user are retained.

The color scale to the right of the model section indicates the colors assigned to specific data values. To select a current value, left-click on the scale; the value is displayed below the color scale. To assign the current value to a cell, left-click on the cell.

The program provides several methods of cell selection – rectangle, ellipse, free-form and by a certain parameter value (the "Magic wand" tool), accessible through the context menu or floating toolbar. If some area of the model section is selected, selecting the current value on the color scale and then left-clicking on any place in the selected area sets all cells of the area to the current value. For example, to give a rectangular area of the model section a certain resistivity value, select the desired area using the rectangular selection tool, then left-click on the color scale at the desired resistivity value, and left-click on any cell of the selected area of the model.

The **Zoom&Scroll** mode is enabled by clicking the  button on the floating toolbar or by selecting the **Setup** option in the context menu brought up by right-clicking on the color scale or on the **Resistivity block-section** title of the model section. To zoom in on a portion of the model area, press the left mouse button and drag the mouse up/down and to the right. To zoom out, drag the mouse in the opposite direction.

In the **Edit mode** (activated with the  button), left-clicking on a cell or selection while holding down the Shift key increases its parameter. Right-clicking on a cell or selection while holding down the Shift key decreases its parameter. The percentage by which the value is changed is specified in the **Model Setup** dialog box (see the **Model section settings** section).

Pressing and holding down a mouse button while holding down the [Ctrl] key allows you to move a selected group of cells within the editing area. With the left mouse button pressed, the contents of the selection are copied to a new location. With the right mouse button pressed, the contents of the selection are cut and copied to a new location.

When switching from the resistivity mode to the polarizability mode, the selections made are retained.

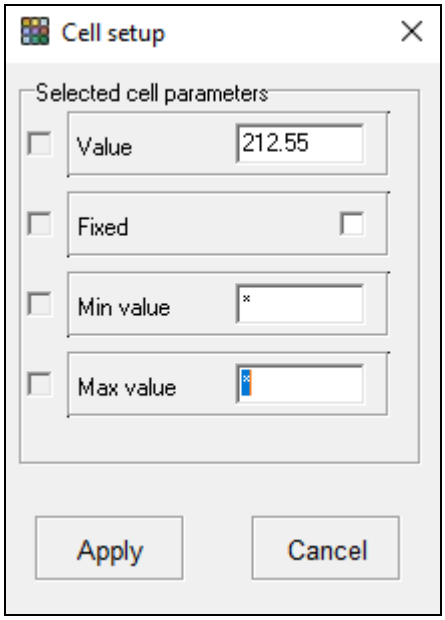
Parameter values for selected cells can be set or edited using the **Cell setup** dialog box. The following options are available:

***Value* –** sets the value of the cell parameter.

***Fixed* –** fixes the cell parameter.

Min value, ***Max value* –** defines the range of variation of the cell parameter for inversion.

If the checkboxes to the left of the option boxes are checked, the changes will be applied to all selected cells.



Right-clicking in various areas of the model editor will bring up context menus with the following options:

Area	***Option***	***Description***
Vertical axis	Set maximum	Set the depth of the bottom layer.
	Redivide	Set the same layer thickness for all model layers.
	Thick mesh	Remove every other node in the vertical direction.
	Thin mesh	Add intermediate nodes in the vertical direction.
Horizontal axis	Redivide	Set the same width for cells located between the unique electrode positions.
	Thick mesh	Remove every other node in the horizontal direction (only removes a node if there is no electrode located in this node).
	Thin mesh	Add intermediate nodes in the horizontal direction.
Model title area	Display model mesh	Show/hide the model mesh.
	Display objects border	Show/hide the object boundaries.
	Setup	Open the Model Setup dialog box.

	Display color bar	Show/hide the color scale.
	Zoom&Scroll CTRL+A	Enable the Zoom&Scroll mode.
	Print preview	Print the model.
Color scale	Setup	Open the Model Setup dialog box.
	Set range	Set the minimum and maximum values of the color scale manually.
	Automatic	Automatically determine the minimum and maximum values of the color scale.
	Log scale	Set the logarithmic scale for the color scale.
	Set half-space value	Set the parameter value of the half-space (the background value).
	Set cursor value	Set the current value of the parameter.
	Colors as histogram	Set the colors based on the distribution of resistivity in model cells.

Arbitrary layered model mode

An arbitrary layered model consists of a set of layers with an arbitrary geometry and parameter variations of each layer. It is a so-called *soft 2D model*, which implies the absence of small local objects of high contrast in the section.

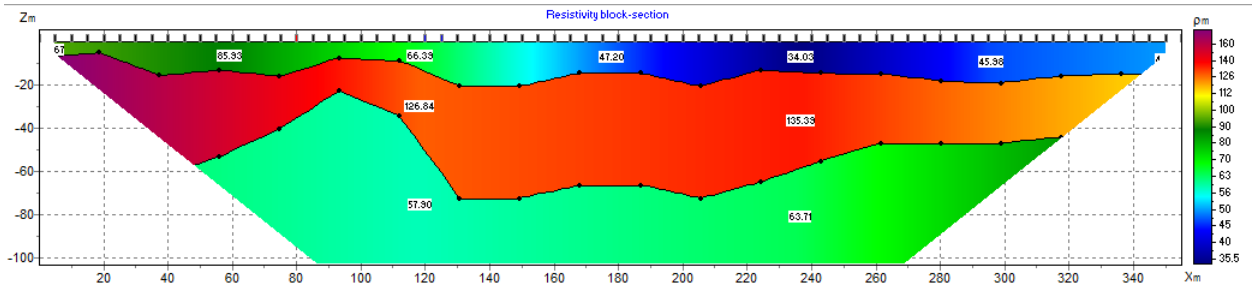
It is similar to the block mode, in which the medium is divided into cells, but in a layered representation. This representation is more structured and often the most reasonable from a geological perspective.

The arbitrary layered models have clear advantages, among which are the easily interpretable "geological" look of the resulting sections and the possibility of conducting joint inversion with data of other geophysical methods.

When the mode is activated with the  button on the toolbar, a **Layered** sub-menu appears in the main window menu.

An initial model in the form of a horizontally layered medium with constant layer parameters can be set using the **Layered / Model constructor** menu command. Setting up the initial model, it

is important to create a good approximation of the real geology for inversion. The initial model can be specified based on the analysis of the block (mesh) model inversion results. The results of the mesh model inversion can be incorporated into the framework of the created layered model using the **Layered / Load from mesh** menu command.



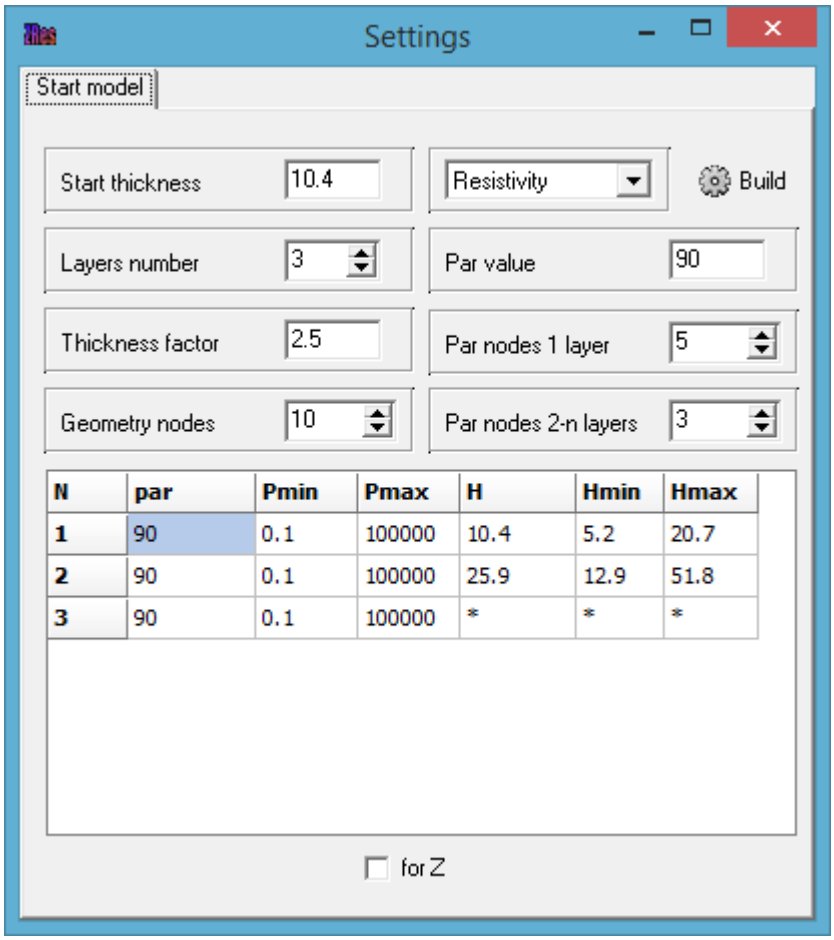
The first set of parameters defining a layered model is the number of layers and their thickness. Next, the number of boundary nodes (geometry nodes) and the number of parameter variation nodes within a layer (parameter nodes) are specified. A boundary node is a point with a changeable vertical coordinate. More complex boundaries can be specified by increasing the number of geometry nodes. On the other hand, overcomplicating the geometry might decrease the reliability of the resulting model. It is recommended to choose the number of geometry nodes based on the number of unique electrode positions on the line. All boundaries have the same number of nodes. In the edit mode (**Layered / Edit mode**), nodes are displayed as circles that can be dragged vertically with the mouse or fixed for inversion. To fix a node, left-click on the node while holding down the Ctrl key; fixed nodes are displayed in red. It is practical to edit boundaries with the **Layered / Transparent** option enabled to display the results of inversion in mesh mode in the background.

Parameter nodes, or parameter variation profile, define the distribution of the parameter within a layer. The parameter variation profile is specified by a set of fixed node values, with linear interpolation between the values. The program implements a different number of nodes for the topmost layer and the subsequent layers. Typically, the topmost layer is characterized by higher variability of geophysical parameters and a more detailed profile should be specified for it (3-10 nodes). In the deeper layers, parameters change more smoothly, and the profile can be limited to 1-3 nodes. In the edit mode (**Layered / Edit mode**), you can edit node parameter values by right-clicking on the value label. To fix the parameter value for inversion, right-click on the value label while holding down the Ctrl key.

Layered models can be used to perform a joint inversion with data of other geophysical methods to obtain a model with common geometry of boundaries and positions of parameter nodes. Joint inversion of resistivity data is possible with the following data types: IP, seismic

refraction, gravity, magnetic, MT, and TEM. Before conducting a joint inversion, it is necessary to specify each method's model individually.

The layered model is configured in the **Layered / Model constructor** window.



Start thickness – sets the thickness of the first layer.

Layers number – sets the number of layers.

Thickness factor – sets the thickness increment for each successive layer.

Geometry nodes – sets the number of geometry nodes for each boundary (from 10 to 50).

– sets the type of parameter from the following list:

Resistivity – resistivity for ERT, VES, MT, or TEM data;

Polarizability – polarizability for IP data;

Gravity – density for gravity data;

Magnetic – magnetic susceptibility for mag data;

Velocity – P or S-wave velocity for seismic data.

If the selected parameter is not available (the data of the corresponding method are not loaded into the project), the type will be set to *Resistivity* automatically.

Values of the selected parameter are displayed in the form of labels on the layered model. At the same time, resistivity or polarizability values, depending on the current mode, are indicated with colors according to the color scale, thus allowing to display values for two different methods simultaneously.

Par value **–** sets the initial value of the parameter in the first layer.

Par nodes 1 layer **–** sets the number of parameter nodes defining the parameter distribution within the first layer. The parameter variation profile is specified by a set of fixed node values with linear interpolation between the values. If set to 1, the parameter in the layer is set to a constant value.

Par nodes 2-n layers **–** sets the number of parameter nodes defining the parameter distribution within the subsequent layers.

The lower portion of the **Model constructor** window contains an editable table. In the table you can manually set the parameter value **par**, parameter variation range during inversion (from **Pmin** to **Pmax**), layer thicknesses **H**, and thickness variation range during inversion (from **Hmin** to **Hmax**). To specify depths instead of thicknesses, check the **for Z** checkbox.

The **Layered / Invert Rho&IP** option is used to invert the resistivity and IP data together within the same model geometry.

The **Layered / Invert boundaries** option allows to invert (determine) layers geometry during inversion. To invert only parameter values (in the case when the boundaries are known and fixed), the **Invert boundaries** option should be disabled.

The **Layered / Draw labels** option shows/hides parameter values.

The **Layered / Transparent** option allows displaying the results of inversion in mesh mode in the background (the colors of the layered model become transparent).

The **Layered / Edit mode** enables the editing mode. The position of boundary nodes can be changed by dragging node points in the vertical direction, parameter values can be edited by right-clicking on the value labels.

A layered model can be saved to and loaded from a text file using the **Save layers** and **Load layers** menu commands.

Polygonal modeling

In many situations, it is more practical to specify the model as a set of polygonal objects. The polygonal modeling mode in **ZondRes2D** provides this opportunity. Clicking the  button on the toolbar enables the mode and brings up a floating toolbar with a set of polygonal modeling tools,

which appears in the upper left corner of the screen. If a polygonal model has already been created within the current project, it will be displayed in the model window.

Creating polygonal models in **ZondRes2D** is similar to the process of creating polygons in vector graphic editors.

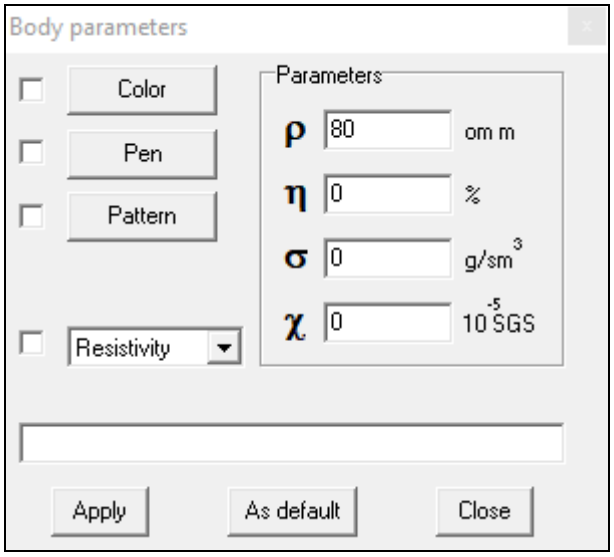
The mode allows the creation of both individual polygons (objects) in a homogeneous medium and a system of connected polygons (objects). Creating and editing a polygon is done using various tools from the floating toolbar:

Tool	***Action***
	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). 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.
	The Zoom mode to change the scale of the model section.

When working with the polygonal modeling tools, remember that **all actions are finalized by right-clicking**.

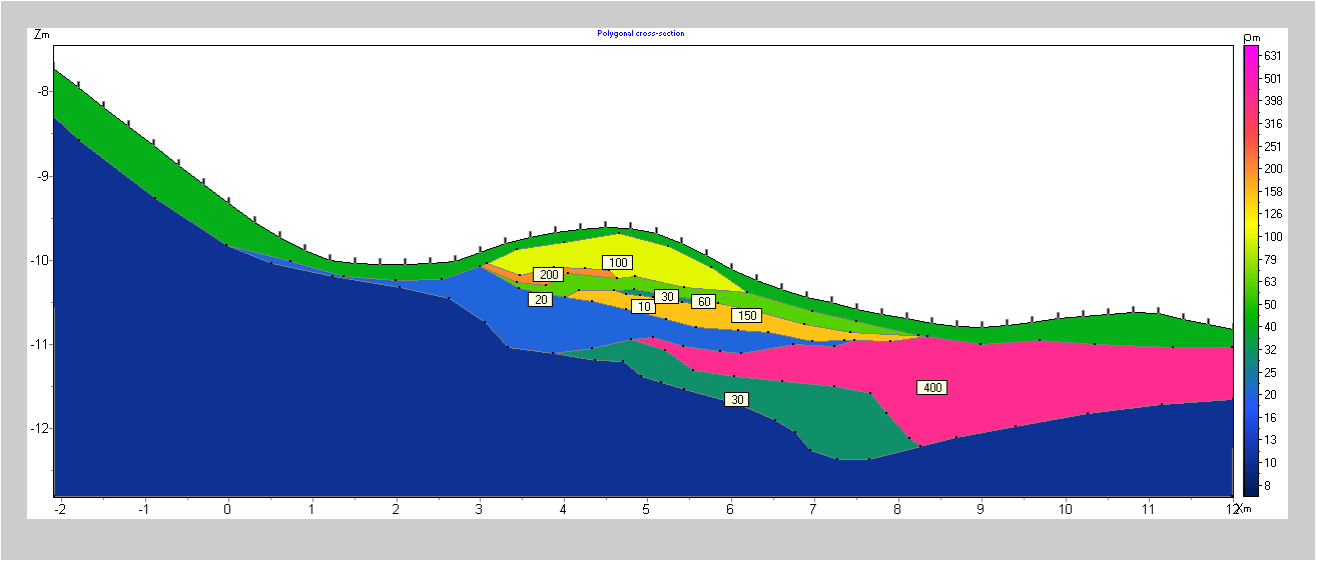
To change the resistivity, polarizability and graphic settings of a polygon, double-click on the polygon. The **Body parameters** dialog box will appear.



The ρ, η, σ, χ input boxes are used to specify the values of resistivity, polarizability, density and magnetic susceptibility. The ***Color***, ***Pen***, ***Pattern*** buttons bring up dialog boxes for setting the polygon fill color (different from the model's color scale), polygon boundary color and hatch pattern, respectively. The drop-down list contains the possible polygon labels – resistivity, polarizability, or a user text that can be entered in the input box below.

When working with self-potential data, you should specify the type of conductivity of the polygon – electronic or ionic. If the SP data are loaded into the project, it is possible to select the electronic conductivity type in the **Body parameters** dialog box, otherwise it will be ionic conductivity.

The exchange of parameter values between polygonal and mesh models is carried out using the **Modeling** / **Get values from mesh** and **Modeling** / **Set values to mesh** menu commands. The first option (embedding with the account of geometry) assigns the block model resistivities to polygons (using the results of preliminary inversion or modeling), and the second option assigns polygon resistivities to the block model.



Polygonal models can be saved using the **Save polygons** and **Load polygons** commands of the **Modeling** menu. With the **Export model to CAD** command, the polygonal model can be exported to the AutoCAD DXF file format.

Calculation of the forward problem for the created model is done by pressing the button on the toolbar or the Space key.

To increase the accuracy of the forward problem solution, it is recommended to set a more detailed model mesh. In particular, after creating a polygonal model, you can switch to the block mode and use the **Thin mesh** option in the properties of each of the axes, then return to the polygonal modeling mode and press the button. After this procedure, the calculation of the forward problem will be carried out using the more detailed model mesh.

The **Polygonal modeling** mode can be used for joint interpretation with seismic and gravity/mag survey data. In this case, the polygonal framework acts as a common part, i.e., the model geometry is fixed and only parameters are inverted. Inversion of seismic velocities is started with the button, and for the inversion of gravity and magnetic data, the **Options / GraviMagnetic / Inversion** command is used.

Saving and reading in the calculated data

To use the data calculated for a created model as observed data, the data should be saved into a text file as ***Zond calculated data [*.z2d]*** type. The saved file can then be opened as an observed data file. Alternatively, the ***Zond model with calculated [*.z2d]*** file type can be used, but after opening the file the imported model has to be cleared before inversion. To add noise to the synthetic data, the **Options / Import/Export / Save synthetic with noise** option should be activated before saving the data.

It is then possible to perform inversion of the data and analyze the differences between the inverted model and the original one. This mechanism allows to conduct various experiments and test different measurement systems.

Data interpretation

The most important part of any geophysical study is the determination of a physical model of the subsurface based on the observed data, i.e. the solution of the inverse problem, or data inversion.

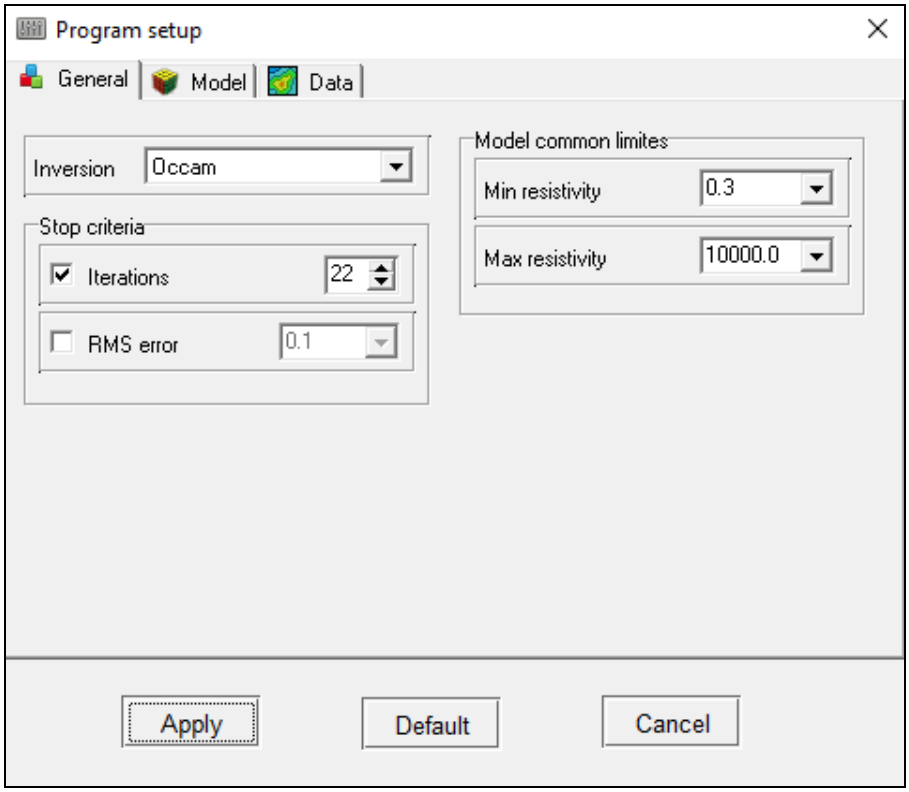
To obtain the most reliable results, it is important to consider all a priori information, evaluate the quality of the data, compare the results of different methods, and choose the most appropriate parameters for inversion.

The program not only performs inversion of resistivity data but also allows joint inversion with data obtained with IP, MT, TEM, refraction seismic, gravity and magnetic methods.

If the opened data file contains IP data,  and  buttons appear on the toolbar, allowing you to switch between the resistivity and polarizability operation modes.

Changing inversion parameters

After opening a data file, processing the data and setting the initial model, the next step is to select the inversion type and set the inversion parameters. The dialog box for setting the parameters of forward and inverse modeling is opened using the  button on the toolbar or the **Option** / **Program setup** menu command.

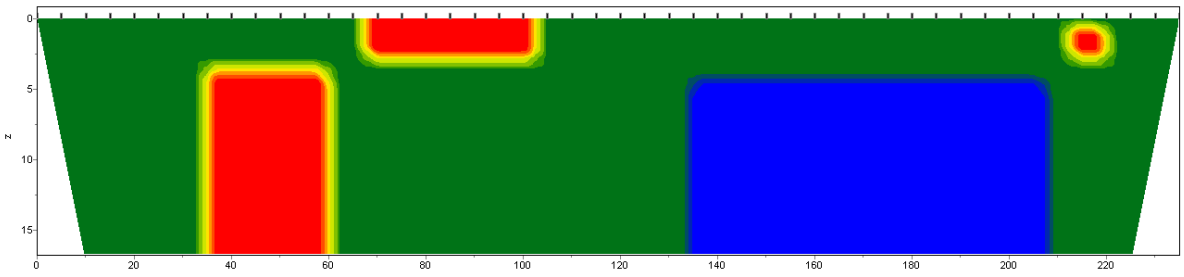


The **Default** button resets the parameters to original settings.

The basic inversion parameters are configured in the **General** tab.

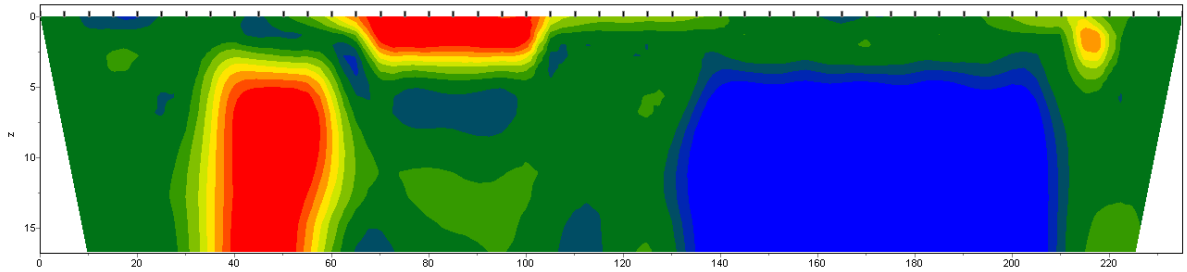
The ***Inversion*** option defines the inversion algorithm.

Let us consider various inversion algorithms on the example of a model consisting of several blocks (see figure below).



To test the algorithms, we calculated the theoretical response for this model and added a five percent Gaussian noise to the result.

Smoothness constrained is a least-squares inversion using a smoothing operator. This algorithm results in a smooth (without sharp boundaries) and stable parameter distribution (see figure below).

