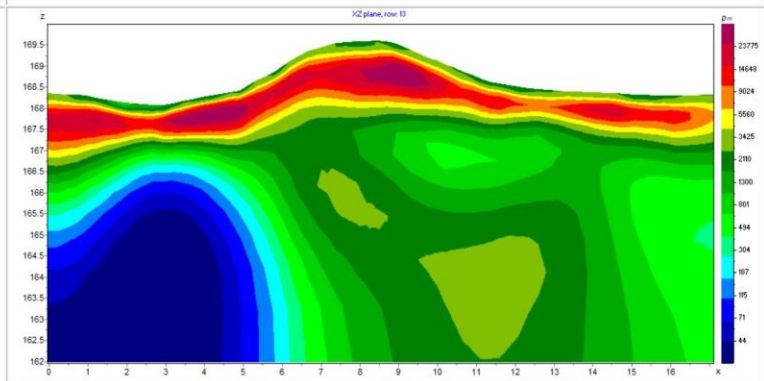
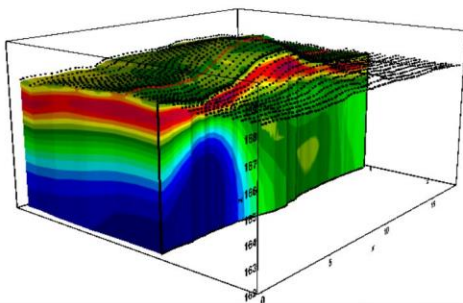
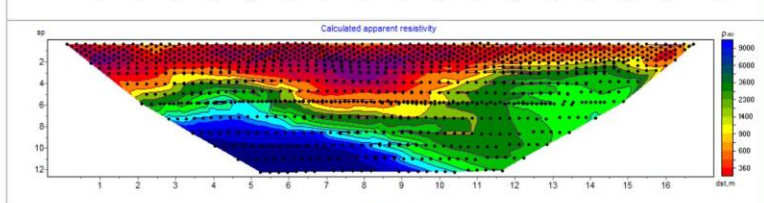
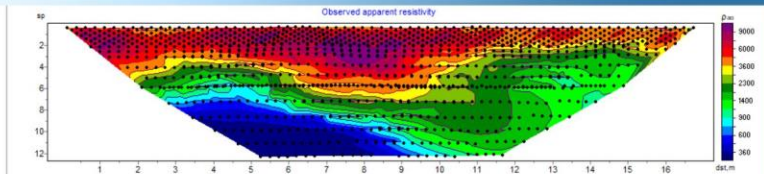
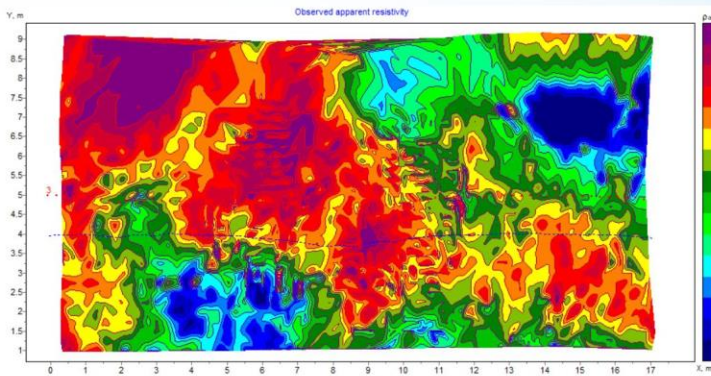


# Zond Software

*Three-dimensional data interpretation program.  
Resistivity and induced polarization methods  
(land, cross-hole and marine surveys).*

# ZondRes3D



# Table of contents

<b>Introduction</b> .....	<b>3</b>
<b>System requirements</b> .....	<b>4</b>
<b>Getting started with the program</b> .....	<b>7</b>
<i>Main window toolbar</i> .....	<i>7</i>
<i>3D settings submenu</i> .....	<i>9</i>
<i>Status bar</i> .....	<i>19</i>
<b>Creating a measurement system</b> .....	<b>19</b>
<b>Opening data files</b> .....	<b>26</b>
<i>Supported formats</i> .....	<i>26</i>
<i>Import data from a text file or Excel spreadsheet</i> .....	<i>32</i>
<i>Line editor</i> .....	<i>33</i>
<i>Creating a mesh model</i> .....	<i>36</i>
<b>Data visualization</b> .....	<b>39</b>
<i>Graphic plot</i> .....	<i>40</i>
<i>Pseudosection</i> .....	<i>41</i>
<i>3D pseudosection</i> .....	<i>46</i>
<i>Quality control module</i> .....	<i>47</i>
<b>Modeling</b> .....	<b>51</b>
<i>Saving and reading in the calculated data</i> .....	<i>55</i>
<b>Data inversion</b> .....	<b>55</b>
<i>Changing inversion parameters</i> .....	<i>55</i>
<i>Inversion of cross-hole tomography data</i> .....	<i>64</i>
<i>Estimating the inversion misfit</i> .....	<i>67</i>
<b>A priori information</b> .....	<b>67</b>
<i>Importing background images</i> .....	<i>67</i>
<i>Creating borehole columns and logs</i> .....	<i>68</i>
<b>Interpretation results</b> .....	<b>73</b>
<i>Visualization of model slices</i> .....	<i>73</i>
<i>3D model visualization</i> .....	<i>74</i>
<i>Trimming the model</i> .....	<i>79</i>
<b>Saving the results</b> .....	<b>80</b>

<b>Graphic Settings .....</b>	<b>81</b>
<i>Export image settings .....</i>	<i>81</i>
<i>Graphic plot settings .....</i>	<i>84</i>
<i>Graph settings .....</i>	<i>85</i>
<i>Axis editor.....</i>	<i>87</i>
<i>Model editor display settings.....</i>	<i>89</i>
<i>Print preview .....</i>	<i>91</i>
<b>List of references .....</b>	<b>92</b>
<b>Additional information .....</b>	<b>93</b>

## Introduction

**ZondRes3D** is a program for three-dimensional (3D) 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 data processing and visualization options. The software can be used on any Windows-operated PC.

The program 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.

**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 offers a wide range of options to account for a priori information when performing the data inversion.

## System requirements

The **ZondRes2D** program can be installed on a computer with a 64-bit Windows 7 or higher operating system. Minimum system parameters: Pentium IV 2 Hz CPU, 4 GB RAM (32 GB recommended), 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](http://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  $40^{\circ}\text{C}$ , 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  $-500^{\circ}\text{C}$ ). 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 [6].

## ***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  $\Delta t$ :

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

## ***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:

$$N(x, z) = \frac{(a + bx + cz)}{2A}$$

$$\frac{\partial}{\partial x} \left( \sigma \frac{\partial \varphi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \sigma \frac{\partial \varphi}{\partial y} \right) - \frac{\partial}{\partial z} \left( \sigma \frac{\partial \varphi}{\partial z} \right) = -I \delta(x) \delta(z)$$

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

where  $\varphi$  – potential value,  $I$  – current value,  $\sigma$  – electrical conductivity of the medium,  $\delta$  – 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]:

$$(A^T W^T W A + \mu C^T R C) \Delta m = A^T W^T \Delta f - \mu C^T R C m,$$

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,  $\mu$  – regularizing parameter,  $\Delta f$  – vector of discrepancies between calculated and measured values,  $R$  – focusing operator.

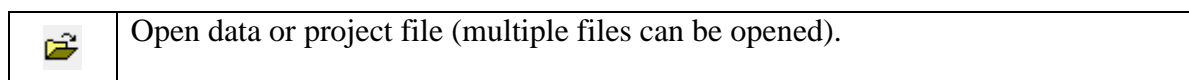
## Getting started with the program








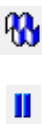




This section provides the description of the main menu commands, toolbar buttons, status bar, and hotkeys.

To bring up a tip about a particular menu command, right-click on that command in the menu.

### *Main window toolbar*

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





	Save data file.
	Open <b>Program setup</b> dialog box.
	Bring up the additional menu for working with the 3D model.
	Go to the previous/next slice of the model.
	Copy the current model slice to the clipboard.
	<p>Paste a model slice from the clipboard into the model. Right-click on the button brings up a menu with the following paste options:</p> <p><i>Current</i> – the model slice from the clipboard replaces the current slice.</p> <p><i>Current&amp;+</i> – the model slice from the clipboard replaces the current and all subsequent slices.</p> <p><i>Current&amp;-</i> – the model slice from the clipboard replaces the current and all previous slices.</p> <p>If the <i>All params</i> option is selected, slices of both model parameters (resistivity and polarizability) are pasted, otherwise, only slices for the currently selected parameter are pasted.</p>
	Calculate the forward problem solution.
	Start/stop inversion.
	Enable the resistivity interpretation mode.
	Enable the IP interpretation mode.
	A drop-down list of 2D survey lines (when a 3D survey consists of a set of individual 2D lines). The selected survey line is displayed in the upper right section of the program window.
	A drop-down list of depth levels (electrode spacings) for plotting a plan view contour map of the data (displayed in the upper left section of the program window). By right-clicking on the list button you can select the number of neighboring levels that will be used to plot the map. This option is used when some electrode spacings are missing in the data, in which case the map appears half-empty when only one level is selected).



### ***3D settings submenu***

The  toolbar button brings up the additional menu containing the following options:

3D view settings	Open the <b>3D view settings</b> dialog box.
Print preview	Open the <b>Print preview</b> dialog box.
Plane cut	Select the orientation of the model slice (XZ, YZ, XY).
Model	Operations with the 3D model. <i>Halfspace value</i> – set a new parameter value for the medium. <i>Clear model</i> – clear the current model. <i>Clear limits</i> – clear the specified parameter variation limits. <i>MinMax auto</i> – determine the parameter variation limits automatically. <i>Smooth/raster model</i> – smooth or coarsen (break up into blocks) the current model. When applying the <b>Rasterization</b> procedure, the cells with close parameter values are grouped depending on the contrast factor value ( <i>Degree</i> ). If <b>Smoothing</b> is applied, parameter values of model cells are averaged depending on the smoothing factor value ( <i>Degree</i> ).
Axes settings	Open the dialog box with the 3D model axis settings (X, Y and Z).
Show	Select the objects to be displayed on the 3D model. <i>Cutting plane</i> – show the position of the slice. <i>Survey points</i> – show electrode locations. <i>Boreholes</i> – show borehole columns and geophysical logs. <i>Background</i> – show the background.
Synchronous cut	Change the position of the slice on the 3D model when a new slice is selected using the   toolbar buttons.
X:Y:Z 1:1:1	Set the same scale for all three axes.

### ***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.

<b>File</b>	
<b>File</b> / Create survey	Open the module for creating a synthetic measurement system for a multi-line survey. The measurement system consists of

	individual 2D survey lines (pseudo 3D). You can select the spread configuration and one of the conventional arrays. The created measurement system can be exported to a file in one of the commonly used measurement protocol formats.
File / 3D sequencer	Open the module for creating a 3D measurement system.
File / Open file	Open a data or project file (multiple files can be opened).
File / Import text/excel	Import data from a multi-column text file or Excel spreadsheet. Column headers in the first row of the table should be specified.
File / Mesh for MN only	If this option is enabled, the model mesh is constructed based on the positions of potential electrodes only. The option should be enabled before opening a data file.
File / Show data filter	If this option is enabled, the <b>Data filter</b> dialog box appears when opening a data file.
File / Save file	Open the <b>Save As</b> dialog box.
File / Edit data	Open the data file opened in the program in the Notepad editor.
File / Print preview	Open the dialog box for printing the workspace.
File / Recent	Show the list of the most recently used files.
File / Exit	Exit the program.
<b>Options</b>	
<b>Options</b> / Project information	Show information about the project that is currently opened. This information can be edited.
Options / Quality control module	Open the quality control and data editing module. The QC module is used to analyze and process the data observed on the current survey line. The module contains a set of options for automatic and semi-automatic analysis of data errors. The <b>Appl&amp;exit</b> and <b>Del&amp;exit</b> buttons should be used to exit the module.
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 / Set survey lines	Open the <b>Line settings</b> window. Specifying individual survey lines does not affect interpretation results but it is essential for data visualization. When a survey consists of a set of individual survey lines, the lines are created automatically by the program. If

	no survey lines are specified, the data are displayed as a point cloud (unstructured) in the upper right section of the main program window.
Options / Program setup	Open the main program settings dialog box. Most of the settings are related to the inversion process.
Options / <i>Data</i> / Pseudosection	Display the calculated and observed data in the form of pseudosections for individual survey lines (if specified).
Options / Data / Graphics plot	Display the calculated and observed data in the form of graphs for individual survey lines (if specified).
Options / Data / 3D pseudosection ON	Display the data in the form of a 3D pseudosection (replaces the plan view contour map). The colors of the data points correspond to the values of the displayed parameter.
Options / Data / Observed data	Display observed data on the contour map.
Options / Data / Calculated data	Display calculated data on the contour map.
Options / Data / Data misfit	Display inversion misfits on the contour map and on the second (lower) pseudosection.
Options / Data / Apparent resistivity	Display apparent resistivity values.
Options / Data / Resistance	Display resistance (V/I) values.
Options / Data / Chargeability	Display IP data values. Available only in the IP interpretation mode and cannot be disabled.
Options / Data / Point position	<p>A set of options for determining the data point X position.</p> <p>For a three-electrode array (<b>3 electrodes</b>):</p> <p><math>(M+N)/2</math> – the data point is at the midpoint between potential electrodes M and N;</p> <p><math>(A+M)/2</math> – the data point is at the midpoint between the current electrode A and the nearest potential electrode M.</p> <p>For a four-electrode array (<b>4 electrodes</b>):</p> <p><i>Auto</i> – automatic determination;</p> <p><math>MN/2</math> – the data point is at the midpoint between potential electrodes M and N;</p> <p><i>AB-O-MN</i> – the data point is at the midpoint between current (AB) and potential (MN) dipoles.</p>

Options / Data / ABMN normed	Apply the additional normalization of the resistance (V/I) by the lengths of potential and current electrode spacings. Available when the <b>Options / Data / Resistance</b> option is selected. The option is useful when the potential electrode spacings vary over a wide range.
Options / Data / Type graphics	Select the method of displaying the data in the graphic plot. <i>Iso-AB</i> – each graph in the graphic plot corresponds to a specific spacing of current electrodes. <i>Iso-MN</i> – each graph in the graphic plot corresponds to a specific spacing of potential electrodes. <i>Iso-psz</i> – each graph in the graphic plot corresponds to a specific pseudodepth (geometric factor).
Options / Data / Error gates	Show measurement errors in the form of confidence intervals on the graphs.
Options / Data / IP units	Set IP measurement units in percentage or mV/V. This option is available only in the IP interpretation mode.
Options / <b>Model</b> / Block-section	Display the model section in the form of blocks (cells).
Options / Model / Smooth section	Display the model section in the form of contours or in the smooth interpolated mode. To display the model in the smooth interpolated mode, activate the <b>Continuous</b> checkbox in the <b>Color palette</b> dialog box accessible by pressing the <b>Palette</b> button in the <b>Colors</b> tab of the <b>Model setup</b> dialog box (right-click on the title of the model section and select <b>Setup</b> ).
Options / Model / 2D mode	This mode is used for two-dimensional interpretation (e.g., when only one survey line is available). In this mode, cells in the Y direction are combined.
Options / Model / Plane	Select the plane for model slices: <i>XZ</i> – the X-Z vertical plane; <i>YZ</i> – the Y-Z vertical plane; <i>XY</i> – the X-Y horizontal plane. If the <i>Autosize</i> option is enabled, the automatic scaling of slices is performed.
Options / Model / Model histogram	Open a chart showing the parameter distribution in the model cells.

Options / Model / MF factor display	Display metal factor in the model section and in the 3D section (resistivity and IP data inversion should be performed).
Options / Model / Cole-Cole model	Display one of the Cole-Cole parameters in the model section and in the 3D section. The option becomes available after the Cole-Cole inversion is performed for IP data in time or frequency domain opened in the project.
Options / <i>Cutting</i> / Cut angle X	Specify the trimming angle of the model in the X direction. As a rule, the shape of the resulting section should be close to the trapezoidal shape of the pseudosection; the trimming angle should depend on the type of array used. The angle is selected experimentally and is applied to the model section and some modes of 3D visualization.
Options / <i>Cutting</i> / Cut angle Y	Specify the trimming angle of the model in the Y direction.
Options / <i>Cutting</i> / Blanking cut	Trim the edges of the model at a specified angle. The trimming can be applied to all types of 3D visualization modes except the <i>Pie-cut</i> .
Options / <i>Inversion</i> / 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 $C^C*(m-m_0)$ .
Options / Inversion / <i>Smoothness</i> / Median	<b>m0</b> (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 <b>Smoothing factor</b> . This option works with the <i>Occam</i> and <i>Focused</i> inversion methods and is best suited for focused inversion and inversion with a priori boundaries.
Options / Inversion / Smoothness / Start model	<b>m0</b> (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 <b>Smoothing factor</b> set in the <b>Inversion</b> tab of the <b>Program setup</b> dialog box. This option works with the <i>Occam</i> and <i>Focused</i> inversion methods.
Options / Inversion /	<b>m0</b> (reference model) is the model obtained at the previous

Smoothness / Previous	iteration. The main goal of inversion in this case is to minimize the RMS error while keeping similarity to the reference model. The convergence rate is controlled by the <b>Smoothing factor</b> . This option works with the <i>Occam</i> and <i>Focused</i> inversion methods. The focused inversion sometimes might not yield the desired result (a piecewise-constant model).
Options / Inversion / Smoothness / Diagonal filter	Apply diagonal smoothing during the inversion. Use this option if there are inclined structures in the section.
Options / Inversion / <b>Cross-gradient</b>	This menu sub-section is available in the resistivity interpretation mode if the project contains IP data.
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.
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 participating models will then be inverted independently.
Options / Inversion / Cross-gradient / CC criteria	Use the common correlation criterion for joint inversion of two models. If this algorithm is activated, the <b>Pushing factor</b> should be set in the 0-2 range.
Options / Inversion / Cross-gradient / IP data	Select IP data for joint inversion. Note that joint inversion of two methods requires twice as much RAM.
Options / Inversion / <b>Resolution</b>	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 / <b>Underwater options</b>	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 (3-10).
Options / Inversion / <b>Optimization</b>	Inversion optimization parameters.
Options / Inversion / Optimization / Line search	Find the optimal damping factor at each iteration (if enabled, the inversion process will take approximately 3-7 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 – <b>Smoothing factor</b> (initial value) and <b>Factor</b> (reducing factor).
Options / Inversion / Optimization / Dec factor	If the <b>Line search</b> option is disabled, the <b>Dec factor</b> controls the degree of damping during the inversion. For the first iteration, the <b>Smoothing factor</b> value is used, and for each subsequent iteration, this value is divided by the user-specified <b>Dec factor</b> . The <b>Smoothing factor</b> can be determined automatically if the checkbox near the <b>Smoothing factor</b> input box in the <b>Inversion</b> tab of the <b>Program setup</b> dialog box is checked.
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 / Groups cut cells	Group cells in trimmed areas into 1D layers for inversion.

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.
Options / <i>Import/Export</i> / Background 2D / Load background	Load a background image for the current model slice. After selecting an image file you will be prompted to specify the coordinates of the image corners.
Options / Import/Export/ Background 2D / Remove current	Remove the background image from the current model slice.
Options / Import/Export / Background 3D / Load background	Load a background image for the 3D model. After selecting an image file you will be prompted to specify the XYZ coordinates of the image corners (by default, the coordinates correspond to the coordinates of the current model slice). You can additionally specify a background color for the image which will have a transparency.
Options / Import/Export/ Background 3D / Remove background	Remove the background image from the 3D model.
Options / Import/Export / Import mod3D	Import a 3D model from a file in the MOD3D format. This option can be used, for example, to load a previously saved version of the model if all 4 buffer slots are occupied by other models, or if model mesh parameters have been changed.
Options / Import/Export / Export mod3D	Export the current 3D model to a MOD3D file.
Options / Import/Export / Import mod2D's	Import a set of 2D models from MOD2D files and embed them into the current 3D model taking into account their XY coordinates. This option can be used to import 2D inversion results.
Options / Import/Export / Export mod2D	Export the current 2D slice of the model to a MOD2D file. The exported model can be opened in the <b>ZondRes2D</b> program or imported to the current 3D model at another slice position.



Options / Import/Export / Save rotations	Export the current 3D model as a set of images displaying the model at different view angles.
Options / Import/Export / Save slides	Export the current 3D model as a set of images representing model slices at different positions.
Options / Import/Export / Save 3D image	Export the current 3D model as an image in the BMP format with specified resolution.
Options / Import/Export/ Export line to z2d	Export the calculated data for the current survey line to the Z2D file of the <b>ZondRes2D</b> program.
Options / Import/Export / Output settings	Open the dialog box for setting up export image parameters.
Options / Import/Export / Draw to surfer	Export the current 3D model as a set of horizontal slices to the Surfer program.
Options / Import/Export / Draw to voxler	Build the 3D model in the Voxler program. Before building, you should specify the path to the Scripiter program exec file (scripiter.exe).
Options / <b>Boreholes</b> / Create/Edit borehole data	Add and edit borehole data (columns and logs).
Options / Boreholes / Load borehole data	Open a file containing borehole columns and/or logging data, or a MOD1D file (one-dimensional sounding results).
Options / Boreholes / Remove borehole data	Remove borehole data from the project.
Options / Boreholes / Set column's width	Set the width of the borehole columns in the model section (in percentage of the survey line length).
Options / Boreholes / Edit positions	Edit borehole positions in plan view.
Options / <b>Extra</b> / Process repeats	If the data file contains repeated measurements, they will be averaged.
Options / Extra / Remove data with big misfit	Exclude data points with misfits exceeding the specified cutoff level from inversion. After selecting this option, a dialog box appears displaying the misfit distribution where you should specify the threshold value of the misfit. Pre-filtering inversion should be run with the <b>Robust reweighting scheme</b> option enabled.
Options / Extra / Remove unstable records	Exclude unstable measurements from the inversion. Unstable measurements are the data points for which geometric factor

	values are very sensitive to errors in the geometry. The distribution of potential errors is shown in percentages of theoretical values. The threshold value is specified in the corresponding input field.
Options / Extra / Remove masked records	Completely remove from the project all data points selected using the previous two options (data points with large misfits and unstable measurements). The forward and inverse problem calculation rates may increase after applying this option.
Options / Extra / Edit remote electrodes	Edit position (coordinates) of "infinity" electrodes. If the infinity electrodes were located far enough from the main spread, it is recommended to relocate them to true infinity using the <b>Remote to infinite</b> option (see below).
Options / Extra / Remote to infinite	If data were collected using "infinity" electrodes, and coordinates of these electrodes were specified in the data file, after applying this option the electrodes will be considered to be at true infinity.
Options / Extra / EM coupling removal	If data were acquired using a relatively high-frequency AC equipment (typically, capacitively-coupled systems), the data can be converted to zero frequency using this option.
<b>Buffer</b>	
The buffer allows you to store up to four models obtained using different inversion parameters. The models can be opened in a separate window for comparison.	
<b>Buffer</b> / Model 1...4	Save/load a 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.
Buffer / Only current	If this option is activated, only the resistivity or polarizability model is saved into the buffer, depending on the selected interpretation mode.
<b>Help</b>	
Help / Manual	Open the program manual.
Help / Check for updates	Check for updates.
Help / Restart library	Restart the dynamic link library (DLL). In case the calculation process stops each time the inversion is started, check your antivirus settings. The antivirus may be blocking the DLL.
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).

### ***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.
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.

## **Creating a measurement system**

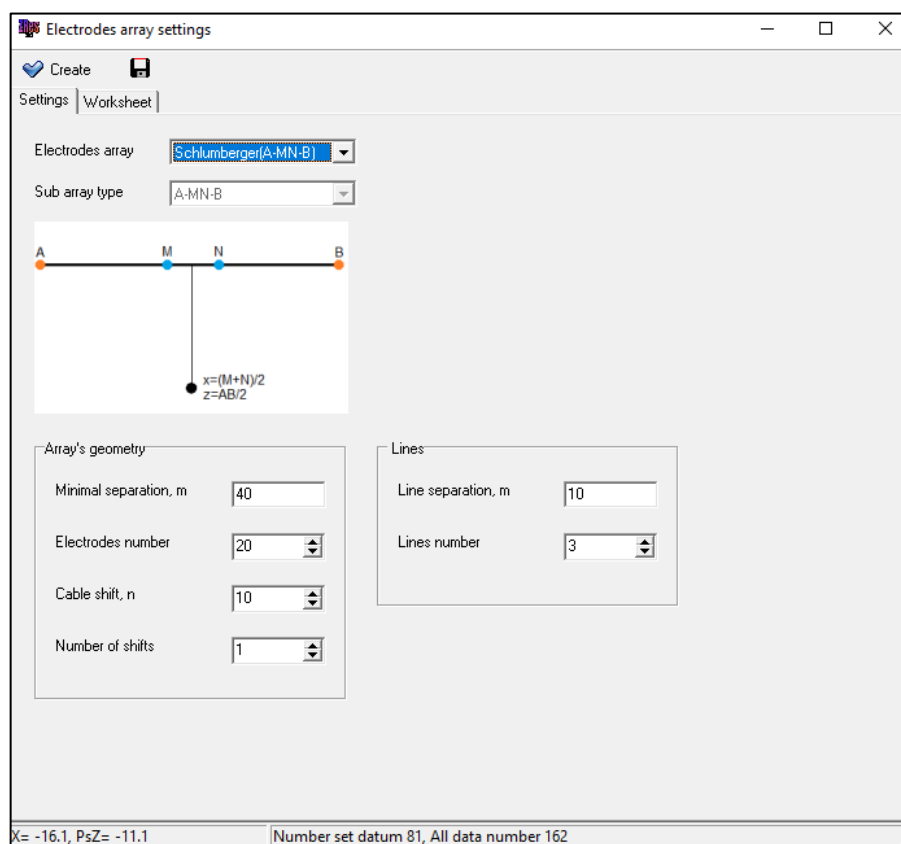
3D surveys are typically carried out using two types of survey configurations. In the first type, the survey is performed using a system of parallel 2D survey lines (*Multi-line 2D measurement*

system or *Pseudo 3D*). In the second type, the electrodes are placed over the whole survey area and measurements are carried out between any combinations of them in arbitrary directions (*True 3D*).

