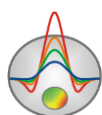


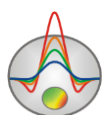
**Program for two-dimensional interpretation of data
obtained by resistivity and induced polarization methods
(land, borehole and marine variants)**

ZONDRES2D

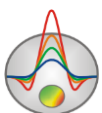
<i>Resistivity and polarizability of rocks</i>	<i>6</i>
<i>Program installation and deinstallation</i>	<i>8</i>
<i>Program registration</i>	<i>8</i>
<i>System requirements.....</i>	<i>8</i>
<i>Program start-up, main options</i>	<i>9</i>
<i>Creation and opening of data file.....</i>	<i>9</i>
<i>Main window toolbar</i>	<i>10</i>
<i>Main menu functions.....</i>	<i>11</i>
<i>“Hot” keys</i>	<i>21</i>
<i>Status bar</i>	<i>22</i>
<i>Starting model setup dialog</i>	<i>22</i>
<i>Main data file format.....</i>	<i>26</i>
<i>1st part of data file: Observed data.....</i>	<i>26</i>
<i>2nd part of data file: Topography data</i>	<i>28</i>
<i>3rd part of data file: Model data.....</i>	<i>30</i>
<i>Input and editing topographic information.....</i>	<i>31</i>
<i>Electrodes editor</i>	<i>33</i>



<i>Working with planned electrode coordinates</i>	34
<i>Apparent parameters visualization</i>	36
Graphics plot.....	36
Pseudosection	38
<i>Additional preparation of data for inversion</i>	39
Module of data quality control	42
<i>Data inversion</i>	45
Inversion parameters setup dialog.....	45
Incorporating a priori information in the inversion	56
Inversion in layered medium	58
Misfit estimation as a result of inversion.....	61
Additional options and data for inversion.....	62
<i>Model visualization modes and parameters</i>	63
<i>Modeling and working with model</i>	66
Survey layout definition	67
Model building in ZondRes2D.....	70
Model editor	71
Working with a block model.....	71
Cell summarization dialog	74
Polygonal modeling.....	76
Working with gravity and magnetic data measurements	79
<i>Working with multiple models in a single project</i>	79
<i>Saving interpretation results</i>	81
<i>Project information</i>	82
<i>Data import and export</i>	83
Additional information visualization	83



Text (tabular) data files – import and export	88
Gravity and magnetic data measurements import for interpretation	90
Data of self potential method import	93
Magnetotelluric data import.....	95
Exporting image setup dialog	96
Logging and lithology data file format	96
<i>Additional features of model visualization</i>	<i>101</i>
<i>Window of geological-geophysical model builder.....</i>	<i>102</i>
<i>Geoelectric models volumetric visualization using several profiles</i>	<i>104</i>
<i>Specifics of work with topography data and marine measurements</i>	<i>108</i>
<i>Specifics of work with borehole data measurements</i>	<i>109</i>
<i>IP parameters inversion</i>	<i>110</i>
<i>Geoelectric monitoring data inversion.....</i>	<i>115</i>
<i>Magnetotelluric data inversion</i>	<i>118</i>
<i>Script-files (batch processing).....</i>	<i>118</i>
<i>Appendix 1: Graphics set editor.....</i>	<i>120</i>
<i>Appendix 2: Graphics editor</i>	<i>121</i>
<i>Appendix 3: Legend editor</i>	<i>124</i>
<i>Appendix 4: Pseudosection (contour section) parameters setup dialog</i>	<i>125</i>
<i>Appendix 5: Axes editor</i>	<i>127</i>
<i>Appendix 6: Dialog of model visualization parameters</i>	<i>130</i>
<i>Appendix 7: Pseudosection points editor.....</i>	<i>132</i>
<i>References.....</i>	<i>133</i>



Program functionality

«ZONDRES2D» is computer program for 2.5D interpretation of electrotomography profile data obtained by resistivity method, induced polarization or excitation-at-the-mass method. Friendly interface and ample opportunities for data presentation allows solving assigned problem with maximum efficiency.