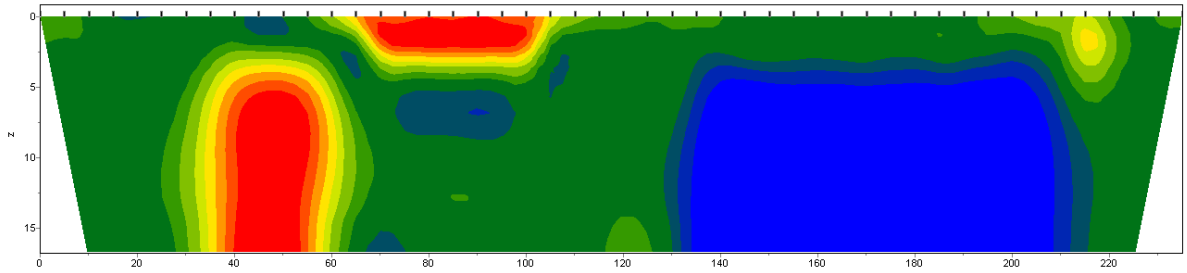


The matrix equation for this type of inversion is as follows:

$$(A^T W^T \mathrm{WA} + \mu C^T C)\Delta m = A^T W^T \Delta f$$

As follows from the equation, the program does not minimize the contrast of the model during the inversion. This algorithm allows achieving minimal misfits and is recommended for use at the initial stages of interpretation, in most cases.

Occam is a least-squares inversion using a smoothing operator and additional contrast minimization [6]. This algorithm results in the smoothest parameter distribution (see figure below).

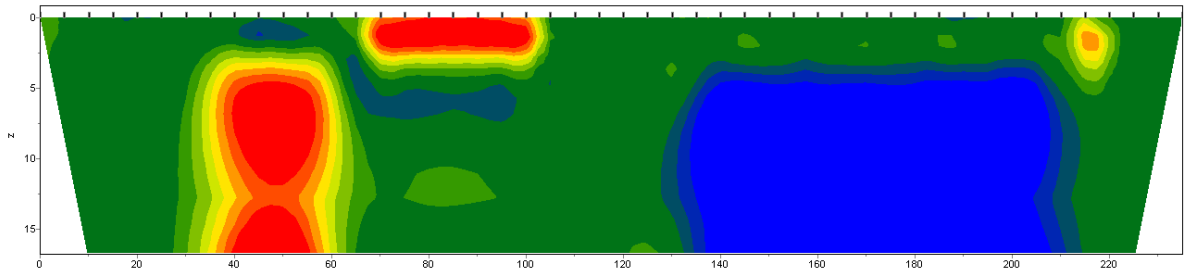


The matrix equation for this type of inversion is as follows:

$$(A^T W^T \mathrm{WA} + \mu C^T C)\Delta m = A^T W^T \Delta f - \mu C^T \mathrm{Cm}$$

The degree of smoothness of the resulting model is directly proportional to the **Smoothness factor** value. It should be noted that too large values of the parameter may lead to higher misfits.

Marquardt is a classical least-squares inversion algorithm with a damping parameter regularization (ridge regression) [7]. If the medium is characterized by a small number of parameters, this algorithm results in inversion models with sharper boundaries (see figure below).



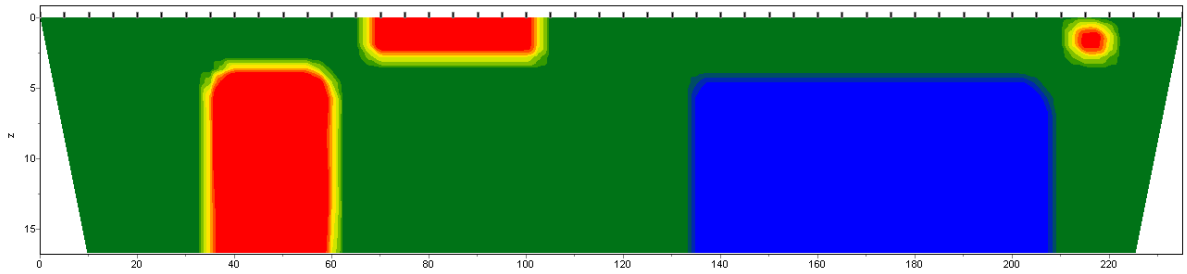
The matrix equation for this type of inversion is as follows:

$$(A^T W^T \mathrm{WA} + \mu I)\Delta m = A^T W^T \Delta f$$

Unconsidered use of this type of inversion can lead to unstable results or an increase in standard deviation.

It is recommended to apply the ***Marquardt*** algorithm as a refinement method (to reduce the misfit) after performing the inversion with the ***Smoothness constrained*** or ***Occam*** algorithms.

Blocks **–** determination of parameters of individual areas that differ in resistivity. Areas with the same resistivities are treated as single blocks (see figure below).

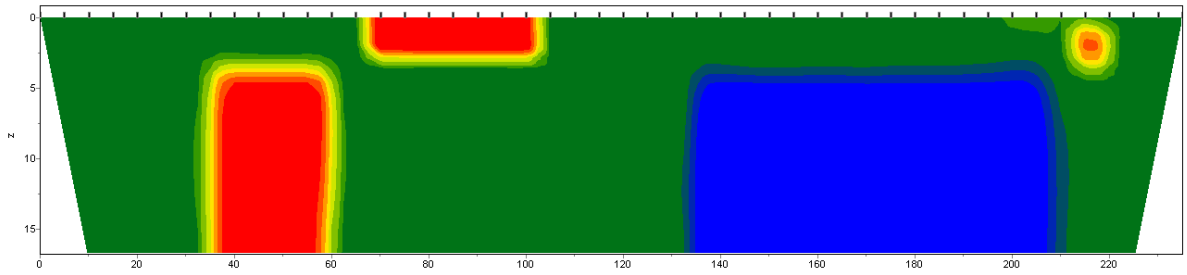


The matrix equation for this inversion type is the same as for the ***Marquardt*** algorithm:

$$(A^T W^T \mathrm{WA} + \mu I)\Delta m = A^T W^T \Delta f$$

It is recommended to use this algorithm at the stage of refining the model obtained with other algorithms (***Focused*** is preferred), having previously combined the cells into blocks using the **Models operations** module accessible by **Options / Extra / Model smooth/raster** menu command. The blocks can also be specified manually using the model editor to assign different parameters (resistivities) to separate areas. Individual blocks are framed with a border while the **Program setup** dialog box is active.

Focused is a least-squares inversion using a smoothing operator and additional contrast focusing [8]. This algorithm results in a piecewise smooth parameter distribution, i.e., in a model consisting of blocks with constant resistivity (see figure below).



The matrix equation for this type of inversion is as follows:

$$(A^T W^T \mathrm{WA} + \mu C^T \mathrm{RC})\Delta m = A^T W^T \Delta f - \mu C^T \mathrm{RCm}$$

Using this type of inversion requires a careful selection of the ***Threshold*** parameter in the **Model** tab. This parameter defines the contrast threshold value of neighboring cells, after reaching which the parameters of the cells are not averaged (i.e., it is considered that there is a boundary between the cells). The degree (weight) R_i of averaging of two neighboring cells is given by:

$$R_i = \frac{e^2}{e^2 + r_i^2},$$

where e is contrast threshold and r_i is contrast between the cells.

ProfileR **–** the ProfileR program designed by Prof. A. Binley is used for inversion.

Statistic(beta) **–** inversion based on the statistical principle of maximum likelihood (requires no additional settings).

The **Stop criteria** group box contains two criteria for inversion termination. The inversion process stops when one of the following criteria is met:

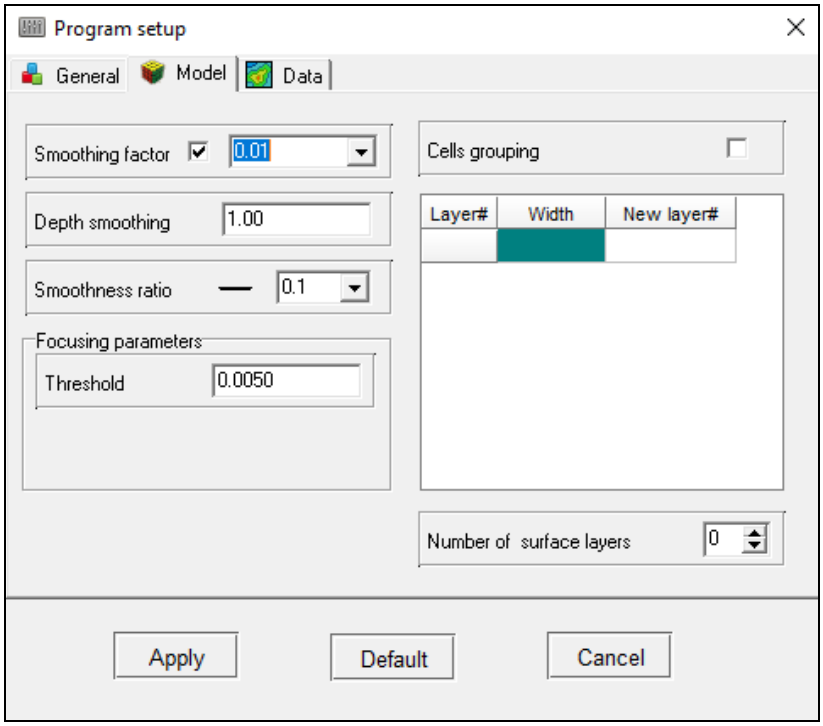
Iterations **–** the set number of iterations is reached.

RMS error **–** the set value of misfit is reached.

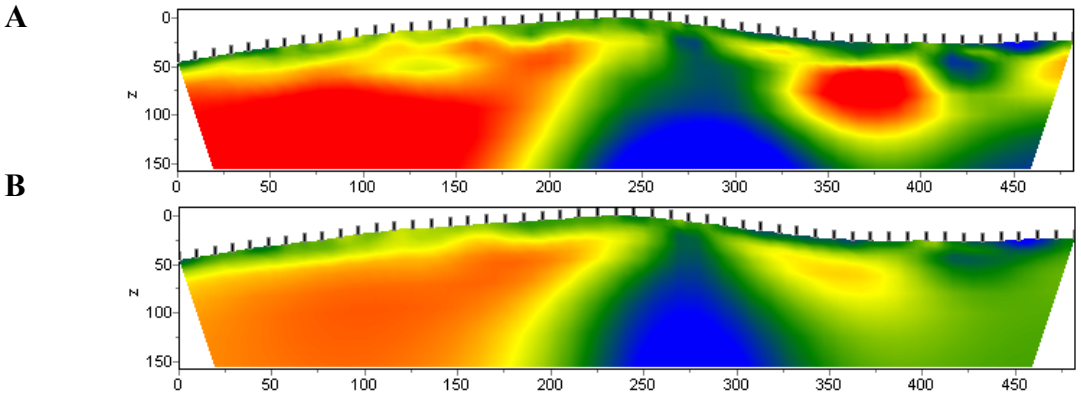
The **Model common limits** group box **–** the **Min resistivity** and **Max resistivity** boxes set the parameter variation limits.

If overall parameter limits or individual cell limits are set too narrow, the inversion will repeatedly try to exceed the parameter limits. This can greatly affect the rate of convergence. In this case, the **Options / Inversion / Optimization / Lim based inv** option should be activated. Activating this option will reduce the contribution of cells exceeding the set limits, at the same time making such an exceeding difficult by applying a special parameter normalization.

The **Model** tab contains a set of inversion parameters related to models.



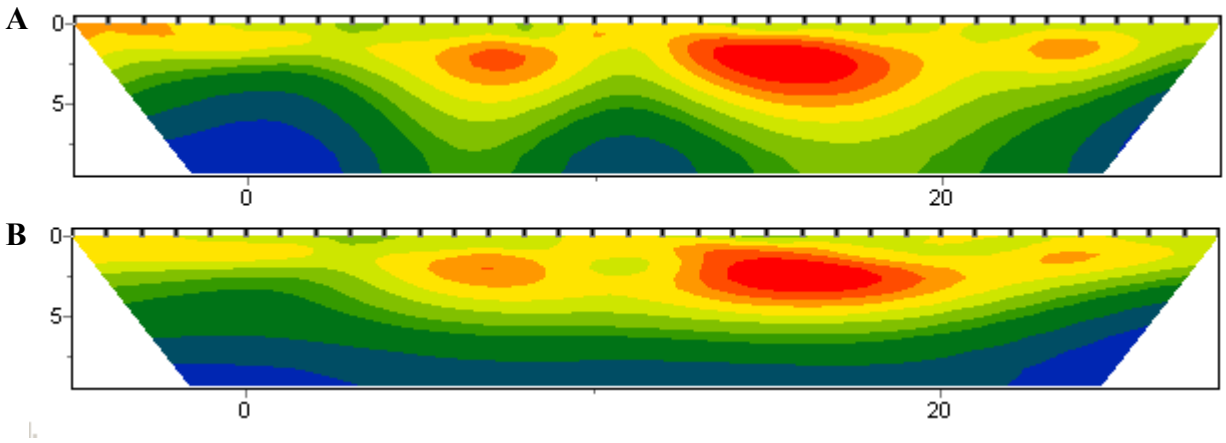
Smoothing factor – sets the ratio between the minimization of data misfit and model misfit. For noisy data or to obtain smoother and more stable parameter distribution, relatively large values of smoothing factor are chosen (0.5-2); for high-quality data, values in the order of 0.005-0.01 are typically used. Large values of the smoothing factor will typically result in larger misfits. The example models in the figure below were obtained using a smoothing factor of 0.01 for model A and 1.0 for model B. The resulting RMS errors were 4.5% and 6%, respectively. The smoothing factor is used in the ***Occam*** and ***Focused*** inversion algorithms. If optimization (Line search) is disabled, the program enables automatic determination of the smoothing factor. For this, the checkbox in the **Smoothing factor** box should be checked.



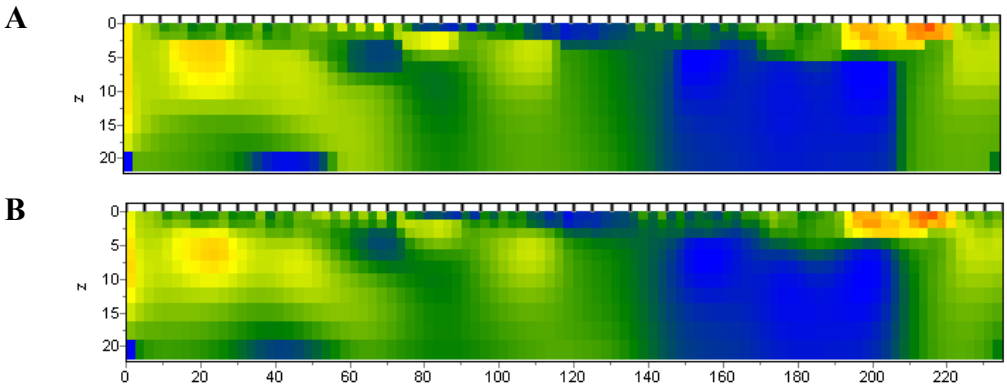
Depth smoothing – sets the coefficient of smoothing with depth. The value of this parameter will depend on the level of noise in the data; with high noise levels and appearance of oscillations

and geologically unrealistic objects in the lower portion of the model, the smoothing value should be increased. The value is selected empirically.

Smoothness ratio – defines the ratio between the degree of smoothness in horizontal and vertical directions. Values of less than 1 should be used for horizontally layered geology and more than 1 for vertically layered geology. Typically, the values range from 0.2 to 1 (in the figure below, model A was obtained with a smoothness ratio of 1, model B – with a smoothness ratio of 0.3).



Threshold – sets the contrast threshold value of neighboring cells, after reaching which the parameters of the cells are not averaged (i.e., it is considered that there is a boundary between the cells). The value of this parameter is chosen empirically, typically in the 0.001 to 1 range. In the figure below, model A was obtained with a threshold value of 0.01, model B – with a threshold value of 0.1. Choosing a very small value of this parameter can lead to an algorithm divergence (in this case the value should be increased). Large values of the parameter lead to a smooth distribution.

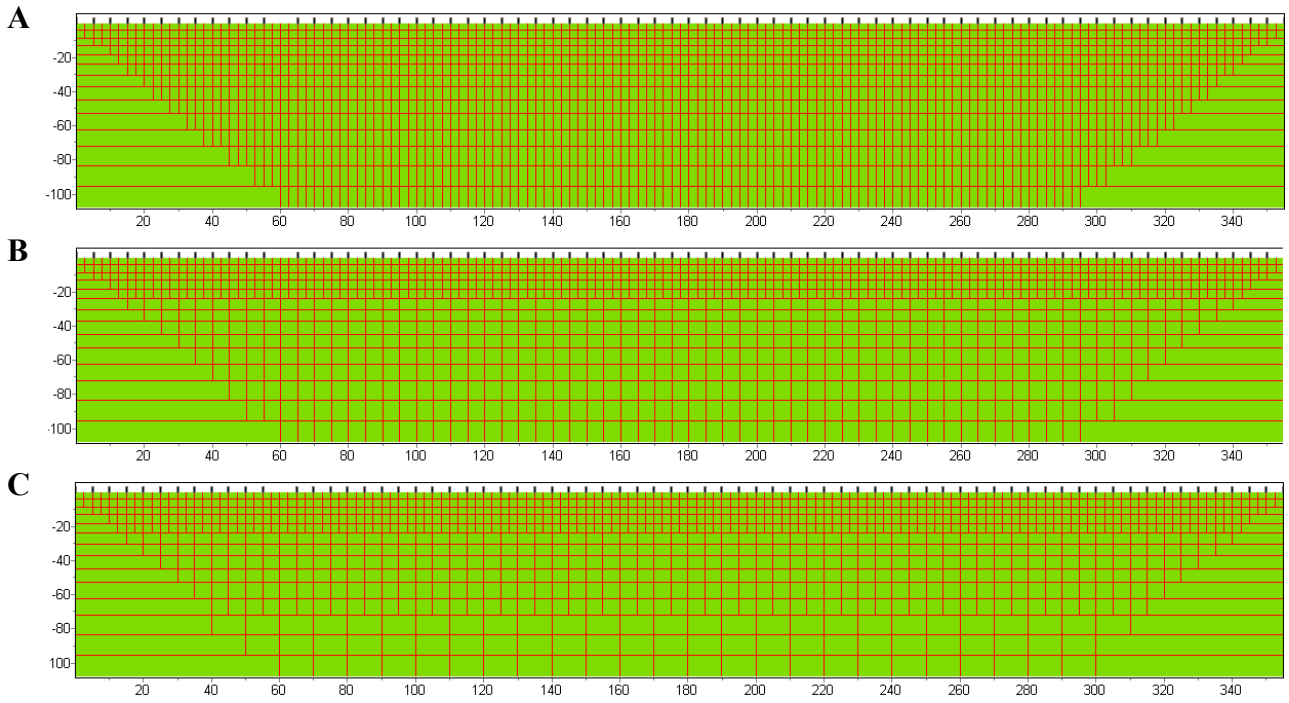


Cell grouping – this option is used in the case of large models (when you open a data file with a large number of data points, the program automatically suggests using it). The option activates the table which allows you to combine adjacent cells (coarsen the grid), resulting in a smaller

number of parameters defined during the inversion. While the number of cells used for inversion decreases, the number of cells used for solving the forward problem remains the same. Ideally, the number of parameters determined during inversion (thus, the number of cells) should be close to the number of data points.

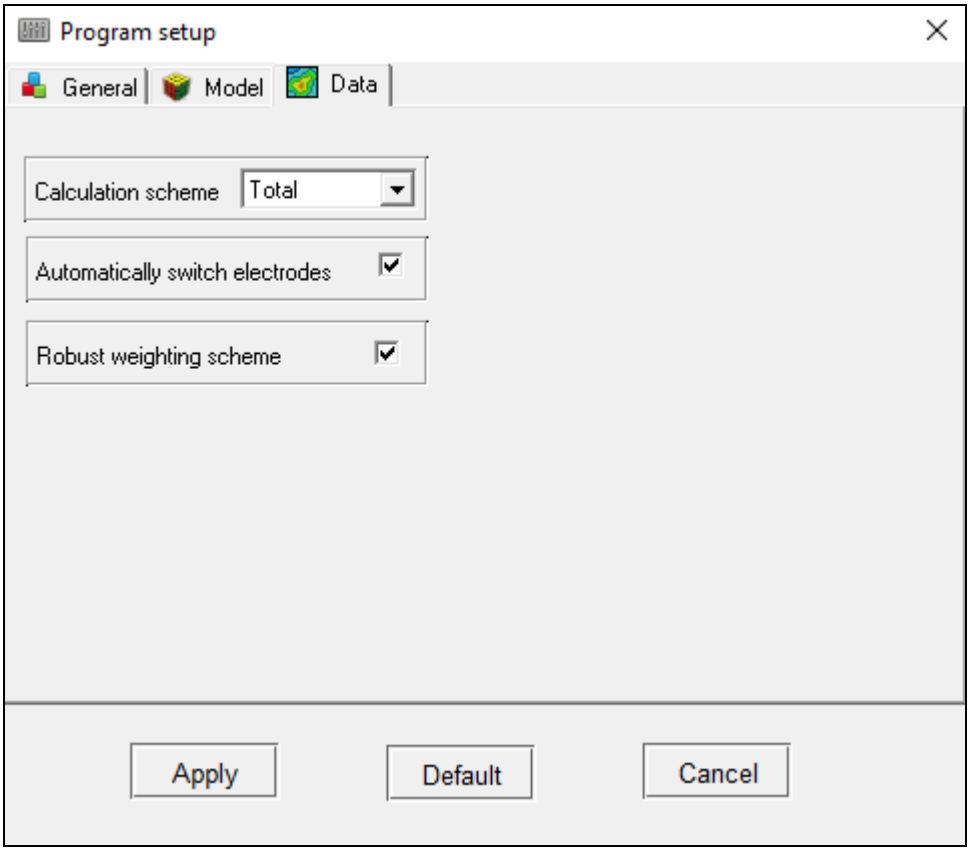
The table contains three columns. The first column (**Layer#**) lists the numbers of the initial model layers. The second column (**Width**) specifies the width (in cells) of cells to be merged for inversion. The third column (**New layer#**) contains the numbers of the modified mesh layers and indicates the thickness of grouped cells. The modified mesh is displayed in the model section during its configuration. Right-clicking on the **Width** or **New layer#** columns brings up a dialog box where you can specify the number of horizontal cells or layers to be merged throughout the model. Uncheck the **Cell grouping** checkbox to reset the table.

The figure below shows three examples of mesh configuration for inversion: **A** is the initial mesh configuration; for **B**, starting from the sixth layer, every two cells are merged in the horizontal direction; for **C**, in addition, starting from the twelfth layer, every two cells are merged in the vertical direction.



Number of surface layers (0-3) – sets the number of near-surface layers where strong parameter variations are allowed. This option should be used in the case of very heterogeneous geology in the upper portion of the section. Also, this option can be used if the medium is of very low contrast, so the main anomalous effects in the data are caused by near-surface heterogeneities.

The **Data** tab contains a set of additional inversion parameters.



Calculation scheme **–** defines the algorithm for calculating the electric potential. If **Secondary** is chosen, the calculation of secondary potential is performed. This method is slower and does not take into account the surface topography, but allows you to obtain fairly accurate results when a coarser model mesh is used. When calculating the total potential (**Total**), it is necessary to use a finer mesh (1-2 nodes between adjacent electrodes) and extend the outer boundaries of the model. This is due to a large error in determining the potential near current electrodes. The **Secondary** option should be used when the survey was carried out using irregular spacings between electrodes, e.g., in the case of VES data.

Automatically switch electrodes **–** if this option is enabled, the program automatically swaps potential electrode locations to obtain a positive value of the geometric factor. It is recommended to disable this option for cross-hole surveys.

Robust weighting scheme **–** this option should be enabled if there are spikes in the data associated with systematic measurement errors. If the number of bad points in the data is comparable to the number of quality measurements, this algorithm may not produce good results.

It is recommended to enter available a priori information before inversion. It can be in a form of an initial model based on geological data, data from other geophysical methods, or results of a previously performed inversion. Various means of entering a priori information in **ZondRes2D** are described in the **A priori information** section.

After setting inversion parameters and entering a priori information, the inversion process is started by pressing the  button. The progress of the inversion process is displayed in the status bar. The inversion optimization progress can be viewed in a separate window using the **Options** / **Inversion** / **Optimization** / **Display process** menu command. The model section is updated during the inversion process.

A one-dimensional data inversion can be performed by pressing the  button on the toolbar.

Inversion of IP data

If the opened data file contains IP data, the  and  buttons appear on the toolbar, allowing you to switch between the resistivity and polarizability operation modes. In the latter case, pressing the  button will invert the data of integral polarizability, taking into account the current resistivity model. If decay curves are available, the program can perform inversion of polarizability for the entire curve and obtain Cole-Cole model parameters.

In practice, IP measurements are carried out using one of the two methods – time-domain or frequency-domain. When working in the time domain, the result of a measurement is the IP decay curve; for the frequency domain data, the measurement results are presented as absolute values and phase shifts of the measured signal at different frequencies.

For IP surveys the program accepts text files with *.z2d extension which have the following structure. The first line after the word **time_#chann** contains the list of delay times for the time-domain or frequencies for the frequency-domain method of measurements. Then follows a table, with each line corresponding to a data point, and columns containing electrode coordinates (**C1**, **C2**, **P1**, **P2**), **Res** (V/I) values, and IP values at corresponding delay times (column headers – **ipi1**, **ipi2**, etc.) or channels (frequencies) (column headers for absolute value – **mod1**, **mod2**, etc., column headers for phase shift – **pha1**, **pha2**, **etc**.). The program will calculate the apparent polarizability automatically.