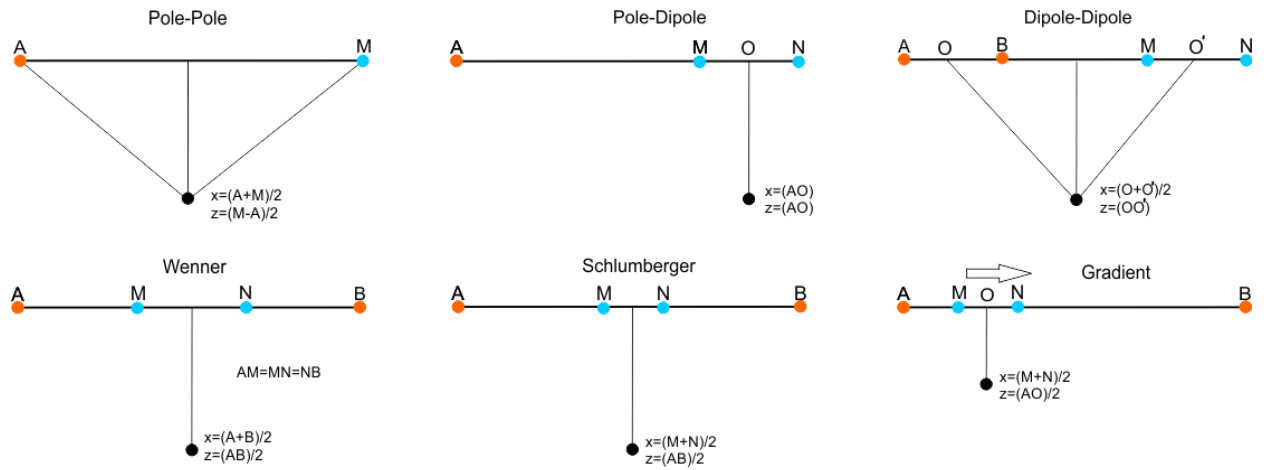
### ***Multi-line 2D measurement system***

Use the **File / Create survey** menu command to open the **Electrodes array settings** dialog box containing settings for configuring the parameters of a multi-line 2D 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;

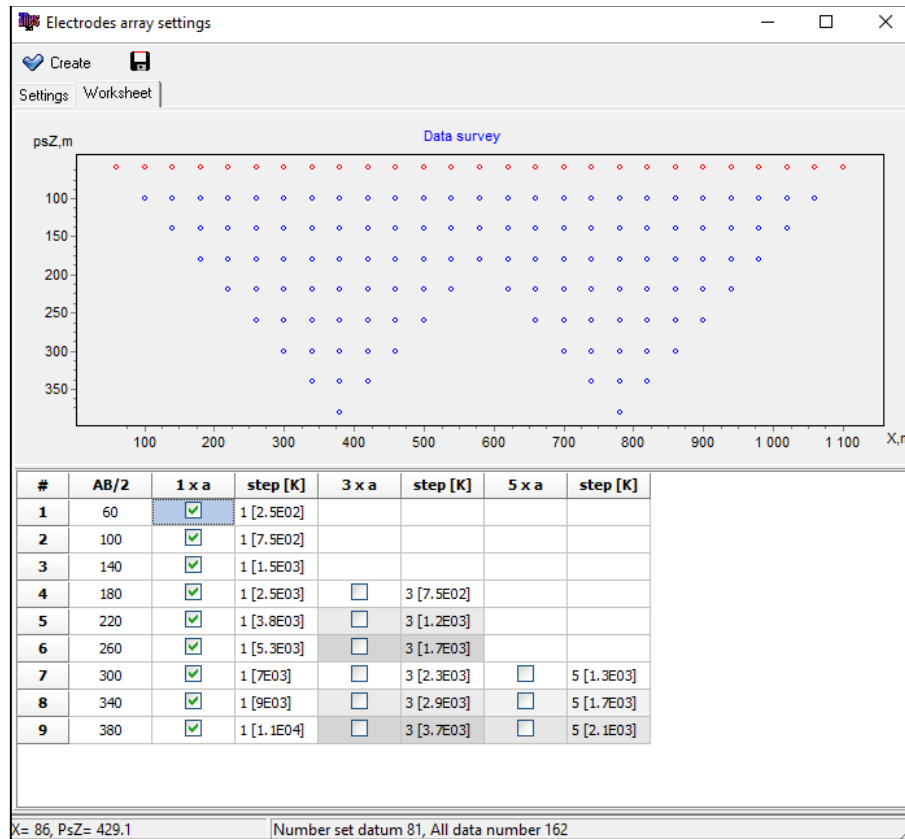
**Electrodes number** – number of electrodes in one spread;

**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.


In the **Lines** group box you can specify the distance between survey lines (**Line separation, m**) and the number of lines (**Lines number**).

The **Worksheet** tab allows you to configure the geometry of the measurement system in more detail. Practically, it is a tool for creating a measurement protocol for the selected array.



The **Data survey** section in the upper portion of the tab contains a graphical representation of the created measurement system. It shows schematically the location of 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 (**AO**, **OO**, **AB/2**, or **AM/2** depending on the selected array type) contains all possible measurement spacings in meters. 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.

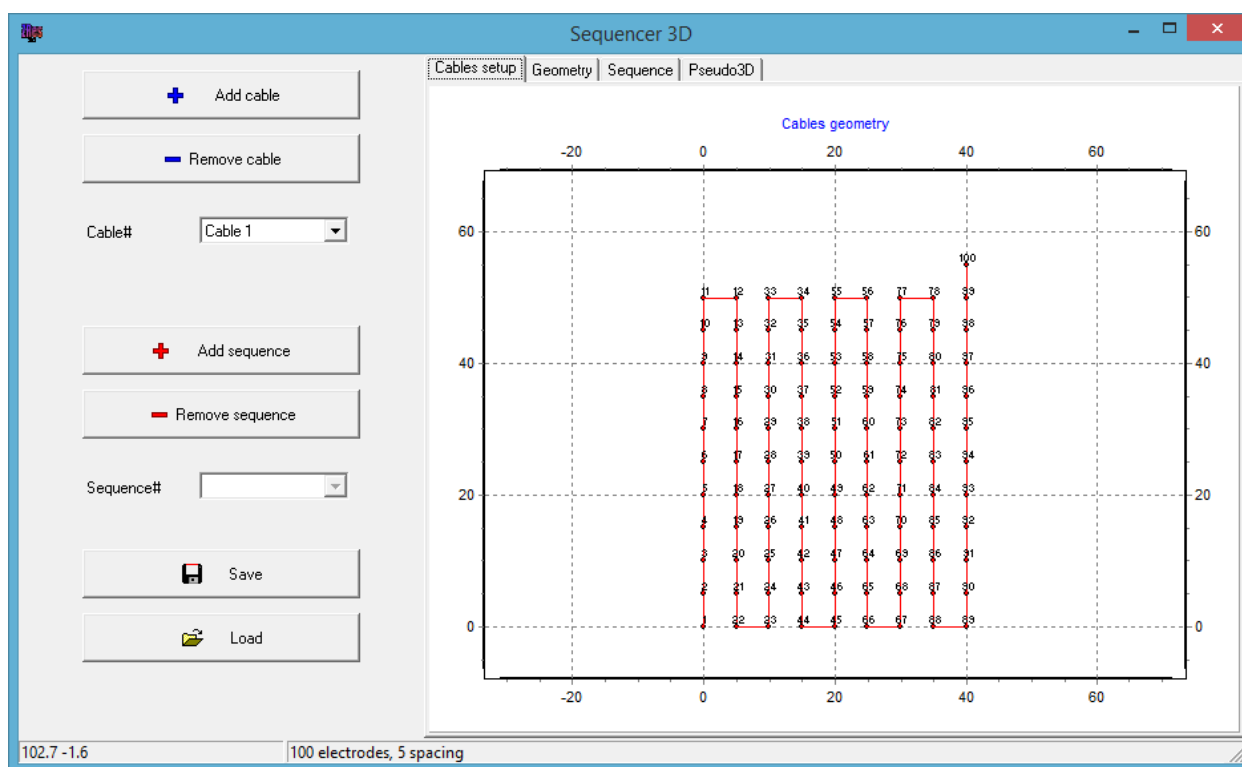
To finalize the creation of the measurement system, press the  button.

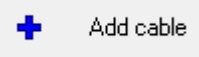
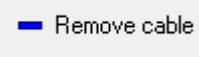
To change the measurement system without changing the model, use the **File / Change array** menu command, which reopens the **Electrodes array settings** dialog box.

### 3D measurement system

Use the **File / 3D sequencer** menu command to open the dialog box containing settings for configuring the parameters of a 3D measurement system.

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

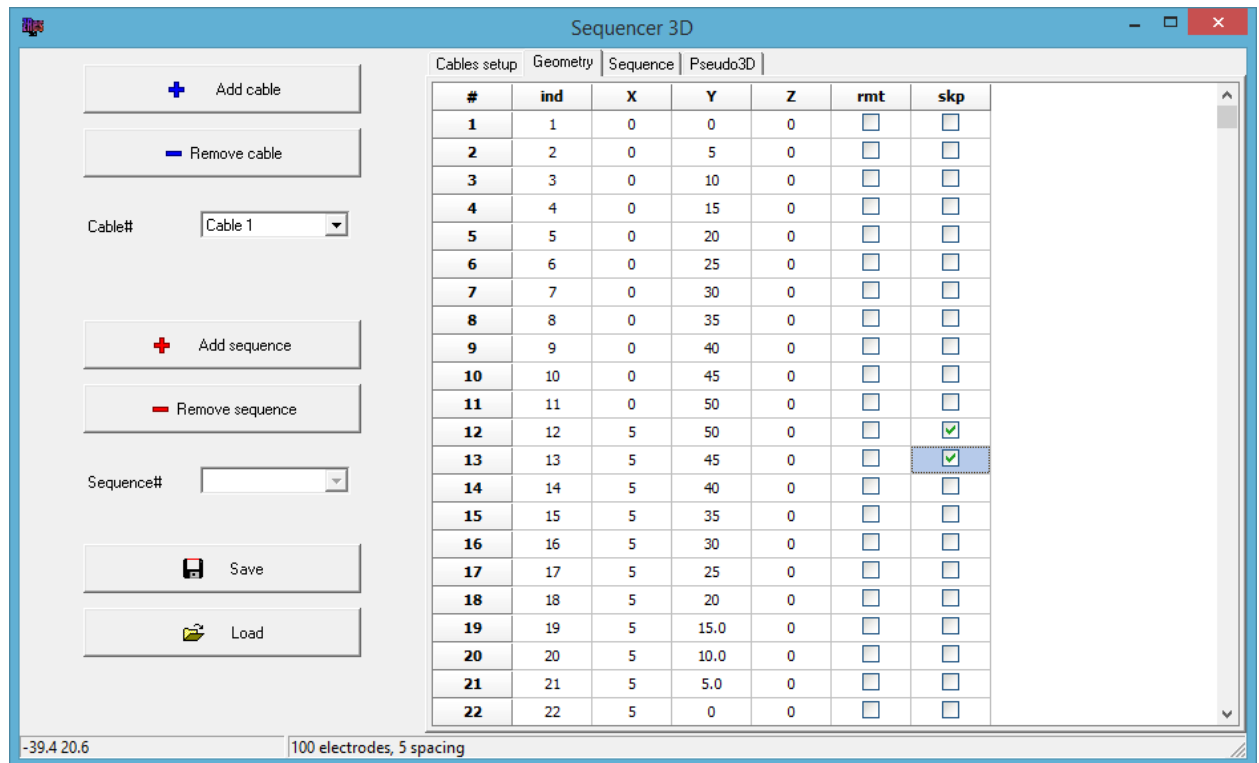


Add a spread (cable) by clicking the  button and then setting the number of electrodes and the spacing between them in meters. Left-clicking on the XY coordinate plane will specify the starting point of the spread; right-clicking will complete the process of creating the spread. Delete the spread by clicking the  button. From the **Cable#** drop-down list, you can select the spread to work with.

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

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

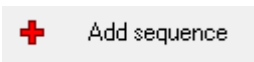
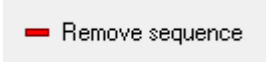
An activated checkbox in the **rmt** column indicates that the electrode is considered "at infinity" (remote). An activated checkbox in the **skp** column excludes the corresponding electrode from the created measurement protocol.



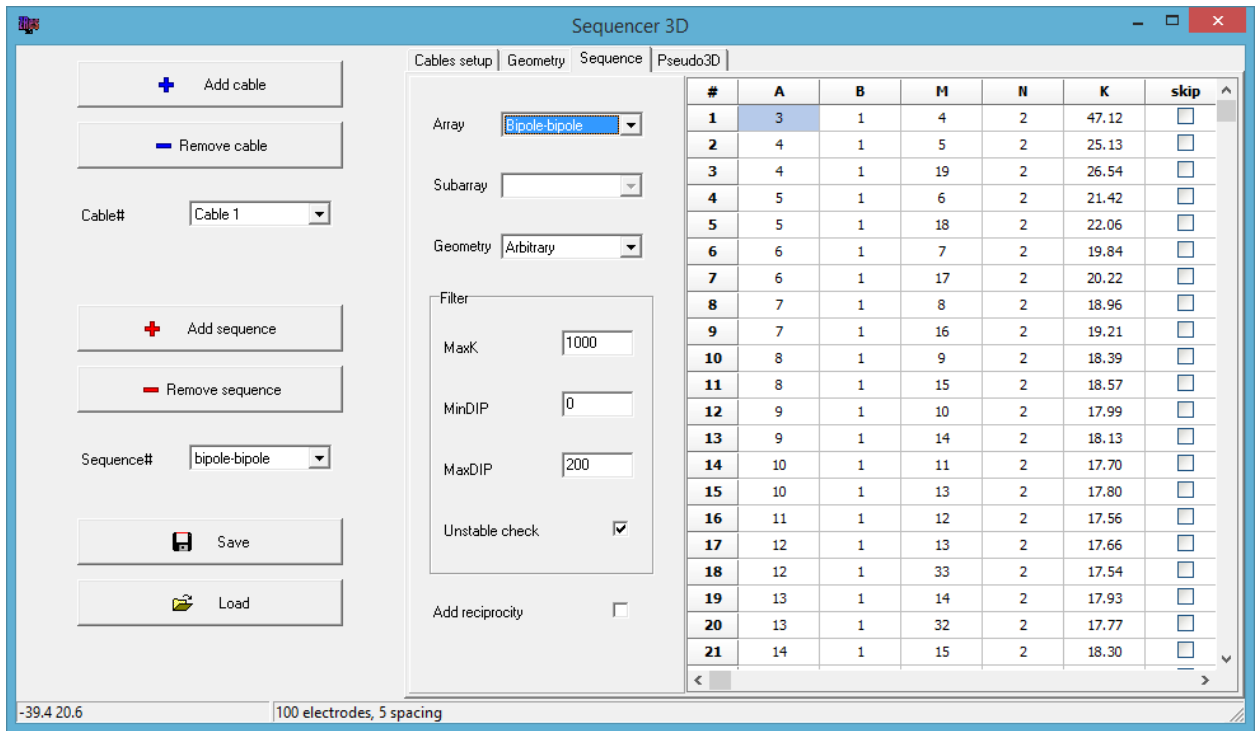
The **Sequence** tab is used for creating a measurement protocol. The **Array** and **Subarray** drop-down lists are used to specify the array type and sub-type. The **Geometry** drop-down list allows you to select the orientation of dipoles.

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.

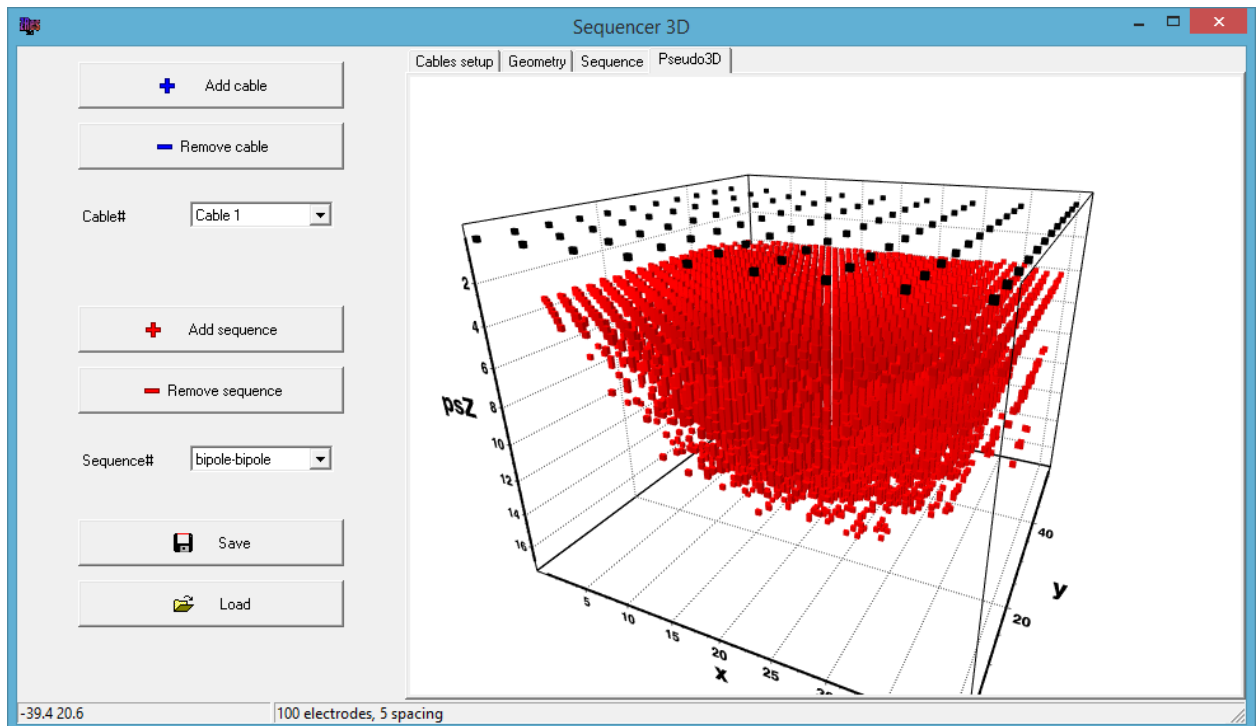
Create a measurement protocol for the selected spread and array type using the  button. Delete the current protocol using the  button. For each spread, you can create several protocols with different settings. The active (current) protocol is selected from the **Sequence#** drop-down list.







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 from the protocol.

In the **Pseudo3D** tab, a cloud of pseudosection data points is visualized. The active spread is displayed in red.



Using the  Save button, the measurement protocol can be saved in the **Sequence project**, **ZondRes3D**, **MAE sequence** and **IRIS sequence** formats.

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

## Opening data files

### *Supported formats*

To start working with **ZondRes3D** in the interpretation mode, you need to open a data file of a certain format containing electrode coordinates, topography and measurement data.

Text data files formatted in the **ZondRes3D** program have the **Z3D** extension (see the **Z3D file format** section for details).

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).

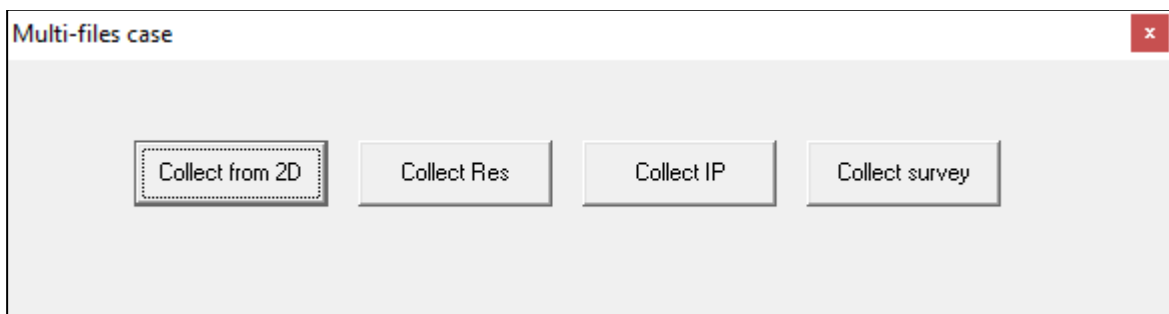
Preferably, the total number of measurements in one data file should not exceed 300 000, and the number of unique electrode positions should not exceed 10 000.

In addition to its internal **Z3D** file format, the program supports the most widely used data formats: **RES3DINV** (Geotomo Software, M.H.Loke, Malaysia), **ABEM** (Guideline Geo AB, Sweden), **AGI** (Advanced Geosciences, Inc., USA), **Syscal** (Iris Instruments, France), and others. When selecting the **File / Open file** menu command you will be prompted to select one of the following file formats:

ZondRes3D files	Open data files or a project file in ZondRes3D format.
ZondRes2D files	Open a ZondRes2D data file or several files to be combined into one project. You will be prompted to specify the XY coordinates for the start and end of each survey line.
Geosoft file	Open the GeoSoft program data file.
Syscal text file	Open a data file in SYSCAL format.
ABEM text file	Open a data file in ABEM format.
AGI data file	Open a data file in AGI format.
Res3dInv file	Open the Res2dInv program data file.
GDD files	Open a data file in GDD format.
Skala 48/64 csv	Open a data file in SKALA-48/64 format.

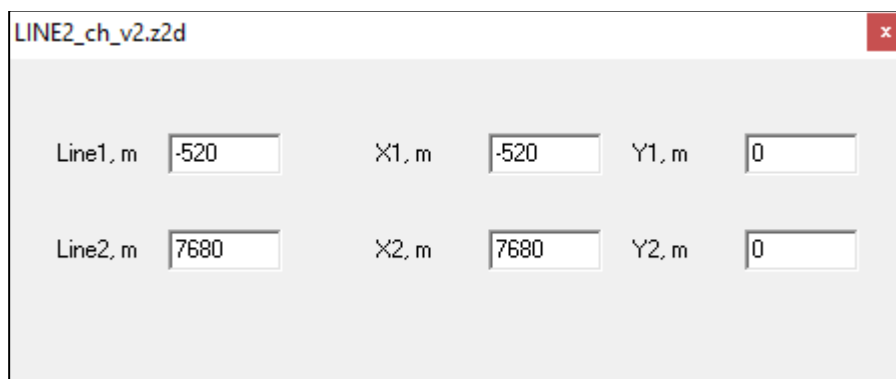
Smartem24 files	Open a data file in SMARTem24 format.
ERTLab data	Open the ERTLab program data file.
LGM device	Open a data file in the LGM device format.

**ZondRes3D** allows creating pseudo 3D survey projects by reading in 2D data from multiple text files in Z2D format. To do this, set the data type to *ZondRes2D files* in the **Open** dialog box and select the files you want to open. In the **Multi-files case** dialog box that appears, select **Collect from 2D**.



This dialog box can also be used to merge multiple Z3D files. The following merging options are available:


**Collect from 2D** – merges data of several 2D survey lines. If the 2D data files contain only X coordinates, after selecting this option you will be prompted to specify the XY coordinates for the start and end of each survey line.



**Collect Res** – this option is used in the case of monitoring (time-lapse) resistivity surveys (repeated measurements at different times using the same survey parameters).

**Collect IP** – this option is used to collect polarizability channels of a multichannel IP survey if they are stored in different files (typically, all polarizability channels are stored in one file).

When multichannel IP or monitoring data are loaded into the project, the **Time lapse** section appears in the main menu of the program that contains inversion settings for the IP and monitoring

data. Switching between channels corresponding to different times (frequencies) or measurement cycles is performed using the drop-down list (  ) that appears on the main window toolbar.

**Collect survey** – this option allows you to merge data in text Z3D format collected in several areas of the survey site. After selecting this option, the program will prompt you to set XY offsets in meters for each of the selected data files.

When working with cross-hole tomography data, the vertical coordinate of the electrodes (depth below ground surface) should be negative.

### ***Z3D file format***

Z3D 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).

Z3D is an internal Zond file format. Z3D 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 data files are text files containing information about the coordinates of current and potential electrodes, topography, and measured values (resistance, apparent resistivity, or apparent polarizability). The project file is binary and can only be read with the **ZondRes3D** program. It contains all the project information, i.e. observed data, calculated data, models, stored a priori information, data from other methods, etc.

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 multichannel IP and monitoring (time-lapse) data the file starts with the following line:

**time\_#chann 1 2 3 ... N**

where numbers denote channels – measurement cycles for monitoring surveys and delay times or frequencies for IP surveys.

The next line (or the first line for conventional ERT surveys) contains headers that specify the type of data in the respective columns.

**Prof** – index of the survey line on which the measurements were carried out (used to separate the data of individual 2D survey lines).

The following headers for electrode coordinates are used:

**C1x C2x C1z C2z C1y C2y** – for current electrodes;

**P1x P2x P1z P2z P1y P2y** – for potential electrodes.

Z electrode coordinates should be present when necessary, for example, when using submerged sources or for cross-hole and marine surveys. When entering Z coordinates of electrodes, remember that a negative value means the distance below the surface of measurements. Positive Z coordinates are only used for marine surveys (where the surface of measurements is the bottom and the electrodes are located above the bottom).

A \* symbol in front of an electrode header (e.g., \*C1x) means that this is a remote (infinity) 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);

for monitoring (time-lapse) data, an index indicating respective measurement cycle is added to the header (e.g., **Ro\_a1 Ro\_a2 ... Ro\_aN** or **Res1 Res2 ... ResN**);

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

theoretical values of resistance (with the account of the sign) calculated during modeling are saved in the file using the **TRes** header;

**Eta\_a** – apparent polarizability in percentage.

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  $V_0$  – 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.

In the frequency domain, the induced polarization parameter is the phase.

The following headers for multichannel IP measurements are used:

**Ipi** – apparent polarizability in percentage for time-domain IP;

**Mod** – absolute value for frequency-domain IP;

**Pha** – phase shift for frequency-domain IP.

Indexes are added to the headers to specify corresponding delay times or frequencies listed in the first line of the data file (e.g., **Ipi1 Ipi2 ... IpiN** or **Mod1 Mod2 ... ModN**).

The error, or measurement weight, specified in the column with the **Weight** (for resistivity) or **WeightIP** (for IP) 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/WeightIP** header in the data file), the program will automatically assign weight 1 to each measurement.

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

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

If necessary, additional horizontal nodes (data points) can be entered to extend the model area beyond the edges of the electrode array, e.g., when steep topography is present outside the array limits. It is also advantageous for cross-hole tomography surveys. The XY coordinates of each node are 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.

Data file example:

```
prof c1x c2x p1x p2x c1y c2y p1y p2y c1z c2z p1z p2z weight res weightip eta_a
0 3800 3600 3550 3575 100 100 100 100 0 0 0 0 1 0.28 1 0.15
0 3800 3600 3525 3550 100 100 100 100 0 0 0 0 1 0.12 1 0.26
0 3800 3600 3275 3375 100 100 100 100 0 0 0 0 1 0.021 1 0.32
.....
1 3800 3600 3550 3575 200 200 200 200 0 0 0 0 1 0.24 1 0.16
1 3800 3600 3525 3550 200 200 200 200 0 0 0 0 1 0.082 1 0.18
1 3800 3600 3475 3525 200 200 200 200 0 0 0 0 1 0.081 1 0.27
.....
```

A data file can have a special structure specifying a trapezoidal grid. In this case, the grid nodes are referenced to specific electrodes. Such a file consists of two blocks: electrode description and data description.