Finite-element method as mathematical apparatus is used to solve forward and inverse problem. It gives best results in comparison with mesh methods [Dey&Morrison, 1979; Lowry, 1989].

For point source field modeling medium is divided into triangle cells grid with different resistivity. Potential behavior inside grid cell is approximated by linear basis function.

$$N(x, z) = \frac{(a + bx + cz)}{2A} \quad (1)$$

Point source field has 3D structure in 2D medium. Using Fourier transform it is possible to transfer problem solving to spatial frequency domain.

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

$$\frac{\partial \phi}{\partial n} + \nu \cdot \phi = 0 \quad (3)$$

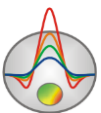
where ϕ – spectral potential value, λ – spatial frequency, I - current strength value, σ – medium electroconductivity, δ - Dirac delta function.

Following solution for spatial frequency set and usage of inverse Fourier transform for obtained spectral potential values gives desired values of point source potential in grid nodes [Xu, 2000].

$$U(x, y, z) = \frac{2}{\pi} \int_0^{\infty} \phi(x, \lambda, z) \cos(\lambda \cdot y) d\lambda \quad (4)$$

Least squares method with regularization is used for inverse problem solution (inversion). Regularization increases solution stability and allows receiving smoother resistivity and potential distribution [Constable, 1987].

$$(A^T W^T W A + \mu C^T R C) \Delta m = A^T W^T \Delta f - \mu C^T R C m \quad (5)$$



where A – the Jacobian matrix of partial derivatives, C – smoothing operator, W – relative error matrix, m – section parameters vector, μ - regularizing parameter, Δf – discrepancy vector between observed and calculated values, R – focusing operator.

During inverse problem solution development special attention was devoted to a priori information accounting (data weights, parameters turn-down).

«ZONDRES2D» has powerful system of profile data visualization, electrodes editor and system of sensitivity and method resolution analysis.

Two types of graphics are used to display observed and calculated data, their discrepancy or measurement weights in the program. They are graphics plan and pseudosection.

User can find array parameters, set data weights (relevance) and correct measured values in electrodes editor.

In resolution analysis system user can study model sensitivity function that is level of cell influence on measuring result.

$$S = \sqrt{\text{diag}(A^T A)} \quad (6)$$

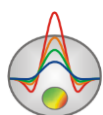
Research of sensitivity allows choosing optimal type and parameters of array in order to solve assigned geologic task.

«ZONDRES2D» uses simple and clear data file which allows combining different types of array on one profile. Apparent resistivity, signal to current strength ratio and apparent polarizability can be used as measured characteristics. Program allows importing and visualizing data using other methods which makes data interpretation process more integrated.

«ZONDRES2D» has modeling system that includes all main array types used in resistivity method. Dialog mode is used for array parameters selection and defining number of gage points.

«ZONDRES2D» is easy-to-use instrument for automatic and interactive electrotomography data interpretation and can be used on IBM-PC compatible PC with Windows system.

Educational materials about using the program are available online on the program website at the section <http://zond-geo.ru/examples/video-lessons/>, as well as on the YouTube channel <http://www.youtube.com/channel/UCGtprIIZkc9CsLfuz4VvmQ>.



Resistivity and polarizability of rocks

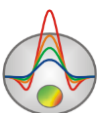
Electrical resistivity (ER) (units are the ohm*meter (Om^*m)) is a measure of how strongly rocks oppose the flow of electric current and is the most universal electromagnetic property. In rocks and ores it varies within wide limits: from 10^{-3} to 10^{15} Om^*m . For the most widespread sedimentary, volcanic, and metamorphic rocks ER depends on mineral composition, physical-mechanical and water properties, salt concentration in groundwater, in a less degree on their chemical composition, and on other factors (temperature, depth of occurrence, metamorphism degree, etc.) [Hmelevskoj, 1997].