A sample of a time-domain data file:

time_#chann		*0.05*	*0.065*	*0.085*	*0.115*	*0.155*			
C1	*C2*	*P1*	*P2*	*res*	*ipi1*	*ipi2*	*ipi3*	*ipi4*	*ipi5*
0	*	*5*	*10*	*0.001*	*0.279*	*0.260*	*0.242*	*0.218*	*0.194*
5	*	*10*	*15*	*0.001*	*0.306*	*0.286*	*0.266*	*0.241*	*0.215*
10	*	*15*	*20*	*0.002*	*0.338*	*0.317*	*0.295*	*0.268*	*0.239*

A sample of a frequency-domain data file:

time_#chann		*0.0001592*		*0.000427*		*0.001145*				
C1	*C2*	*P1*	*P2*	*res*	*mod1*	*mod2*	*mod3*	*pha1*	*pha2*	*pha3*
0	*	*5*	*10*	*0.001*	*0.001*	*0.001*	*0.001*	*-0.004*	*-0.006*	*-0.010*
5	*	*10*	*15*	*0.001*	*0.001*	*0.001*	*0.001*	*-0.005*	*-0.007*	*-0.012*
10	*	*15*	*20*	*0.002*	*0.002*	*0.002*	*0.002*	*-0.005*	*-0.009*	*-0.014*

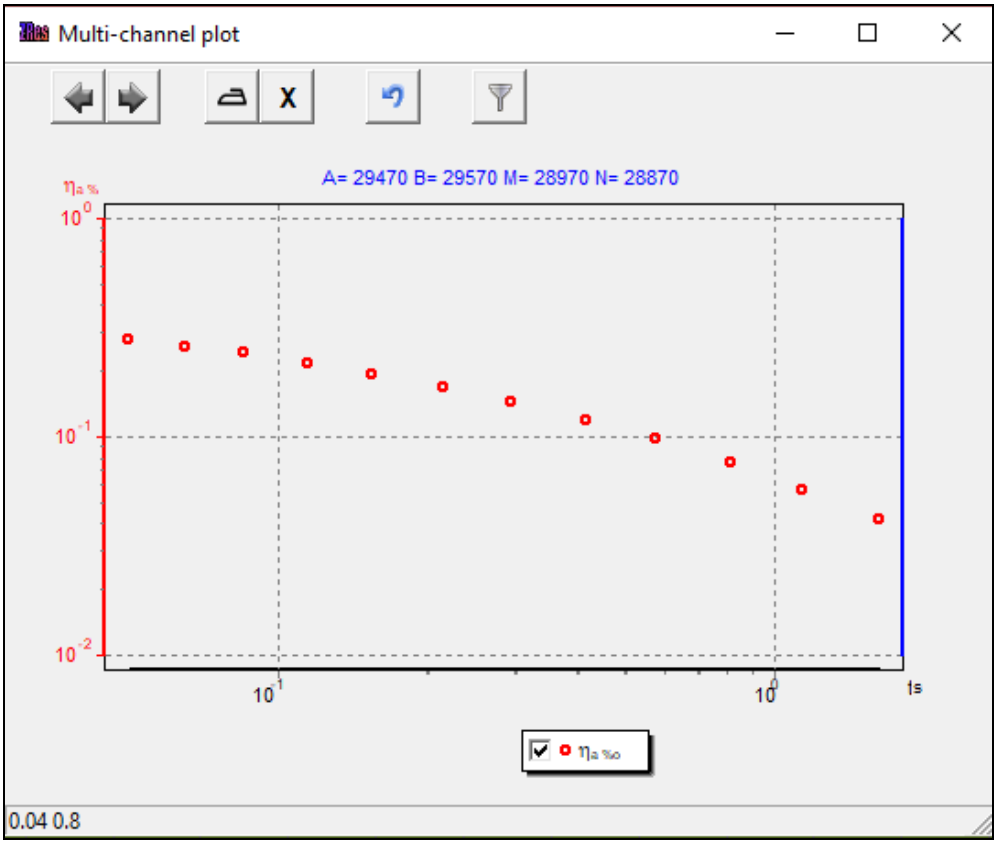
If the input data are multichannel IP or time-lapse measurements, the **Time lapse/domain** section appears in the main menu of the program. Switching between channels corresponding to different times (frequencies) is performed using the drop-down list on the main window toolbar.

The **Time lapse / domain / Channels** option opens the channel setup table. Here you can edit the time delay or frequency values and subset titles which are displayed in the drop-down list on the toolbar. Some program options require the times/frequencies to be entered beforehand to function properly.

The **Time lapse / domain / Display channel plot** command opens a window displaying a decay curve (or time/frequency function) for the current measurement point. Switching to another measurement is done by left-clicking on the pseudo-section or graph, or using the  and  buttons; the coordinates of the corresponding current and potential electrodes are displayed at the top of the graphic area. An individual point of the curve can be edited using commands of the context menu brought up by right-clicking on the point. You can delete the point (***Delete point***), shift it to a smoothed value (***Smooth point***), restore the original value (***Origin point*)** or delete all points (***Delete All***). Right-click on the  button brings up a context menu with the following options: smooth the current curve (***Smooth current***), smooth all data curves (***Smooth All***), apply a filter for the current curve (***Apply filter (current*)**), apply a filter for all data curves (***Apply filter (all*)**).

The  tool allows you to specify a filter in both time and frequency domains.

The  tool allows you to remove individual points from a curve, remove the current curve, and remove spikes and absurd values by applying a filter to the current curve, or all curves. The last three options are available by right-clicking on the tool button.



After editing the curves of the time-domain data it is necessary to use the **Time lapse / domain/ Recalculate chargeability** option to recalculate the integral IP parameter (chargeability) values taking into account the changes made in the data. IP units of measurement (percentage or mV/V) can be selected using the **Options / Extra / IP Units** menu command.

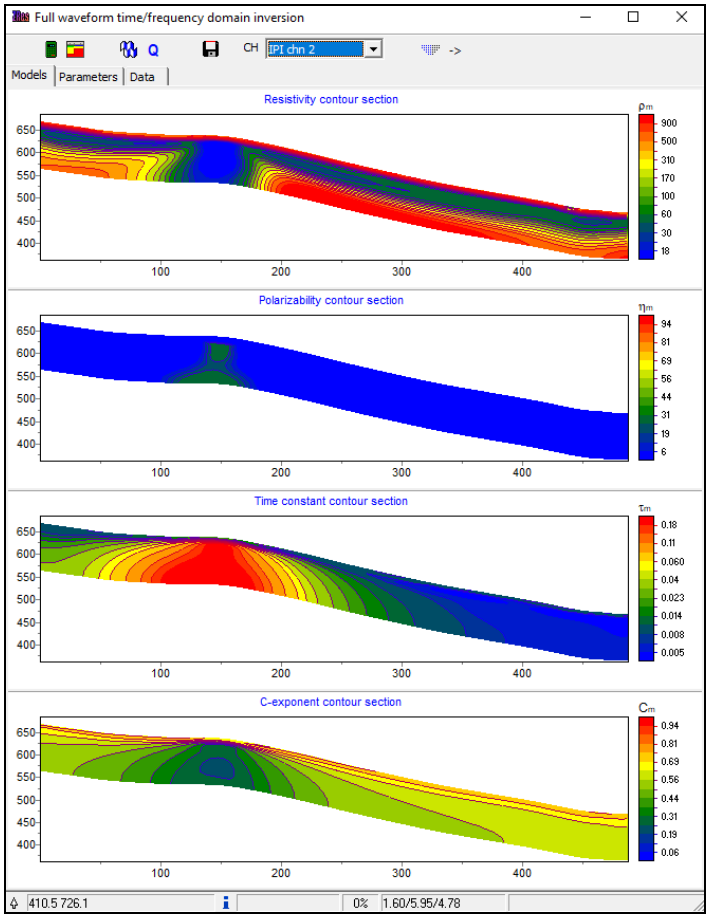
The **Time lapse/domain / Time lapse inversion** command starts the consecutive inversion of each subset of the IP data set (in this case the standard calculation scheme is used, same as for the integral IP parameter). If the **Time lapse/domain / Next from previous** option is enabled, each subsequent iteration uses the results of the previous iteration as a reference model. If this option is disabled, the current model of the given subset is used as a reference model for each subsequent iteration. The resulting models are displayed in the model section. Switching between models of different subsets is performed using the drop-down list on the main window toolbar.

Since in most cases the model obtained for the previous subset is used as a starting model for each following subset, a much smaller number of iterations is needed to achieve acceptable RMS error. The number of iterations for the subsets following the first one is set using the **Time lapse/domain / Max iter number for next** command.

The **Time lapse/domain / Save time shots** option saves the results of the data inversion for all subsets into a single text file.

The module for full waveform inversion of IP data is opened using the **Time lapse/domain / Full waveform inversion** menu command. The window contains three tabs. The **Models** tab

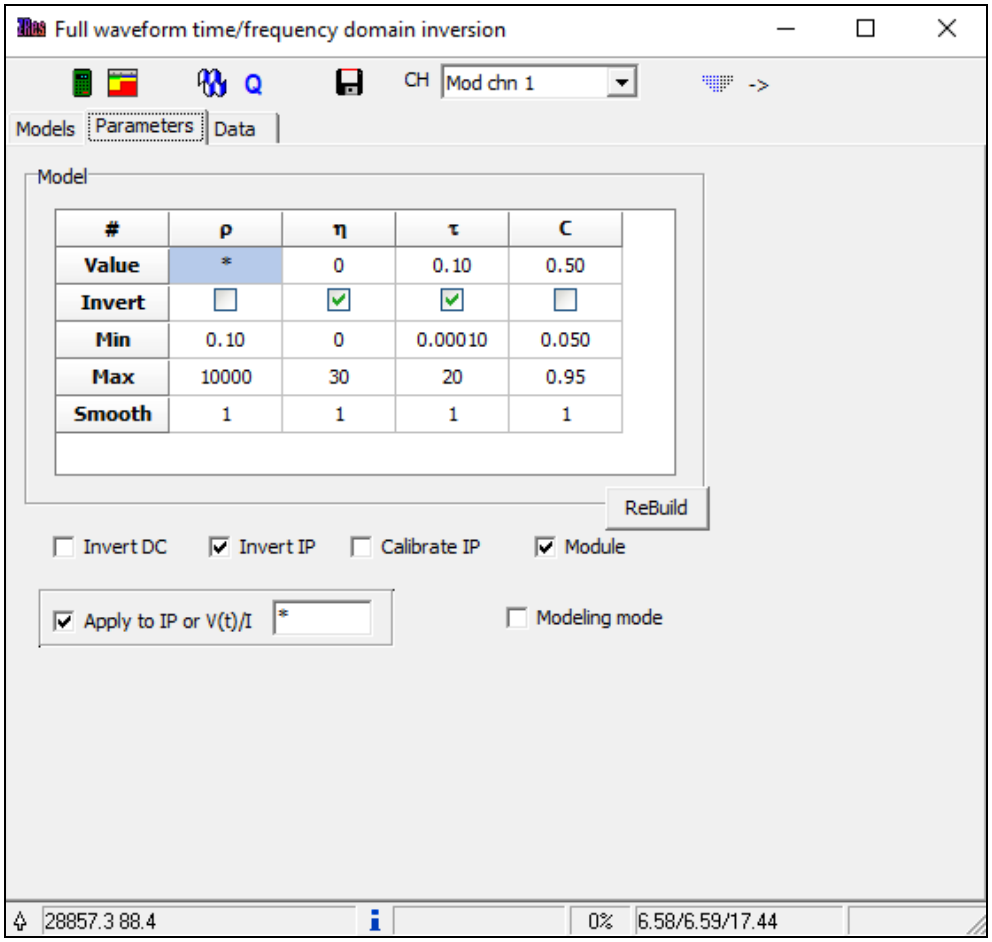
displays contour sections of resistivity and Cole-Cole polarizability parameters (η, τ and c) obtained through the inversion.



The  button allows editing of the four models.

The  button starts the calculation of the forward problem.

In the **Parameters** tab, you can select the parameters to be inverted and specify the variation ranges for these parameters. The values set in the *Smooth* row of the **Model** table indicate the degree of smoothness of the parameters.



The ***ReBuild*** button assigns each model parameter a value from the *Value* row of the table. If a cell in the *Value* row contains the * symbol, the value will not be assigned (or will be taken from the main program module).

The ***Module*** option, available when working with frequency-domain data, enables inversion of absolute values (moduli) of current-normalized signal additionally to the phase. As a rule, within the utilized frequency range the modulus varies very little, and, unlike the phase, has little effect on the inversion results.

The ***Invert DC*** option allows inversion of apparent resistivity. It is not recommended to use this option because the resistivity model is typically pre-determined in the main program module.

The ***Invert IP*** option allows inversion of apparent polarizability drops. It is recommended to leave this option enabled.

The ***Modeling mode*** option appears when the model has been changed in the model editing mode (activated by the  button).

The ***Apply to IP or V(t)/I*** option specifies whether the apparent polarizability or the current-normalized voltage drop is used in the inversion (if checked, the apparent polarizability is used). The input box next to it allows setting a coefficient for pseudo-logarithmic normalization of the

data. Its value should be in the order of the background IP values. If set to -1, the coefficient will be determined automatically. If set to *, the linear normalization of the IP data will be used.

The inversion procedure is started by pressing the  button. The inversion is performed simultaneously for all selected parameters. The  button provides the alternative Born approach, in which the inversion is performed for each cell independently using the "true" IP curves obtained through standard inversion of each channel.

The **Data** tab displays the observed and calculated polarizability data. The drop-down list on the toolbar, available for all tabs, allows switching between subsets corresponding to different decay times (frequencies).

The **Time lapse/domain / Digital filter** dialog box contains the filter settings for the Fourier transform from frequency to time domain. It also allows setting the length of the current pulse. The option is only available for time-domain data.

Inversion of time-lapse (monitoring) data

Monitoring involves repeated measurements at different times using the same survey parameters. For monitoring (time-lapse) surveys the program accepts files with *.z2d extension which have the following structure: the first four columns contain the electrode coordinates; the following columns with headers res1, res2, etc. contain the **Res** (V/I) values for the first, second, etc. measurement cycles. It is also possible to create a time-lapse project by opening several files at once.

Below is a sample of a three-electrode time-lapse survey data file:

C1	*C2*	*P1*	*P2*	*res1*	*res2*	*res3*	*res4*	*res5*
0	*	*2*	*4*	*5.57*	*5.57*	*5.57*	*5.60*	*5.63*
0	*	*4*	*6*	*1.55*	*1.55*	*1.56*	*1.59*	*1.62*
0	*	*6*	*8*	*0.84*	*0.84*	*0.85*	*0.88*	*0.91*
0	*	*8*	*10*	*0.48*	*0.48*	*0.49*	*0.51*	*0.52*
0	*	*10*	*12*	*0.44*	*0.44*	*0.45*	*0.49*	*0.46*
0	*	*12*	*14*	*0.25*	*0.26*	*0.26*	*0.29*	*0.24*

Working with the time-lapse data is similar to working with the full waveform IP data, but instead of the IP decay function, a set of resistivity measurement cycles is processed.

When a data file containing time-lapse measurements is opened, the **Time lapse/domain** section appears in the main menu of the program. Switching between subsets and models corresponding to different measurement times is performed using the drop-down list on the main window toolbar.

The **Time lapse/domain / Channels** command opens the channel (measurement cycle) setup table. Information about the time of the measurement cycles can be entered in the **time, s** column. Arbitrary text information (title) entered in the **name** column will be displayed in the drop-down list on the toolbar.

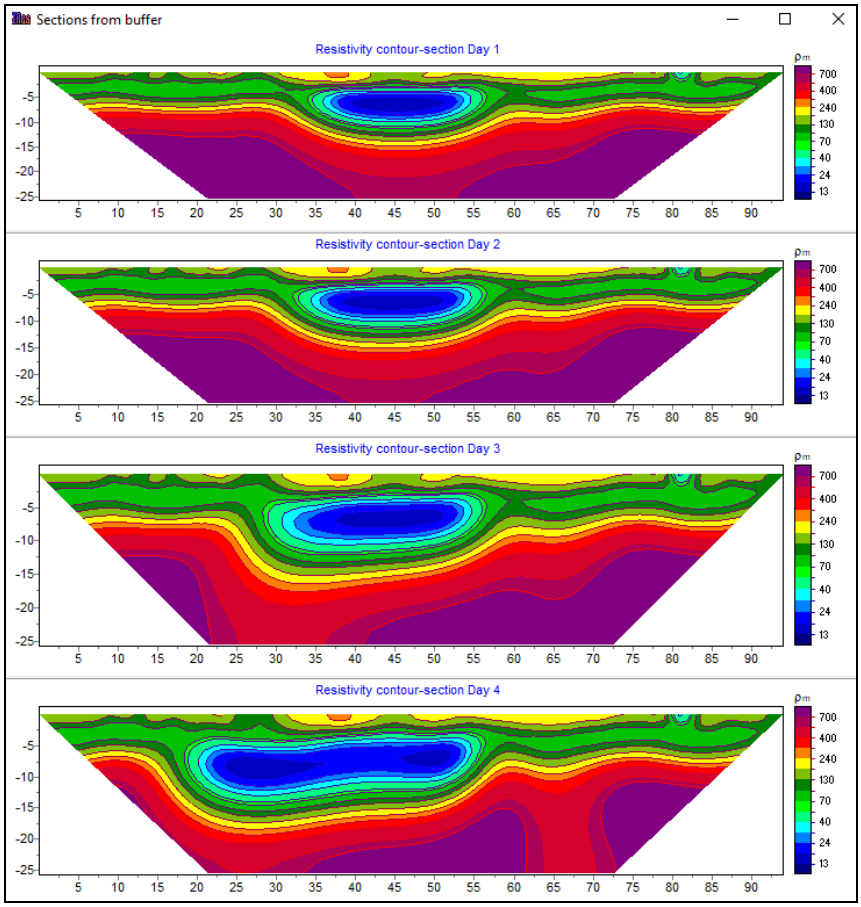
To delete a channel (measurement cycle), uncheck the checkbox in the **inv** column, then right-click anywhere within the window and select **Delete unchecked** in the context menu.

The **Time lapse/domain / Time lapse inversion** command starts the consecutive inversion of each measurement cycle (subset) of the time-lapse data set. If the **Time lapse/domain / Next from previous** option is enabled, each subsequent iteration uses the results of the previous iteration as a reference model. If this option is disabled, the current model of the given subset is used as a reference model for each subsequent iteration. The resulting models are displayed in the model section.

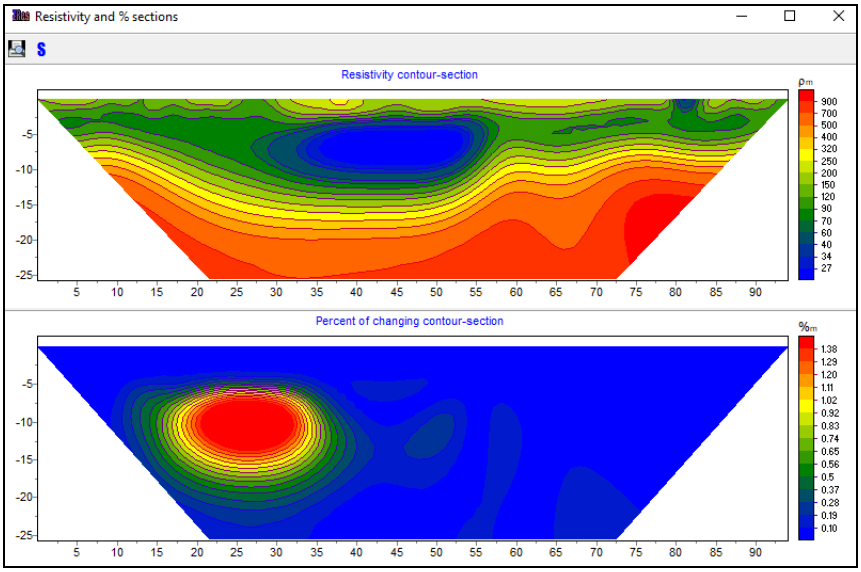
The **Time lapse/domain / Display channel plot** command opens a window displaying a time function for the current measurement point.

The **Time lapse/domain / Save time shots** command saves the results of time-lapse data inversion for all measurement cycles into a single text file.

The figure below shows an example of a time-lapse resistivity survey carried out at a water reservoir over a period of four days.



The **Options / Extra / Monitoring summary** command opens a window containing time-lapse survey summary sections. Two sections are displayed: 1) a section of the parameter's average values; 2) a section of the parameter's variation over time (in percentage).



Joint inversion with MT data

The magnetotelluric (MT) method uses natural time variations of the Earth's electromagnetic field. If magnetotelluric measurements have been conducted on the same line, a joint inversion of MT data with DC resistivity and IP data can be performed. The results of joint interpretation will be more reliable, provided that the depth ranges of both methods overlap significantly. The joint inversion of MT and ERT data allows the determination of both longitudinal conductivity (from MT data) and transverse resistivity (from ERT data) of geological layers, significantly reducing the non-uniqueness of the inverse problem solution and increasing the amount of useful geological information the survey can provide.

In addition to MT data, the joint inversion can be carried out with the data of the magnetovariational (MV) method, which is based on measuring the vertical component of the Earth's natural magnetic field. **ZondRes2D** performs inversion of one of the functions of the MV method – tipper.

It should be noted that **ZondRes2D** is not designed for full-scale processing and analysis of MT data. For this, you should use the **ZondMT2D** program. It is supposed that the input file for joint inversion contains already processed data in the desired (trimmed) frequency range for the magnetotelluric impedance tensor components and tipper. The components of the magnetotelluric impedance tensor are aligned with the profile direction (for the TM-mode) and across the profile direction (for the TE-mode).

Operations with MT data are performed using the **Options / MT Data** menu subsection. The MT data file format is M2D. The data in the M2D file format are imported into the project using the **Options / MT data / Load MT data** command.

The M2D file format has the following structure. The first line contains a list of periods (in seconds). Then follows the data in the form of blocks containing descriptions of each data point. The description of a data point (the block) starts/ends with a line containing an opening/closing curly bracket. The next line after the opening curly bracket contains spatial information – distance along the line (km), elevation (km) and XY coordinates of the data point. Then follow four lines with the data itself. Each line corresponds to a certain parameter type and starts with a key: **ro_a_tm** (apparent resistivity for the TM mode), **ro_a_te** (apparent resistivity for the TE mode), **phi_tm** (phase for the TM mode), **phi_te** (phase for the TE mode). After the key, the values for each period are written in the order corresponding to the list of periods in the first line of the file.

Below is an example of a data file with two data points and three periods:

```
1,0526E-06      1,1765E-06      1,3333E-06      !periods(s)
{
0,00    0  ! pos(km) elevation(km)
ro_a_tm 64,9    62,5    57,8    55      !apparent resistivity
phi_tm  50,1    51,8    53,7    51      ! phase(degree 0-90)
ro_a_te 62,3    57      56,3    54,1
phi_te  51,1    52,4    50,5    49,4
}
{
0,010   0
ro_a_tm 62,3    57      56,3    54,1
phi_tm  51,1    52,4    50,5    49,4
ro_a_te 64,9    62,5    57,8    55
phi_te  50,1    51,8    53,7    51
}
```

After opening a data file, the **Enter shift of stations in meters** dialog box appears, where you can specify line distances for all data points. This operation is required to align the MT data profile with the ERT profile, if necessary.

If the **Options / MT data / Invert MT data** option is enabled, the program will perform the joint inversion of the MT and ERT data. The total weight of the MT measurements can be set using the **Options / MT data / Set weight of MT** menu command.

Before running the inversion, the MT data components to be inverted should be selected. This is done using the **Options / MT data / Components** menu command. The available components are:

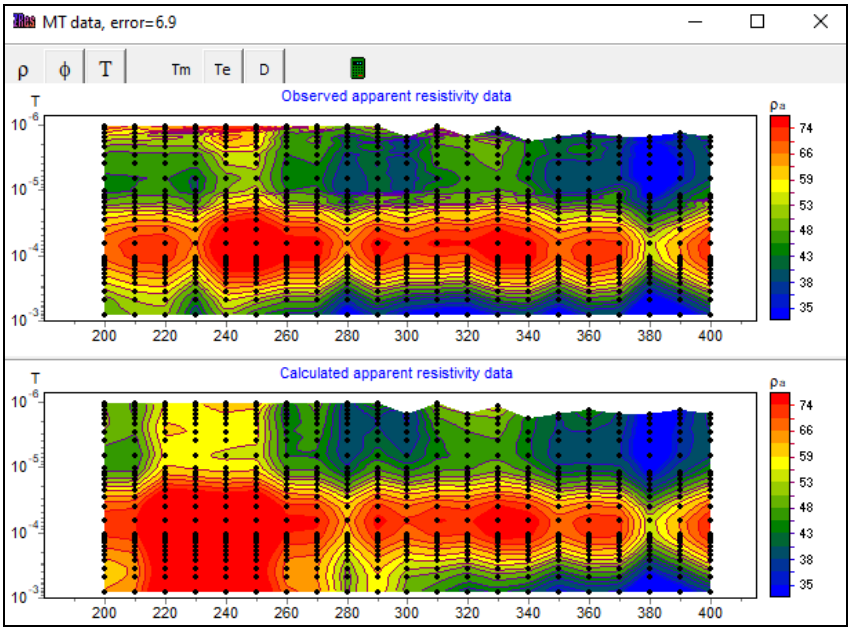
TM inversion – curves calculated along the profile;

TE inversion – curves calculated across the profile;

Tipper inversion – tipper data;

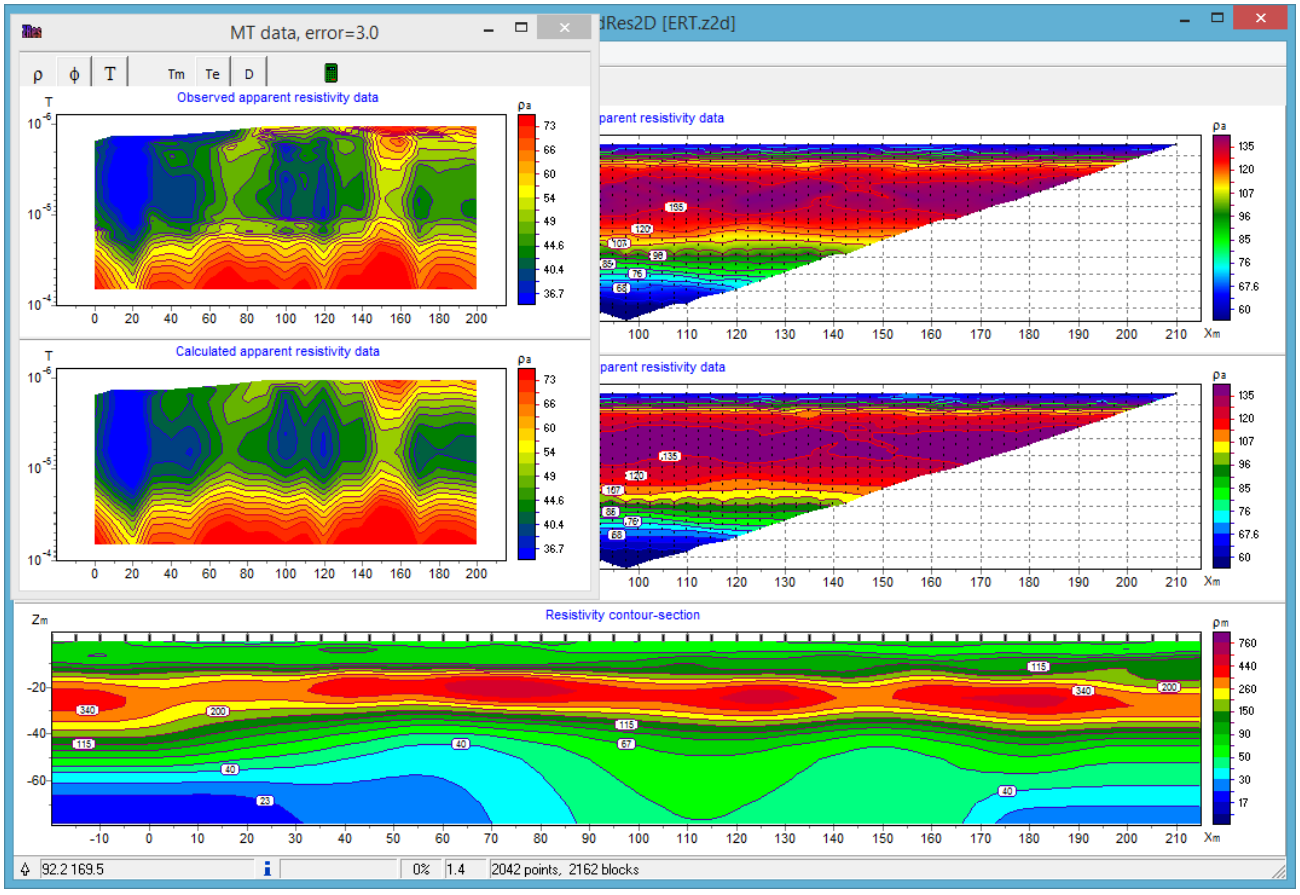
DET inversion – invariant (effective) curves.