The first block starts with a line containing the **#electrodes** key. The second and subsequent lines contain data in five columns. The first column is the X coordinate, the second – Y coordinate, the third – elevation, and the fourth and fifth columns contain indexes of the grid nodes in the X and Y directions, respectively. Each line after the first line represents a particular electrode with a number (index) corresponding to the line number in the block.

The second block starts with a line containing data headers C1, C2, P1, P2, Ro\_a, etc. The following lines contain the measured data as described above, but instead of real electrode coordinates (X, Y, Z), the electrode numbers from the first block are entered.

File example:

```
#electrodes
0      0      0      0      0
2      0      0      1      0
4      0      0      2      0
6      0      0      3      0
8.1    0      0      4      0
10     0      0      5      0
12     0.1    0      6      0
14     0      0      7      0
16     0      0      8      0
18     0      0      9      0
20     0      0      10     0

.....
c1      p1      p2      ro_a
1       2       3       100
1       3       4       100
1       4       5       100
```

### ***Part II of the data file: topography data***

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

The following keys other than **topo** can be used to describe topography.

**topow** – this key 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 land surveys is carried out. 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 relative to the bottom elevations.

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

File example:

```
prof c1x c2x p1x p2x c1y c2y p1y p2y c1z c2z p1z p2z weight res
```

```
.....
10 7280 7680 7160 7080 2000 2000 2000 2000 0 0 0 0 1 0.23
10 7280 7680 7080 7000 2000 2000 2000 2000 0 0 0 0 1 0.093
10 7280 7680 7000 6840 2000 2000 2000 2000 0 0 0 0 1 0.076
10 7280 7680 6840 6680 2000 2000 2000 2000 0 0 0 0 1 0.030
```

Topo

```
.....
-5.200000000000000E+0002 0.000000000000000E+0000 5.576400000000000E+0002
-4.800000000000000E+0002 0.000000000000000E+0000 5.546300000000000E+0002
-4.400000000000000E+0002 0.000000000000000E+0000 5.514800000000000E+0002
-4.000000000000000E+0002 0.000000000000000E+0000 5.484700000000000E+0002
-3.600000000000000E+0002 0.000000000000000E+0000 5.459000000000000E+0002
```

The electrode elevations can be specified in the C1z ... P2z columns; in this case, the **topo** block is not necessary. Note that for surveys with submerged electrodes, the electrodes that are on the ground surface or the bottom of a water body have Z coordinates equal to 0; the electrodes in a borehole have negative Z coordinates. If the measurement system consists of both submerged and surface electrodes, the topography should be described in the **topo** block, Z coordinates of the submerged electrodes – in the C1z ... P2z columns.


### ***Import data from a text file or Excel spreadsheet***

Importing data from a text file or Excel spreadsheet is performed using the **File / Import 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; after pressing the button the respective line in the table turns grey).



N	None	None	None	None	None	None	None	None
1	prof	None	c2x	p1x	p2x	c1y	c2y	p1y
2	0	C1x	*	10.40	14.02	4.46	*	4.37
3	0	C2x	*	9.26	12.76	4.46	*	4.25
4	0	P1x	*	8.06	11.53	4.46	*	4.15
5	0	P2x	*	6.84	10.40	4.46	*	4.28
6	0	C1y	*	5.53	9.02	4.46	*	4.41
7	0	C2y	*	4.17	7.56	4.46	*	4.47
8	0	-0.051	*	5.33	6.39	4.46	*	4.44
9	0	-0.051	*	4.41	5.53	4.46	*	4.47
10	0	-0.051	*	3.43	4.65	4.46	*	4.50
11	0	-0.051	*	2.44	3.68	4.46	*	4.46
12	0	-0.051	*	1.94	3.18	4.46	*	4.41
13	0	-0.051	*	1.44	2.68	4.46	*	4.45
14	0	-0.051	*	1.94	2.18	4.46	*	4.41
15	0	-0.051	*	1.69	1.94	4.46	*	4.44
16	0	-0.051	*	1.44	1.69	4.46	*	4.45
17	0	-0.051	*	1.19	1.44	4.46	*	4.47
18	0	-0.051	*	0.95	1.19	4.46	*	4.49
19	0	-0.051	*	0.70	0.95	4.46	*	4.48
20	0	-0.051	*	0.44	0.70	4.46	*	4.47
21	0	-0.051	*	0.20	0.44	4.46	*	4.47
22	0	0.20	*	9.50	13.02	4.47	*	4.24
23	0	0.20	*	8.32	11.78	4.47	*	4.15
24	0	0.20	*	7.08	10.64	4.47	*	4.24
25	0	0.20	*	5.75	9.26	4.47	*	4.38
26	0	0.20	*	4.41	7.81	4.47	*	4.47

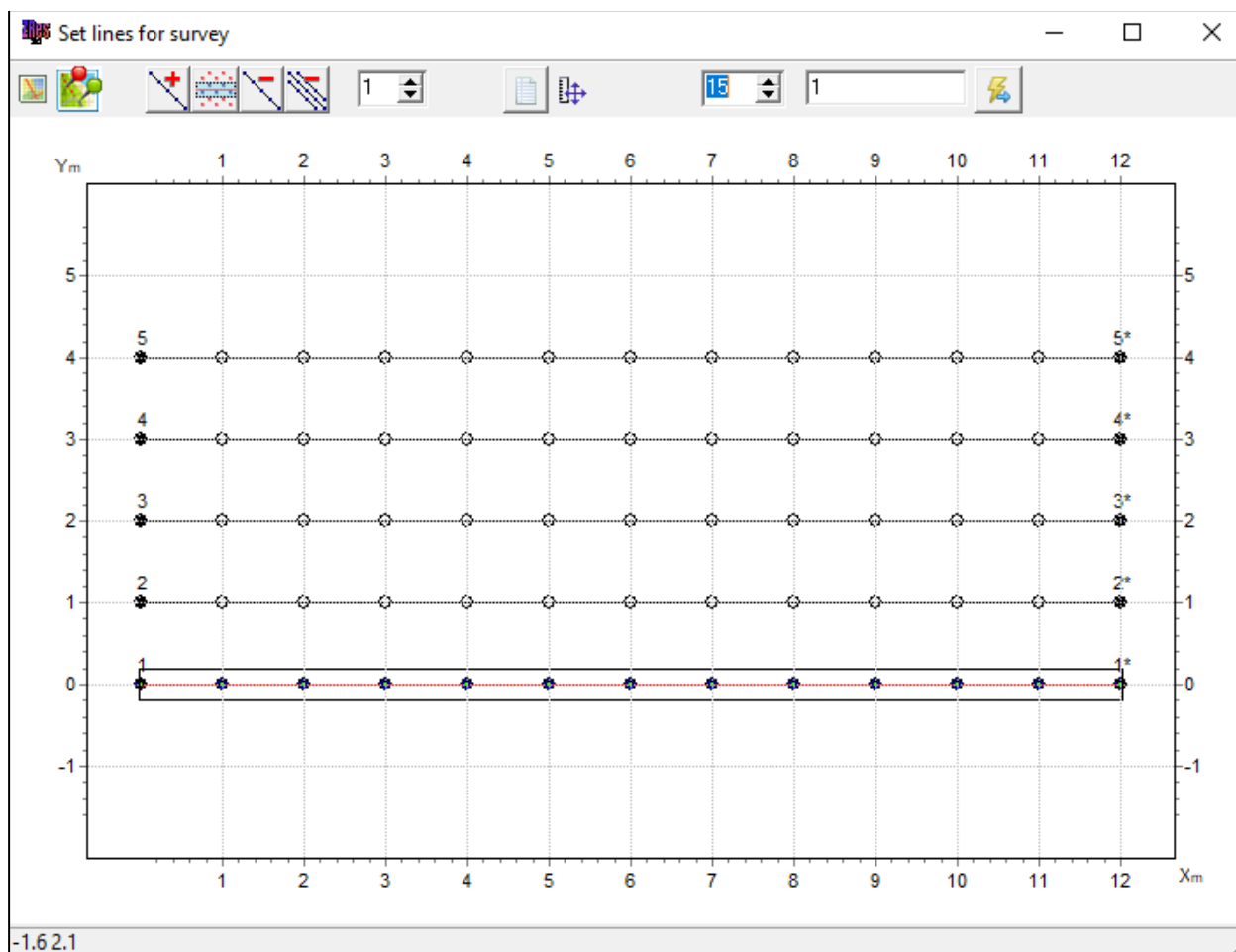
Two drop-down lists on the top panel are used to select the type of data (**Land** – land survey data, **Underwater** – marine survey data, **Cross-boreholes** – cross-hole survey data) and units of polarizability (**IP %** – percentage in the 0-100 range; **IP mV/V** – mV/V in the 0-1000 range; **IP deg** – phase shift in degrees; **IP mrad** – phase shift in radians, always negative).

Cells in the uppermost row of the dialog box table contain a drop-down list of headers (**line** – survey line number; **C1x**, **C2x**, **C1y**, **C2y**, **C1z**, **C2z** – current electrode coordinates; **P1x**, **P2x**, **P1y**, **P2y**, **P1z**, **P2z** – potential electrode coordinates; **ro\_a** - apparent resistivity; **res** – absolute value of resistance; **IP** – apparent polarizability). For each column to be imported, it is necessary to specify the corresponding header. To import the data into the project, press the  button.





### *Line editor*







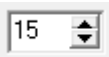


After opening a data file or creating a synthetic measurement system, the **Line settings** module is started automatically. In this module, you can set, remove and edit survey lines, change electrode coordinates, display a raster image or plot a topography contour map in the background.


The line editor can be opened at any time while working with a project using the **Options / Set survey lines** menu command.




The toolbar of the module window contains the following options (left to right):

	Import a raster image as a base map or plot topography contours in the background.
	Import a Bing satellite image as a base map (the coordinates in the input data file should be in the UTM coordinate system; the UTM zone should be specified).
	Enable/disable the line drawing mode. Points are added to the line by left-clicking on the XY plane; right-click on the plane finalizes the line plotting.  After the line drawing mode is enabled, right-click on the  button brings up a context menu with the following options: <i>Set line coordinates</i> – specify the line coordinates manually; <i>Cursor to near point</i> – snap the line points to the nearest electrode locations; <i>Invert line</i> – invert the survey line; <i>Create lines auto</i> – plot survey lines automatically.

	Include into the active survey line the points (electrodes) that fall into the rectangular selection area surrounding the line. The width of the area is specified in the input box (see below).
	Delete the active survey line.
	Delete all survey lines.
	Select the active survey line.
	Open an editable table of electrode coordinates.
	Switch between proportional (1:1) XY scale and the scale that fits the plot into the plan view area.
	Set the width of the rectangular selection area.
	Specify the name for the active survey line.
	Proceed to the data inversion mode.

Electrode coordinates can be edited in the **Locations** dialog box (the  toolbar button). The coordinates can be imported from a text file containing columns with the point numbers and XYZ coordinates or copied directly from an Excel spreadsheet.

name	X	Y	Z	v
	0	0	0	✓
	1	0	0	✓
	3	0	0	✓
	2	0	0	✓
	4	0	0	✓
	5	0	0	✓
	6	0	0	✓
	7	0	0	✓
	8	0	0	✓
	9	0	0	✓
	10	0	0	✓
	11	0	0	✓
	12	0	0	✓
	0	1	0	✓
	1	1	0	✓
	3	1	0	✓
	2	1	0	✓
	4	1	0	✓
	5	1	0	✓
	6	1	0	✓

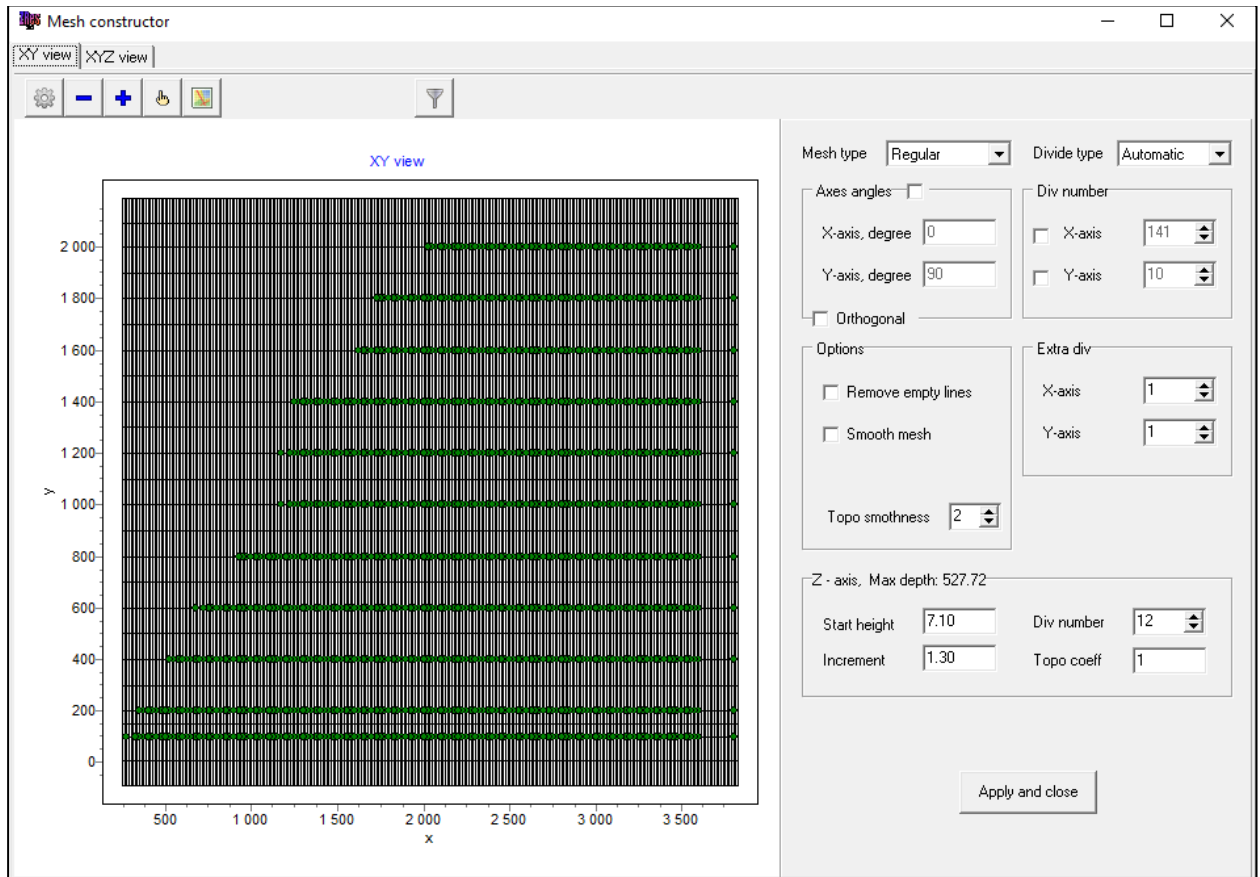
To exit the line editor module and proceed to the interpretation mode, press the  button.

### *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.

The **Mesh constructor** window contains two tabs – **XY view** and **XYZ view**. In the **XY view** tab, the model mesh parameters are configured.



The **XY view** tab consists of a toolbar, plan view plot of the survey grid on the left, and mesh configuration parameters on the right.

The toolbar contains the following buttons:

	Rebuild the mesh with new parameters.
	Remove a mesh line. Left-click on the line in the plan view area to remove it. Right-click on the toolbar button brings up a context menu where you can choose an X or Y direction of lines to be removed.
	Add a mesh line. Left-click in the specific location in the plan view area to add a line. Right-click on the toolbar button brings up a context menu where you can choose an X or Y direction of lines to be added.
	Enable/disable the mesh node editing mode.
	Left-click on the button shows/hides topography contours. Right-click on the button allows you to import topography data from an XYZ text file.
	Check for the presence of potential and current electrodes that produce a measurement in the same cell (this situation should be avoided).

In the right section of the tab, the mesh parameters are specified.

The **Mesh type** drop-down list specifies the mesh type. The following mesh types are available: *Regular* – suitable for most standard survey grids when the electrodes are arranged more or less evenly across the area; *General* – for relatively disordered survey grids; *One line* – for cases when the survey is conducted along a single survey line.

The **Axes angles** group box sets the local angles for the X and Y axes of the grid (**X-axis, degree** and **Y-axis, degree**, respectively). The **Orthogonal** checkbox, if checked, keeps the axes orthogonal.

The **Div number** group box sets the number of divisions along the X and Y axes when the corresponding checkboxes are checked.

The **Topography** group box contains the **Topo smoothness** option which sets the degree of smoothing for topography contours. The values are selected from 0 to 5, where 5 is the maximum smoothness; check the **Krigging topo** checkbox to apply.

The **Divide type** drop-down list allows you to select *Automatic* or *Manual* modes of mesh configuration.

When the **Divide type** is set to *Automatic*, the **Extra div** group box is available, where you can set the number of additional nodes between unique electrode positions in the X and Y directions.

When the **Divide type** is set to *Manual*, the **Axes Range** group box is available, where you can specify the minimum and maximum X and Y coordinates.

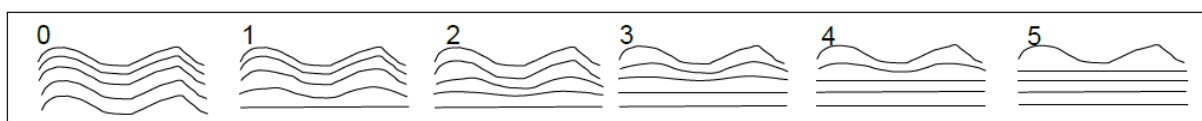
The **Z-axis** group box contains options that allow you to set the parameters of the model mesh in the vertical direction.

**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.

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

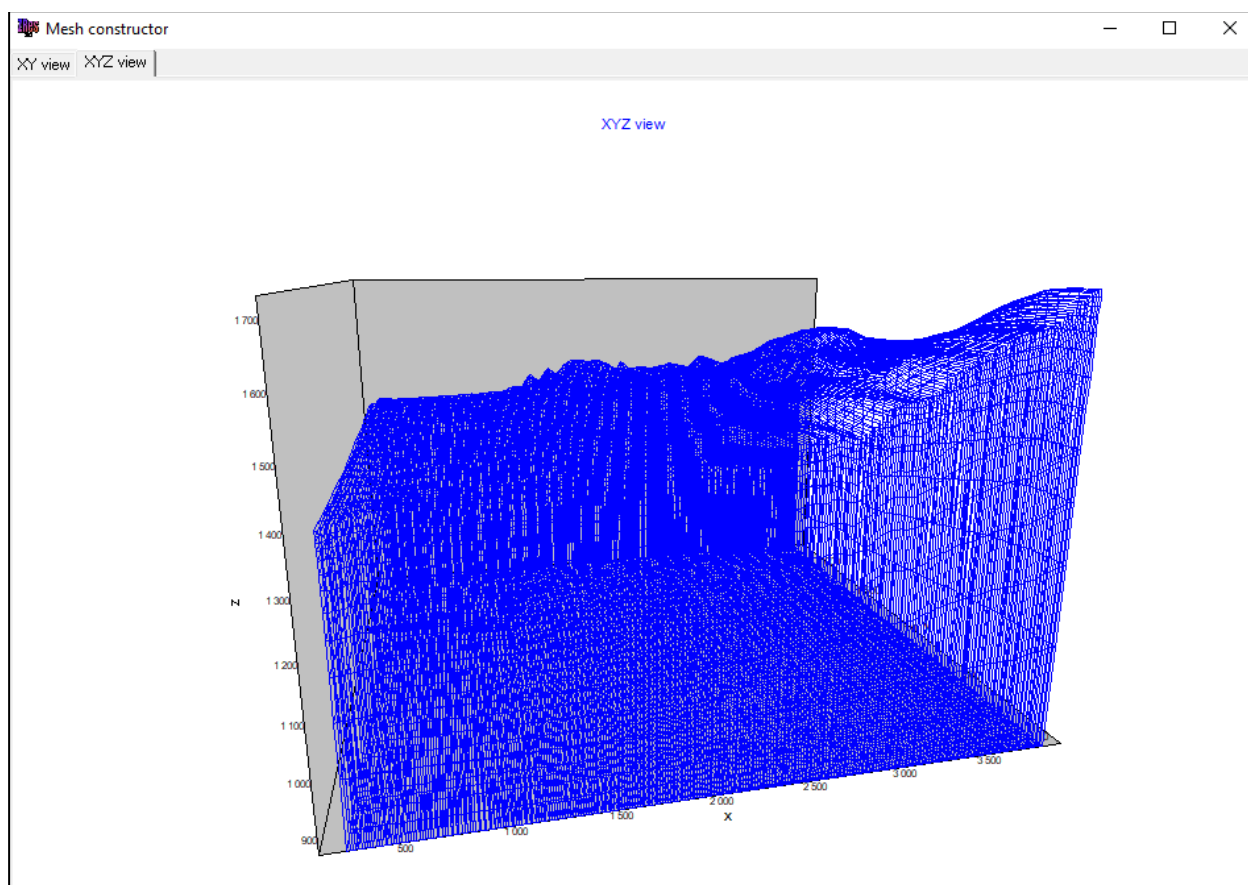
**Div number** – sets the number of model layers. Typically, 12-20 layers are enough for a comprehensive model. A large number of layers will significantly reduce the speed of calculations.

**Topo coeff** – sets the coefficient of topography distortion with depth (values from 0 to 5). If set to 0, the topography 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. If set to 5, the boundary between the first and second layers will be flat (see figure below).



The **Max depth** value 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.


The **XYZ view** tab allows viewing the configured mesh in 3D. Use the left mouse button to rotate the image. To scale the plot, spin the mouse wheel. Press and hold the right mouse button to move the plot in any direction. Right-click on an axis while holding down the Shift key to access the axis configuration parameters.



After configuring the mesh, press the **Apply and close** button to proceed to the working mode of the program. A brief information about the new project (the number of data points and 3D model cells) will appear in the status bar of the main window.

## Data visualization

In **ZondRes3D** data can be visualized in the form of a pseudosection (**Options / Data / Pseudosection**) or graphs (**Options / Data / Graphics-plot**) for each individual survey line, and in the form of a contour map or 3D pseudosection (**Options / Data / 3D pseudosection ON/OFF**). Survey lines

to be displayed are selected from the  drop-down list on the toolbar; line IDs correspond to the entries in the **Prof** column of the data file.

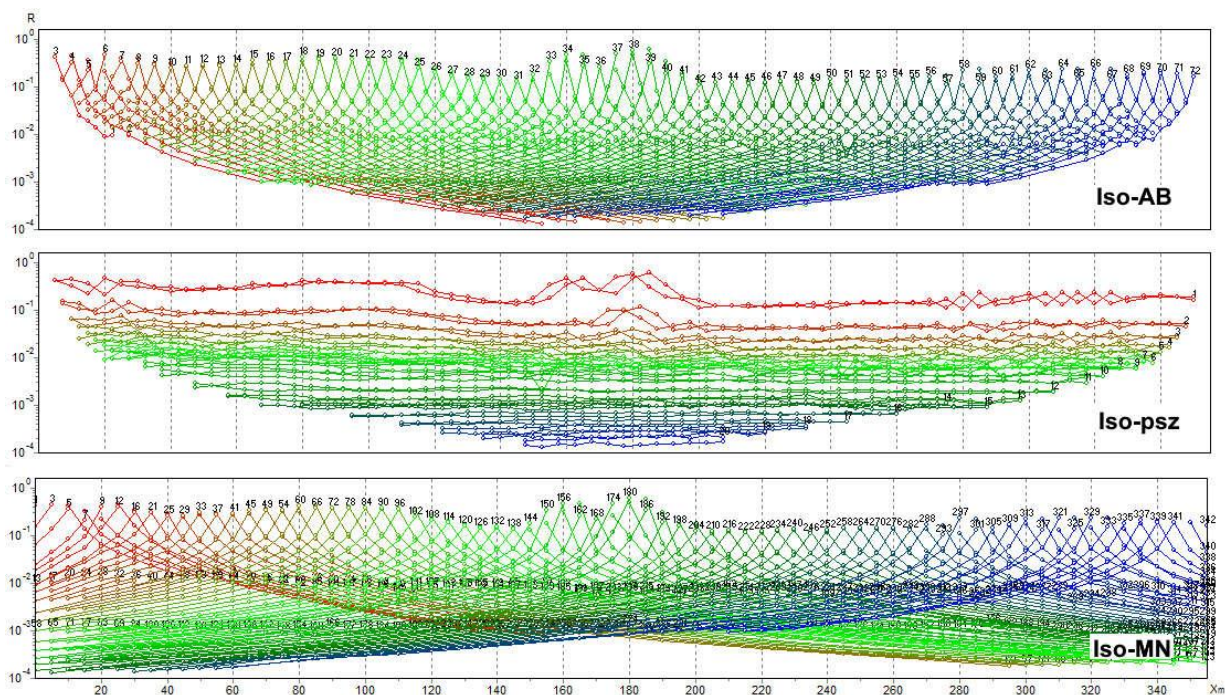
For ERT surveys, the data can be displayed in the values of:

- apparent resistivity (**Option / Data / Apparent resistivity**);
- resistance V/I (**Option / Data / Resistance**);
- resistance (V/I) normalized by the lengths of potential and current electrode spacings (**Option / Data / ABMN normed**). This option is available when the *Resistance* display is selected.

### 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 / Type graphics** menu subsection you can choose to plot graphs for a specific geometric factor or pseudodepth (**Options / Data / Graphics-plot / Iso-psz**), for a specific position of current electrodes (**Options / Data / Graphics-plot / Iso-AB**), or potential electrodes (**Options / Data / Graphics-plot / Iso-MN**).



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** dialog box brought up by right-clicking on a graph while holding down the Shift key. The **Axis editor** is opened by right-clicking on an axis while holding down the Shift key.

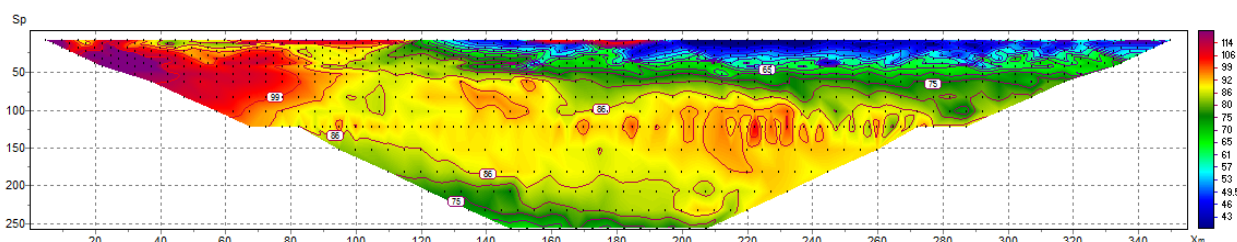
To display only one graph and hide the rest, left-click on the specific graph's index in the legend while holding down the Shift key. If you click again, the operation is reversed. To scroll through the graphic plot, select one or several adjacent graphs in the legend and spin the mouse wheel, with the mouse pointer hovering over the legend. The indexes of the active graphs will change.