Electrical resistivity of minerals depends on their crystal bonds. Dielectric minerals (quartz, micas, feldspars, etc.) which mostly have covalent forces are characterized by very high resistivity (10^{12} - 10^{15} Om^*m). Semiconductor minerals (carbonates, sulfates, haloids, etc.) which mostly have ionic bonds are characterized by high resistivity (10^4 - 10^8 Om^*m). Clay minerals (hydromicas, montmorillonite, kaolin, etc.) have ion-covalent bonds and are characterized by quite low resistivity.

Ore minerals (native and some oxides) have electronic conduction and carry current very well. First two groups of minerals create “rigid” matrix solid material. Clay minerals create “plastic” matrix solid material that is able to adsorb bound water whereas rocks with “rigid” minerals can adsorb only solutions and free water (water that can be extracted from rock).

Electrical resistivity of free groundwater changes from Om^*m unit fractions in case of high total salt content to 1000 Om^*m in case of low one. Chemical composition of dissolved salts does not really matter that is why electrical exploration allows assessing only total salt content. Electrical resistivity of bound water that is adsorbed by solid particles of rocks is very low and does not change greatly (from 1 to 100 Om^*m). Its constant mineralization (3-1 g/l) explains this fact. Average mineralization of ocean water is 36 g/l.

Pore water (bound and free) has very low electric resistivity in comparison to matrix of the majority rocks, that is why electric resistivity of rocks is almost independent of their mineral composition but depends on porosity, fracturing, and water saturation. Increase of their values



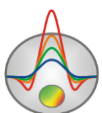
causes decrease of electric resistivity because ion content in groundwater grows. This is the reason why electroconductivity of the majority of rocks is ionic (electrolytic).

Rise of temperature in 40° causes resistivity decrease in half. It can be explained by ion mobility increase. Resistivity of rocks increases unevenly at freezing as free water becomes almost dielectric and electroconductivity is determined only by bound water that freezes at very low temperatures (below -50° C). Increase of resistivity varies in different rocks: several times in clays, up to 10 times in hard rocks, up to 100 times in clay and sandy loams, and up to 1000 times and more in sands and coarse rocks.

Despite the fact that resistivity depends on numerous factors and varies within wide limits in different rocks, main laws of ER are well determined. Volcanic and metamorphic rocks are characterized by high resistivity (from 500 to 10000 Om^*m). Among sedimentary rocks high resistivity (100 – 1000 Om^*m) can be found in rock salts, gypsums, limestones, sandstones, and some other rocks. As a rule, in detrital sedimentary rocks the more grain size is the higher resistivity rock has, that is ER depends on clayiness on the first place. In passing from clays to clay and sandy loams and sands resistivity changes from unit fractions and first Om^*m to tens and hundreds of Om^*m [Hmelevskoj, 1997].

Polarizability coefficient expresses ability of rocks to polarize that is to accumulate charge during passage of current and discharge then after current interruption. Coefficient η is measured in percents as ratio of voltage that remains in potential circuit some time after current interruption (usually 0.5-1 sec.) to voltage in this circuit during passage of current.

Polarization is complex electrochemical process that progresses in rocks during passage of direct or low-frequency alternating (up to 10 Hz) current. Ores with electronic conduction (sulphides, sulphosalts, some native metals, individual oxides, graphite, and anthracite) are characterized by the highest polarizability. These IP potentials' origin is connected with so called concentrated and electron polarization of ore minerals. Polarizability coefficient has up to 2-6 % value above water-encroached loose sedimentary rocks where clay particles are present. Their polarizability is caused by deformation of external plates of double electrical layers which form at solid-liquid contact. The majority of volcanic, metamorphic, and sedimentary rocks saturated with mineral water have low polarizability [Hmelevskoj, 1997].



Program installation and deinstallation

«ZONDRES2D» program is supplied on CD or by internet. Current manual is included in the delivery set. Latest updates of the program can be downloaded from website: www.zond-geo.ru/english

To install the program copy it from CD to necessary directory (for example, Zond). To install updates rewrite previous version of the program with the new one.

Secure key SenseLock driver must be installed before starting the program. To do that open SenseLock folder (the driver can be downloaded from CD or website) and run InstWiz3.exe file. After installation of the driver insert key. If everything is all right, a message announcing that the key is detected will appear in the lower system panel.