The **Options / MT data / Show MT plot** command opens a module for visualizing the MT data. In this module, you can switch between the components of the MT data and visualize both the apparent resistivity and the phase of magnetotelluric impedance. In the title bar of the window the discrepancy (in percentage) between the observed and computed MT data is displayed, e.g., "error=6.9". The buttons for switching between the components being visualized are located on the toolbar. The  button calculates the forward problem solution. The main window area contains two pseudo-sections corresponding to the observed and computed data and color scales for the displayed parameter.



With the **Options / MT data / With static shift** option enabled the program will determine static shifts of the MT curves during the inversion. It is recommended to enable this option after several iterations.

The figure below is an example of the results of joint inversion of ERT and MT data.



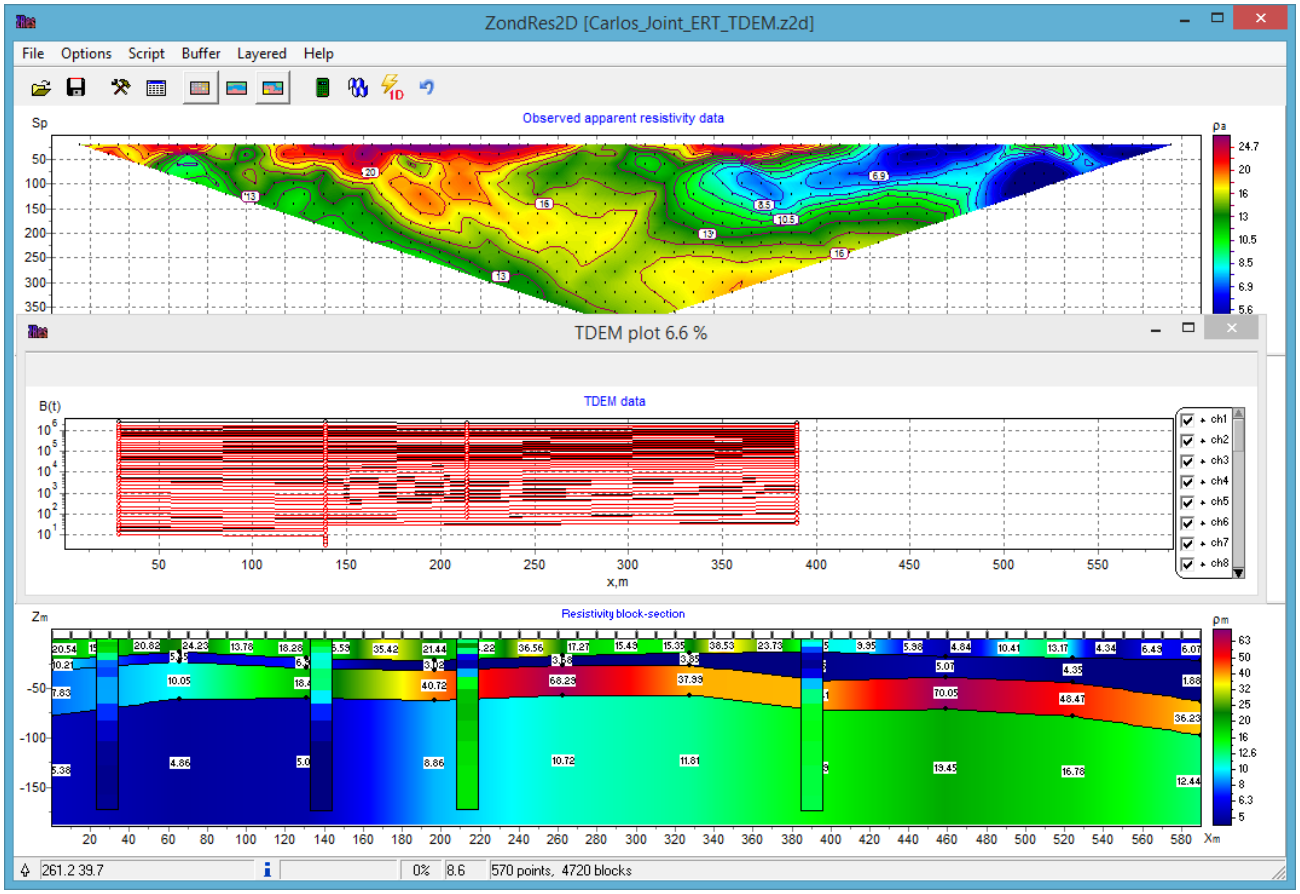
Joint inversion with TEM data

When working in the layered model mode (the button on the main window toolbar), data of the transient electromagnetic (TEM) method in time domain (TDEM) and frequency domain (FDEM) can be imported into the project.

When enabling the layered mode, the **Options / TDEM data** menu subsection appears.

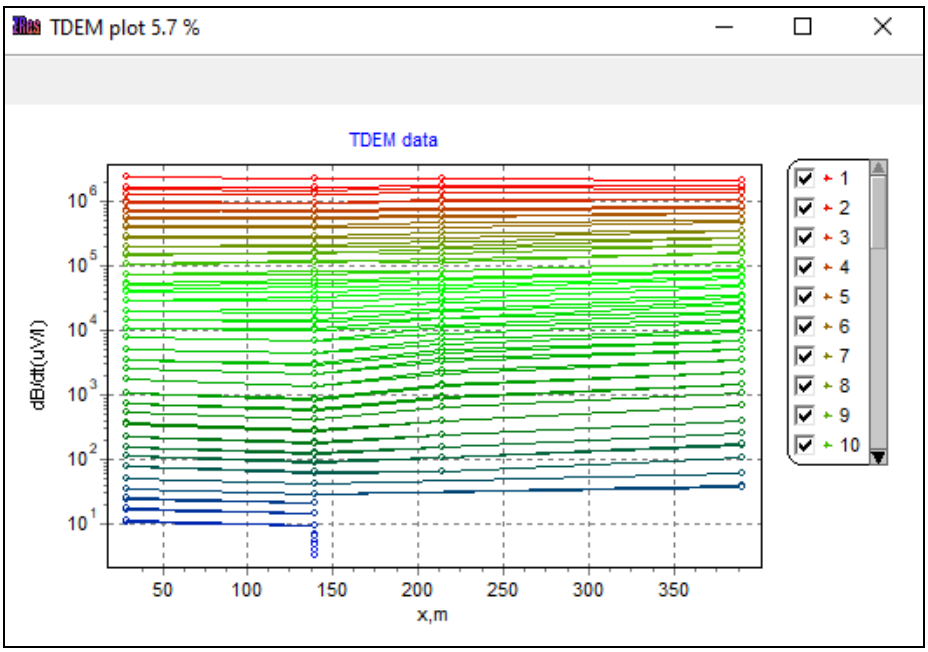
The **TDF** (the **ZondTEM1D** program file format) and **USF** (Universal Sounding Format) text files are used to import the TEM data. The coordinates (line distances) should be in the same coordinate system as the current model.

If the **Options / TDEM data / Invert TDEM data** option is enabled, the program will perform the joint inversion of the TEM and ERT data. An example of the joint inversion results is shown in the figure below. Results of independent 1D interpretation of the TEM data are displayed in the form of columns in the model section.



The total weight of the TEM data can be set using the **Options / TDEM data / Set weight of TDEM** menu command. Assigning weights allows you to control joint inversion misfits.

The **Options / TDEM data / Show TDEM plot** command opens the window containing the TEM data plots. In the case of time-domain soundings (TDEM), the plots represent isochrones of measured and computed values of the transient electromagnetic field.



Joint inversion with seismic data

Joint inversion with seismic data can be performed in the mesh model mode (cross-gradient inversion) or arbitrary layered model mode.

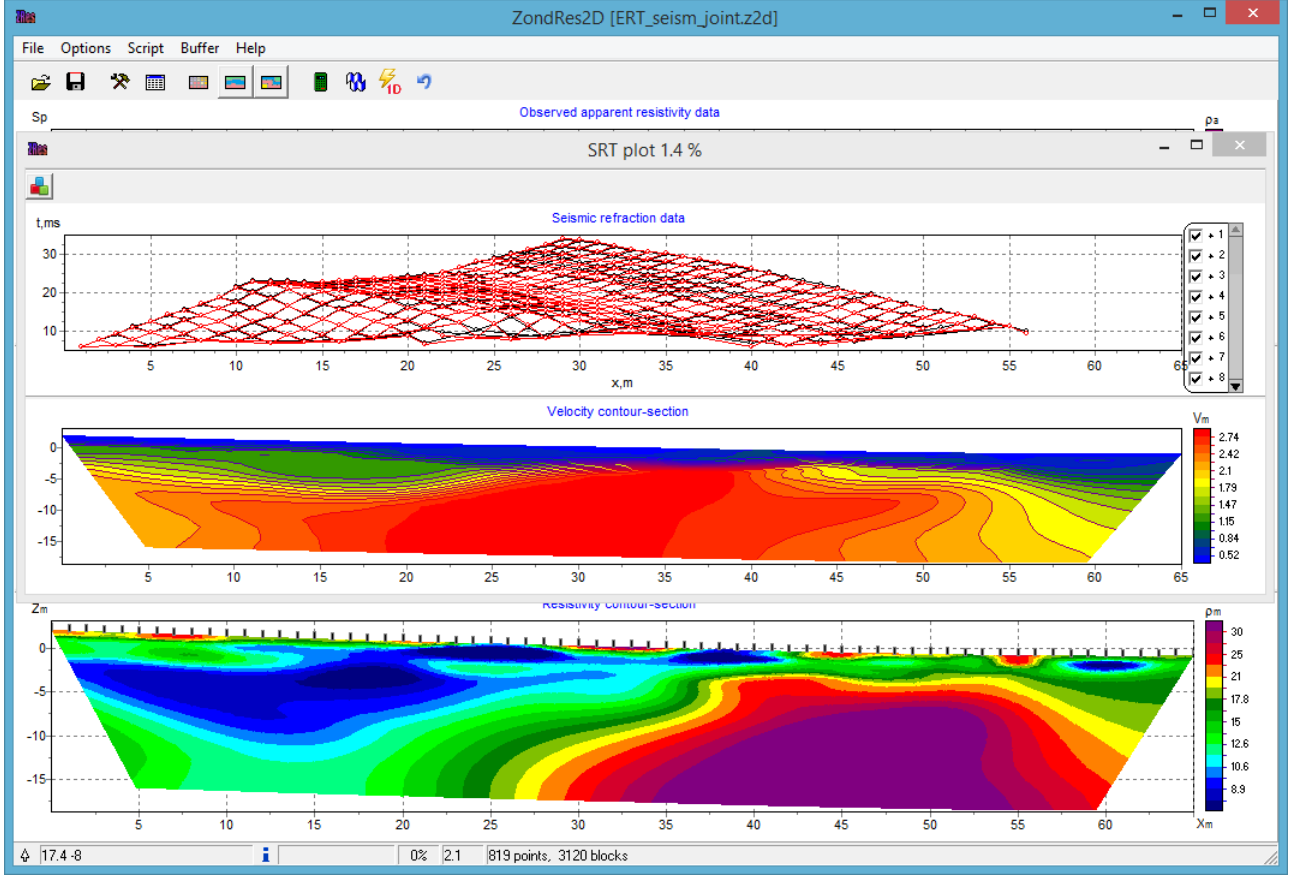
Operations with seismic data are performed using the **Options / Seismic data** menu subsection. The data is imported into the project using the **Options / Seismic data / Load SRT data** command.

A simple text file format is used to import the data, containing three columns with the following headers: **sx** – source coordinate (m), **rx** – receiver coordinate (m), **fb** – first arrival time (ms). The coordinates (line distances) should be in the same coordinate system as the current model.

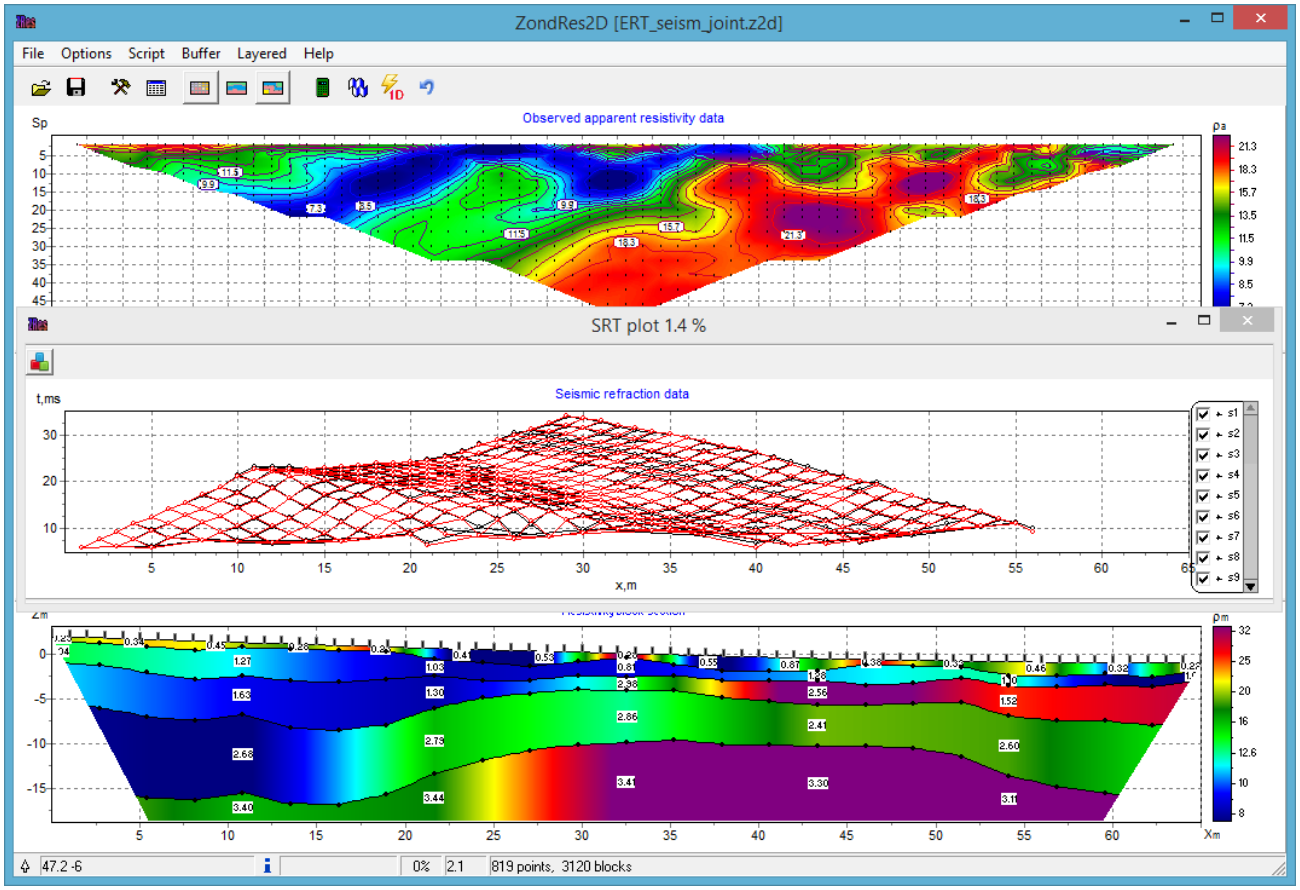
Below is a sample of a seismic survey data file:

```
sx  rx    fb          ! source X pos (m), receiver X pos (m), first arrival (ms)
0   1     19.8
0   2     30.1
0   3     41.8
```

The **Options / Seismic data / Show SRT plot** command opens the window with the seismic data visualization. If joint inversion is performed in the mesh model mode, the velocity model will be shown in the lower section of the window.



If the **Options / Seismic data / Invert SRT data** option is enabled, the program will perform the joint inversion of the seismic and ERT data in the arbitrary layered model mode. An example of the joint inversion results in the form of a layered model is shown in the figure below.



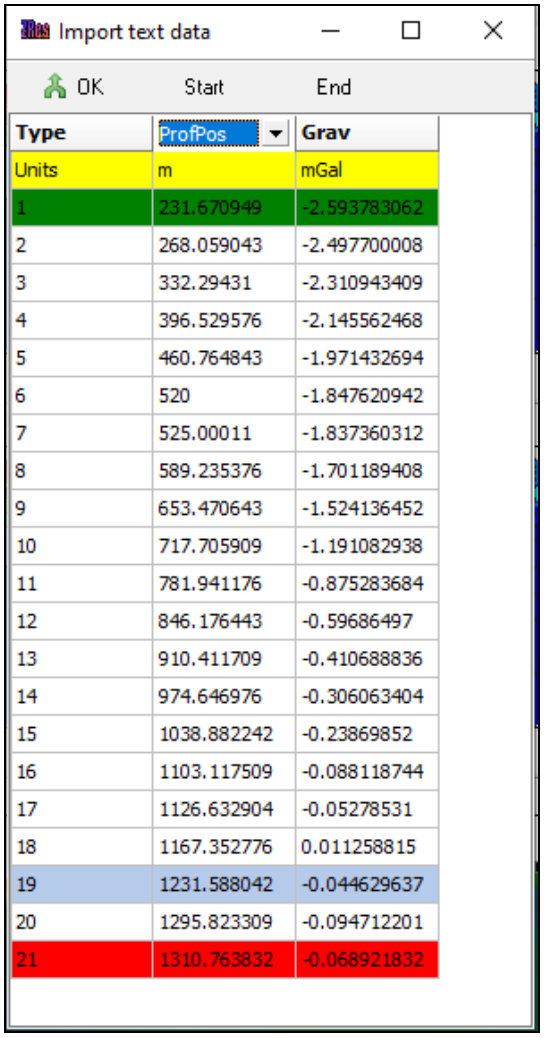
The total weight of the seismic data can be set using the **Options / Seismic data / Set weight of SRT** command. Assigning weights allows you to control joint inversion misfits.

Joint inversion with gravity and magnetic data

Inversion of gravity and magnetic data in **ZondRes2D** can be performed in all modeling modes.

For the gravity data, the program can invert measurements of the vertical component of the gravity field and its vertical gradient, for the magnetic data – measurements of the total vector and vertical gradient of the magnetic field.

Data of gravity/magnetic surveys are imported into the program from a text file or Excel spreadsheet containing two columns, with distances along the line in one column and observed values in another. To import the data, select the **Options / GraviMagnetic / Load new data** menu command. An example of data being imported is provided in the figure below.

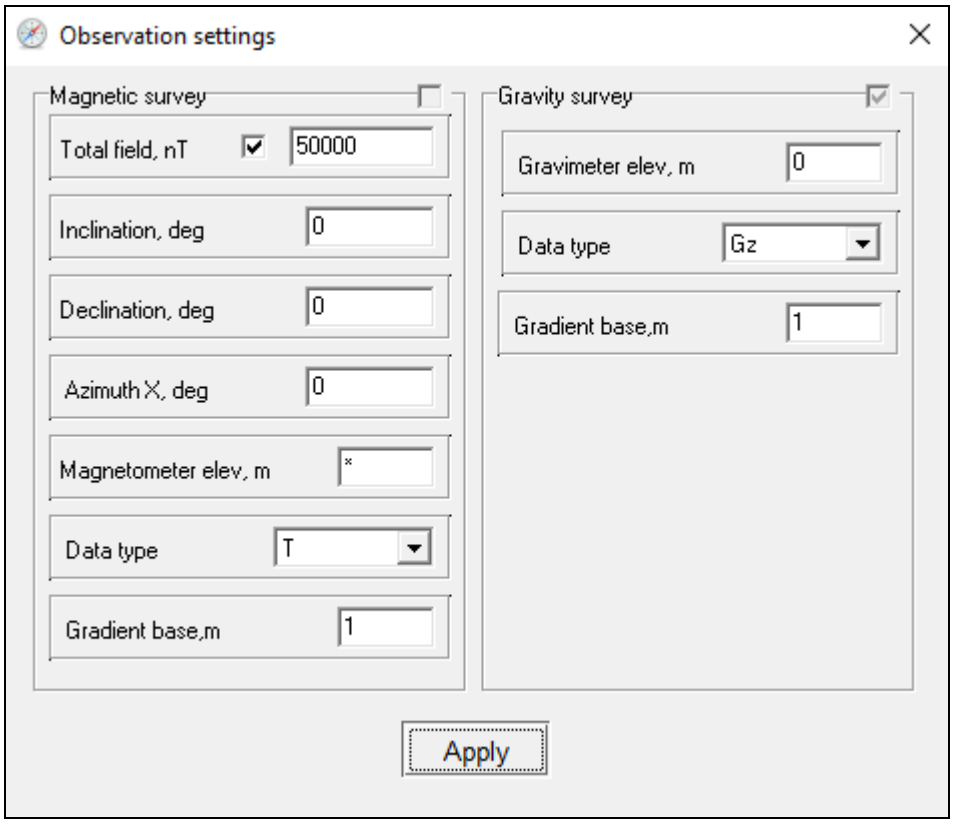


Type	ProfPos	Grav
Units	m	mGal
1	231.670949	-2.593783062
2	268.059043	-2.497700008
3	332.29431	-2.310943409
4	396.529576	-2.145562468
5	460.764843	-1.971432694
6	520	-1.847620942
7	525.00011	-1.837360312
8	589.235376	-1.701189408
9	653.470643	-1.524136452
10	717.705909	-1.191082938
11	781.941176	-0.875283684
12	846.176443	-0.59686497
13	910.411709	-0.410688836
14	974.646976	-0.306063404
15	1038.882242	-0.23869852
16	1103.117509	-0.088118744
17	1126.632904	-0.05278531
18	1167.352776	0.011258815
19	1231.588042	-0.044629637
20	1295.823309	-0.094712201
21	1310.763832	-0.068921832

If the survey data are in different files (e.g., mag data in one file and gravity data in another), you can use the **Options / GraviMagnetic / Add new data** command to append data to the project.

After selecting the file, the data import window appears (see above). Cells in the uppermost row of the table contain a drop-down list of headers: ***ProfPos*** (distance along the line), ***Grav*** (gravity data), and ***Mag*** (mag data). For each column, it is necessary to select the corresponding header. The second row specifies the units of measurement: for gravity data – milligals (mGal) or microgals (uGal), for magnetic data – nanoteslas (nT). Use the ***Start*** and ***End*** buttons at the top of the dialog box to specify the range of data to be imported (the ***Start*** button specifies the first row, the ***End*** button specifies the last row).

After setting the import criteria and pressing the OK button, the **Observation settings** dialog box appears. The dialog box contains settings for survey parameters and parameters of the normal magnetic field, and can also be opened at any time using the **Options / GraviMagnetic / Field settings** command.



The **Gravity survey** group box contains the following options:

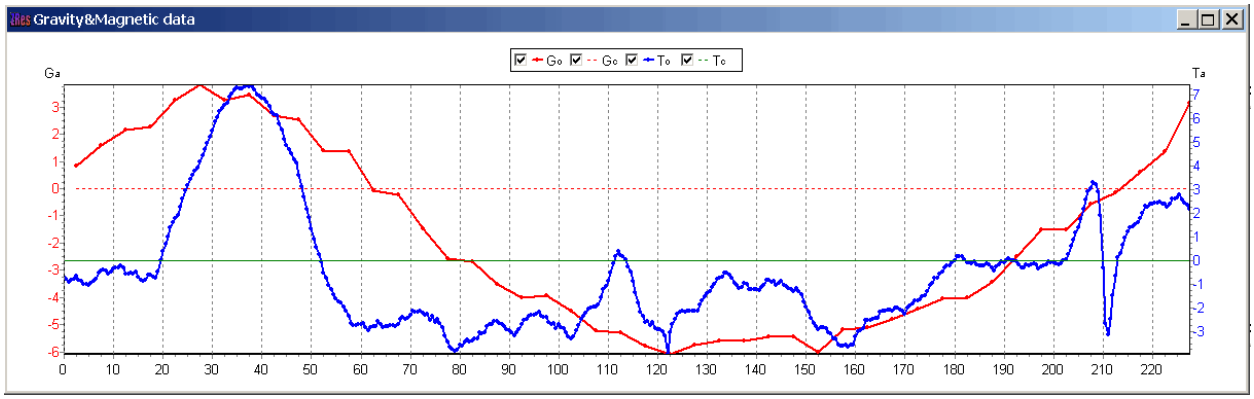
Gravimeter elev, m
Height of gravimeter above the ground (m).