### *Pseudosection*

The pseudosection is a distribution of data with depth along a 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 in the area surrounding the pseudosection 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 <b>Contour-section setup</b> dialog box.
Print preview	Print the pseudosection.
Save picture	Export the pseudosection to a BMP or PNG file.
Save XYZ file	Export the pseudosection to a DAT text file.
X:Y=1:1	Set proportional (1:1) scaling for horizontal axes.

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  $\frac{1}{4}$  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:

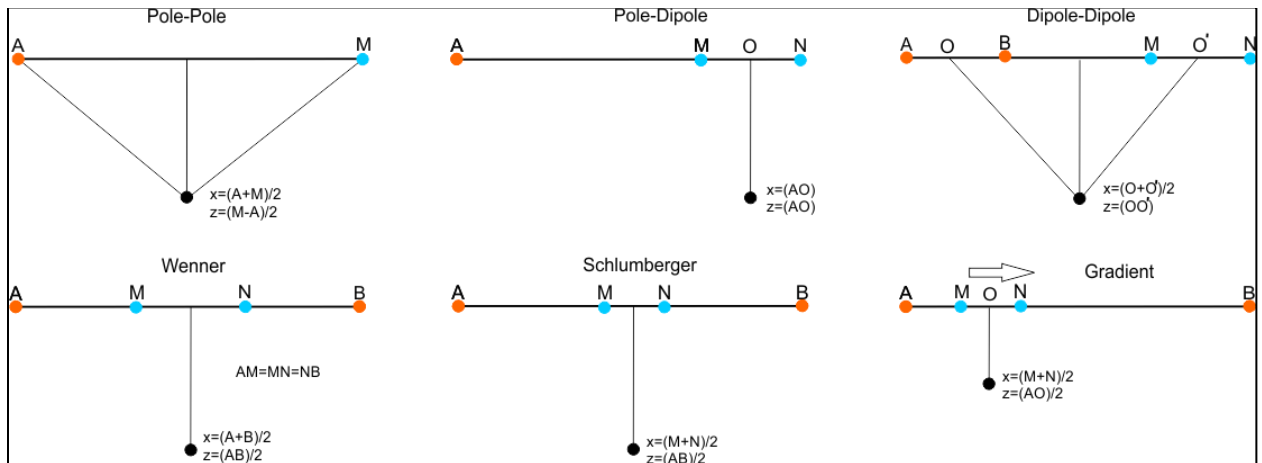
Options / Data / Point position / 3 electrodes	<p>For a three-electrode array:</p> <p><math>(M+N)/2</math> – the data point is at the midpoint between potential electrodes M and N;</p> <p><math>(A+M)/2</math> – the data point is at the midpoint between the current electrode A and the nearest potential electrode M.</p>
--	--

- For a *Dipole-Dipole* array, the Z coordinate corresponds to  $\frac{1}{2}$  the distance between the centers of the current and potential dipoles. For the X coordinate, the program provides three options:

Options / Data / Point position / 4 electrodes	<p>For a four-electrode array:</p> <p><i>Auto</i> – automatic determination;</p> <p><math>MN/2</math> – the data point is at the midpoint between potential electrodes M and N;</p> <p><i>AB-O-MN</i> – the data point is at the midpoint between current (AB) and potential (MN) dipoles.</p>
--	--

- For a *Gradient* array, the X coordinate corresponds to the midpoint between the potential electrodes; the Z coordinate corresponds to  $\frac{1}{2}$  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  $\frac{1}{2}$  the distance between the electrodes.

- For a *Schlumberger* array, the X coordinate corresponds to the center of the array; the Z coordinate corresponds to  $\frac{1}{2}$  the distance between the outermost (current) electrodes.





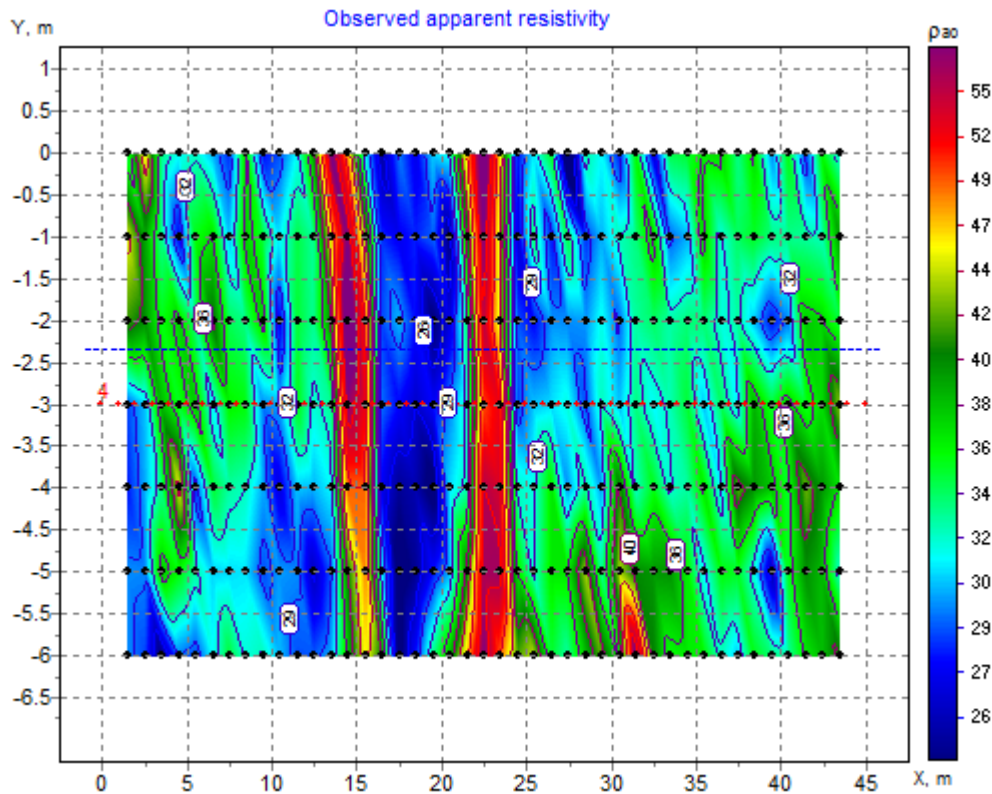
In the second (lower) pseudosection which displays the calculated data, you can display relative misfits (RMS errors) for each measurement (**Options / Data / Data Misfit**).

### *Plan view data visualization*

In the upper left section of the program workspace a plan view contour map of the data for a certain depth level (which corresponds to a pseudodepth, or electrode spacing) is displayed. The following data types can be plotted on the map:

- observed data (**Options / Data / Observed**);
- calculated data (solution of the forward problem) (**Options / Data / Calculated data**);
- the discrepancy between the observed and calculated data (**Options / Data / Misfit**).

The depth level can be selected using the  drop-down list on the toolbar. By right-clicking on the  button, you can select a number of neighboring levels that will be used to plot the map in the case when some electrode spacings are missing in a portion of the data.



The dotted red line on the map shows the position of the survey line for which graphs or pseudosections are plotted in the top right section of the program workspace. The dashed blue line shows the current position of the 3D model slice.

The position of a borehole is shown on the map with a black cross.

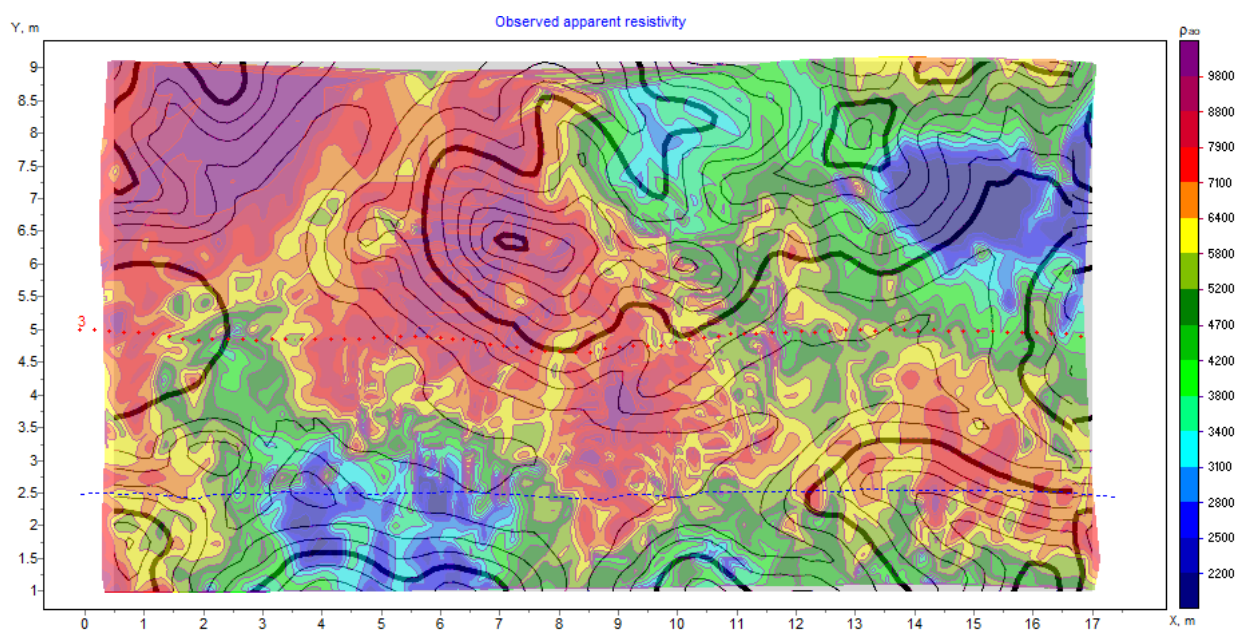
If you click on a data point on the map, the position of the current and potential electrodes for this point will be displayed with red and blue dots, respectively.

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

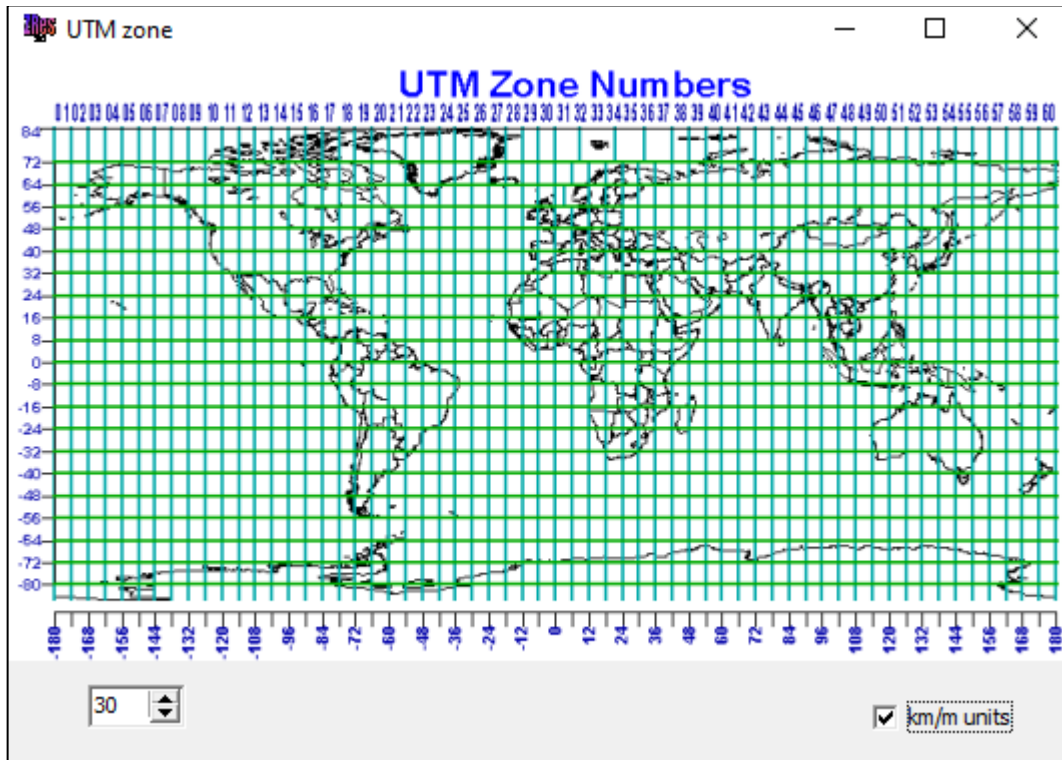
Log data scale	Use the logarithmic scale for the color scale.
Smooth mode	Use a smooth interpolation palette.
Display grid point	Show/hide measurement points.
Display ColorBar	Show/hide the color scale.
Setup	Open the <b>Contour-section setup</b> dialog box.
Print preview	Print the contour map.
Save picture	Export the contour map to a BMP or PNG file.
Save XYZ file	Export the contour map to a DAT test file.
X:Y=1:1	Set proportional (1:1) scaling for horizontal axes.
Load background	Import an image and display it in the background.
Load bingmap	Import a Bing satellite image and display it in the background.
Remove background	Remove the background.

Draw in Surfer	Plot the contour map in Surfer.
Copy BG from model	This option is available if an XY model slice is selected. In this case, the graphical representation of the slice can be used as a base (background) layer.

The *Load background* option allows you to import any raster image to be displayed in the background; the contour map becomes semi-transparent. This option is useful for comparing measured data with topographic or geological maps, and with data of other methods (geophysical, geochemical, etc.). The figure below shows an example of apparent resistivity data with a topographic map in the background.



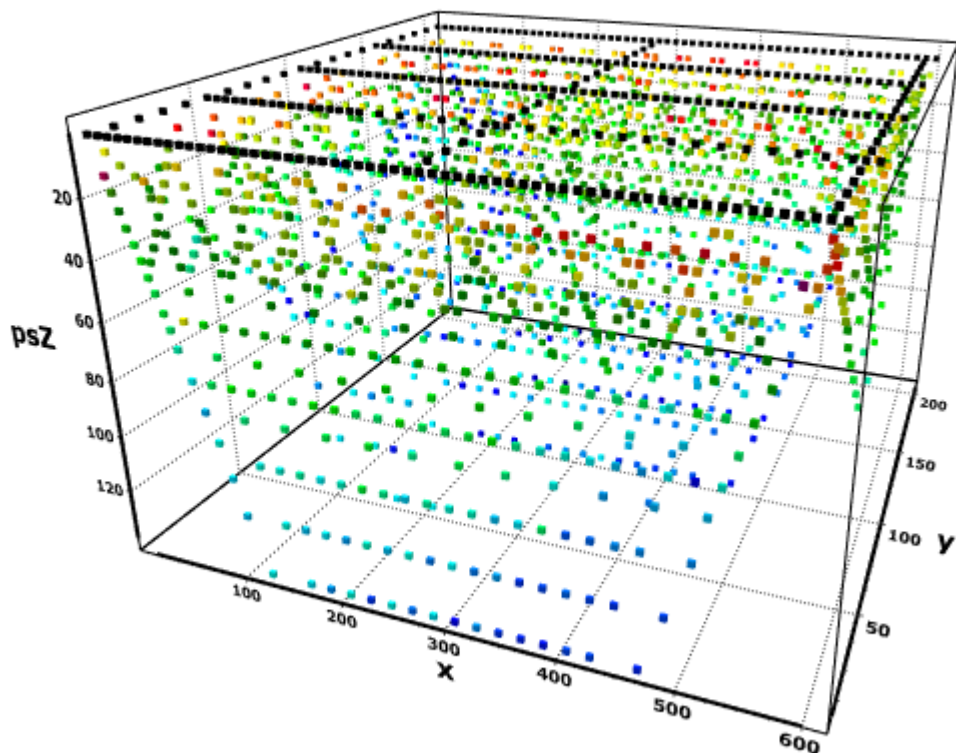
The *Load bingmap* option allows you to import a Bing satellite image in the case when electrode coordinates are in the UTM coordinate system. After selecting this option, the UTM zone selection window appears where it is necessary to specify the UTM zone of the survey site.



### *3D pseudosection*

The 3D pseudosection is used to display the measurement data of a 3D survey or to display the data of all survey lines at the same time.

To display the data in the form of a 3D pseudosection, use the **Options / Data / 3D pseudosection ON/OFF** menu command; the 3D pseudosection replaces the plan view contour map in the upper left section of the program workspace. The colors of the data points correspond to the values of the displayed parameter.



## Preparing the data for inversion

The processing of the data before running the inversion includes assessing the quality of field measurements and removing poor-quality data points.

### *Quality control module*

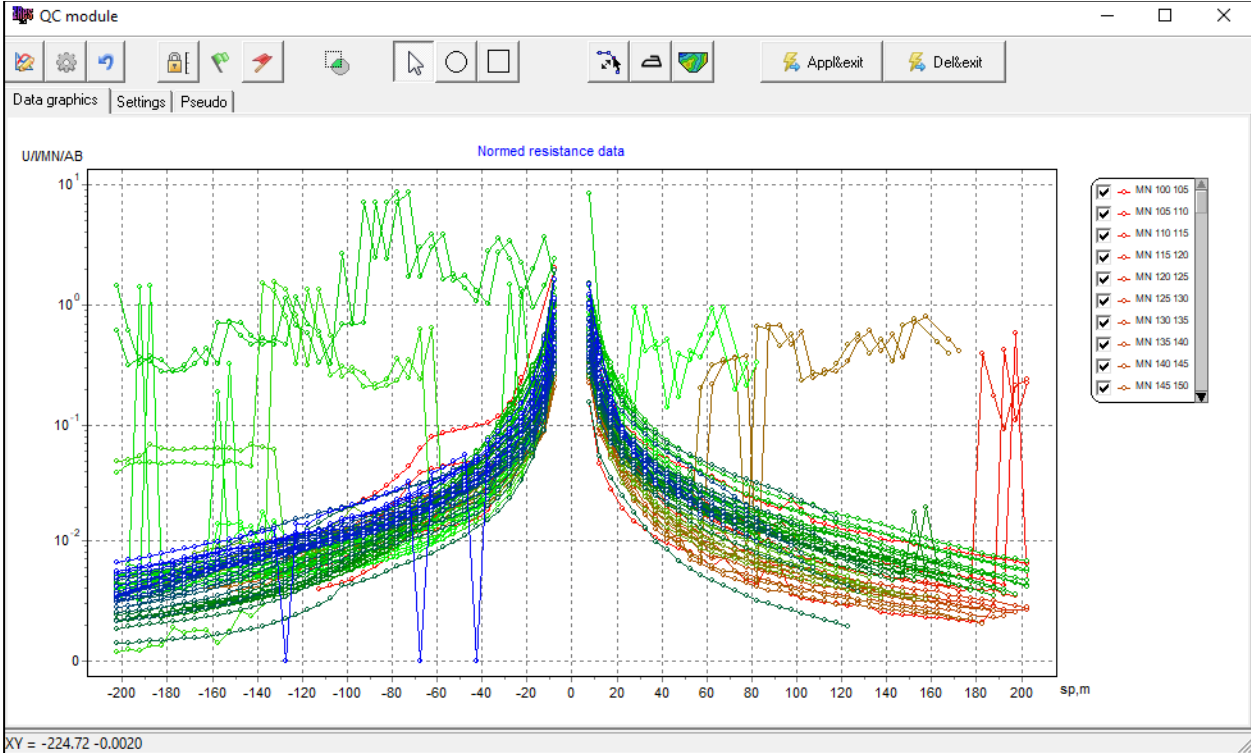
For quality control and data editing, a special module is implemented in **ZondRes3D**, which can be opened with the **Options / Quality control module** command. The module allows you to perform the following operations with data:

- 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;
- remove bad data points from the graphs.

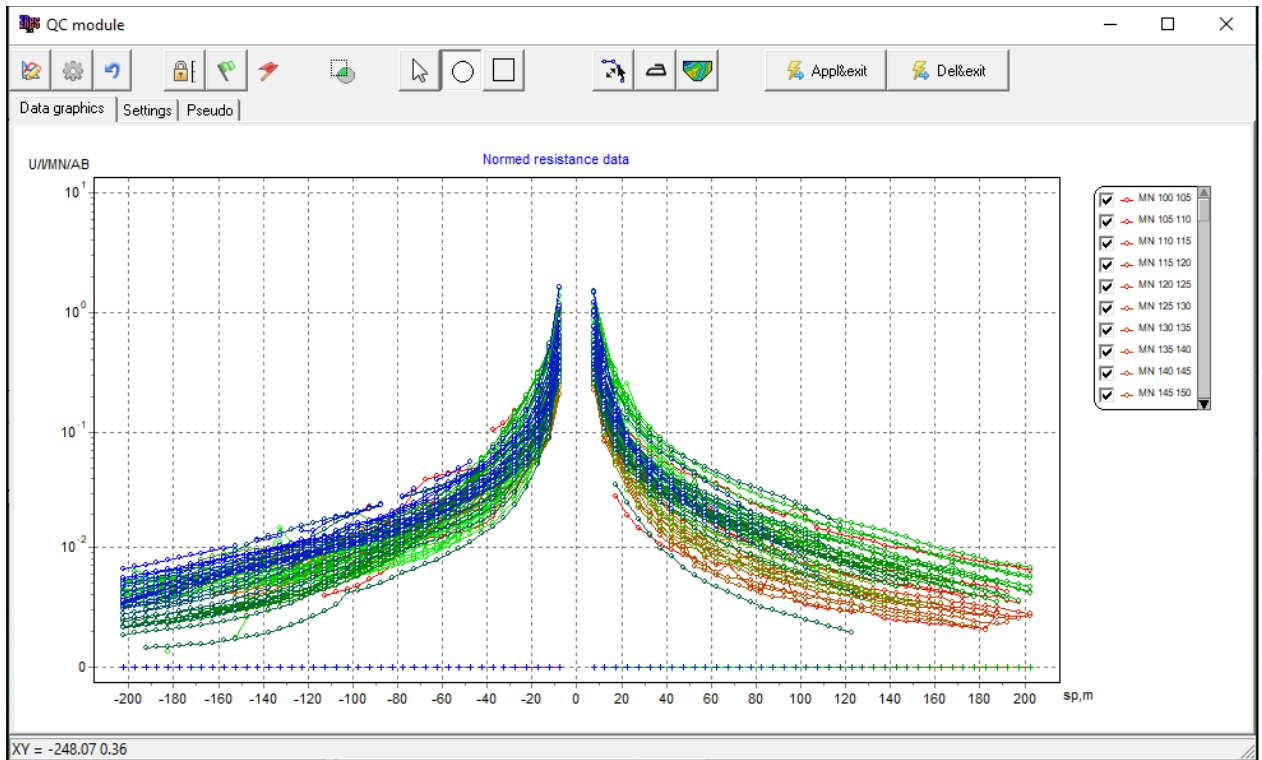
The ability to sort data by the type of array, measured signal, various geometric parameters and measurement weights, combined with manual and automatic bad points removal options, allows you to carry out the entire quality control sequence to prepare the data for inversion.

The features of the module described in this section are available for 2D and pseudo 3D surveys. For 3D surveys, the functionality of the QC module is limited.

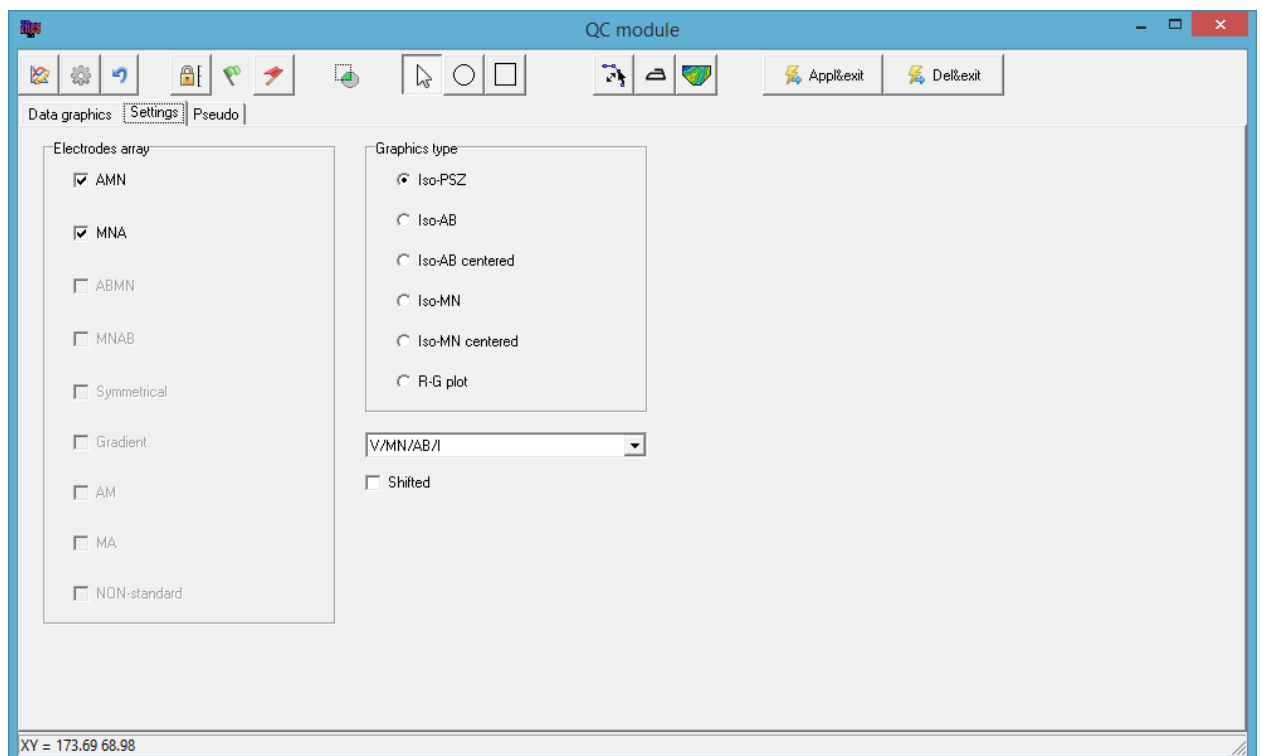
The **QC module** window consists of a toolbar and three tabs: the **Data graphics** tab displaying graphs of selected parameters (the figure below shows the original (top) and edited (bottom) data sets), the **Settings** tab for selecting the parameters to display, and the **Pseudo** tab displaying 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.






In the **Data graphics** 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.


Other operations like zooming in and out, selecting one or several graphs for display are performed the same way as it is done in the *Graphic plot* section of the main window (see the **Graphic plot** section above).



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 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:


- The measured data is susceptible to noise interference. In this case, the noisy data should be deleted.
- The resistivity variation range is set too narrow or the task is over-regulated. In this case, you can change the model parameter constraints (or do not use any) or inversion settings and repeat the inversion.


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  button, 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 selected points (not recommended); set weights for selected points; change the sign (positive or negative) of selected points (sometimes required for cross-hole surveys).

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 updates the pseudosection plot in the **Pseudo** tab.

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).