Data type
Data type – vertical component of gravity field or its vertical gradient.
Gradient base, m
In the case of vertical gradient data, the measurement base (m).

The **Magnetic survey** group box contains the following options:

Total field, nT
Amplitude of the normal field (nT).
Inclination, deg
Magnetic inclination at the time of survey (degrees).
Declination, deg
Magnetic declination at the time of survey (degrees).
Azimuth, deg
Azimuth of the survey line (degrees) (from the north direction clockwise).
Magnetometer elev, m
Height of the magnetometer sensor above the ground (m).
Data type
Data type – total vector of magnetic field or its vertical gradient.
Gradient base, m
In the case of vertical gradient data, the measurement base (m) (the distance between sensors).

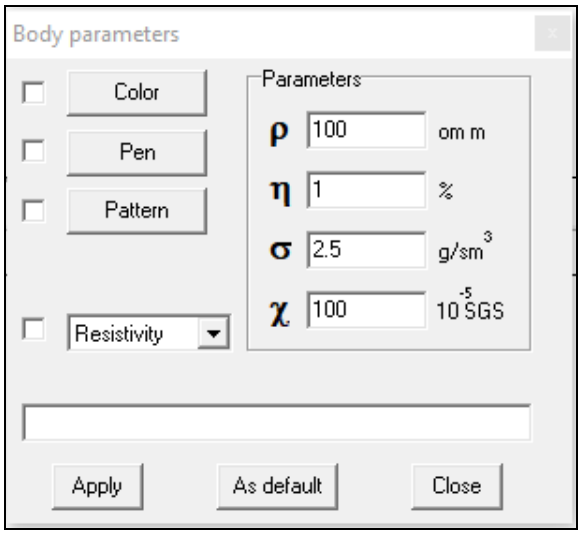
After setting the parameters and pressing the **Apply** button, the gravimagnetic data will appear in a new window. The window can also be opened at any time using the **Options / GraviMagnetic / Display GM window** command.



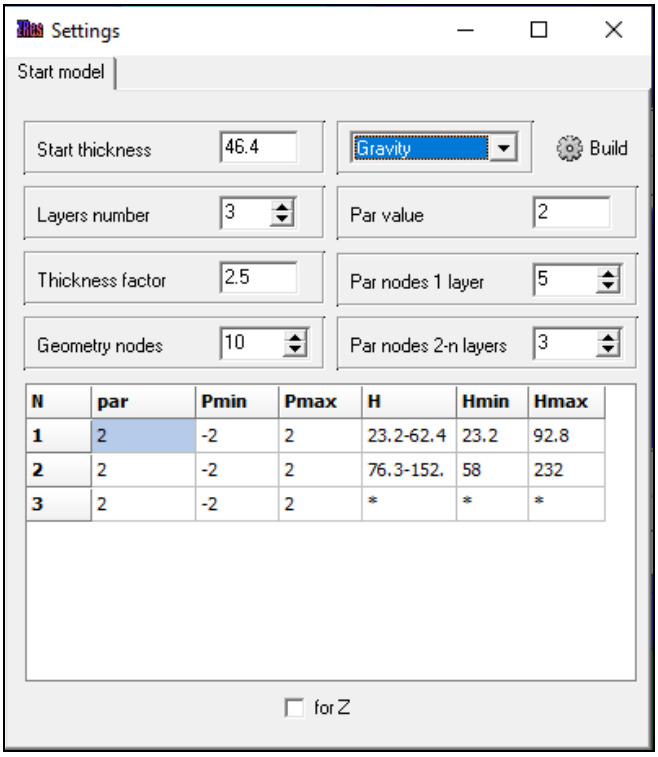
The **Options / GraviMagnetic / Substract median grav** and **Substract median mag** options perform the subtraction of the median value from the observed field, i.e., reduce the field to the anomalous component.

After creating a polygonal model in the model editor, you can either calculate the forward gravity/magnetics problem (perform modeling in the polygonal mode) or perform the data inversion. Double-clicking on the gravimagnetic data window recalculates the forward problem for a current polygonal model. To start the inversion of density and/or magnetic susceptibility for a given framework of polygons, use the **Options / GraviMagnetic / Inversion** command.

To set the density and/or magnetic susceptibility of a polygonal object, use the **Body parameters** dialog box (see the **Polygonal modeling** section for details).



In the layered model mode, the density or magnetic susceptibility can be set by selecting the **Gravity** or **Magnetic** option in the model constructor settings dialog box **(Layered / Model constructor)**.



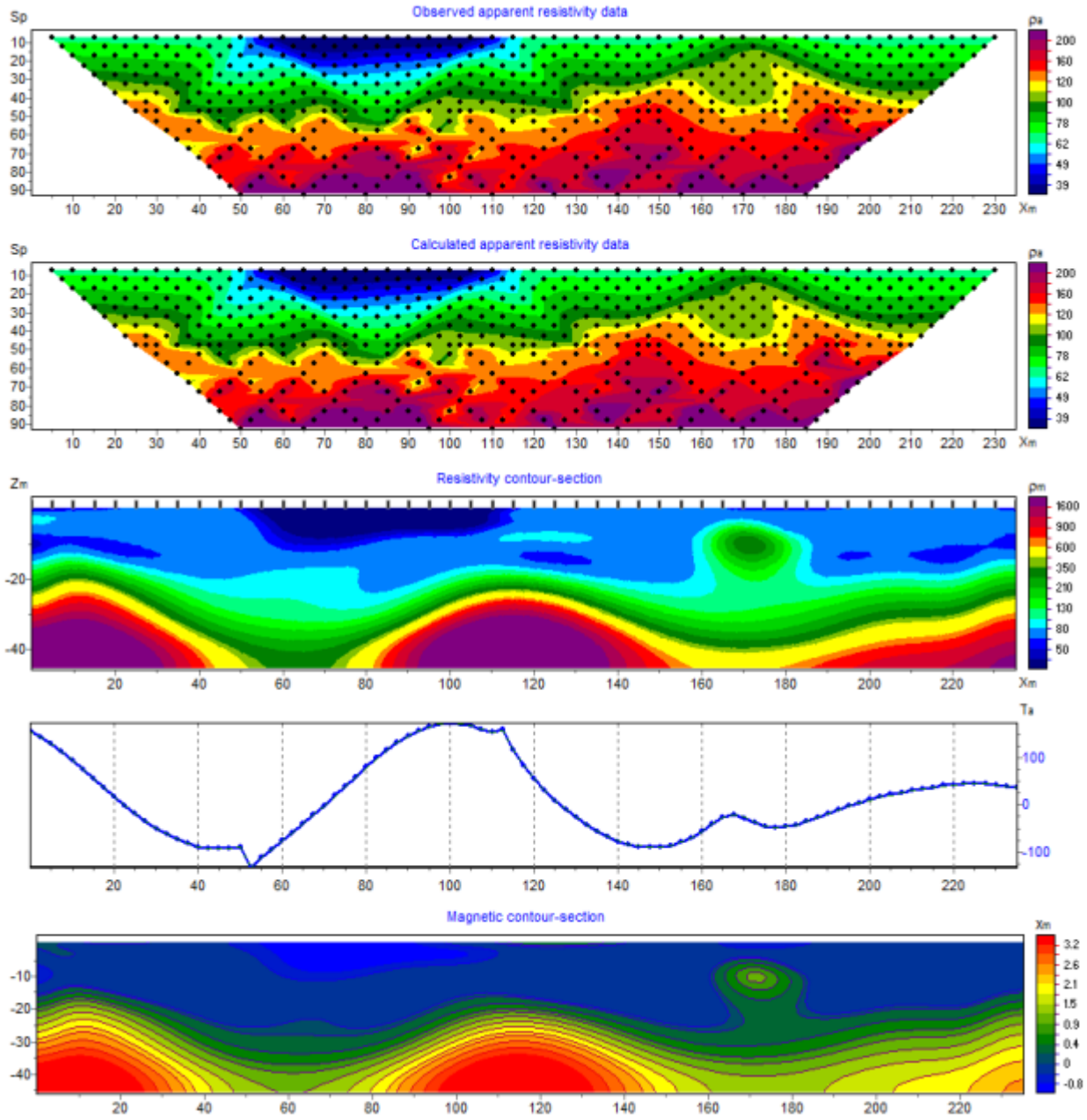
Joint inversion can be performed in all three modeling modes:

Block (mesh) mode. A joint inversion of ERT data with gravimagnetic data is performed. The **Options / Inversion / Cross-gradient** menu subsection is used.

Polygonal mode. The inversion is performed independently for ERT data and gravimagnetic data within the same polygonal model framework. The **Options / GraviMagnetic / Inversion command** is used.

Layered mode. A joint inversion of ERT data with gravimagnetic data is performed based on the common geometry of boundaries. The **Options / GraviMagnetic / Invert gravity** and/or **Options / GraviMagnetic / Invert magnetic** options are used.

The figure below shows an example of cross-gradient joint inversion of ERT and magnetic survey for a block (mesh) model.



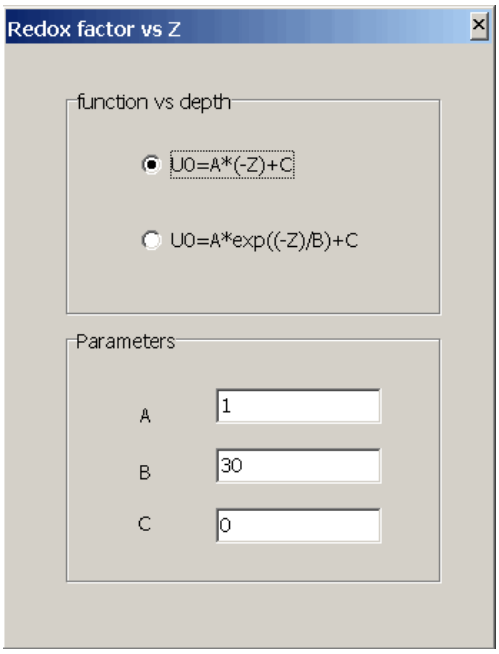
Interpretation of self-potential (SP) data

ZondRes2D provides a solution for the forward problem of the SP method based on the resistivity of the medium and specified redox potential depth distribution function, as well as the algorithm of inversion of volumetric sources in a specified polygon. The SP interpretation options in **ZondRes2D** are intended mainly for solving mineral exploration problems and are not suitable for the interpretation of filtration potentials. For the latter, we recommend using the **ZondSP2D** program which features more advanced options.

The **Options / Self potential** menu subsection is available when working in the polygonal modeling mode.

The input file of the SP data consists of four columns: 1) the data point location (the survey line coordinate to which the measured value belongs) (m); 2) M electrode coordinate (m); 3) N electrode coordinate (m); 4) measured value (mV). Thus, the program can work with the data collected with both the potential and gradient methods. The file is imported using the **Options / Self potential / Load SP data** command.

The **Options / Self potential / Display SP window** command opens the window displaying the observed and calculated SP data plots. The calculated data plot will be displayed after the parameters of the redox potential depth distribution are specified in the **Options / Self Potential / Redox factor vs Z** dialog box and the forward problem is solved (by pressing the  button or the Space key).



The variable parameters of the redox potential are the redox potential depth distribution function (linear or exponential) and the corresponding coefficients.

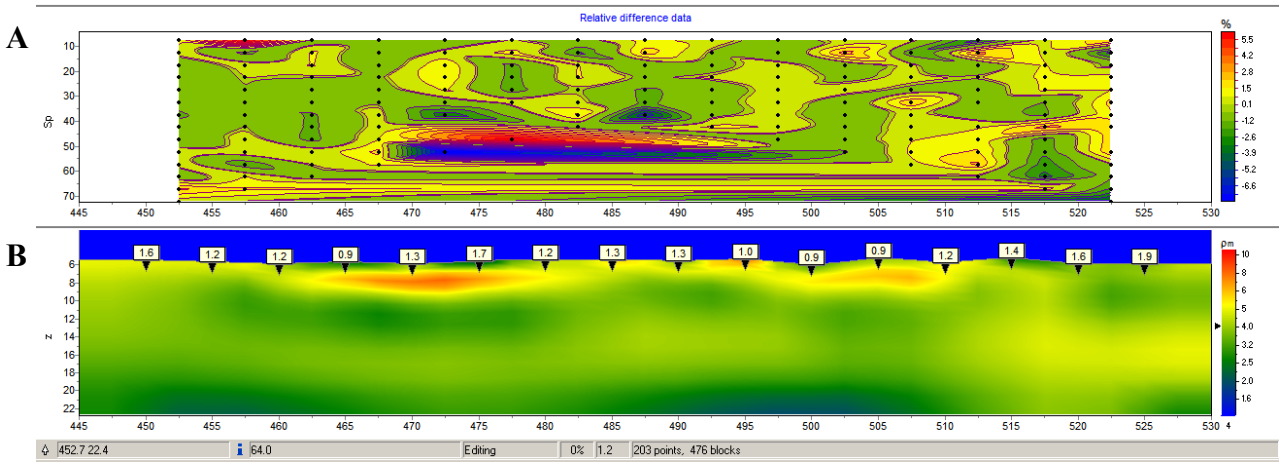
The option to specify the ionic or electronic type of conductivity in the properties of polygons is available when SP data are loaded into the project.

Estimating the inversion misfit

A quick estimation of the inversion results can be given by the RMS error. The percentage value of the RMS error is displayed in the central part of the main window status bar. As a rule, if the data quality is satisfactory, the value should not exceed 5%.

The convergence between the observed and calculated values for each measurement can be evaluated using the pseudo-section of relative misfits which is displayed when the **Options / Data / Data Misfit** option is enabled (section A in the figure below).

To display the RMS error for each electrode, the **Options / Extra / Display electrode RMS** option is used (section B in the figure below). The ***Visible*** checkbox in the **Marks** tab of the **Graphic editor** dialog box should be activated (the editor is opened by right-clicking on an electrode in the model section while holding down the Shift key); recalculating the forward problem might be required.



The estimation of misfits allows you to re-assess the data quality and remove data points with larger misfits. This can be performed using the **Options / Extra / Remove data with big misfit** option.

The **Options / Inversion / Invert only visible graphs** option can be used to exclude a portion of the data from inversion. To do this, switch the data display into the graphic plot mode and use the legend to disable the graphs you do not want to include in the inversion.

A priori information

Utilizing additional a priori information provides a comprehensive data interpretation and increases the reliability of survey results.

Incorporating a priori information into the inversion is realized, most often, in two ways: by setting the starting model (the presumed distribution and limits of its parameters) and by setting the position of sharp boundaries.

By default, the starting model for inversion in **ZondRes2D** is the current model displayed in the model section. It is updated by the program with each iteration during the inversion.

The known position of sharp boundaries available from borehole or geophysical data is most often used as a priori information.

The boundaries are plotted in the boundary editing mode accessible through the **Options / Inversion / Set boundaries** menu command, which brings up a floating toolbar containing the following buttons:

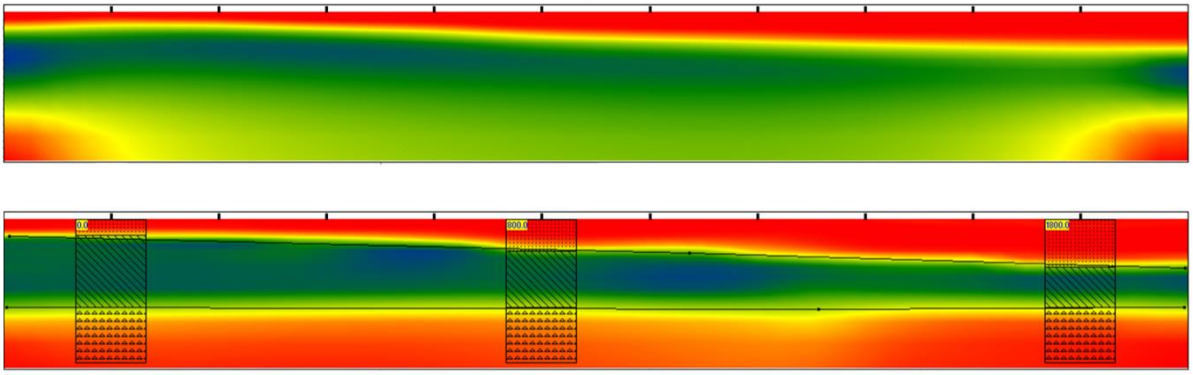
	Enable/disable boundary editing mode
	Add new boundary
	Remove all boundaries
	Save boundaries to file
	Load boundaries from file

Introducing a priori geological boundaries into the inverse problem is an essential method to improve the quality of interpretation. It increases the stability of the problem while reducing the area of non-uniqueness and allows obtaining a more consistent structure. In the areas of the model where the parameters are only slightly sensitive to changes in the section, the incorporation of a priori boundaries is practically the only way to obtain acceptable results.

It is preferred to use the ***Occam*** algorithm when a priori boundaries are incorporated into the model. Typically, only 1 or 2 boundaries are used; it is also worth noting that geological and geophysical boundaries do not always coincide.

It is recommended first to perform the ***Occam*** inversion and then plot the boundaries over the resulting model. When the boundary editing mode is enabled, boundary nodes are added by left-clicking on the model. Right-click adds the last node and finalizes the boundary. Avoid using too many nodes and plotting highly undulating boundaries; try to set the boundaries near the nodes of the model mesh. After plotting the boundaries, the inversion should be run again, this time taking into account the incorporated boundaries.

The top section in the figure below was obtained using the Occam algorithm without a priory information; the bottom section was obtained after performing the inversion with incorporated boundaries.

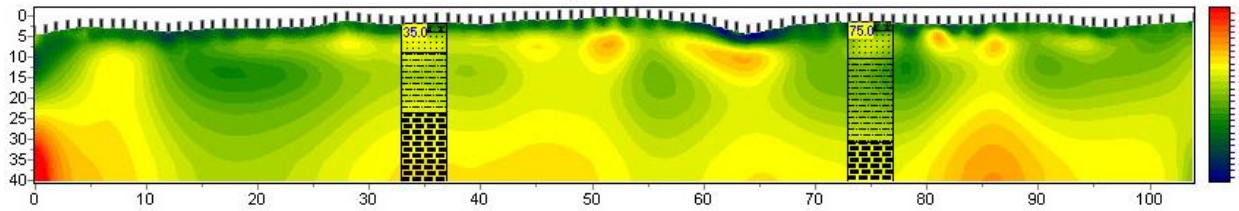


Another way of incorporating a priory information is realized in the **Image guided inversion** algorithm (**Options / Inversion / Cross-gradient / BG image**) which performs the inversion based on the similarity of the inversion model to a loaded graphic image.

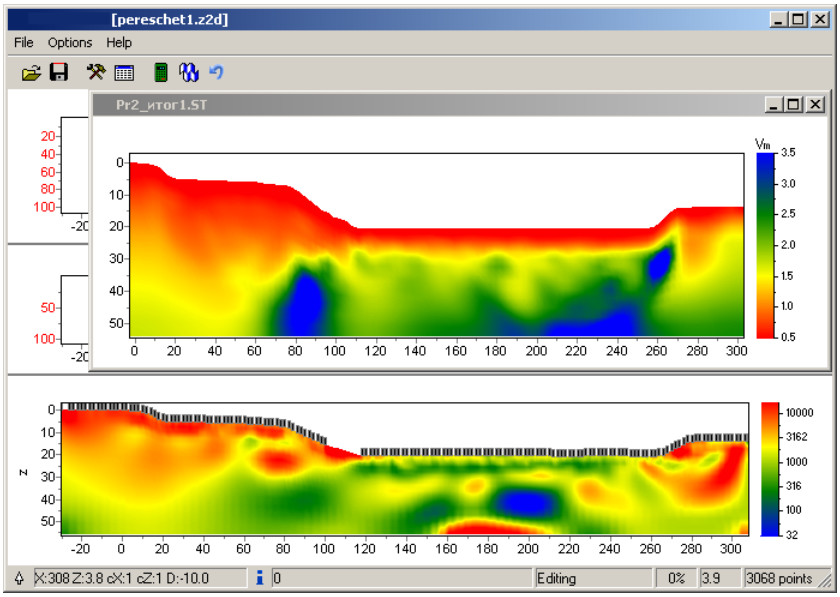
Several methods of visualizing a priori information are implemented in the program. Using the **Options /Import/Export** menu subsection, you can import a variety of geological and geophysical information:

- borehole columns;
- geophysical logs;
- horizontal profiling data in the form of graphs;
- models from other **Zond** programs;
- background graphic images (e.g., a geological or seismic section).

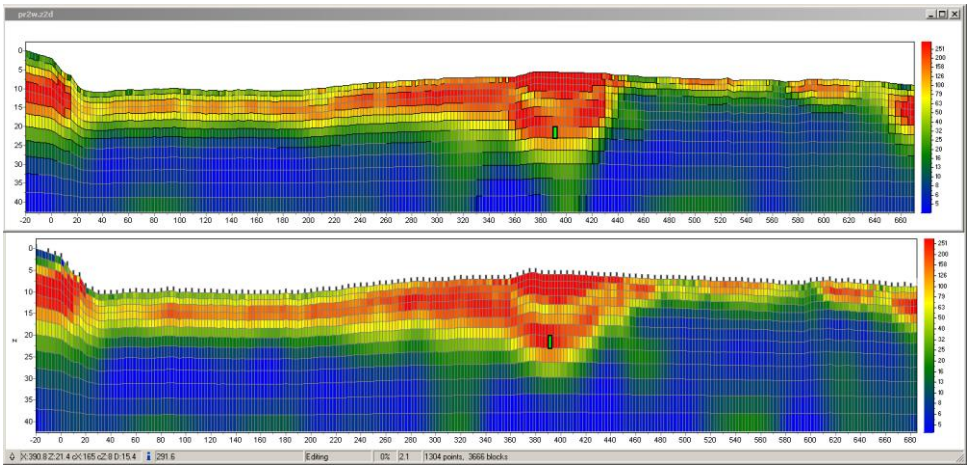
If borehole data are available, the columns and geophysical logs can be loaded into the model section using the **Options** / **Borehole** / **Load borehole data** command or created directly in the program interface using the editor.



The **Options** / **Import/Export / Import model/data** option allows you to open models created in **Zond** projects in separate windows. The option can be used to compare interpretation results of neighboring survey lines or when interpreting data from different methods.



When moving the cursor in the model area of the main program window, the corresponding cells of imported models opened in separate windows will also be highlighted, according to the size of the active cell.



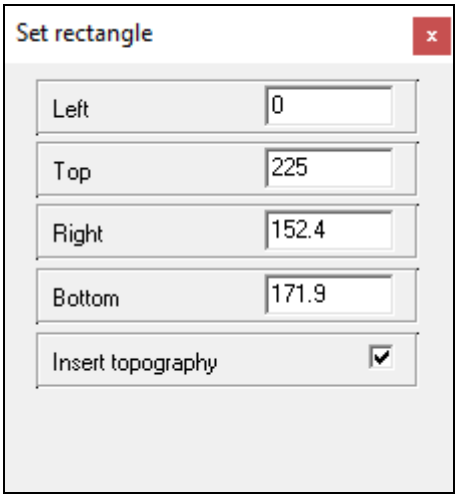
It is possible to save and load a fragment of the mesh model using the **Options** / **Import/Export / Model parts / Save selection** and **Load selection** commands. To save a fragment, select the desired fragment using appropriate selection tools and click **Save selection**. To load a model fragment, select a small area of the current model; the upper left corner of the selection will be a reference point for the fragment to be embedded. If there is no selection, the fragment will be inserted in the upper left corner of the model.

You can export and import a depth profile of the model parameter at a given horizontal location by using the **Options** / **Import/Export / Model parts / Extract 1d log** and **Load 1d log** commands. When exporting, the X coordinate of the depth profile should be specified in the export dialog box. When importing a depth profile, a range of X coordinates can be specified. This option

can be used, for example, for incorporating borehole resistivity logs or to compare inversion results at the intersection of two survey lines.

A background image can be imported into the model section using the **Options / Import/Export / Background image** command. It could be a geological or geophysical section, e.g., a seismic section or a resistivity model obtained at a neighboring survey line. Graphic files in PNG and BMP formats and geophysical data files in the SEG-Y format are supported. Auxiliary spatial reference files in the internal SEC format can be used.

After selecting a graphic file (BMP or PNG), a dialog box appears in which you should specify coordinates of the image boundaries according to the coordinate system of the model section.



In this dialog box, you can manually set the coordinates of the top left and bottom right corners of the image.