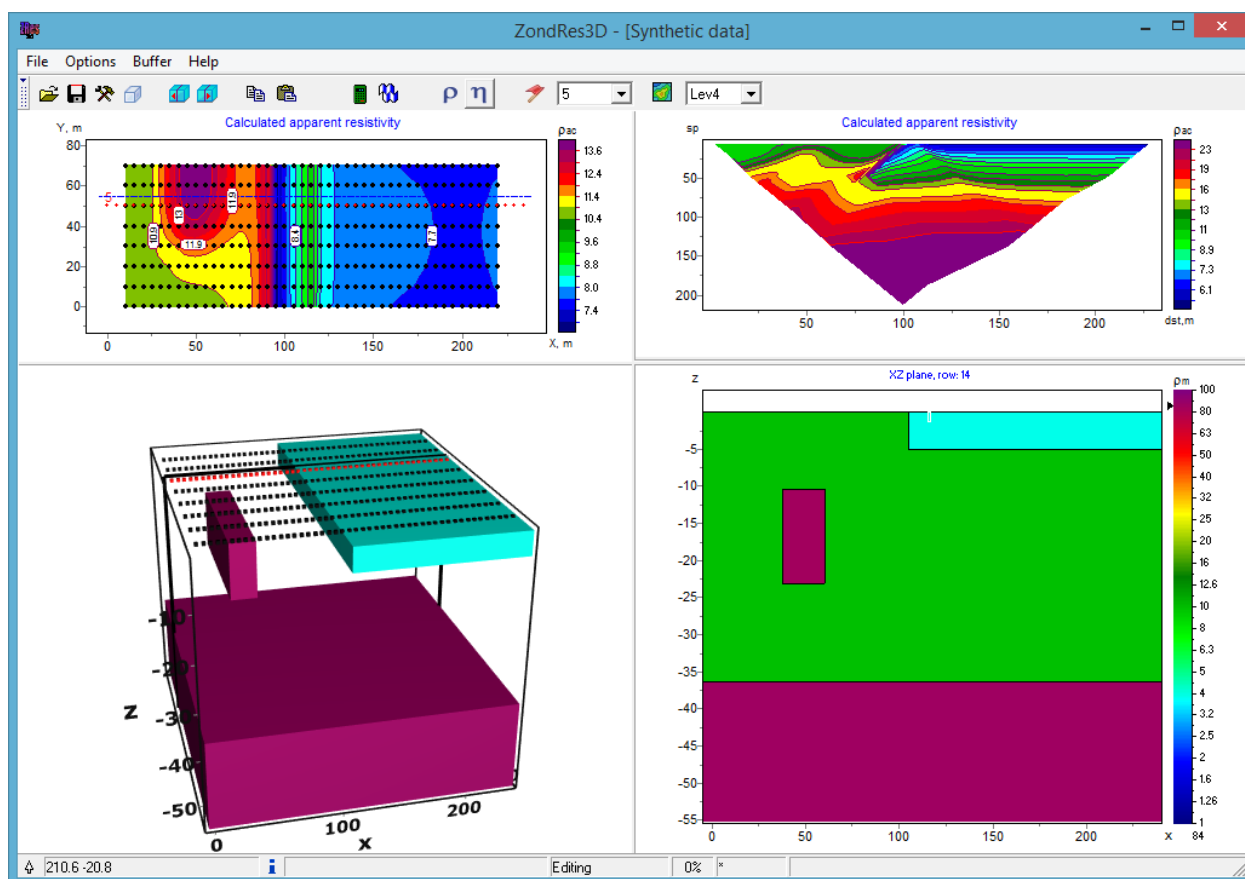
The quality control module is typically used to process consecutively all survey lines of the project.

## 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.

**ZondRes3D** allows you to model the data of three-dimensional (3D) ERT and IP surveys, with the account for the topography of the survey site. The input data for the modeling are the geometry

of the measurement system and the resistivity model. It is possible to create measurement systems and resistivity models in **ZondRes3D** or use measurement systems created in third-party programs. It is also possible to build geoelectric models based on other models of the **Zond** package or geological cross-sections in the form of raster images.



### *Editing the model*

Resistivity modeling is performed in the *Model editor* – the lower right graphical section of the main program window – in the *Block-section* mode (**Options / Model / Block-section**).

In the model editor, you can change the parameters of individual model cells using the mouse. 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.

Using the **Options / Model / Plane** option, you can select the plane for model slices (XZ, YZ or XY). The slice plane can be changed using the hotkeys:

- "Alt+1" – slice in the XZ plane;
- "Alt+2" – slice in the YZ plane;
- "Alt+3" – slice in the XY plane.

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 cell is indicated with a fill of white dots.

There is a set of commands and several selection tools for working with the model in the program. They are accessible through the context menu brought up by right-clicking in the model area:

Display cell setup	Open the <b>Cell Setup</b> dialog box.
Cell to cursor value	Use the parameter value 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.
Selection \ Elliptical selection	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 <b>Model setup</b> dialog box (right-click in the area surrounding the model section and select <b>Setup</b> ). This action can also be performed by pressing the X key.
Selection \ Remove selection	Remove selection.
Clear model	Clear the current model slice.

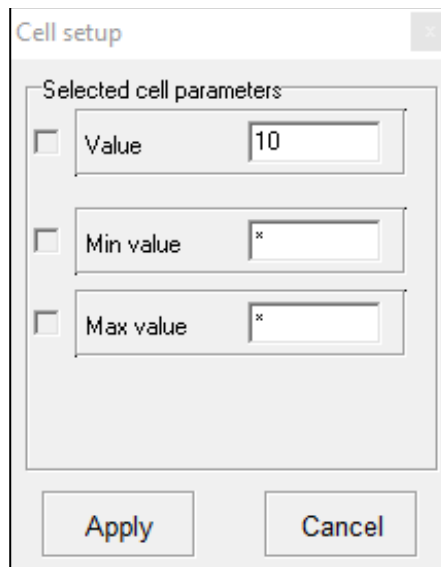
Pressing and holding down a mouse button while holding down the Ctrl key allows you to move a selected group of cells within the editing area. With the left mouse button pressed, the contents of the selection are copied to a new location. With the right mouse button pressed, the contents of the selection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

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.

**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:

<i>Area</i>	<i>Option</i>	<i>Description</i>
Model title area	Display model mesh	Show/hide the model mesh.
	Display objects border	Show/hide object boundaries.
	Display color bar	Show/hide the color scale.
	Setup	Open the Model Setup dialog box.
	Zoom&Scroll	Enable the Zoom&Scroll mode.
	Print preview	Print the model.
	X:Y=1:1	Set proportional (1:1) scaling for horizontal and vertical axes.
Color scale	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 based on the slice parameters (the minimum and maximum values of the model parameter are set as the scale limits).
	Log scale	Set the logarithmic scale for the color scale.
	Smooth image	Display the model in the smooth interpolated mode.

	Set cursor value	Set the current value of the parameter.
--	------------------	---

In the **Zoom&Scroll** mode, press the left mouse button and drag the mouse up/down and to the right to zoom in on a portion of the model area. To zoom out, drag the mouse in the opposite direction.



### *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 **Calculated data [\*z3d]** type. The saved file can then be opened as an observed data file. Alternatively, the **Project with calculated [\*z3d]** file type can be used, but after opening the file the imported model has to be cleared before inversion. To export the results of 3D modeling to 2D, use the **Options / Import/Export / Export line to z2d** command.


It is then possible to perform inversion of the data and analyze the differences between the inverted model and the original one.

## **Data inversion**

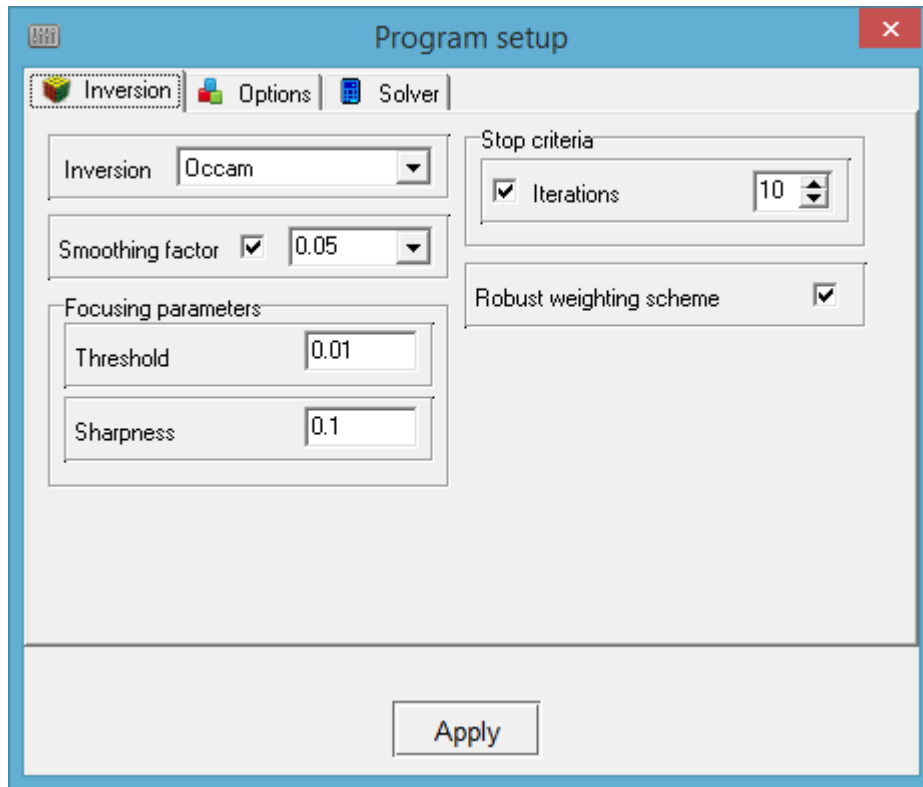
The program performs inversion of resistivity and IP data. It allows you to carry out inversion of the resistivity data separately or perform a joint inversion of the apparent resistivity and apparent polarizability data (**Options / Inversion / Cross-gradient** option).

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*

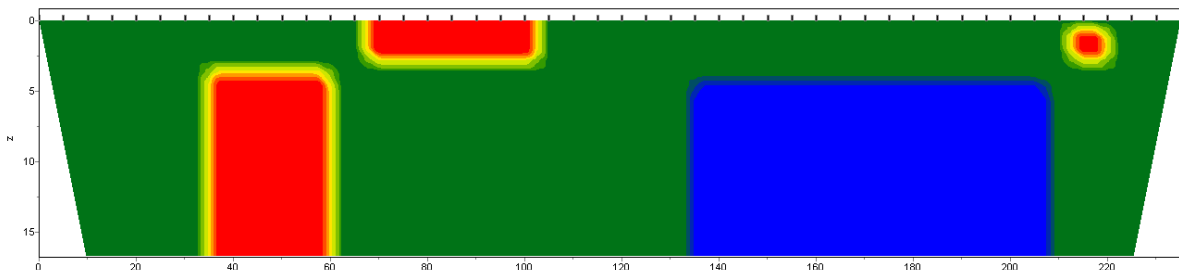
After opening a data file, processing the data and setting the initial model, the next step is to select the inversion type and set the inversion parameters. The dialog box for setting the parameters of forward and inverse modeling is opened using the  button on the toolbar or the **Option / Program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

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



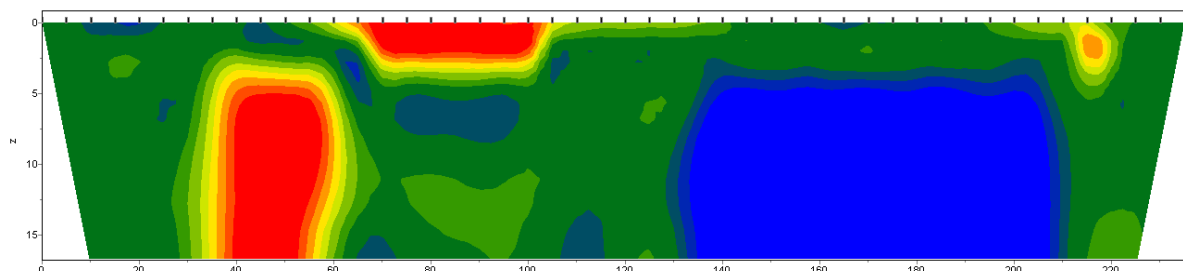
The Inversion option defines the inversion algorithm.

Let us consider various inversion algorithms on the example of a two-dimensional model slice 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).



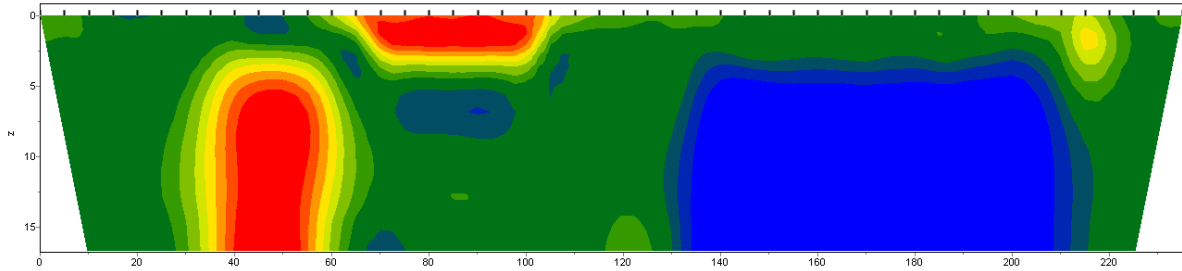


The matrix equation for this type of inversion is as follows:

$$(A^T W^T W A + \mu C^T C) \Delta m = A^T W^T \Delta f$$

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 [3]. This algorithm results in the smoothest parameter distribution (see figure below).

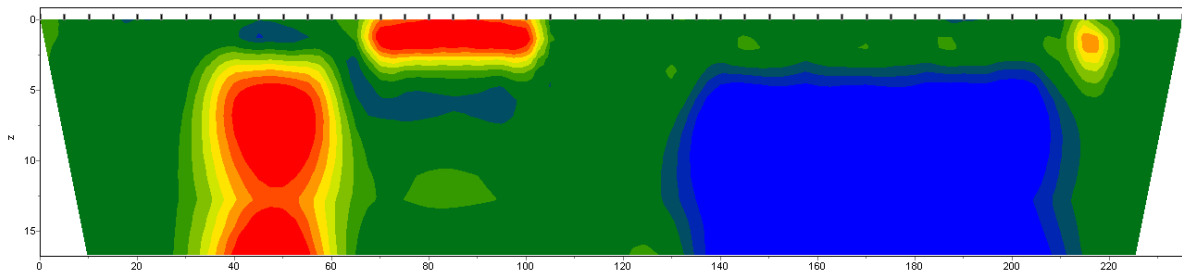


The matrix equation for this type of inversion is as follows:

$$(A^T W^T W A + \mu C^T C) \Delta m = A^T W^T \Delta f - \mu C^T C m$$

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) [4]. If the medium is characterized by a small number of parameters, this algorithm results in inversion models with sharper boundaries (see figure below).



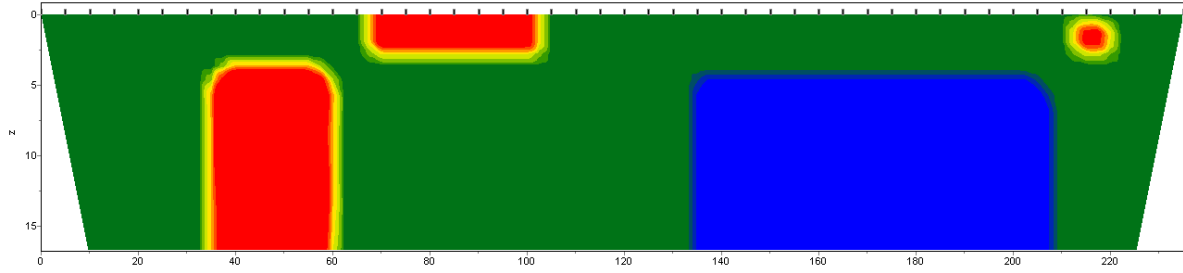
The matrix equation for this type of inversion is as follows:

$$(A^T W^T W A + \mu I) \Delta m = A^T W^T \Delta f$$

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).

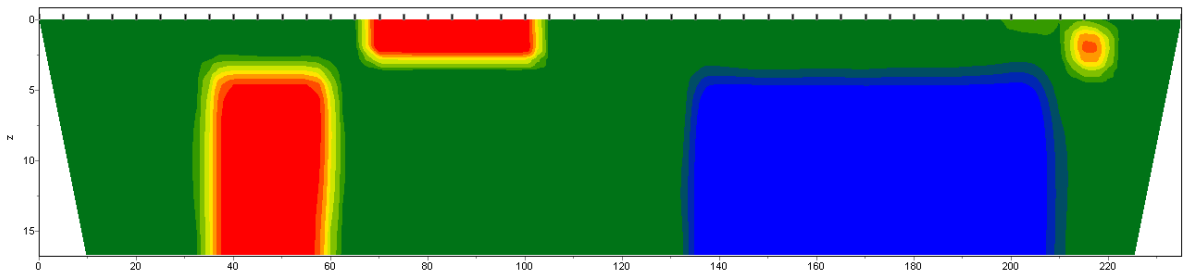


The matrix equation for this inversion type is the same as for the *Marquardt* algorithm:

$$(A^T W^T W A + \mu I) \Delta m = A^T W^T \Delta f$$

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 **Model / Smooth/raster** option accessible through the  toolbar button. 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 [5]. 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:

$$(A^T W^T W A + \mu C^T R C) \Delta m = A^T W^T \Delta f - \mu C^T R C m$$

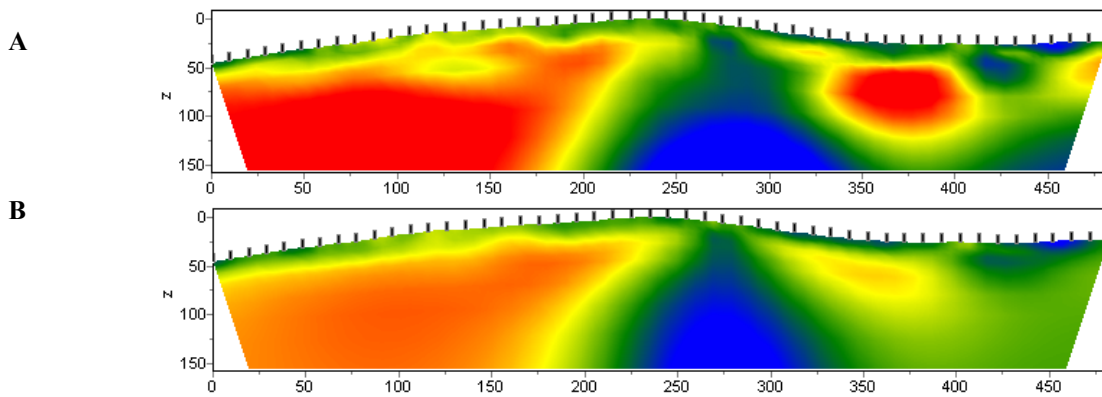
Using this type of inversion requires a careful selection of the **Threshold** parameter. This parameter defines the contrast threshold value of neighboring cells, after reaching which the parameters of the cells are not averaged (i.e., it is considered that there is a boundary between the cells). The 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.

**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 the model A

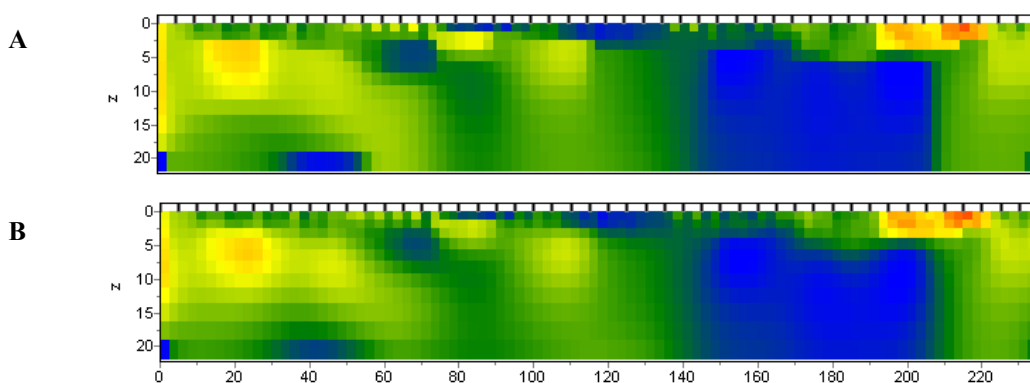
and 1.0 for the model B. The smoothing factor is used in the *Occam* and *Focused* inversion algorithms.



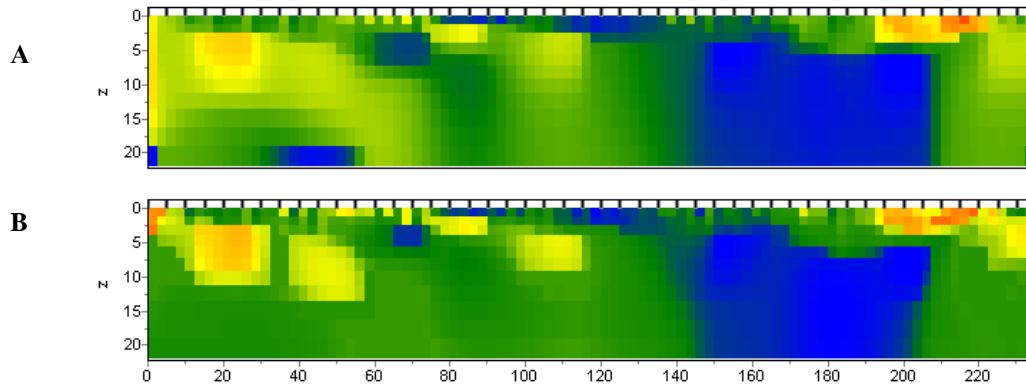
**Robust weighting scheme** – this option should be enabled if there are spikes in the data associated with systematic measurement errors. If the number of bad points in the data is comparable to the number of quality measurements, this algorithm may not produce good results.

In the **Stop criteria** group box, the maximum number of inversion iterations is specified.

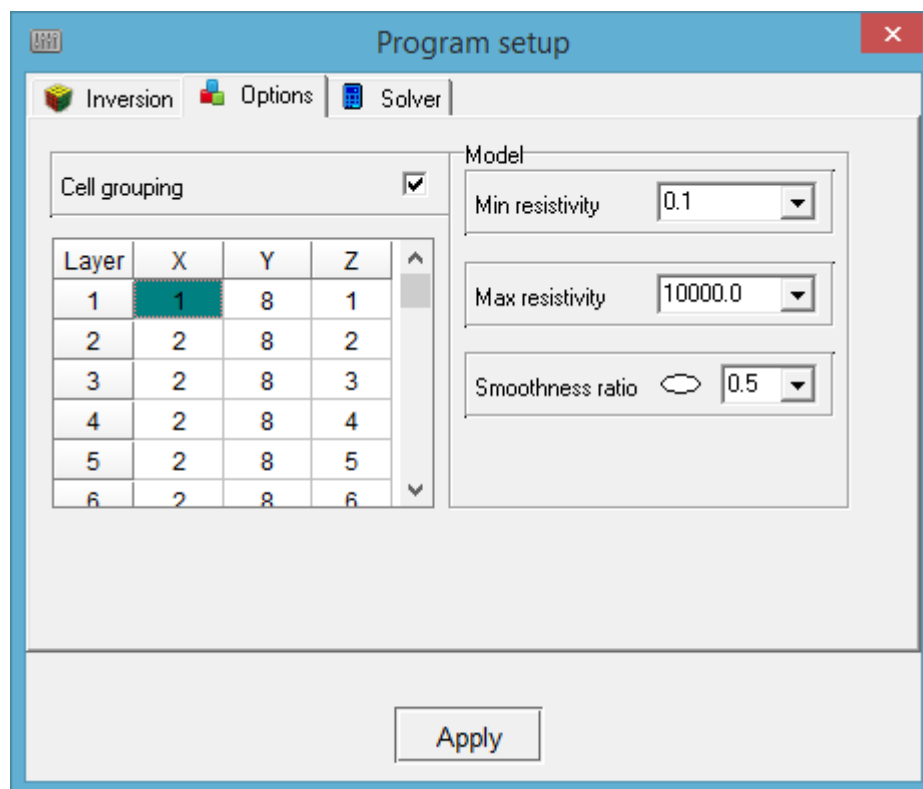
**Threshold** – sets the contrast threshold value of neighboring cells, after reaching which the parameters of the cells are not averaged (i.e., it is considered that there is a boundary between the cells). The value of this parameter is chosen empirically, typically in the 0.001 to 1 range. In the figure 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.



**Sharpness** – sets the ratio between the minimization of the volume of anomalous objects (0) and obtaining a piecewise smooth parameter distribution (1). The value of this parameter is chosen empirically in the 0 to 1 range.. In the figure below, model A was obtained with a sharpness value of 0.8, model B – with a sharpness value of 0.2.



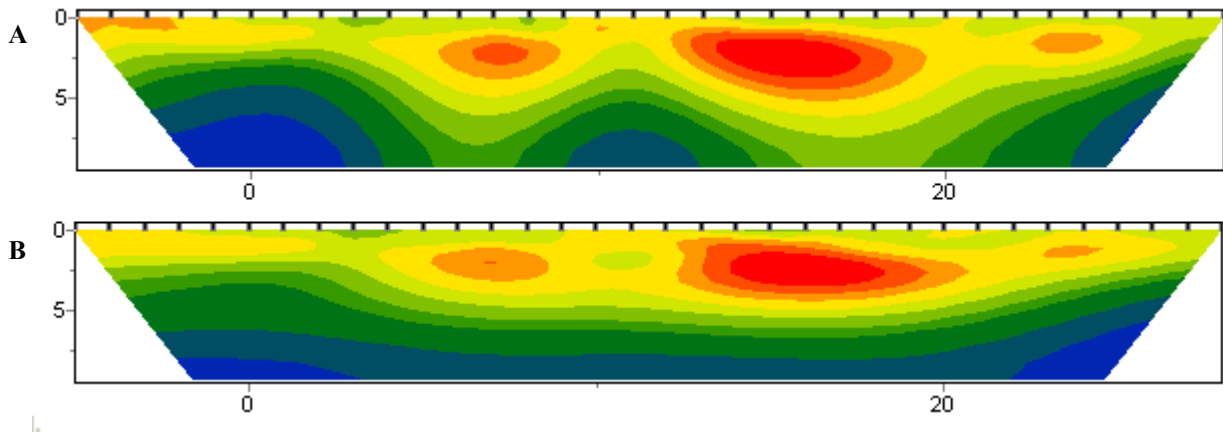
The **Options** tab contains a set of additional inversion parameters.



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

the **Min resistivity (polarizability)** and **Max resistivity (polarizability)** boxes set the parameter variation limits. If the set variation limits are known to be accurate enough, it is recommended to enable the **Options / Inversion / Optimization / Lim based inv** option.

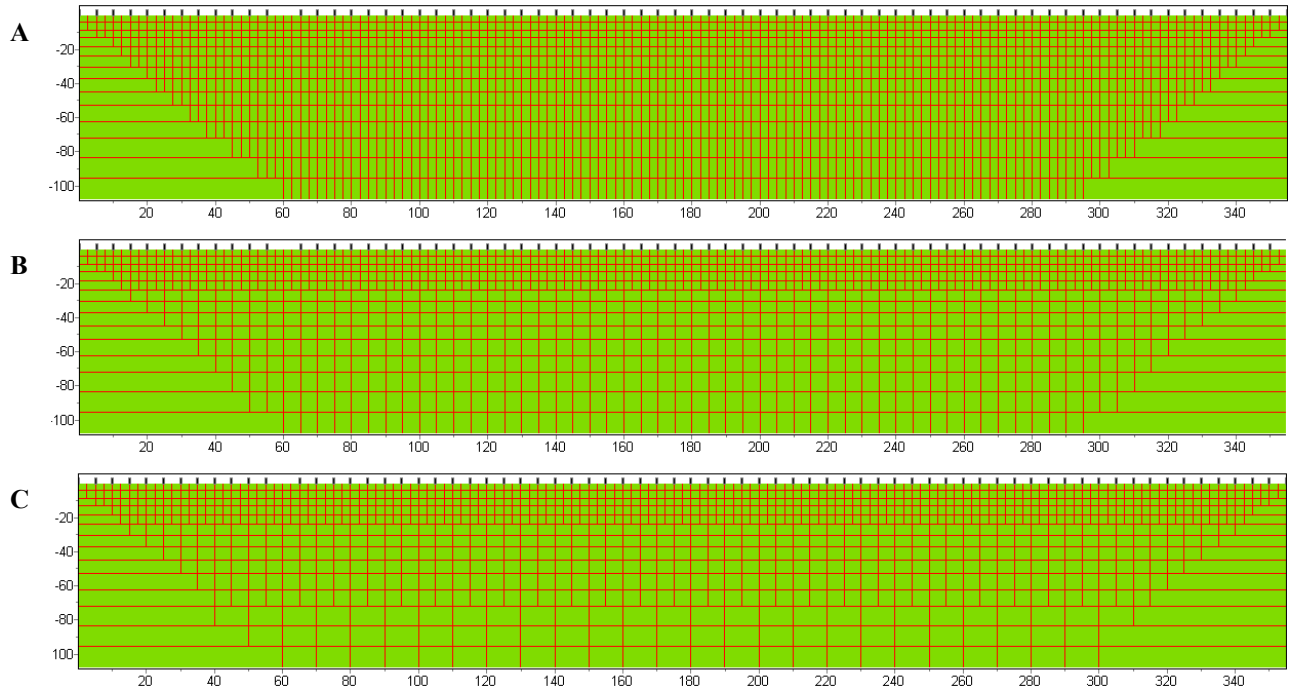
**Smoothness ratio** – defines the ratio between the degree of smoothness in horizontal and vertical directions. Values of less than 1 should be used for horizontally layered geology and more than 1 for vertically layered geology. Typically, the ratio values range from 0.2 to 1 (in the figure below,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



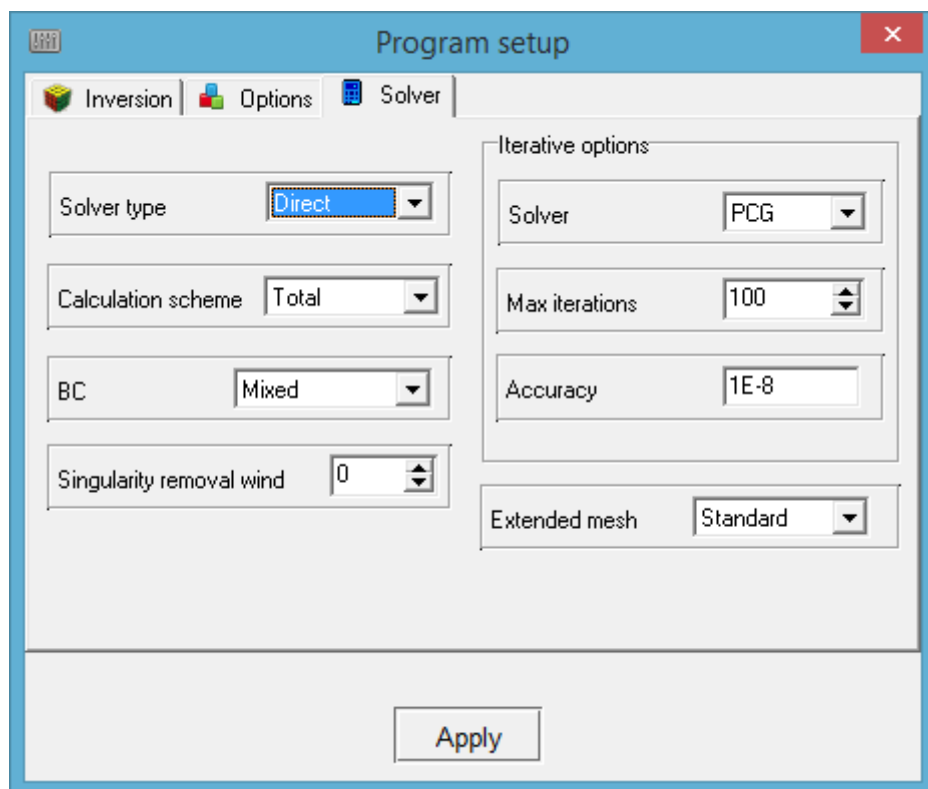
**Cell grouping** – this option is used in the case of large models. The option activates a table that allows you to combine adjacent cells (coarsen the grid), resulting in a smaller number of parameters defined during the inversion. While the number of cells used for inversion decreases, the number 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 four columns. The first column (**Layer**) lists the numbers of the initial model layers. The next three columns (**X**, **Y** and **Z**) specify the number of cells in the corresponding direction to be merged for inversion. The modified mesh is displayed in the *Model editor* and *3D model* sections during its configuration. Right-clicking on a cell in the **X** and **Y** columns brings up a dialog box where you can specify the number of horizontal model cells for the specific layer and all underlying layers to be merged throughout the model. Uncheck the **Cell grouping** checkbox to reset the table.

The figure below shows three examples of XZ slice 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 (X) direction; for **C**, starting from the twelfth layer, every four cells are merged in the horizontal (X) direction.



The **Solver** tab contains options for setting the parameters of forward modeling.



The **Solver type** box sets the method of solving the system of equations of the forward problem:

**Direct** – the direct method of solving the system of equations is applied; this method is recommended for most cases.

**Iterative** – the iterative method of solving the system of equations is applied; this method is recommended for very large models and measurement systems with a relatively small number of 62-

current electrode positions. When this option is selected, the **Iterative options** group box becomes available for configuring the iterative method parameters.

The **Calculation scheme** box defines the algorithm for calculating the electric potential:

**Total** – the total potential is calculated; this option is set by default. It is the fastest and most stable calculation scheme but it can result in large errors in determining the potential near current electrodes due to the singularity of the source; thus, a finer mesh near the electrodes might be required.

**Secondary** – the calculation of secondary potential is performed (relative to the half-space). This method significantly reduces the singularity of the sources but performs slower than the total potential method.

The **BC** box defines the type of boundary conditions of the forward problem:

**Mixed** – classic mixed boundary conditions are used.

**Infinite element** – "infinite" radial element at the edges of the modeling area is used. This variant allows increasing the rate of calculations by reducing the modeling area, however, in our opinion, it produces less accurate results.

The **Singularity removal window** box is an experimental original option for suppressing the singularity near sources in the *Total* calculation scheme. It allows reducing the error of calculations near sources by half for potential arrays. The size of the window is determined by the distance (in mesh nodes) from the source (a current electrode) to the nearest potential electrode. If there is one mesh node between the adjacent current and potential electrodes, it is recommended to choose the value of this parameter equal to three or more. In case of large errors in determining the potential near current electrodes, it is recommended to use the **Secondary** calculation scheme instead of using this option, because then it will not perform effectively and will require additional calculations.

The **Double** option in the **Extended mesh** box should be enabled if geological structures are known to be stretched along one direction. This option extends the external mesh which leads to more accurate results but the drawback is the lower rate of calculations.

The **Iterative options** group box contains options for configuring the iterative method parameters.

The **Solver** box defines the type of iterative method of solving the system of equations:

**PCG** – sets a simple preconditioned conjugate gradient method. It typically requires 60-100 iterations to achieve an acceptable level of error ( $10^{-8}$ - $10^{-9}$ ).

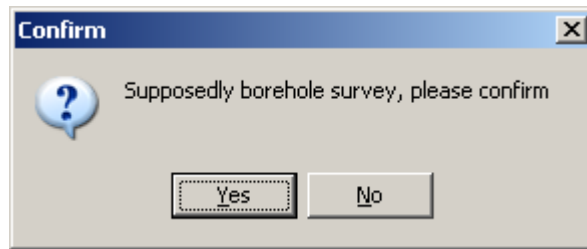
**biPCGstab** – sets a stabilized, preconditioned biconjugate gradient method, which is more stable and has better convergence but requires double the computational time per iteration (for a total of 40-80 iterations).

**Max iterations** – sets the maximum number of iterations. If the set level of accuracy (see below) is reached faster, the process is terminated.

**Accuracy** – sets the level of accuracy of the forward problem solution ( $10^{-8}$ - $10^{-10}$ ). The modeling process stops when the set value of accuracy is reached.

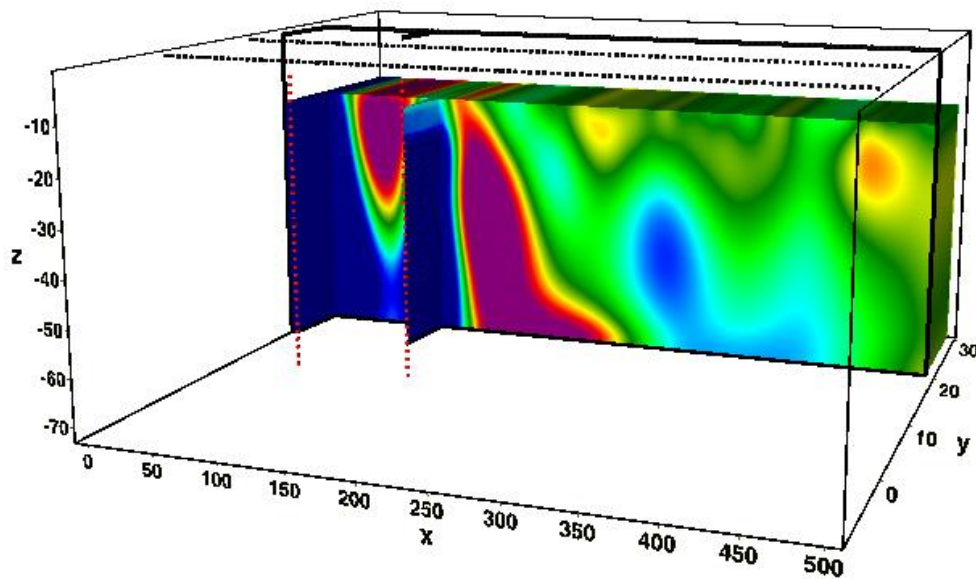
### *Inversion of cross-hole tomography data*

In a cross-hole survey data file, submerged electrodes should have negative Z coordinates (depths below ground surface). When the file is opened, the program will prompt you to confirm that it contains borehole data:






In the **Mesh constructor** dialog box that appears, the *Borehole* axis type should be set in the Z-axis group box. The program will automatically configure the model mesh in vertical direction based on the position of borehole electrodes.

An example 3D model for a project containing cross-hole tomography data is shown in the figure below. To display all electrodes, the **Show / Survey points** checkbox in the 3D model settings should be checked.





## *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.


In the polarizability operation mode, the **Options / Model** submenu contains the **MF factor display** option which allows displaying the metal factor in the model area after inversion of polarizability data.

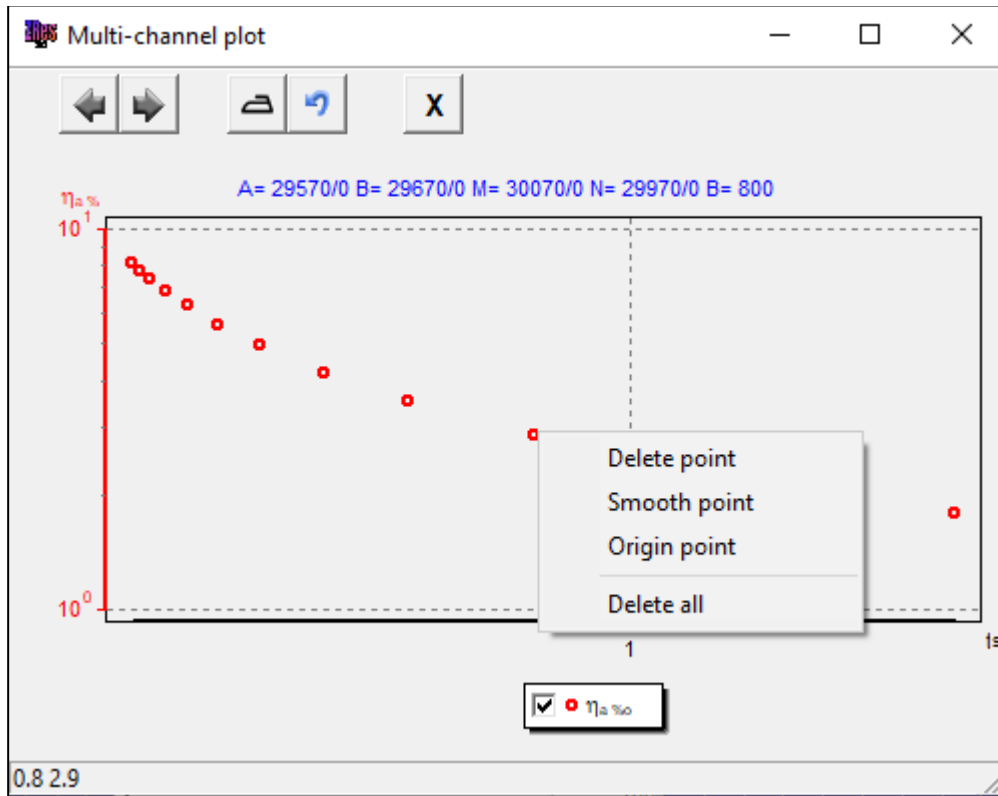
If the input data are multichannel IP (in the time or frequency domain) or time-lapse measurements, the **Time lapse** section appears in the main menu of the program. Switching between channels corresponding to different times (frequencies) is performed using the drop-down list on the main window toolbar.

The **Time lapse / Cole-cole inversion** command allows you to perform a Cole-Cole inversion. This menu item appears in the **Time lapse** menu section after inversion of all channels, for example, using the **Time-lapse inversion** procedure. The Cole-Cole model parameters can be displayed in the model section after the calculations are completed: in the **Options / Model / Cole-cole model** menu subsection, select the desired parameter to be displayed – **Chargeability**, **Time constant** or **C exponent**.

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 plan view map 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**). The  toolbar button smooths the current curve.

The  tool allows you to remove individual measurement points from the curve.



### *Inversion of time-lapse (monitoring) data*

When a data file containing time-lapse measurements is opened, the **Time lapse** 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** menu section contains the following commands:

Time lapse / Channels	Open the channel (measurement cycle) setup table. Arbitrary text information (title) entered in the <b>name</b> column will be displayed in the drop-down list on the toolbar. By unchecking a checkbox in the <b>inv</b> column, you can exclude a specific channel from inversion.
Time lapse / Channel plot	Open a window displaying time function plots for the measured parameter.
Time lapse / Time lapse inversion	Start the consecutive inversion of each measurement cycle (subset) of the time-lapse data set.
Time lapse / Next from previous	If enabled, each subsequent measurement cycle (subset) uses inversion results of the previous subset as an initial model. If disabled, the current model of the given subset is used as an initial model for each subsequent iteration.
Time lapse / Max iter numb for next	Set the maximum number of iterations for each subsequent measurement cycle.

Time lapse / Clear all models	Clear models for all channels.
-------------------------------	--------------------------------

The **Options / Model / Time lapse summary** command (available when the project contains models for several measurement cycles) opens a window displaying time-lapse survey summary sections, where you can estimate the parameter's variation over time.

### *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 on the plan view map or pseudo-section of relative misfits which are displayed when the **Options / Data / Data Misfit** option is enabled.

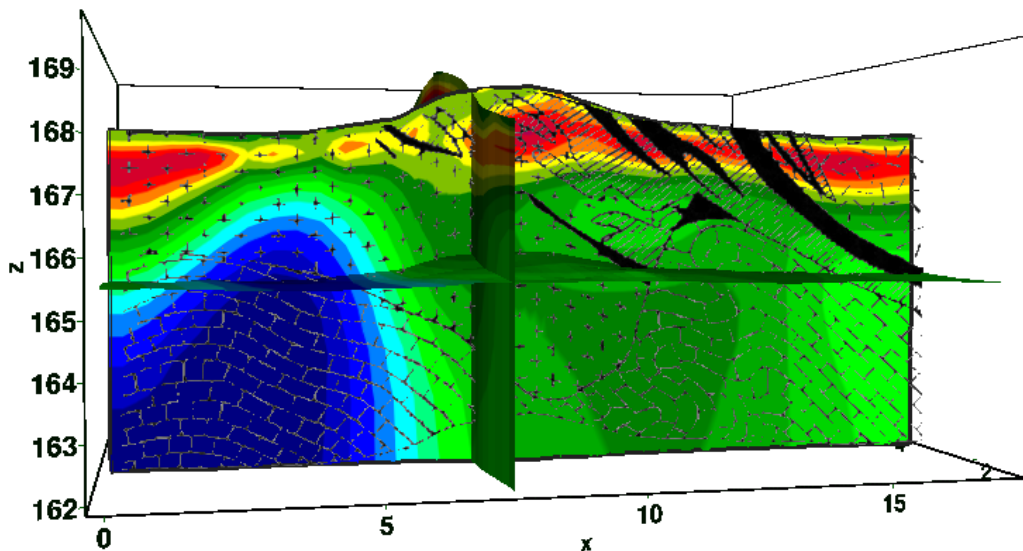
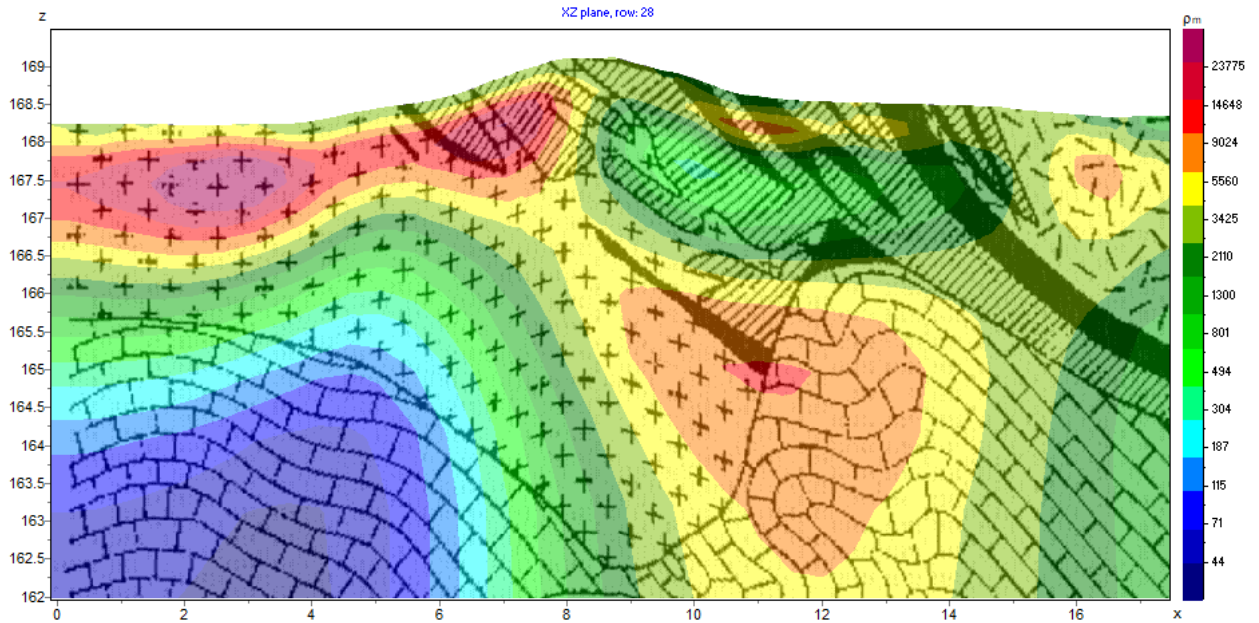
The estimation of misfits allows you to re-assess the data quality and remove data points with larger misfits. This can be performed using the **Options / Extra / Remove data with big misfit** command.

## **A priori information**

### *Importing background images*

The most effective ways to improve the quality of data interpretation are the combination of geophysical methods and use of a priori information. Several methods of visualizing a priori information are implemented in the program. Using options from the **Options / Import/Export** menu subsection, a variety of geological and geophysical information can be imported into the project in the form of models created in other Zond programs and graphic images that can be displayed in the model's background (for example, geological or seismic cross-sections).

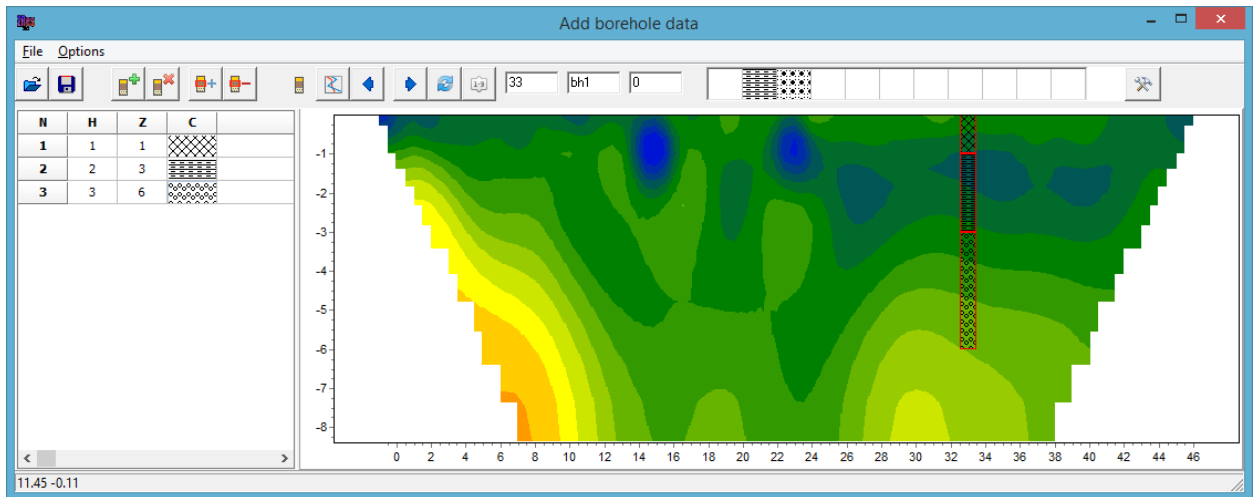
The figures below provide examples of a background image imported using the **Options / Import/Export / Background 2D** and **Background 3D** commands.



To compare the results of different geophysical methods, the **Options / Import/Export / Built-in mod2d's** and **Built-in mod3d** commands can be used. These options allow importing 2D and 3D models saved in other Zond programs in MOD2D and MOD3D file formats into the current model.

### *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 a survey line.





The module 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
<input type="text" value="30"/>	Set horizontal coordinate
<input type="text" value="bh1"/>	Borehole/log label (no more than 5 characters)
<input type="text" value="1"/>	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.

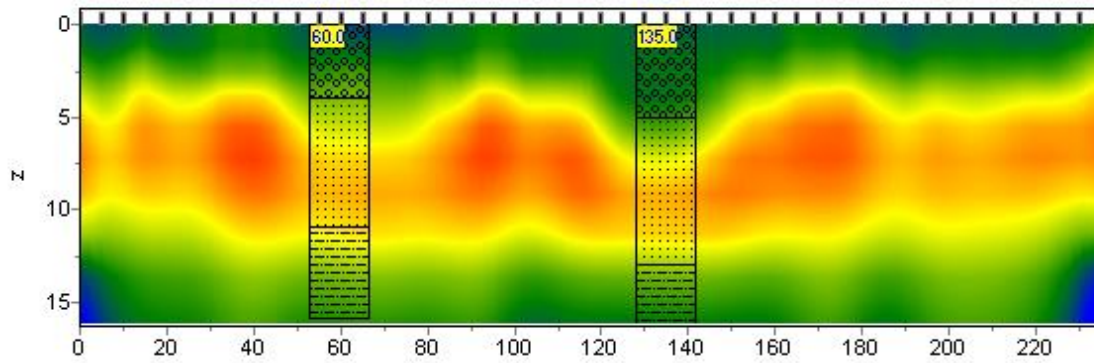
To start creating a borehole column, click the  button on the toolbar. A new table will appear in the left section. Use the  button to set the required number of layers, then specify the thickness or depth of each layer and choose the hatch pattern or fill color according to the borehole geology. Values 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 and the borehole will appear in the graphic section. The active borehole is displayed in red color. You can specify the borehole ID and horizontal coordinate using input boxes on the toolbar.

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.

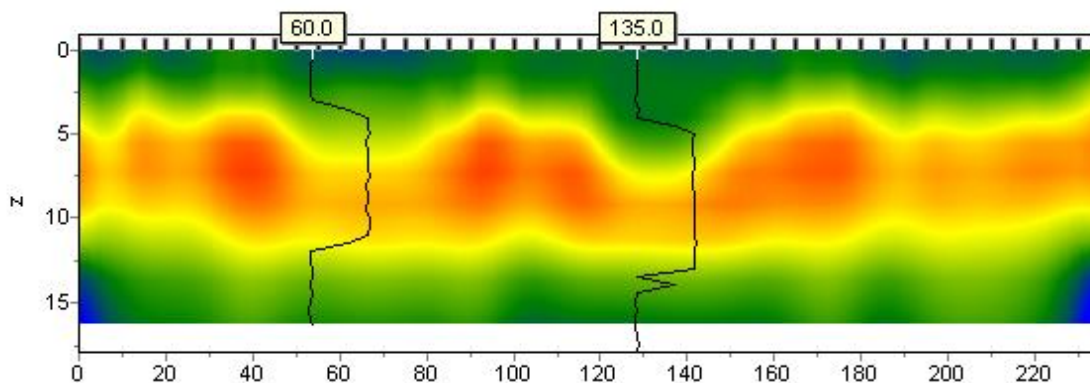
The **Set borehole width** command in the  context menu sets the borehole column width as a percentage of the survey line length.




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 **ZondST2D** 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.