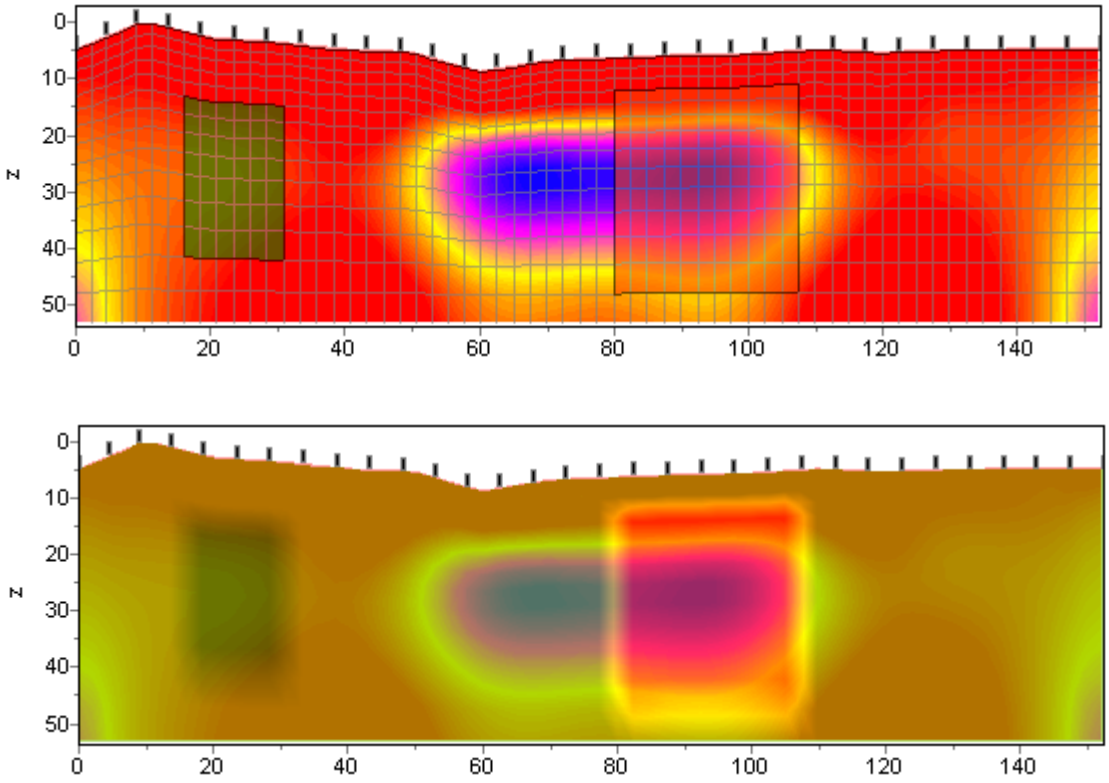
If the **Insert topography** checkbox is checked, the imported image will be transformed to reflect the topography of the model (the top boundary of the image will have the same geometry as the topography profile).

SEC is a reference file format, which is used to assign coordinates to imported images in accordance with the coordinate system of the model section (an example can be found in the sample_with_sectfile directory). The file has the following structure: the first line contains the name of the image file, including the file format; the second line contains the four coordinates of the top left and bottom right corners of the image in the X1 Y1 X2 Y2 order, separated by space.

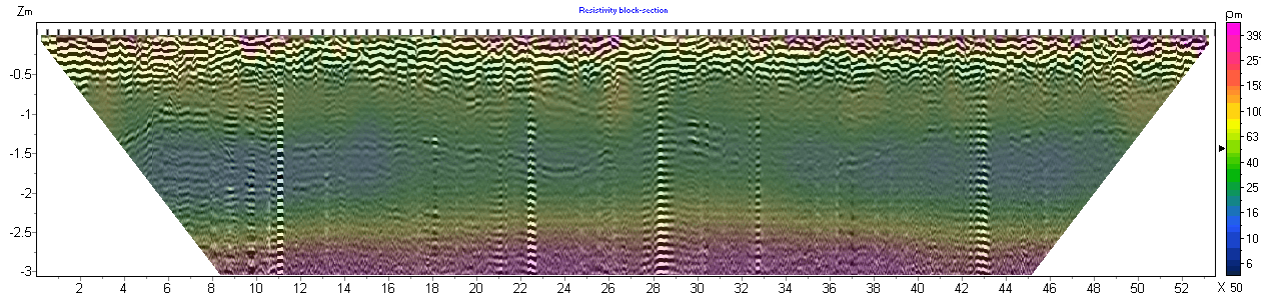
To change the transparency of the image, use the **Transparency** option in the **Model setup** dialog box (right-click on the title area of the model section, the **Setup** option in the context menu).

In the *Block-section* display mode, the cells with values differing from the host medium will stand out from the background image (the top section in the figure below). In this way, it is possible to model anomalous objects over the background. In the *Smooth-section* display mode, the

colors of the background image and the current model will blend, giving you the opportunity to observe the features of the two sections at the same time (the bottom section in the figure below).



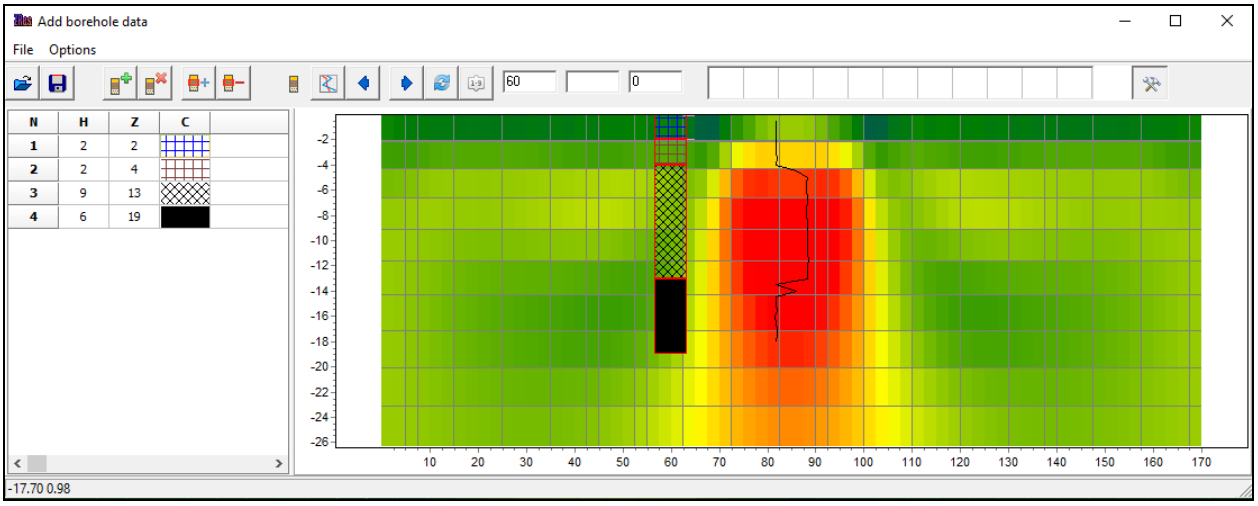
Importing a background image allows performing the ERT data interpretation taking into consideration results of other surveys. For example, using sections obtained through seismic surveys, GPR or geological mapping, a priori boundaries can be specified prior to ERT data inversion. The figure below shows an example of a GPR image underlying a resistivity model.



Models created in **ZondRes2D** can be exported to the most widespread program formats: AutoCAD, GeoSoft and SEG-Y. The model can also be exported as a raster image of a user-defined resolution and size set in the **Picture settings** dialog box (**Options / Graphics / Bitmap output settings**).

Creating borehole columns and logs

Creating borehole columns and geophysical logs is performed in a special module opened with the **Options** / **Borehole** / **Create/Edit borehole data** command. Using this module you can create, edit and visualize the borehole data along the survey line.



The dialog box toolbar contains the following buttons:

	Open borehole file
	Save borehole file
	Create new borehole column or log
	Remove borehole column or log
	Add borehole column layer
	Remove borehole column layer
	Borehole column mode
	Logging data mode
	Go to previous borehole/log
	Go to next borehole/log
	Re-plot data plotting area

	Sort boreholes/logs by coordinate
	Set horizontal coordinate
	Borehole/log label (no more than 5 characters)
	Borehole inclination angle
	Additional options

The module window is divided into two main sections. The left section contains a table with the following columns: **N –** layer number; **H –** layer thickness in meters; **Z –** depth to the bottom of the layer in meters; **C –** fill color / hatch pattern. The right section displays the graphical representation of the borehole data.

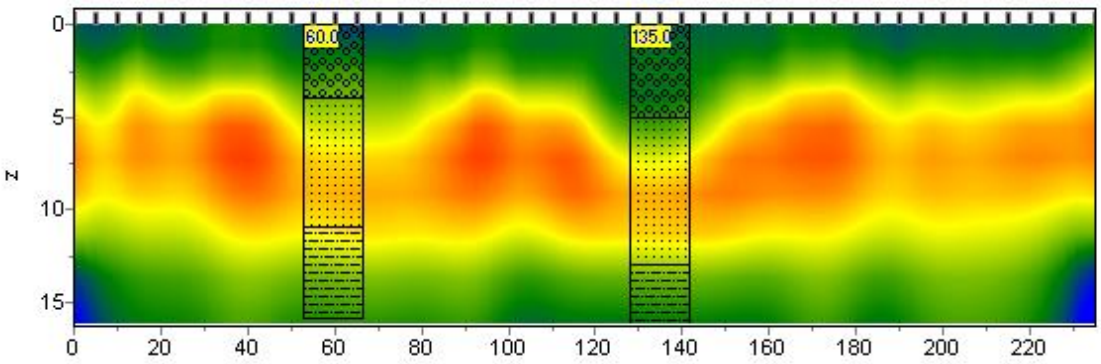
To start creating a borehole column, click the  button on the toolbar. A new table will appear in the left section. Use the  button to set the required number of layers, then specify the thickness or depth of each layer and choose the hatch pattern or fill color according to the borehole geology. Values in the **H** and **Z** columns can be edited by double-clicking on the desired cell. Double-click on a cell in the **C** column opens the **Pattern Color Editor** dialog box for selecting the hatch pattern. The program offers a vast selection of preset patterns; the **Color** button allows you to select a solid fill color.

After filling in the table, press the  button. The active borehole is displayed in red color in the graphic section.

To facilitate working with a large number of boreholes, the program offers the possibility to create a palette. To create a palette, select the desired fill in the **C** column of the table, then right-

click on a cell in the Fill area on the toolbar (a row of empty cells). You can create a set of fills, which can be saved and loaded using the **Save default palette** and **Load default palette** commands of the  context menu.

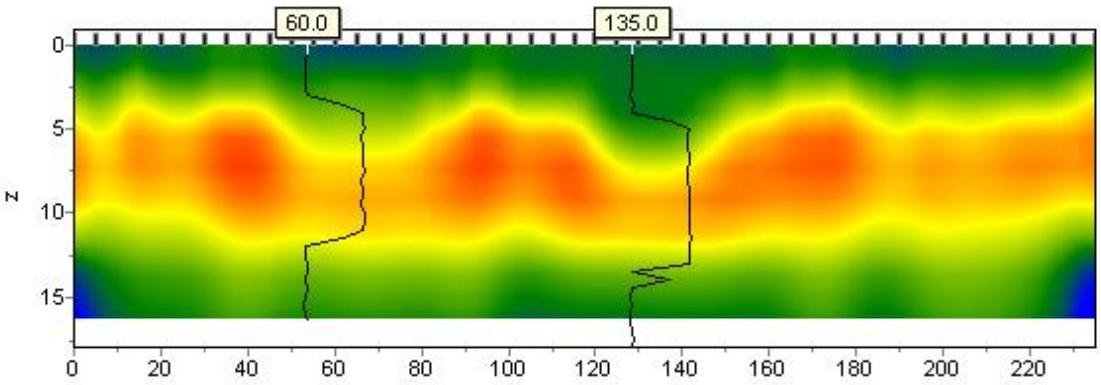
The **Set borehole width** command in the  context menu sets the borehole column width as a percentage of the survey line length.



Logging data are loaded using the  toolbar button (prior to this, a new borehole should be created using the  button). The supported file formats are TXT and LAS.

The text file should have the following structure: the first column contains depths from the ground surface, the second column contains measurement values, the third and fourth columns contain zeros. The columns are separated by Tab.

Below is an example of logging data displayed in the graphic section.



The created borehole columns and uploaded logs can be saved in the internal CRT file format using the  button. The created CRT project file can then be opened in the main module of **ZondRes2D** using the **Options / Borehole / Load borehole data** command.

Although the module provides convenient tools for the interactive creation of borehole columns, there is also a possibility to import borehole data in text format. For this, several text files should be created – one for each of the boreholes and one file that has a CRT format structure. When a project is saved in the module, the created CRT file is a binary file that cannot be opened in a text editor, however, it is possible to create a text file, change its extension to CRT and then open it in the module.

The following is a description of the CRT file structure for importing any number of borehole columns or logs into the module.

2280.txt // The first line is the name of the file containing a column or a log.

Skv2280 // The second line is the borehole label (will be displayed in the module).

18 2 2 1 0 1 0 0// The third line contains the following parameters:

18 – X coordinate of the borehole;

2 – image width (as a percentage of the survey line length, typically 1-20);

2 – type of data (0-3);

0 – logging data (graph);

1 – logging data (interpolated color column) (the color scale of the section is used to display the data);

2 – borehole column;

3 – logging data (color column) (the color scale of the section is used to display the data);

1 – normalization parameter for logging data (0-2);

0, 1 – the same minimum and maximum values are used for all logs;

1, 2 – each log is normalized by its mean value;

0 – logging method index (if data of several methods needs to be displayed, enter different indexes for each of the methods starting with 0);

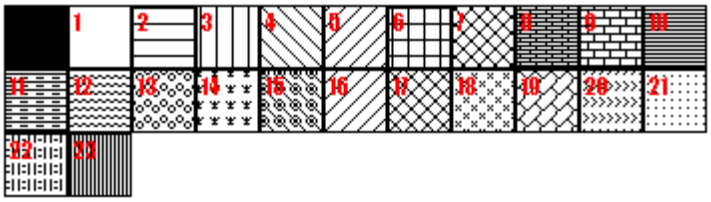
1 – graph color;

0 – scale (0 for logarithmic, 1 for linear);

0 – vertical displacement of the borehole relative to the ground surface.

The text file structure for logging data has been described above. For borehole columns, the following structure is used: the first column contains depths of layers (from the ground surface), the second column contains zeros, the third column contains values indicating fill colors for layers, the fourth column contains values indicating the type of hatch. The columns are separated by space.

Below is a list of fill colors and hatch patterns that can be used when describing a borehole column in a text file.



Model smooth/raster dialog box

After editing the model or performing an inversion, several useful operations can be applied to the mesh model.

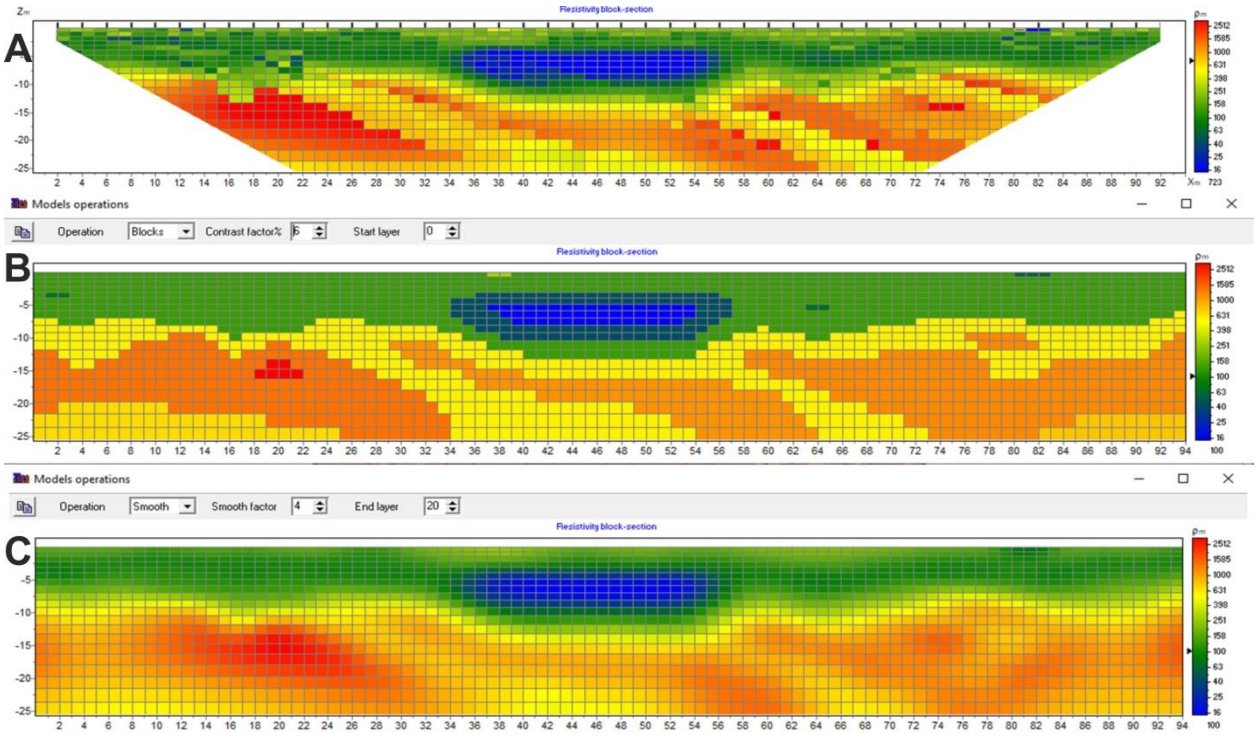
The **Model operations** module (**Options** / **Extra** / **Model smooth/raster**) allows you to smooth or coarsen (break up into blocks) the current model. The block model can be used for the ***Blocks*** inversion type. In this case, the model parameter is inverted for each block. Before breaking up the model into blocks, it is recommended to use a focusing inversion.

When using the **Blocks** operation in the **Model operations** module, the cells with close values of the parameter are grouped depending on the ***Contrast factor*** value. The ***Start layer*** option specifies the first layer of the range of layers this operation will be applied to.

When using the ***Smooth*** operation, the parameter values of neighboring cells are smoothed (averaged) depending on the ***Smooth factor*** value. The ***End layer*** option specifies the last layer of the range of layers this operation will be applied to.

The  button copies the resulting model into the model editor.

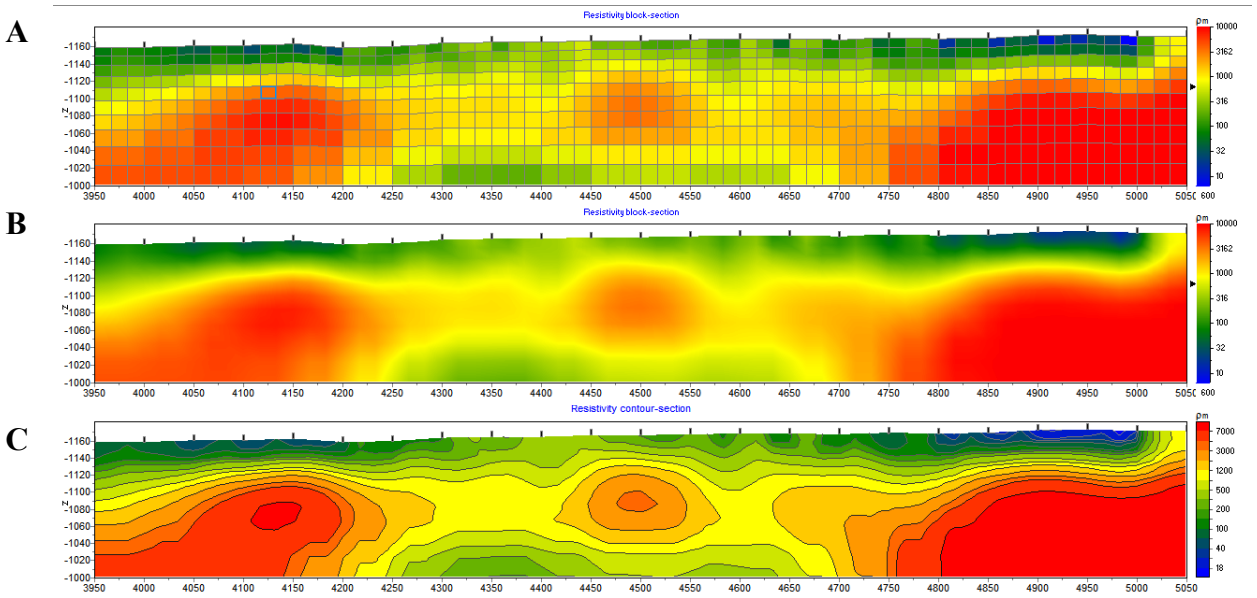
In the example below, **A** is the initial model, **B** is the model after the **Blocks** operation was applied and **C** is the model after smoothing.



Interpretation results

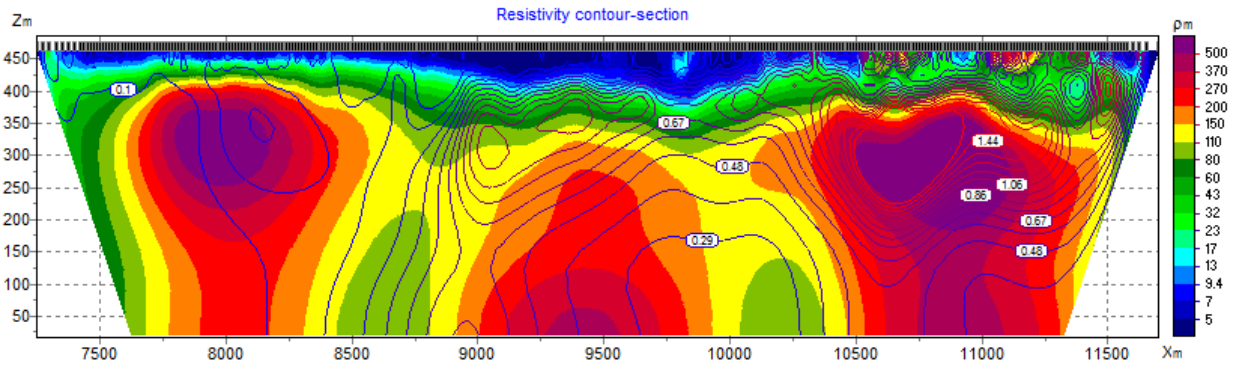
Visualizing the model

In the mesh mode, the model section can be displayed in the form of blocks (cells), in the smooth interpolated palette mode or in the form of color contours (in the figure below, sections A, B and C, respectively). The **Options / Model / Block-section**, **Smooth-section** and **Contour-section** commands are used to switch between the visualizing modes.

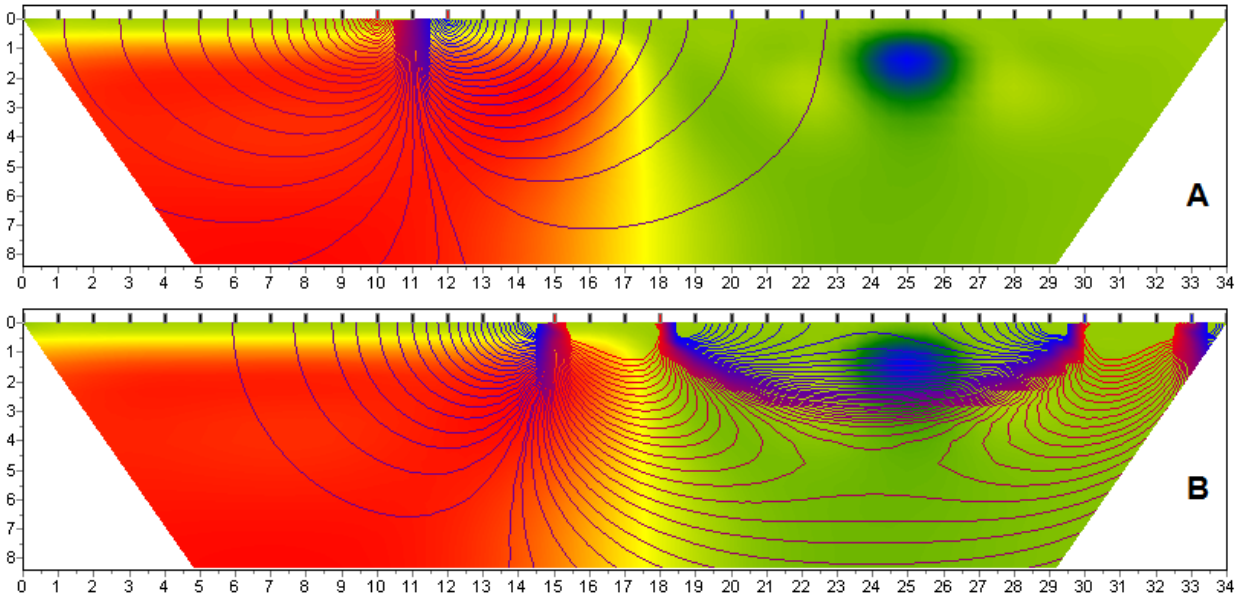


Additionally can be visualized:

- *Two parameters simultaneously* – display resistivity and polarizability. When the *Contour-section* display mode is enabled, the second parameter's isolines can be overlaid on the main parameter color contour plot using the **Options / Extra / Display Rho& IP together** option. The isoline settings can be specified in the **Options / Extra / Extra isolines color settings** dialog box.



- *Potential distribution* – display isolines of potential for a selected source (electrodes) overlaid on the main parameter color plot. When the model is displayed in the *Block-section* or *Smooth-section* mode, the potential distribution is enabled using the **Options / Extra / Potential&sensitivity in model / Potential isolines** command (section A in the figure below). The position of the source is selected in the **Data editor** table. The isoline settings can be specified in the dialog box opened with the button in the **Data editor** window.
- *Sensitivity function distribution* – display sensitivity isolines for a selected measurement overlaid on the main parameter color plot. When the model is displayed in the *Block-section* or *Smooth-section* mode, the sensitivity distribution is enabled using the **Options / Extra / Potential&sensitivity in model / Sensitivity isolines** command (section B in the figure below). The measurement for which the sensitivity function will be displayed is selected in the **Data Editor** table. The isoline settings can be specified in the dialog box opened with the button in the **Data editor** window.



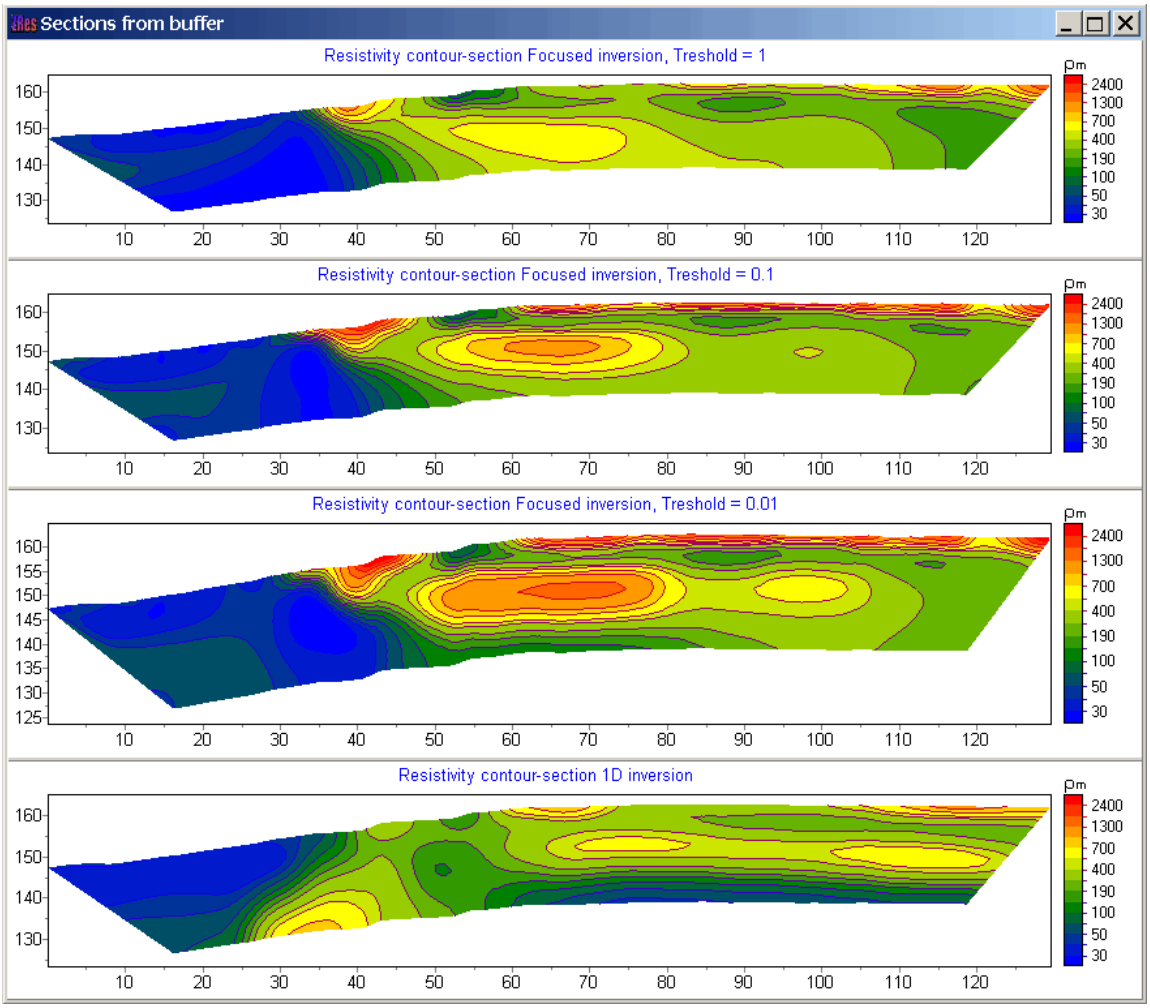
Analyzing the distribution of potentials and sensitivity in the medium gives a better understanding of the ERT method fundamentals.

Working with several models

It is often necessary to store several models within one project and display them simultaneously for comparison, e.g., when determining optimal inversion parameters, or when solving the forward problem for several similar models.

In **ZondRes2D** the current model can be saved into a buffer and displayed at any time using the **Buffer** section of the main menu. The **Model 1** through **Model 5** options correspond to the five buffer models that can be stored within one project.

To save the current model into the buffer, select one of the buffer models in the Buffer menu. If the selected buffer model is empty, the current model will be saved into this buffer slot. In the dialog box that appears you can enter the name of the buffer model which will be displayed in the **Buffer** menu and in the buffer models display window (see figure below) as a model title.



The buffer slots that contain stored models are indicated by checkmarks.

If the selected buffer slot is not empty (checked), in the dialog box that appears you can choose to load the buffer model (**From Buffer**) or to save a current model to this buffer slot (**To buffer**). If you choose **From Buffer**, the buffer model will replace the current model in the model section.

The **Buffer / Open** command opens a separate window where you can view and compare all buffer models.

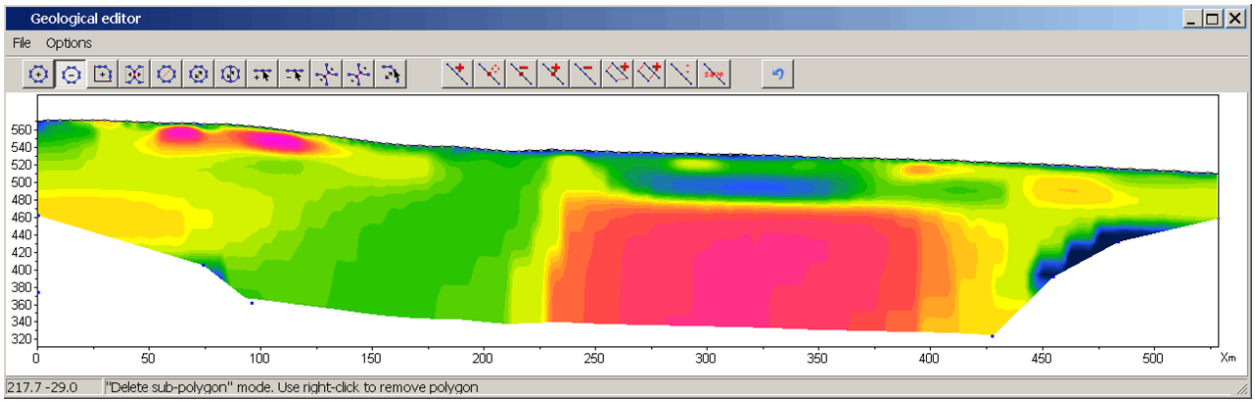
Geological editor module

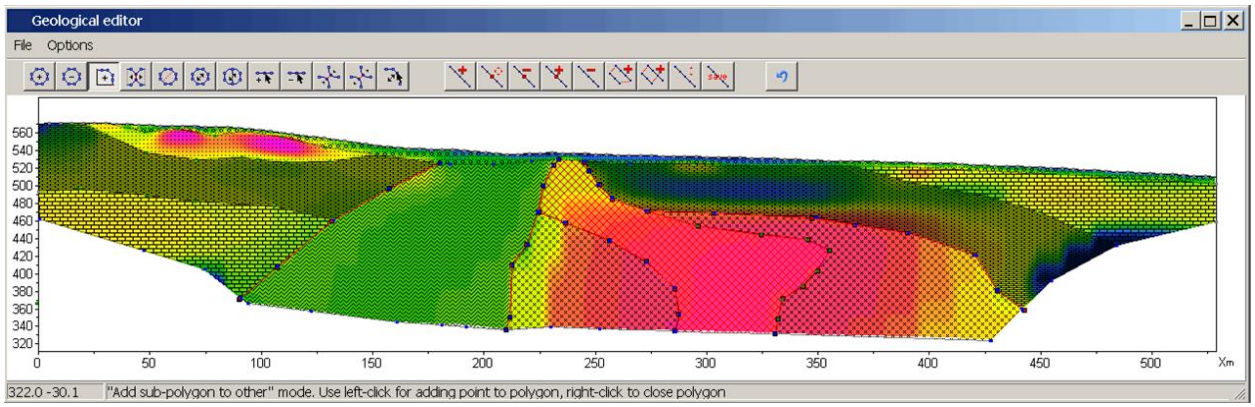
To build geological models (perform geological interpretation), a special **Geological editor** module (**Options / Geological editor**) is implemented in **ZondRes2D**. The editor allows to interactively create a geological model based on the current project model, borehole data, background images and 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.

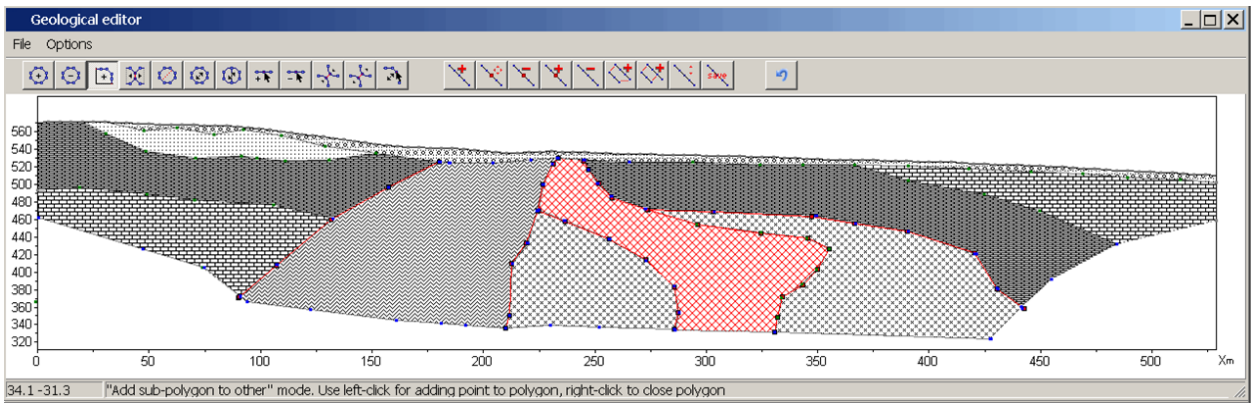
In the editor the current inversion model is used as a background over which the geological model is plotted. When creating the geological model, local objects (polygons) and layers are

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.

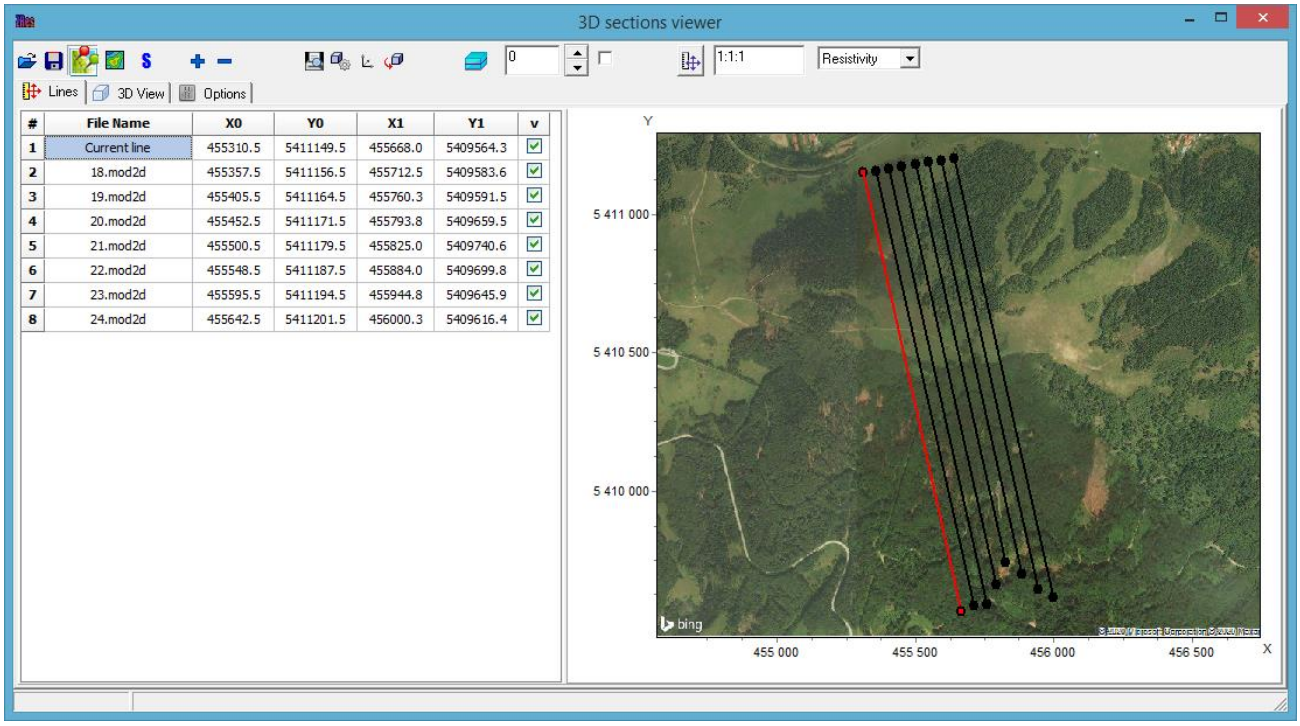
Options / Remove all polygons **–** remove all polygons.

3D visualization of several sections

If the survey was conducted on a grid consisting of several closely spaced lines, a joint visualization and interpretation of these lines is recommended. This allows you to study the distribution of observed structures in the plan view and to facilitate the interpretation of each line by simplifying the identification of the most stable elements of the model.

In **ZondRes2D** the 3D visualization of resistivity or polarizability models is implemented in the **3D section viewer** module, opened with the **Options / 3D fence diagram** command. The module performs the interactive visualization of several models with their topography in a three-dimensional view and plotting of a parameter distribution at a user-defined depth/elevation in the plan view.

The **3D section viewer** window consists of three tabs – *Lines* for adding the 2D models and specifying survey grid coordinates, *3D View* for displaying the models in 3D and *Options* for image settings and scaling. The window toolbar provides access to various visualization options and commands for loading, saving and exporting 3D models.



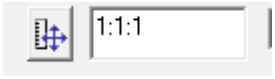
The toolbar contains the following buttons:

	Download a Bing satellite image from the Internet. The survey grid coordinates should be in the UTM coordinate system. If the image was not downloaded automatically, you should enter an actual Bing API key in the Help / Bing maps api_key dialog box.

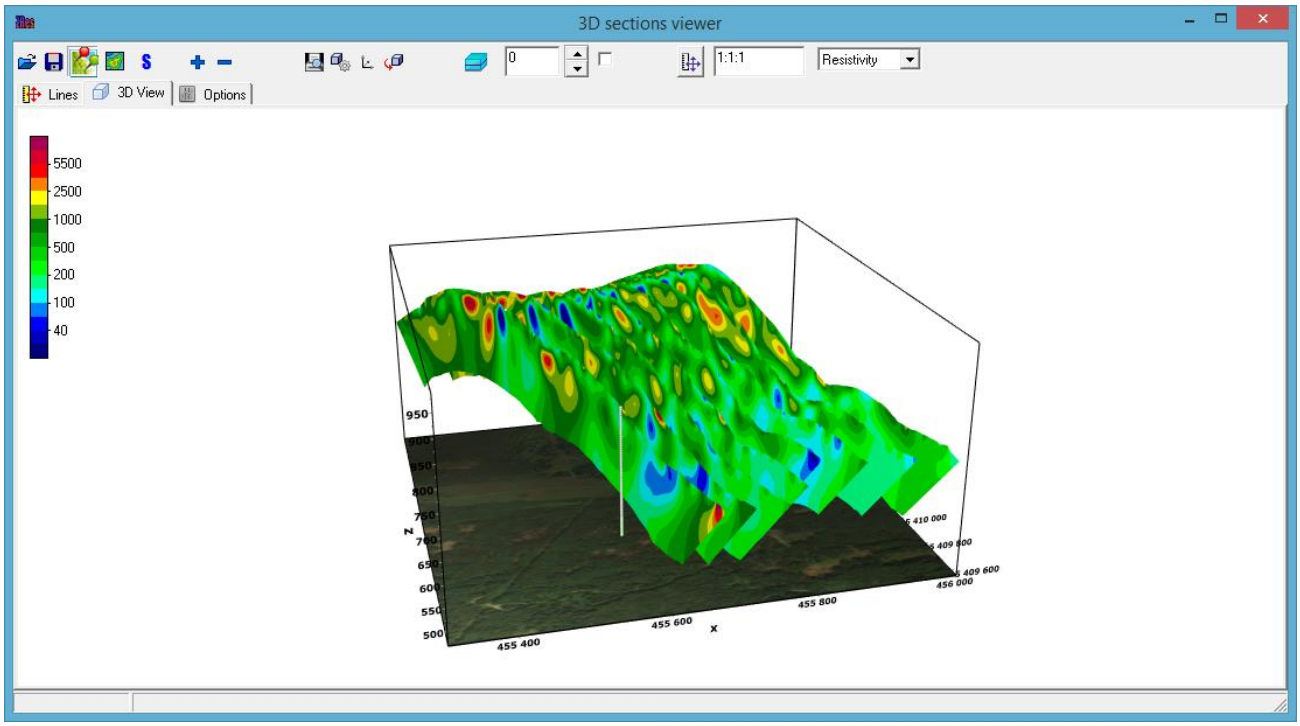
	Build a horizontal (XY) slice at a specified depth.
	Build a horizontal (XY) slice in Surfer.
	Print preview.
	Open the 3D settings dialog box.
	Axes settings (see the Axis editor section).
	Start 3D animation.
	Show a model slice in plan view. The depth of the slice is specified in kilometers in the input box to the right of the button:  .
	Set proportional scaling for all axes (1:1:1). The input box to the right of the button allows you to specify the axes aspect ratio.
	Specifies the parameter to be plotted (resistivity or polarizability).

The **Lines** tab is used for adding survey lines to the 3D plot and specifying their coordinates. Each survey line corresponds to a row in the table. To add a survey line to the table, right-click on a cell in the **File Name** column and select the model file to be loaded in the dialog box that appears. For the 3D visualization, the program uses files with the MOD2D extension which are created automatically when you save a project in the **ZondRes2D** format. You can add an empty row to the table or delete a survey line using the  and buttons on the toolbar. The *X0, Y0, X1, Y1* columns of the table contain the rectangular coordinates for the start and end of the corresponding line (a curved line can be rendered if the loaded model contains XY coordinates in the topography profile). Any orientation of the survey lines is allowed – they can be parallel or non-parallel, intersecting or non-intersecting. The plan view plot of the survey grid is displayed in the right portion of the tab. The last column of the table (**v**) allows you to exclude lines from the 3D plot by unchecking the checkboxes.

The **Options** tab contains options for changing the display parameters – the color scale of the model and the scale for each of the axes. The ***Continuous*** option specifies whether the individual sections are plotted using contours or in the smooth (continuous) form. The axes aspect ratio can

also be set using the  input box on the toolbar. The  button allows switching between the proportional (1:1:1) and user-defined scaling.

The 3D model is displayed in the **3D View** tab.

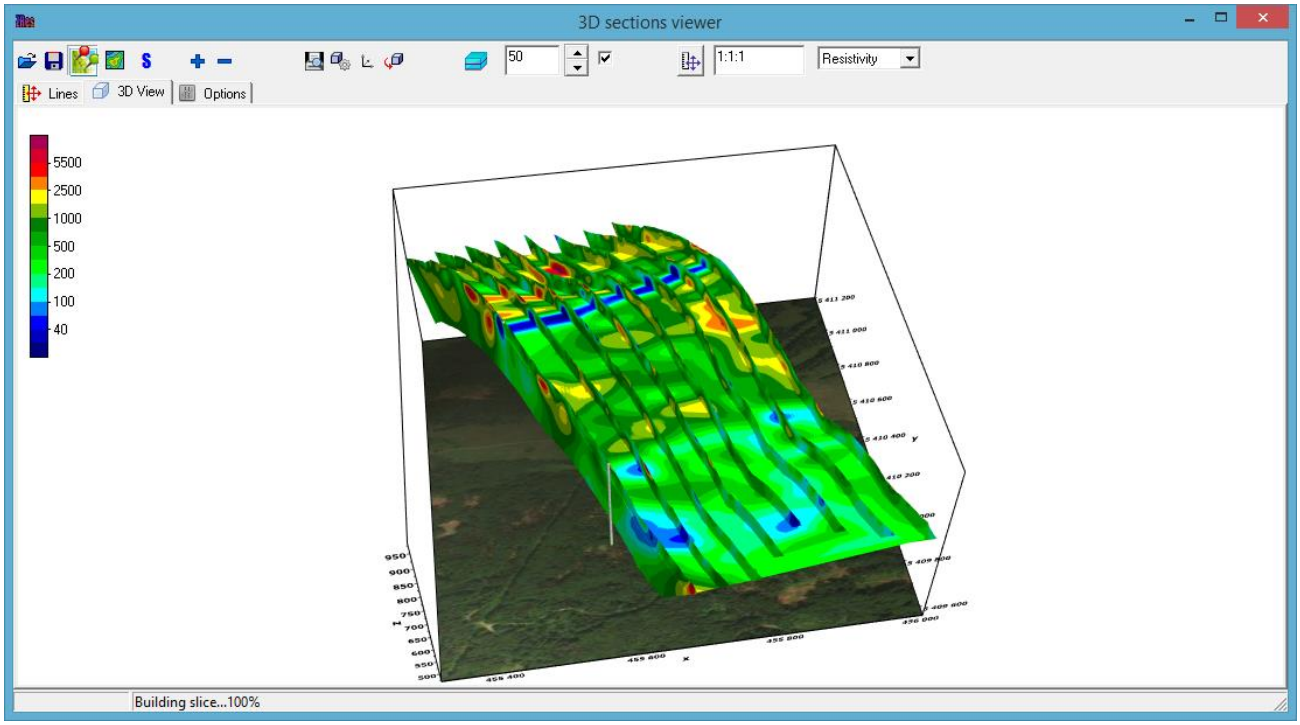


For each axis, the axis properties can be changed in the dialog box opened using the  toolbar button. You can configure the axis labels and grid lines, change the axis title, etc.

The  button gives access to the **3D View** display settings (projection types, scale, rotation angle, the position of the image on the screen, etc.).

You can rotate the 3D plot by pressing and holding down the left mouse button; the mouse wheel can be used to zoom in and out. Press the  toolbar button to start the automatic rotation of the 3D plot around its geometric center.

The  box on the toolbar allows you to build and display a horizontal slice. The checkbox on the right defines whether the depth from the ground surface (when checked) or the elevation (when unchecked) of the slice is specified in the input box.



The button builds a horizontal slice for the specified depth/elevation.

The button builds a horizontal slice for the specified depth/elevation and exports the results to Surfer.

The button allows you to download a Bing satellite image from the Internet and use it as a background. The survey grid coordinates should be in the UTM coordinate system.

The current set of models in the **3D section viewer** can be saved and opened again later using the and buttons. If you select the ***XY plane*** type in the ***Save As*** dialog box, the program will create a DAT text file for the current horizontal slice which can be used in other data mapping programs such as Surfer. When saving in the ***Voxler 3d grid*** format, the program creates a DAT text file that contains the entire model data. The 3D model workspace can be printed by pressing the toolbar button.

Saving the results

The project data are stored in a binary file of the internal **ZondRes2D** format with the Z2D extension. The file stores the field data, measurement weights, the current model and much more. Thus, the current model is restored from the file's data after it is opened in the program.

The project data can be saved by pressing the  button on the toolbar or selecting the **File / Save file** menu command. In the dialog box that appears you can choose to save the whole project, its parts in the form of observed or calculated data values, or an image of the entire workspace or model only in the BMP format. The model image scale and resolution can be changed in the **Picture settings** dialog box opened with the **Options / Graphics / Bitmap output settings** command. In the dialog box, you can specify the ***Vertical scale***, ***Horizontal scale*** (in meters per centimeter), ***Print resolution*** (in DPI) and ***Font size*** for the export image. To apply the settings, disable the ***Automatic*** option, otherwise, the exported image will have the same dimensions as it appears on the screen.

The table below contains the description of file formats for saving and exporting the data:

Zond project data [*.z2d]	Save the project to a project file.
Zond calculated data [*.z2d]	Save calculated data to a text file.
Zond observed data [*.z2d]	Save observed data to a text file.
Zond model with calculated [*.z2d]	Save calculated data values, current model and all program settings to a project file. In this mode, the observed values are replaced by calculated values. This might be convenient for testing the inversion routine on different models.
ProfileR observed data [*.in]	Save observed data in ProfileR format.
ProfileR calculated data [*.in]	Save calculated data in ProfileR format.
Res2dInv observed data [*.dat]	Save observed data in the Res2dInv program format.
Res2dInv calculated data [*.dat]	Save calculated data in the Res2dInv program format.
Worksheet [*.bmp]	Save the workspace of the program window in the BMP image format.
Model [*.bmp]	Save the model section in the BMP image format. To change the scale and resolution, use the Picture settings dialog box (Options / Graphics / Bitmap output settings).

Program configuration [*.cfg]	Save the program inversion parameters.
Grid file [*.dat]	Export the model section to the Surfer program format: data values to a DAT text file, contours to a BLN file and the color scale to an LVL file.
Section file [*.sec]	Export the current model to the internal Zond graphic format consisting of an image file and a spatial reference file in the SEC format (see the A priori information section for details).

When you save the project, the program creates a MOD2D file containing the current model data. This model can be imported into another **ZondRes2D** project (**Options / Import/Export / Import model/data**) or into a project of another program of the **Zond** package.

Additional Excel export options

To use the following data export options, the Excel program needs to be installed on the computer.

Exporting observed data to an Excel spreadsheet

Export of the observed data to an Excel spreadsheet is performed using the **Options** / **Import/Export** / **Export to Excel** / **Data levels** command. The data are grouped based on the geometric factor values. Three rows contain the data corresponding to each geometric factor value: the row with the ***Level (K, 1/m)*** header contains the geometric factor value; the row with the ***X, m*** header contains the X coordinates (distances) along the line for each data point; the row with the ***Appres*** header contains the corresponding apparent resistivity values.