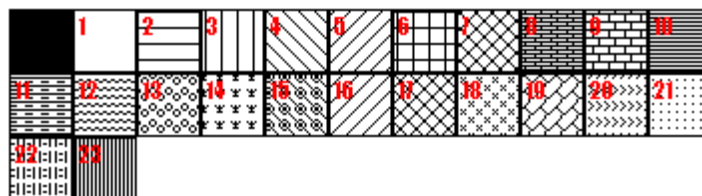
The following is a description of the CRT file structure for importing any number of borehole columns or logs into the module.

**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.

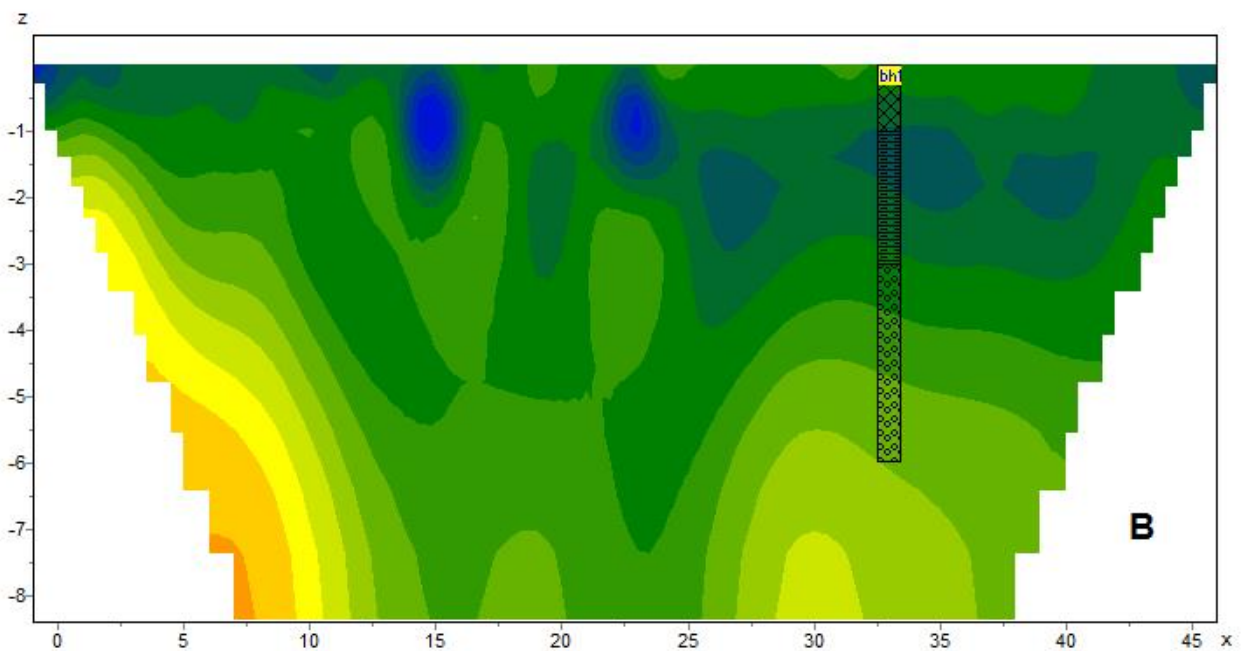
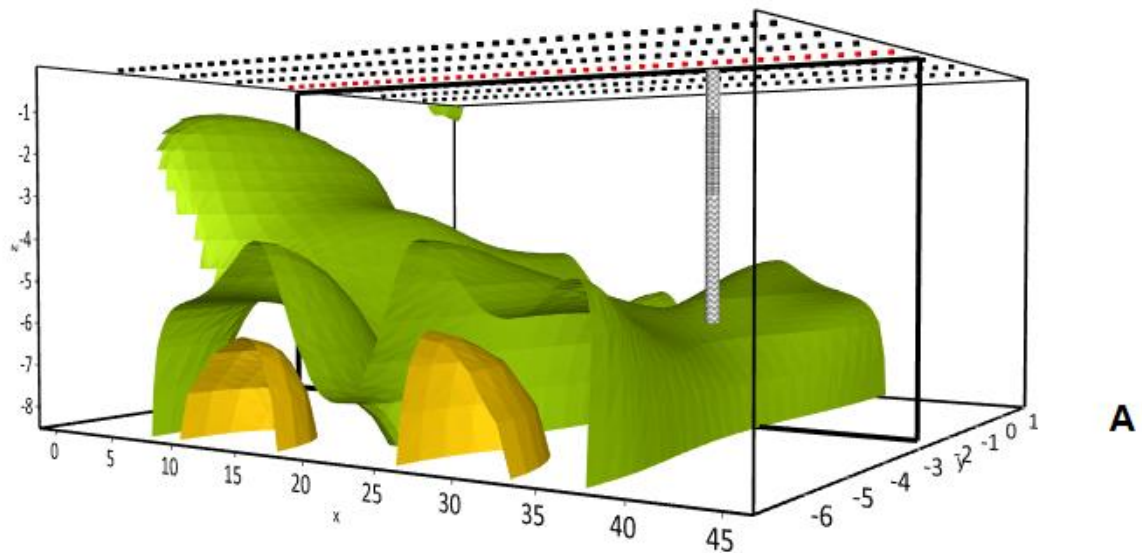
The text file structure for logging data has been described above. For borehole columns, the following structure is used: the first column contains depths of layers (from the ground surface), the second column contains zeros, the third column contains values indicating fill colors for layers, the fourth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



The figure below shows a borehole column displayed in a 3D model (A) and slice (B).

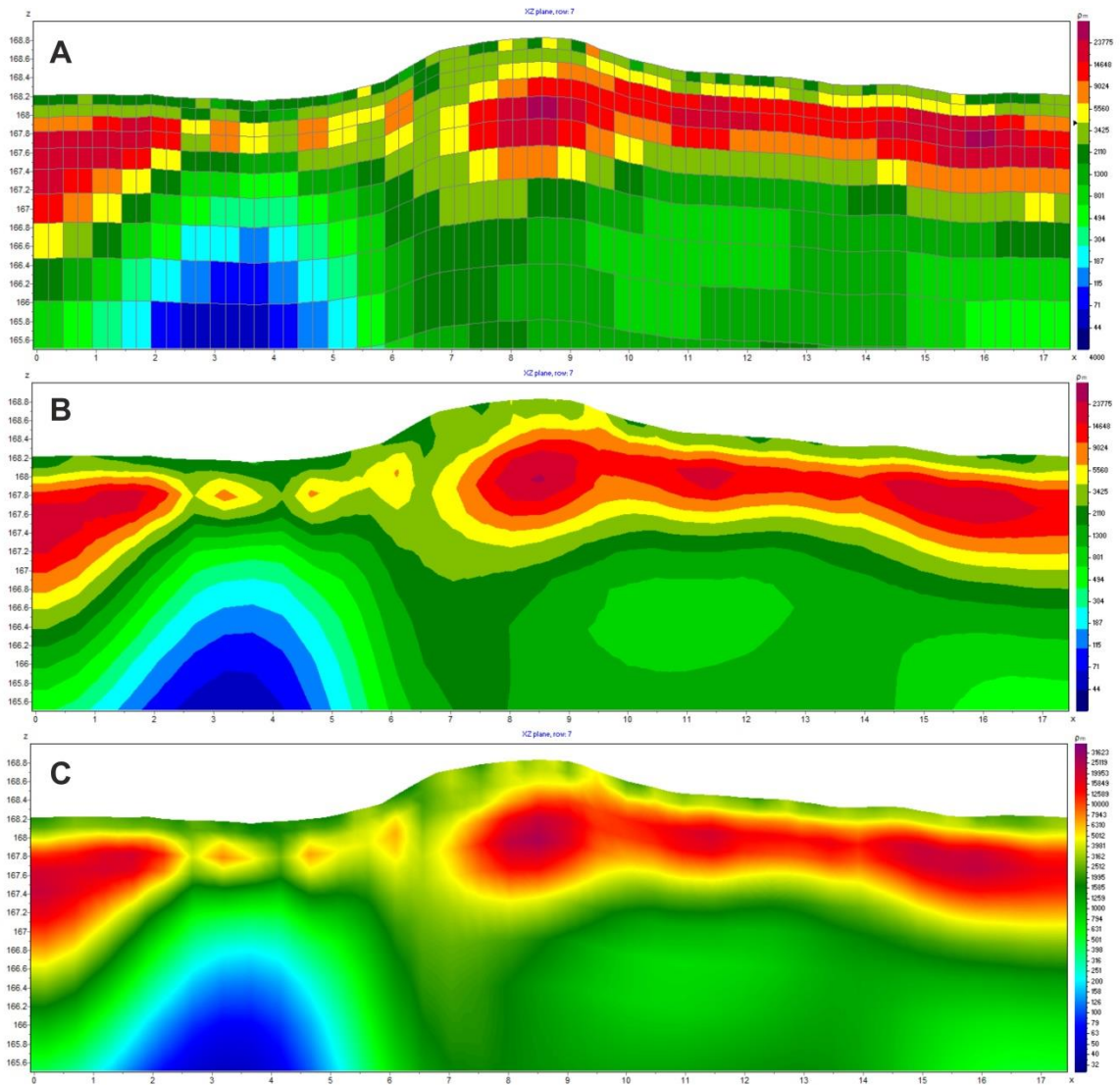




## Interpretation results

### *Visualization of model slices*

Model slices can be displayed in the form of blocks (cells), color contours or in the smooth interpolated palette mode (in the figure below, sections A, B and C, respectively). The **Options / Model / Block-section** and **Contour-section** commands are used to switch between the visualizing modes. To display the contour section in the smooth interpolated mode, check the **Continuous** checkbox in the **Color palette** dialog box (to access, right-click on the title of the model section and select **Setup**, then press the **Palette** button in the **Colors** tab).




### *3D model visualization*

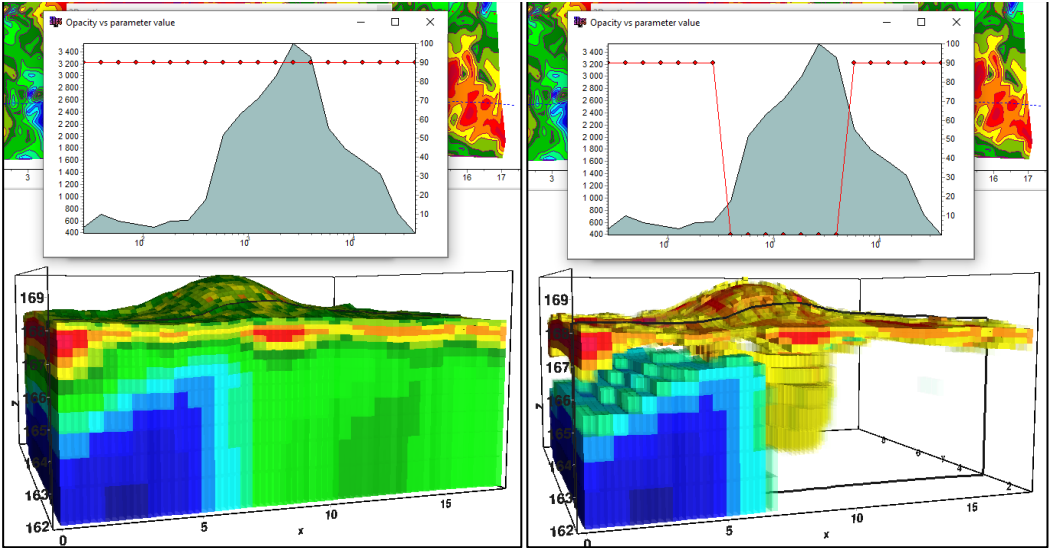
The 3D model interactively displays the survey results in a three-dimensional form. You can rotate the 3D plot by pressing and holding down the left mouse button; the mouse wheel is used to zoom in and out.

The **3D Options** dialog box containing visualization options of the 3D model is brought up by right-clicking in the 3D model area.

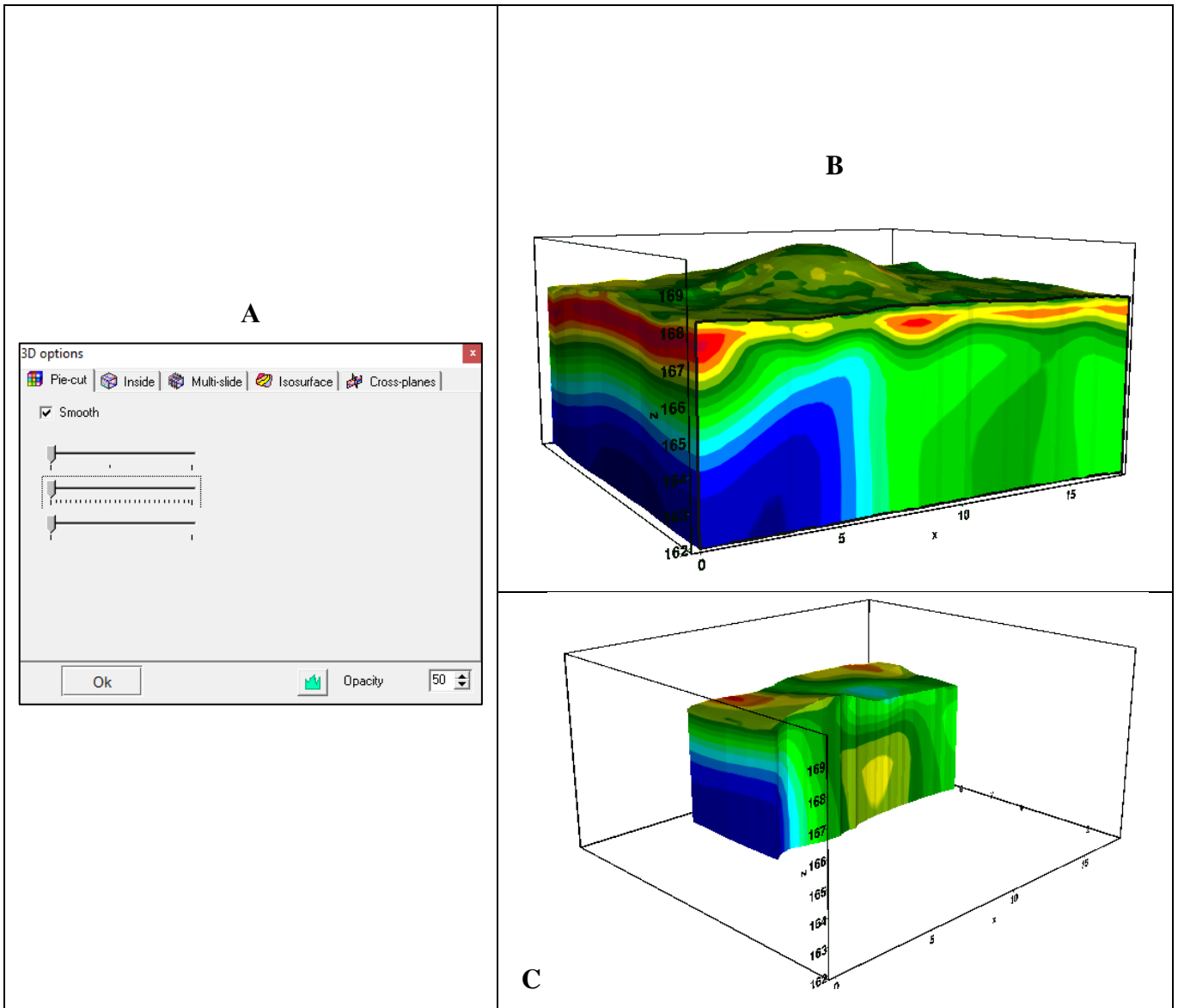
The dialog box consists of five tabs. By switching between the tabs, the model can be visualized in various forms: as a filled 3D volume (the **Pie-cut** tab), blocks (the **Inside** tab), a set of slices (the **Multi-slide** tab), isosurfaces (the **Isosurface** tab) and orthogonal planes (the **Cross-planes** tab).

The **Opacity** option in the bottom right corner allows you to set the overall transparency (in percentage) of the model. A more selective transparency can be configured in the **Opacity vs parameter value** dialog box opened using the  button. In the dialog box, the distribution of the model's parameter is displayed in the form of a graph with filled area. The red line shows the

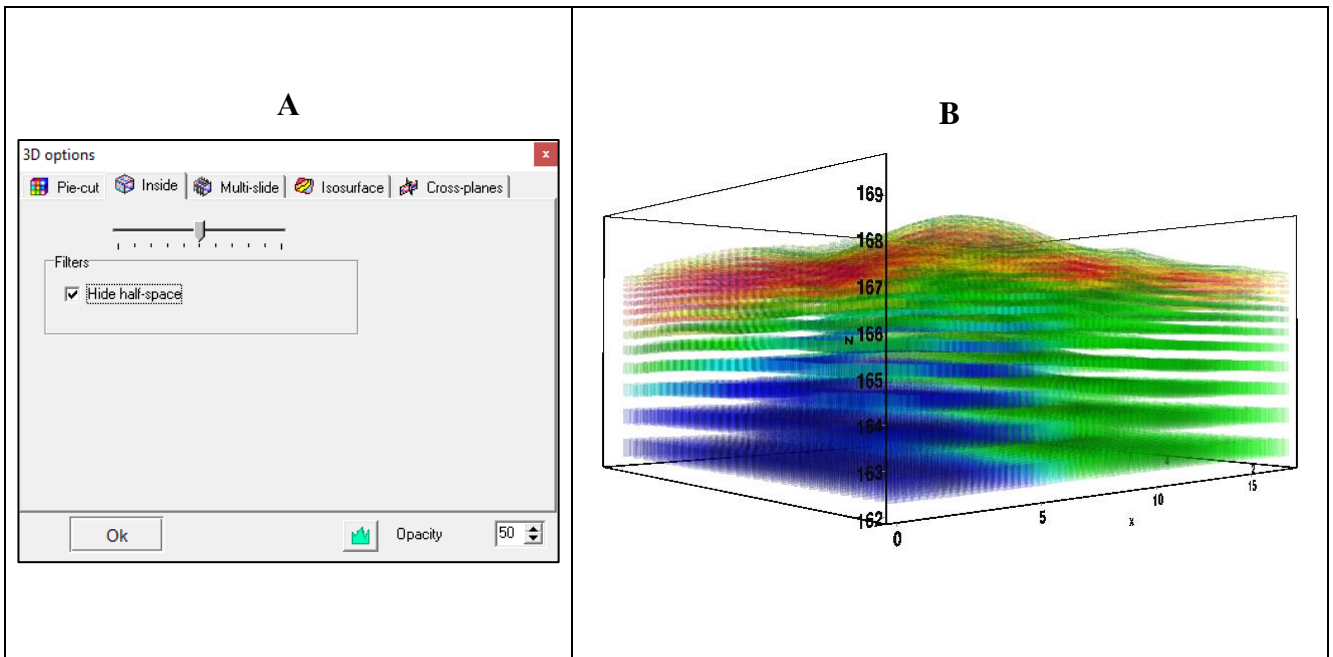
transparency profile. Transparency values corresponding to a specific range of parameter values can be manually adjusted by dragging the line points with the mouse. Typically, the maximum of the parameter distribution graph corresponds to the mean parameter value of the host medium (a background value). If it is necessary to show only anomalous objects, the model parts representing the host medium can be made transparent by reducing the transparency of the corresponding parameter range (see figure below).



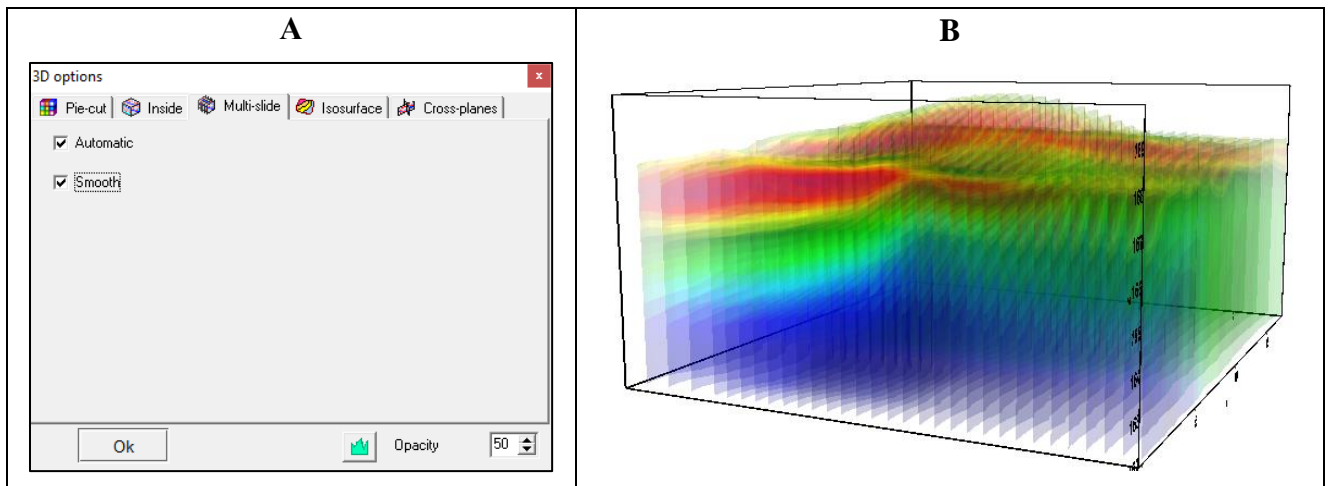
In the **Pie-cut** tab, the rendered model volume can be reduced in all three dimensions by adjusting the positions of the sliders (see figure below). The opacity of the model cannot be controlled in this visualization mode. The **Smooth** option enables the smooth color interpolation between the model cells.



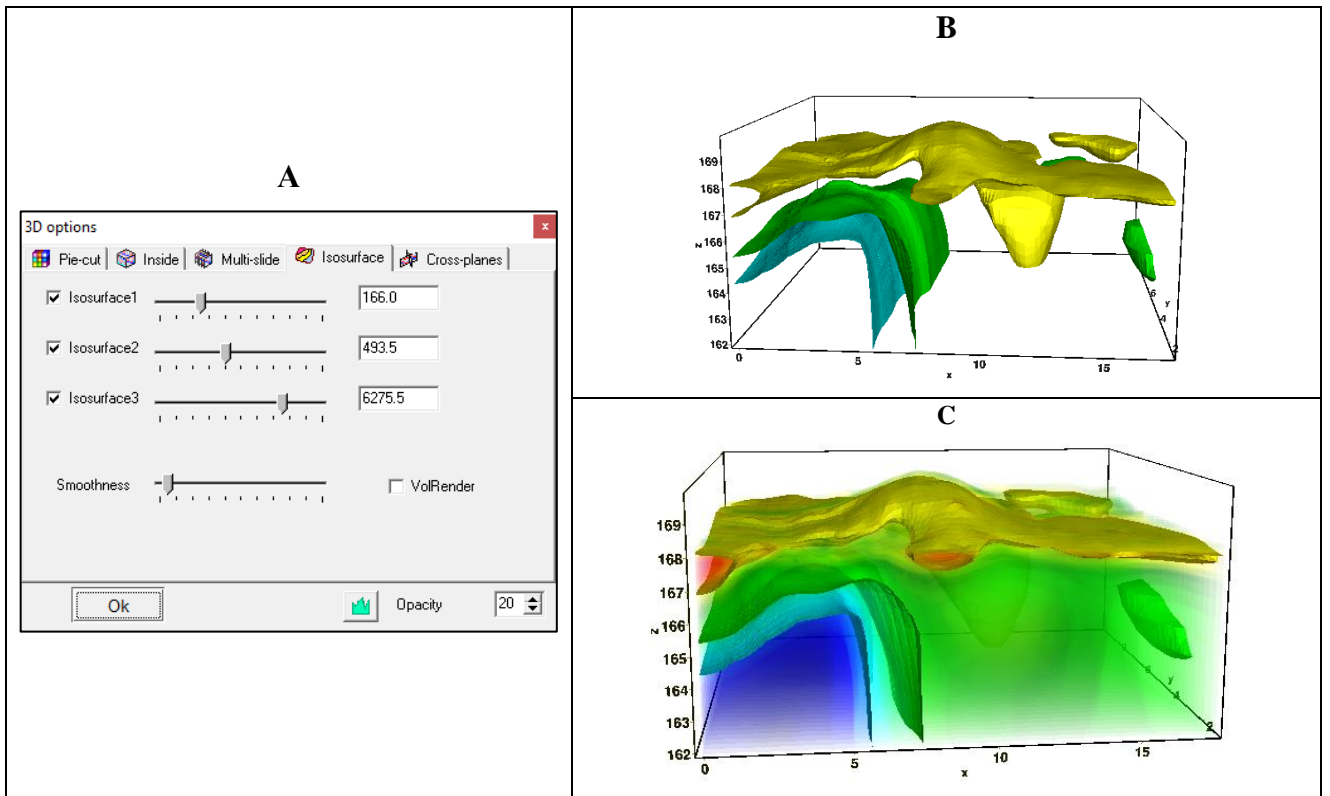
By switching to the **Inside** tab, the model is plotted in the form of blocks (see figure below). The transparency of the blocks is controlled by the *Opacity* value. The slider sets the relative cell size. If the **Hide half-space** checkbox is checked, the blocks with parameter values equal to the half-space value will not be plotted (this option is convenient for modeling).



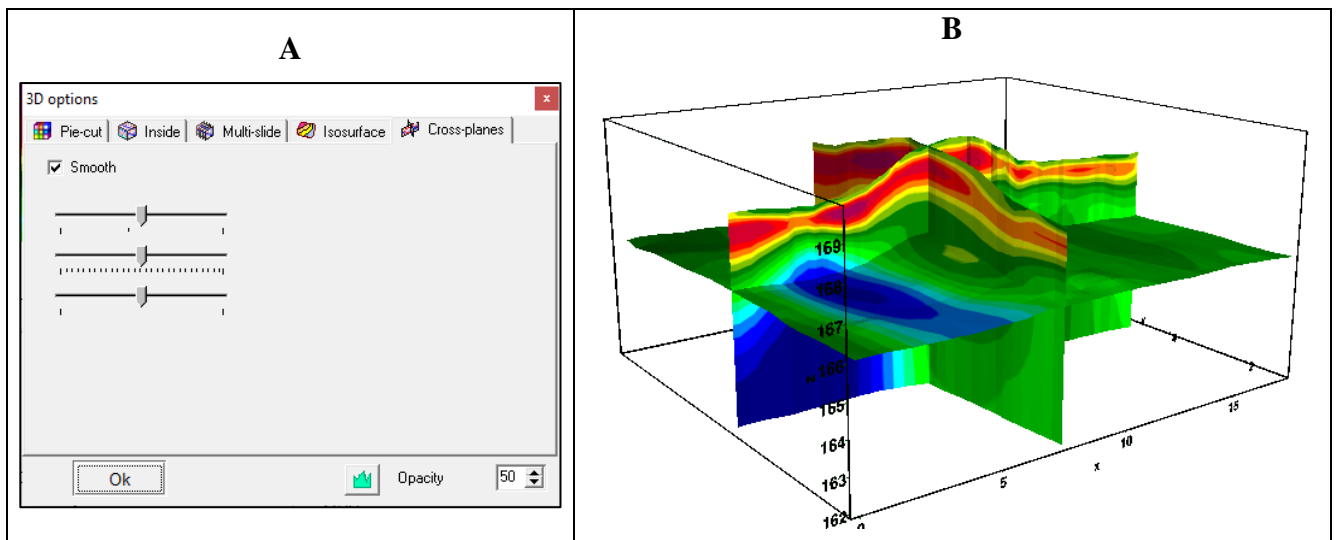
By switching to the **Multi-slide** tab, the model is displayed as a set of model slices along one of the directions (see figure below). The transparency of the slices is controlled by the *Opacity* value. The **Smooth** option enables the smooth color interpolation between the model cells. If the **Automatic** checkbox is checked, the direction of the slices is changed automatically depending on the view angle when the model is rotated.