Exporting a model to an Excel spreadsheet

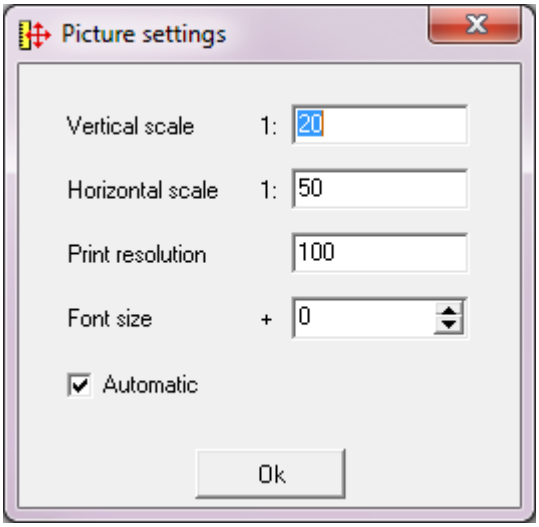
The current model can be exported to an Excel spreadsheet using the **Options** / **Import/Export** / **Export to Excel / Model** command. In the created spreadsheet the model is written in the form of a matrix with rows corresponding to horizontal cell coordinates and columns corresponding to vertical cell coordinates.

The first column contains vertical coordinates of the cell centers; the row with the ***X*** header – horizontal coordinates of the cell centers; the row with the ***Elev*** header – topography values (elevations) for the corresponding cells.

Graphic Settings

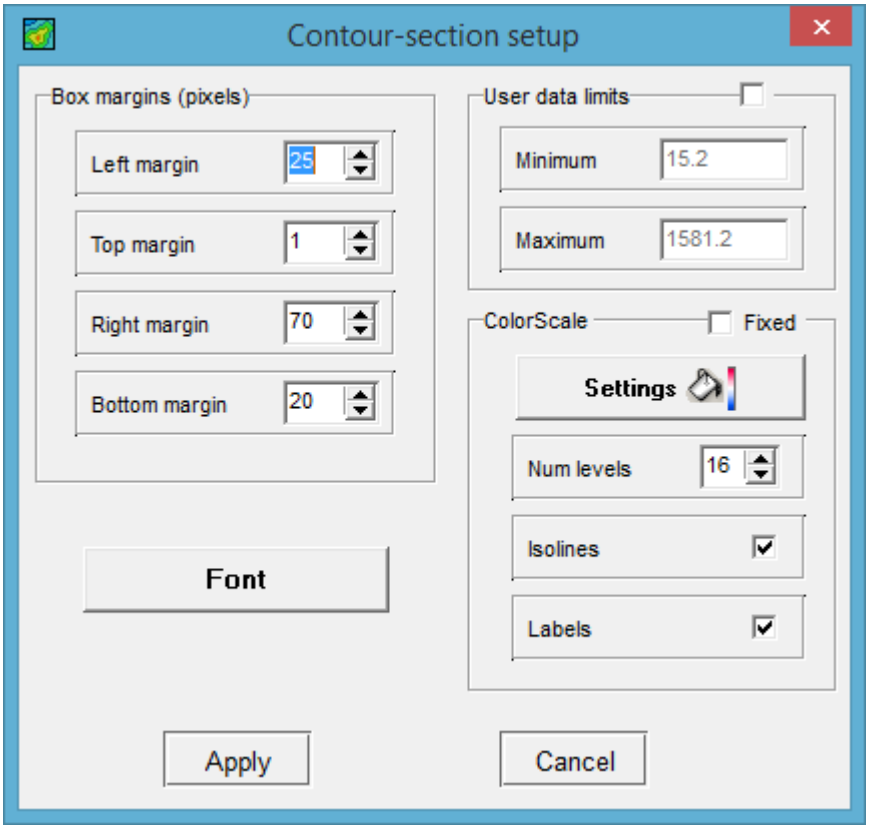
Export image settings

The **Picture settings** dialog box is opened with the **Options / Graphics / Bitmap output settings** command. With the **Automatic** option disabled, you can change the **Vertical scale**, **Horizontal scale**, **Print resolution** (DPI) and **Font size** of the export image.



Contour section and pseudosection settings

To open the **Contour-section setup** dialog box, right-click on the title area of the pseudosection or model contour section and select the **Setup** option in the context menu.



The dialog is used to configure the contour section properties.

The **Box margins (pixels)** group box contains the following options:

Left margin – sets the margin area (in pixels) on the left side of the image.

Right margin – sets the margin area on the right side of the image.

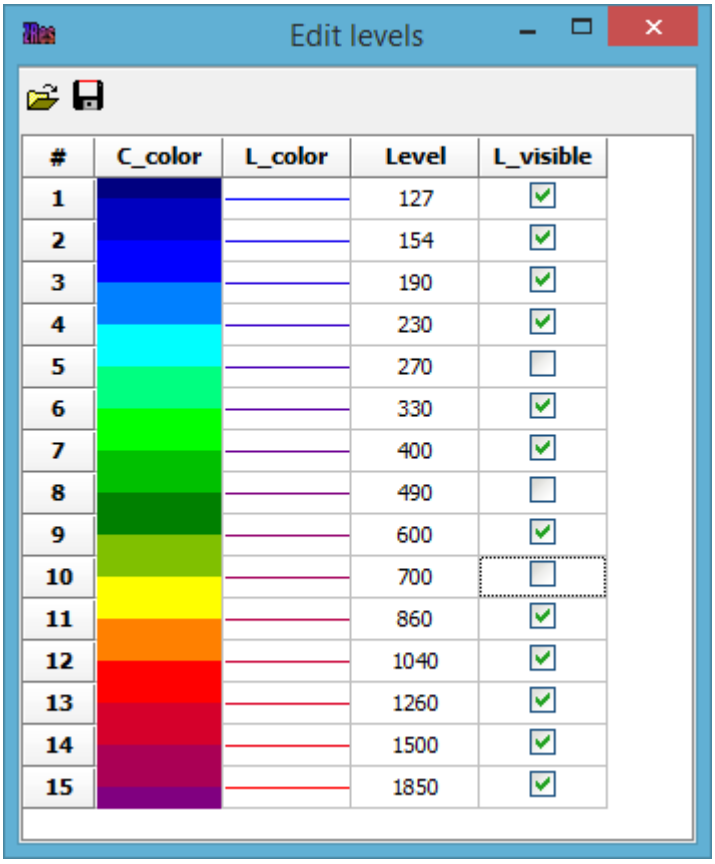
Top margin – sets the margin area on the top of the image.

Bottom margin – sets the margin area on the bottom of the image.

If the ***User data limits*** option is enabled, the values in the ***Minimum*** and ***Maximum*** input boxes are used to set the range of values for the contour plot. If disabled, the data limits are used.

The **ColorScale** group box contains the following options:

Settings – sets the color palette.



Right-click on the table headers performs the following actions:

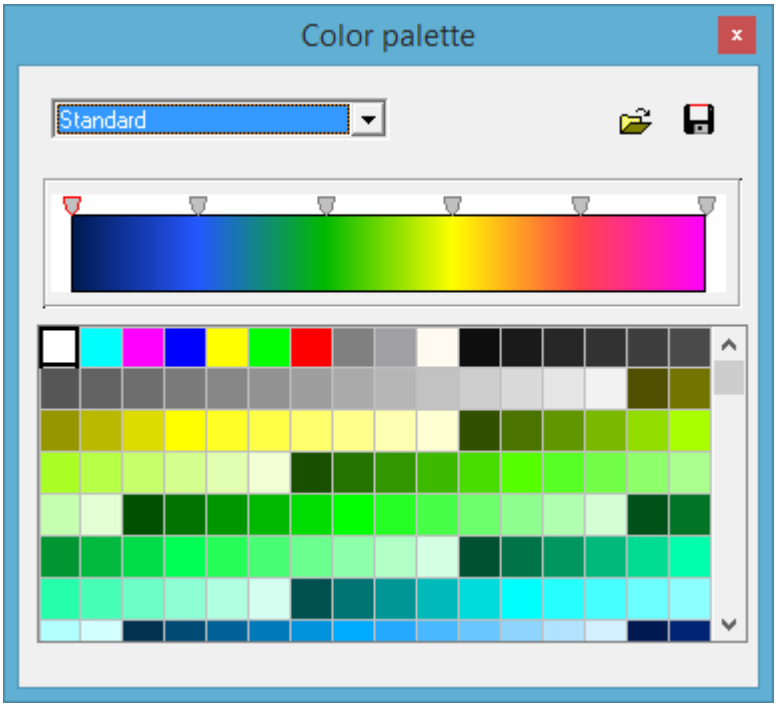
C_color – opens the dialog box for editing the fill color palette.

L_color – opens the dialog box for editing the line color palette.

Level – opens the dialog box for specifying the range of values.

L_visible – uncheck/check all boxes.

The **Color palette** dialog box allows you to choose a preset palette or create a custom one. To add a slider to the palette, left-click on the slider area while holding down the Ctrl key. To delete a slider, select it and press the *Delete* key. Custom palettes can be saved and loaded using the and buttons.



You can save and load color palettes in the Surfer color scale format (CLR).

Num levels – defines the number of contour levels. The contour level method can be set to linear or logarithmic, depending on the data type.

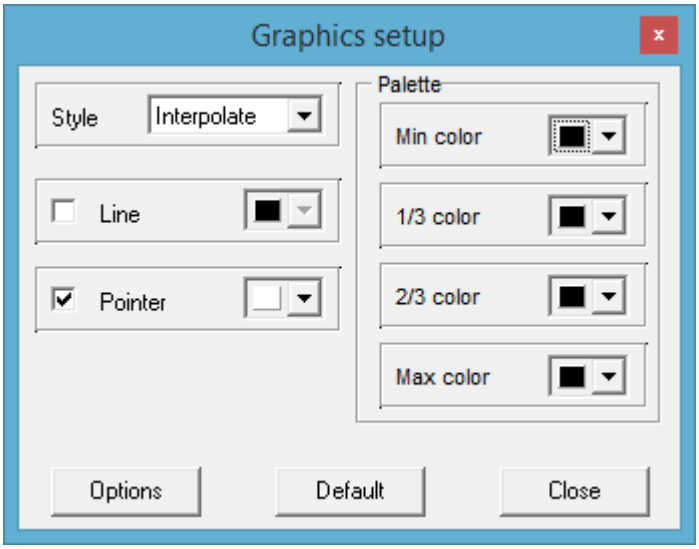
Isolines – shows or hides isolines.

Labels – shows or hides isoline labels.

The ***Font*** button opens a dialog box for changing the color scale font.

Graphic plot settings

To open the dialog box for changing properties of a graphic plot (a set of graphs) displayed in the main program window, select the **Options / Graphics / Observed graphics** or **Calculated graphics** menu command. In the **QC module**, press the  toolbar button.



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.

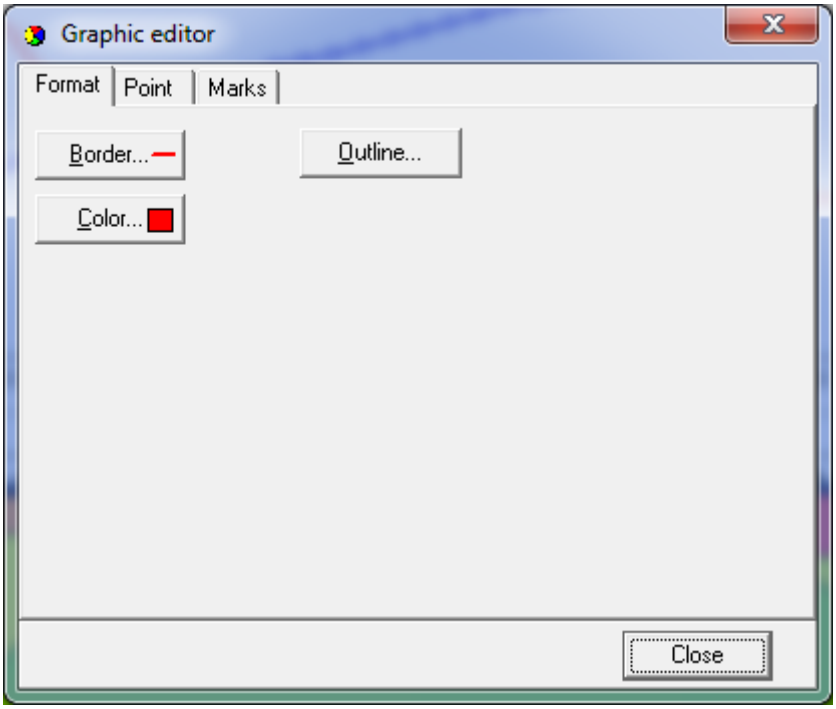
Pointer – specifies the graph point fill color. If the checkbox is checked, the points are filled with the color set in the ***Pointer*** box, otherwise, colors from the **Palette** group box are used.

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

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

Graph settings

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.

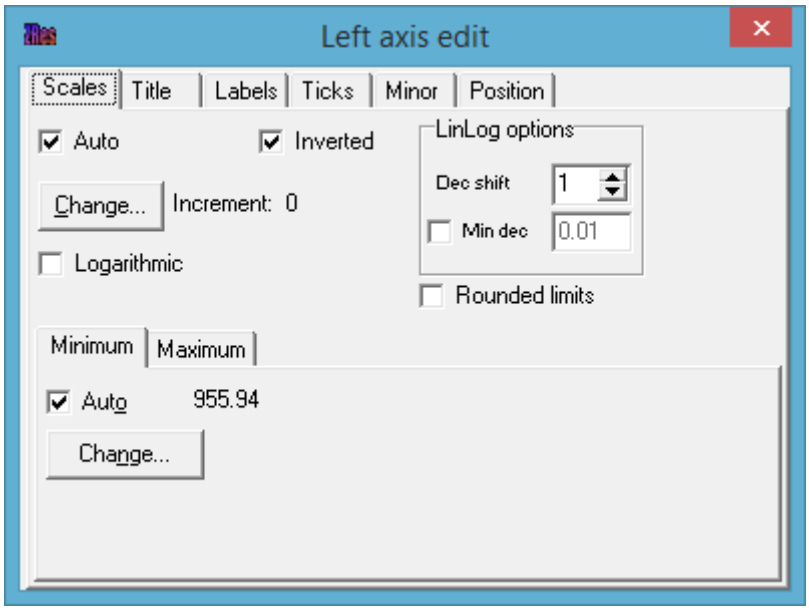
The **Text** sub tab contains the following settings:

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.

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).

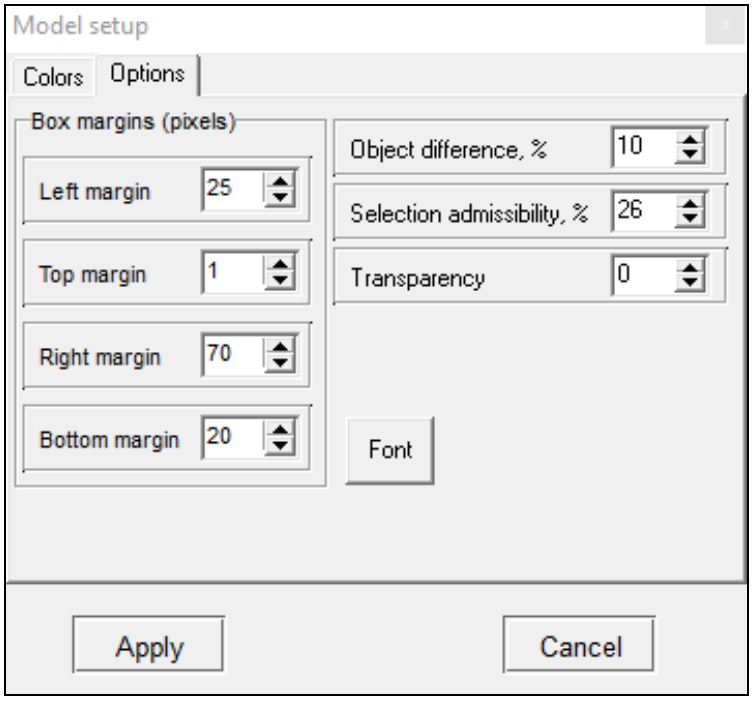
Start % – sets the offset of the start of the axis on the graph relative to the default position (as a percentage of the total graph size).

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.

Model section settings

To open the **Model setup** dialog box, right-click on the title area of the model section (when not in the *Contour-section* display mode) and select the **Setup** option in the context menu.



The **Options** tab of the dialog box contains the following options:

The **Box margins (pixels)** group box specifies the model section margins.

Left margin – sets the margin area (in pixels) on the left side of the image.

Right margin – sets the margin area on the right side of the image.

Top margin – sets the margin area on the top of the image.

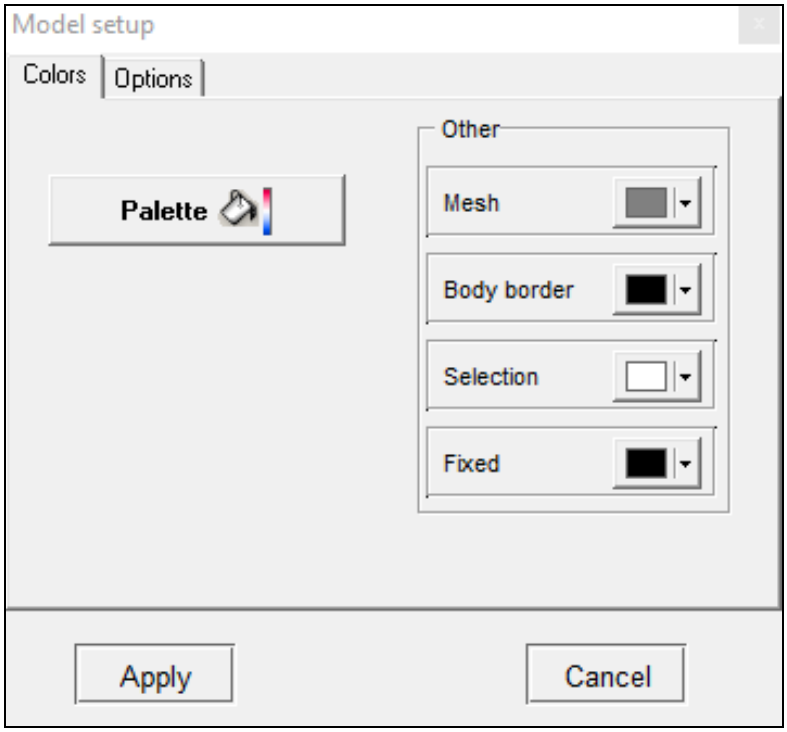
Bottom margin – sets the margin area on the bottom of the image.

Object difference, % – sets the ratio (in percentage) between parameter values of adjacent cells, above which a boundary is plotted between the cells.

Selection admissibility, % – sets the degree of similarity for adjacent cells to be selected as a single object with the "Magic wand" selection tool.

Transparency – sets the transparency of the background image.

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



The **Colors** tab contains the following options:

The **Palette** button opens the dialog box for configuring the color palette.

Mesh – specifies the mesh color.

Body border – specifies the color of the boundary between adjacent cells.

Selection – specifies the color of the selection fill pattern.

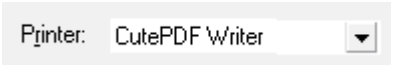
Fixed – specifies the color of the fill pattern of fixed cells.

Print preview

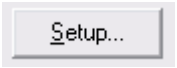
The **Print preview** window is opened with the **File / Print preview** menu command.

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

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

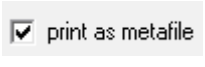


– select a printer.

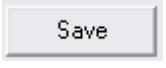


– open a standard print setup dialog box.

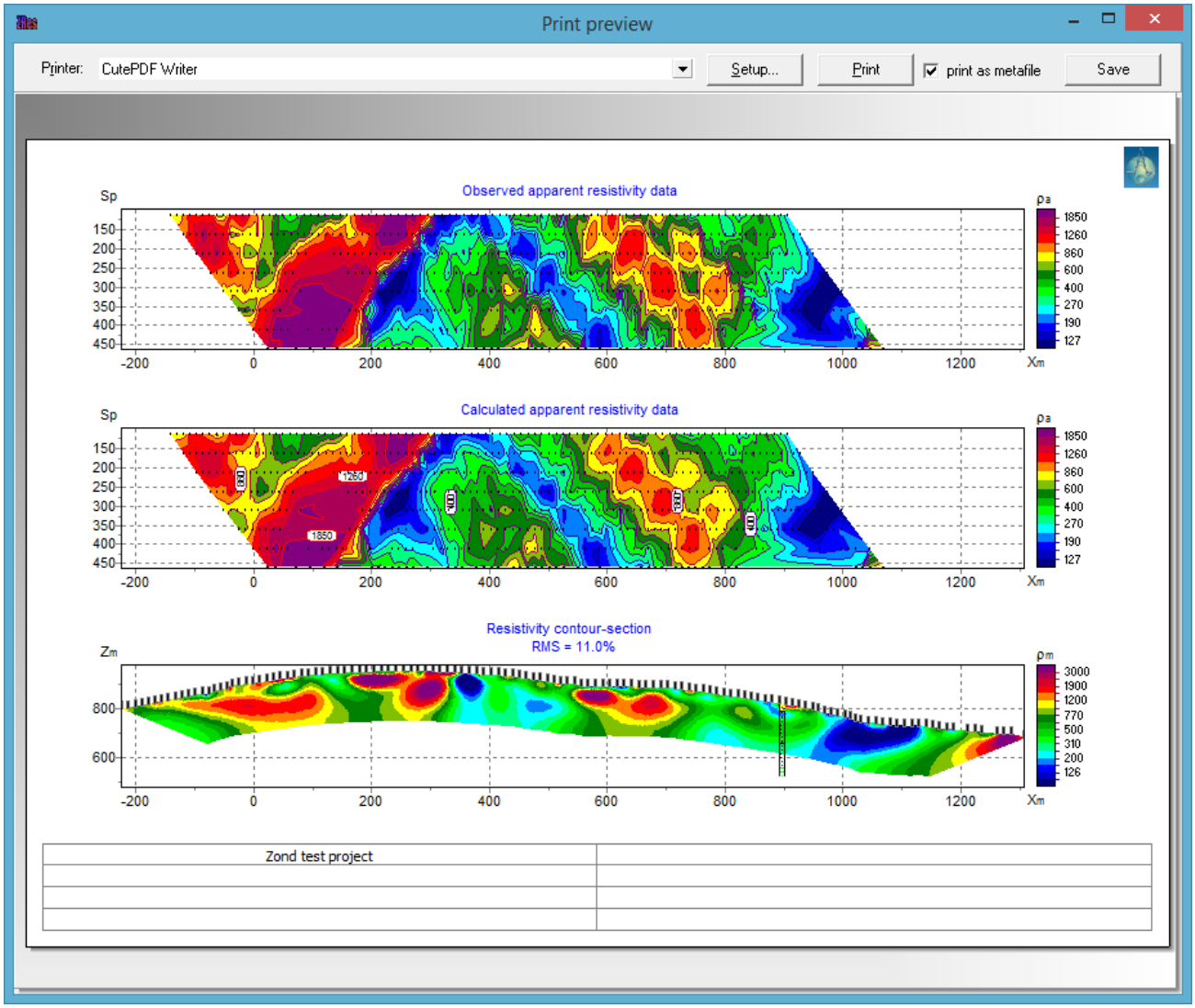
– print the drawing.



– 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.

List of references

1. Dey, A. and Morrison, H.F. [1979] Resistivity modeling for arbitrary shaped two-dimensional structures. Geophysical Prospecting 27, 106-136.
2. Lowry, T., Allen, M.B. and Shive, P.N.. [1989] Singularity removal: A refinement of resistivity modeling techniques. Geophysics, 54(6), 766-774.
3. Xu, S.Z., Duan, B. and Zhang D. [2000] Selection of the wave numbers k using an optimization method for the inverse Fourier transform in 2.5D electrical modeling. Geophysical Prospecting, 48, 789-796.
4. Tsourles, P.I., Symanski, J.E. and Toskas, G.N. [1999] The effect of terrain topography on commonly used resistivity arrays. Geophysics, 64, 1357-1363.
5. Loke, M.H.. [2000] Topographic modeling in resistivity imaging inversion. 62nd EAGE Conference & Technical Exhibition Extended Abstracts, D-2.
6. Constable, S.C., Parker, R.L., and Constable, C.G.. [1987] Occam's inversion: a practical algorithm for generating smooth models from electromagnetic sounding data. Geophysics, 52, 289-300.
7. Marquardt, D.W. [1963] An algorithm for least squares estimation of nonlinear parameters: J. Soc. Indust. Appl. Math., 11, 431-441.
8. Portniaguine, O.N. and Zhdanov M.S. [1999] Focusing geophysical inversion images. Geophysics. V. 64. P. 874–887.
9. Khmelevskoy, V.K., Geofizicheskie metody issledovaniya zemnoi kory (Geophysical Menods of the Earth’s Crust Study), Dubna: Mezhd. Univ. Prir., Obshch., Chel., 1999, Book 2.

Additional information

Youtube video lessons

https://www.youtube.com/channel/UCGtprIIZkc9CsLfiuz4VvmQ?view_as=subscriber

LinkedIn support group

https://www.linkedin.com/groups/6667336/

Demo Zond projects

ftp://zond-geo.com/

Username: download@zond-geo.com

Password: 12345

USB dongle troubleshooting

If the program does not work with a USB dongle:

- The dongle driver is not installed or has not been installed correctly. In some operating systems the dongle is recognized as an HID device and there is no need to install a driver, but in some systems, it is not recognized correctly and the driver needs to be installed. Driver download link: http://senselock.ru/files/senselock_windows_3.1.0.0.zip. The dongle should appear in the device manager as “Senselock Elite”.
- The period of free updates has ended. In this case, you have to use the latest working version of the program or purchase additional 2 years of updates.
- Sometimes, when the dongle turns into HID mode, the system might not recognize it as an HID device. In this case, you need to switch it back to USB mode using a small application that can be downloaded from the following link: http://www.zond-geo.com/zfiles/raznoe/SenseSwitch.zip. The senseswitch.exe has to be started from the command prompt using the command senseswitch.exe usb.