By switching to the **Isosurface** tab, up to three isosurfaces of the model parameter can be displayed. The top three sliders (or input boxes to the right of the sliders) set the specific values of the model parameter. The **Smoothness** slider determines the degree of smoothness of the isosurfaces. The **VolRender** option adds a semi-transparent fog which color is determined by the distribution of model parameter (section C in the figure below). The *Opacity* option controls the transparency of the fog but not the transparency of the isosurfaces.



By switching to the **Cross-planes** tab, three orthogonal slices (in XZ, YZ and XY planes) are plotted (see figure below). The sliders define the positions of the planes. The opacity of the slices cannot be controlled in this visualization mode. The **Smooth** option enables the smooth color interpolation between the model cells.

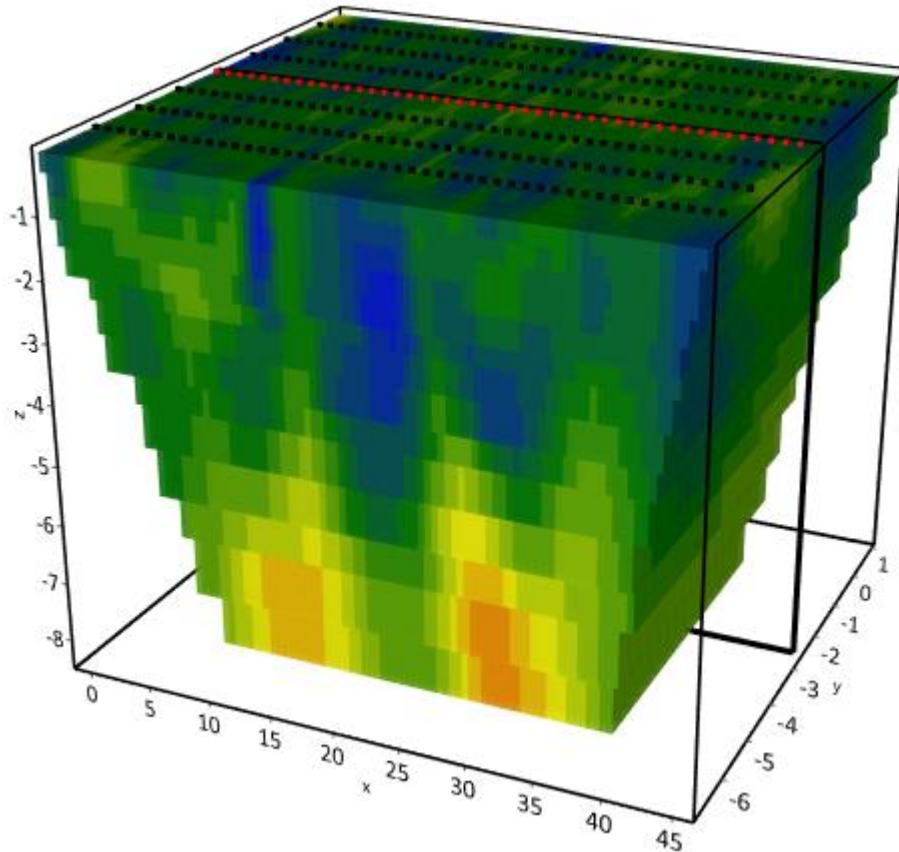


## *Trimming the model*

The edges of the model are typically characterized by low sensitivity and resolution. Very often these areas contain false anomalies. Options in the **Options / Cutting** submenu allow trimming the model edges by setting the values (in degrees) of the trimming angles.

The trimming angles are specified separately for X and Y directions using the **Cut angle X** and **Cut angle Y** commands. To trim the edges of the model, enable the **Blanking cut** option.

The trimming options can be applied to all types of 3D visualization modes except the **Pie-cut**.



## *Working with several models*

It is often necessary to store several models within one project and display them simultaneously for comparison, e.g., when comparing models obtained using different inversion parameters. In **ZondRes3D** 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 4** options correspond to the five buffer models that can be stored within one project. Several models can be displayed in the same color palette and scale.

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 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 the current model to this buffer slot (**To buffer**). If you choose **From Buffer**, the buffer model will replace the current model in the model editor.

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

When the **Only current** option is enabled, the program will save only the resistivity or polarizability model into the buffer, depending on the current operating mode.

## Saving the results

The project data are stored in a binary file of the internal **ZondRes3D** format with the Z3D extension. The file stores the field data, measurement weights, the current model and much more.

The project data can be saved by pressing the  button on the toolbar or selecting the **File / Save file** menu command.

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

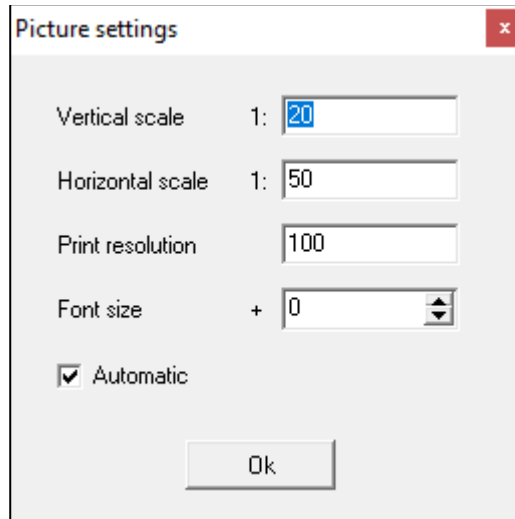
Project file [*.z3d]	Save the project to a project file.
Project with calculated data [*.z3d]	Save calculated data values, current model and all program settings to a project file. In this mode, the observed values are replaced by calculated values. This might be convenient for testing the inversion routine on different models.
Observed [*.z3d]	Save observed data to a text file.
Calculated data [*.z3d]	Save calculated data to a text file.
XYZV [*.dat]	Save the 3D model to a text file. This file can be used in a third-party software to visualize the 3D model.
XYZ [*. dat]	Save the current slice to a text file. This file can be used in a third-party software to visualize the 2D model slice.



# Graphic Settings

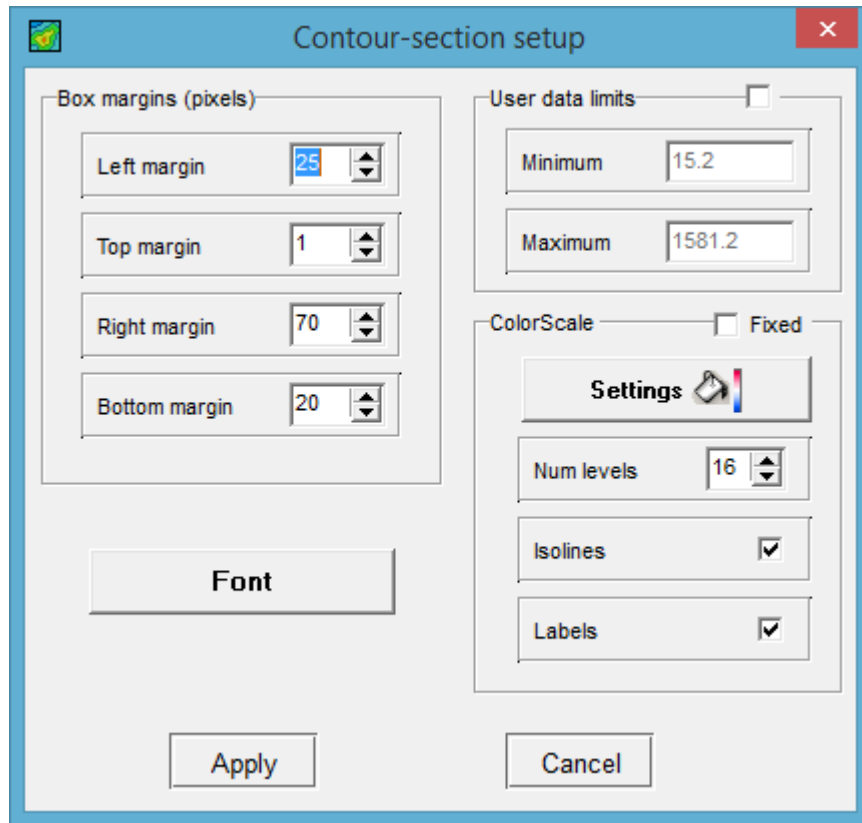
## *Export image settings*

The **Picture settings** dialog box is opened with the **Options / Import/Export / 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 map and pseudosection settings*

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



The dialog is used to configure the contour plot 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.

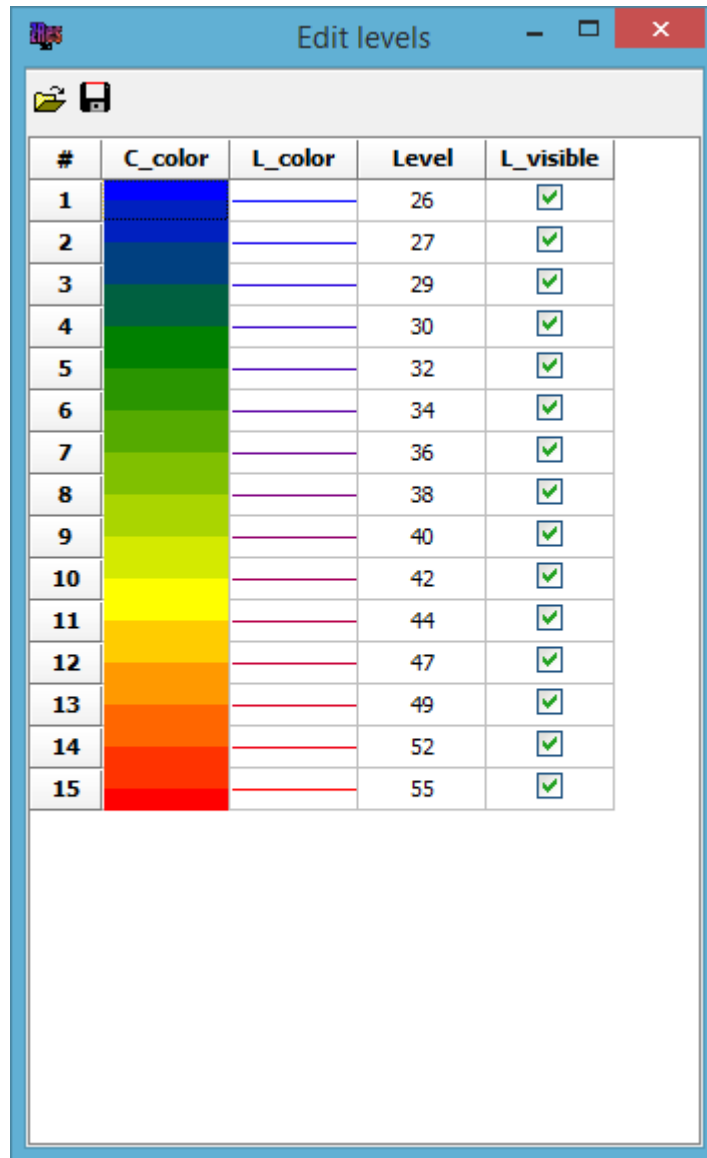
**Top margin** – sets the margin area on the top of the image.

**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.



In the **Edit levels** dialog box, 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.

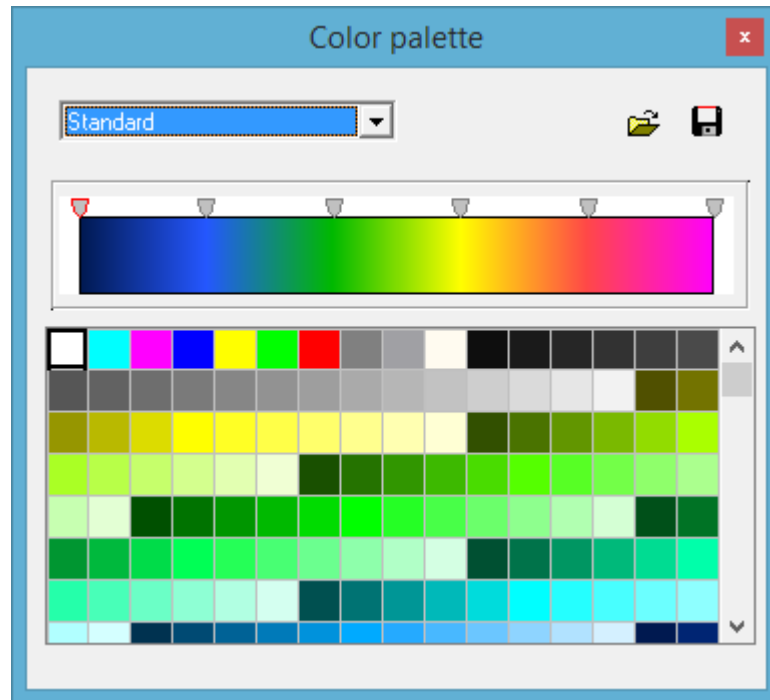
*L\_visible* – uncheck/check all boxes.

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.

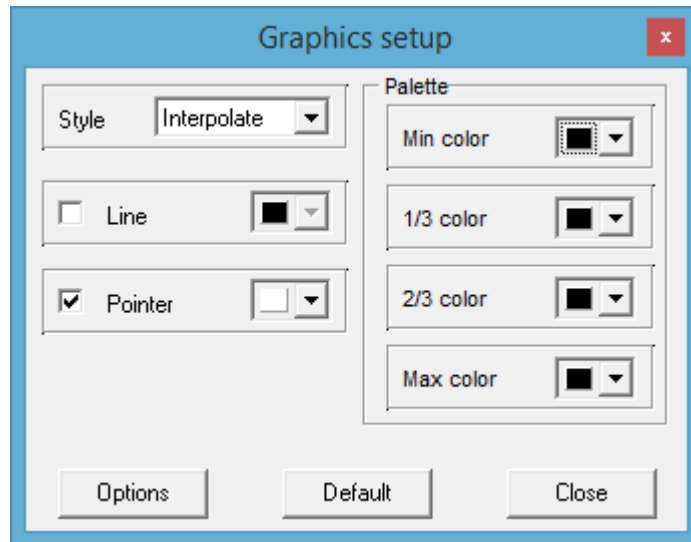
**Labels** – shows or hides isoline labels.

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, right-click on a graph point while holding down the Shift key. 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.

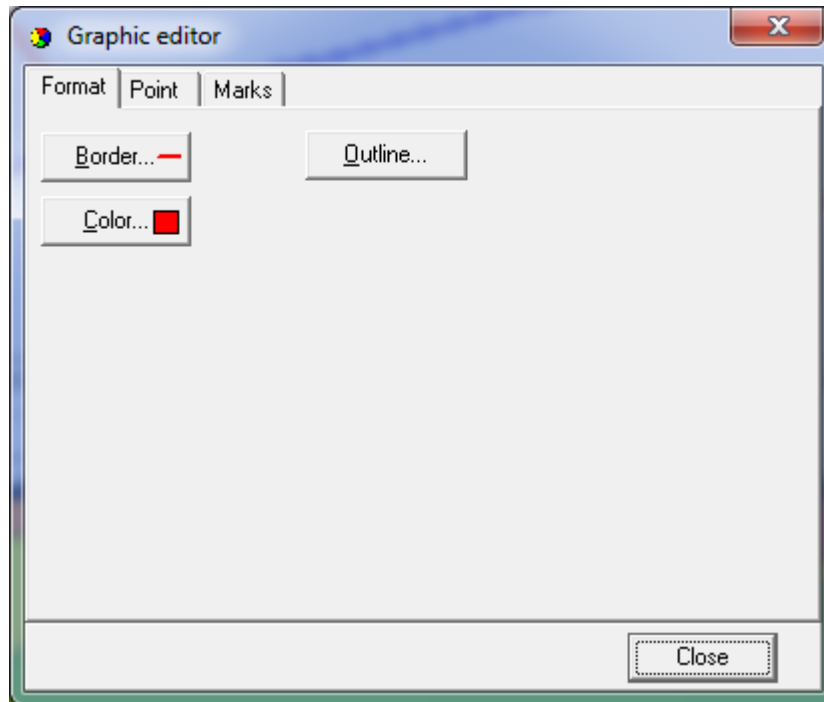
**Pointer** – specifies the graph point fill color. If the checkbox is checked, the points are filled with the color set in the **Pointer** box, otherwise, colors from the **Palette** group box are used.

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.

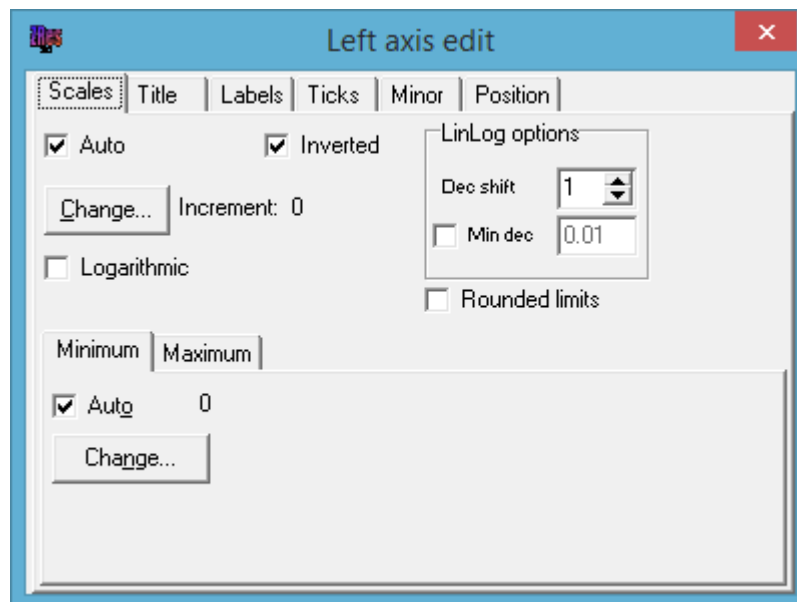
The **Text** sub tab contains the following settings:

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).

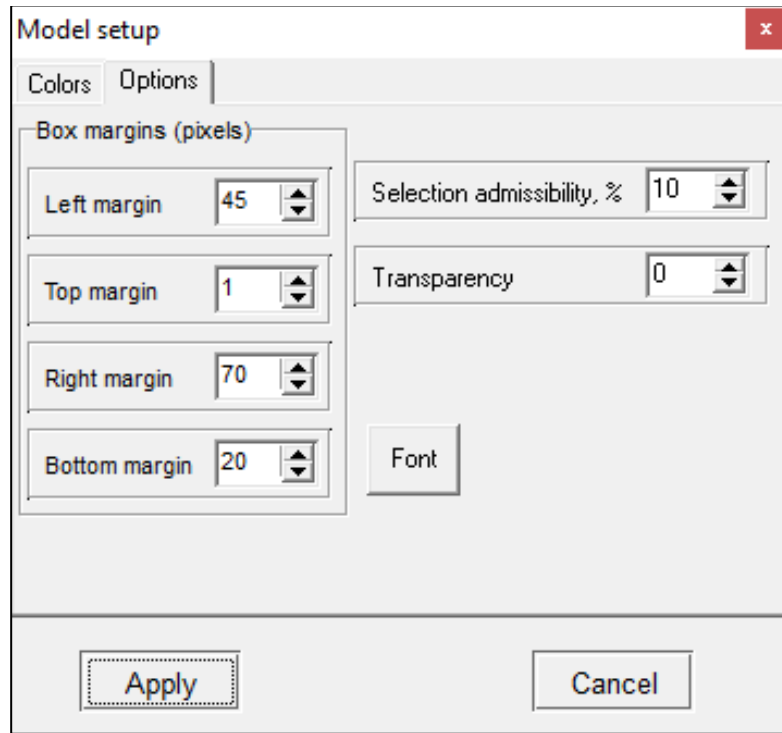
**Start %** – sets the offset of the start of the axis on the graph relative to the default position (as a percentage of the total graph size).

**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 editor display settings*

To open the **Model setup** dialog box, right-click in the area surrounding the model editor section 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.

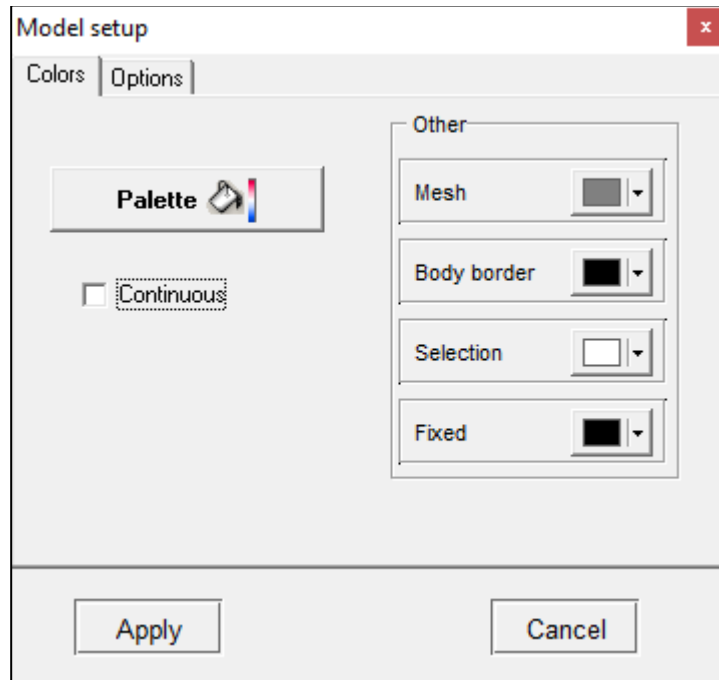
**Top margin** – sets the margin area on the top of the image.

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

**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.

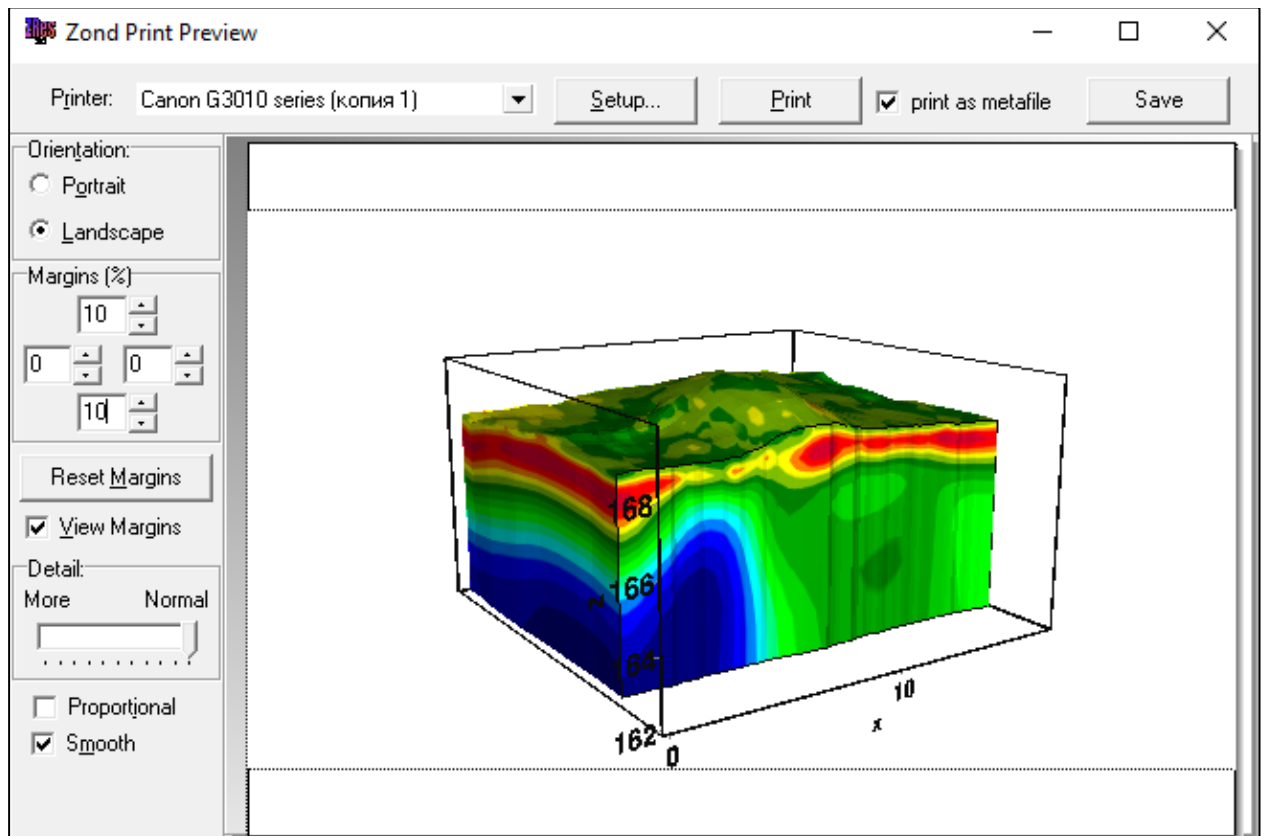
**Body border** – specifies the color of the boundary between adjacent cells.

**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 for printing the program workspace is opened with the **File / Print preview** menu command. To print individual program objects (e.g., a pseudosection), the **Print preview** command is selected from the context menu brought up by right-clicking in the area surrounding the object.



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

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

Printer: Canon G3010 series  – select a printer.

– open a standard print setup dialog box.

– print the drawing.

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

– save as a bitmap image.

## 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. 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.
4. Marquardt, D.W. [1963] An algorithm for least squares estimation of nonlinear parameters: *J. Soc. Indust. Appl. Math.*, 11, 431-441.
5. Portniaguine, O.N. and Zhdanov M.S. [1999] Focusing geophysical inversion images. *Geophysics*. V. 64. P. 874–887.
6. Khmelevskoy, V.K., *Geofizicheskie metody issledovaniya zemnoi kory* (Geophysical Methods 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](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:

1. 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](http://senselock.ru/files/senselock_windows_3.1.0.0.zip). The dongle should appear in the device manager as “Senselock Elite”.
2. 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.
3. 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**.