To uninstall the program delete work directory of the program.

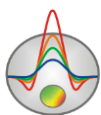
Program registration

For registration click “Registration file” item of the main menu of the program. When a dialog appears, fill in all the fields, select file name, and save it. Created file is transmitted to specified in the contract address. After that user receives unique password which depends on HDD serial number. Input this password in “Registration” field. The second option is to use the program with supplied SenseLock key inserted in USB-port while working.

System requirements

«ZONDRES2D» can be installed on PC with OS Windows 98 and higher. Recommended system parameters are processor P IV-2 GHz, memory 512 Mb, screen resolution 1024 X 768, color mode – True color (screen resolution change is not recommended while working with data).

As far as the program is actively using the registry, it is recommended to launch it as administrator (right click on program shortcut – run as administrator), when using systems higher than Windows XP.



Program start-up, main options

Creation and opening of data file

To start up «ZONDRES2D» it is necessary to create data file of certain format which contains electrodes coordinates, topography and measuring results. «ZONDRES2D» supports also most popular data formats: RES2DINV (Geotomo Software, M.H. Loke, Malaysia), SENSINV2D (Geotomographie, T. Fleschner, Germany), ABEM (Sweden), AGI (Advanced Geosciences, Inc., USA), Syscal (Iris Instruments, France), ProfileR (Lancaster Environment Centre, A. Binley) etc. It is also possible to import text data file of arbitrary tabular format.

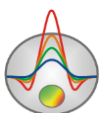
One profile data usually corresponds to one file. Text data files of «ZONDRES2D» format have «*.z2d» extension (see «[Main data file format](#)» for more details).

Zond data file [*.z2d]	Open Zond data or project file.
ProfileR data file [*.in]	Open ProfileR format data file.
ABEM/AGI/SYSCAL text file	Open ABEM/AGI/SYSCAL format data file.
Res2dInv [*.dat], ARES	Open Res2dInv format data file.
Sens2dInv [*.gem, *.imp, *.sen]	Open Sens2dInv format data file.
Zond1d file conversion [*.zlf]	Import data from Zond-IP 1D file.
Program configuration [*.cfg]	Open program parameters file.
Omega-48 data files	Open Prosys format data file (*.txt)
Skala-48/64 csv files	Open Skala-48/64 data file
SmartTEM-24 file [*.ip]	Open SmartTEM-24 data file
GDD instruments [*.gdd]	Open GDD instruments data file
MAE instruments [*.tsv]	Open MAE instruments data file

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













- incorrect symbols of records separator (TAB and SPACE use only);
- absurd data values (for example, negative values of apparent resistivity).

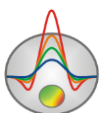
Desirably, there must be no more than 20000 observed data values and total unique electrodes position - no more than 500 for one file.



Main window toolbar

The toolbar serves for quick run of the most frequently used functions. It contains the following functional buttons (from left to right):

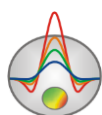
	Open data file (it is possible to load several files).
	Save data.
	Run inversion parameters setup dialog.
	Run electrodes editor.
	Run forward solution process.
	Turn ON mode of working with block model.
	Turn ON mode of working with layered model.
	Turn ON/OFF mode of polygonal modeling.
	Start (one click) or cancel (second click) inversion process.
	Run data interpretation mode of resistivity method (DC).
	Run data interpretation mode of induced polarization method (IP).
	Cancel previous step of model changing.



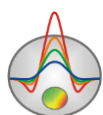
Main menu functions

The following table lists items found in the Menu with their corresponding functions:

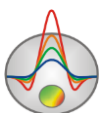
File /Open file	Open data file (it is possible to load several files).
File/Save file	Save data.
File/Edit data	Open current data file in Notepad editor.
File/Create synthetic survey	Run synthetic measurement system generation dialog.
File/Change electrode array	Run dialog of changing electrode array (usually used in modeling mode)
File/Print preview	Print main window of the program.
File/Recent	Recently used files.
File/Russian	Work with program in Russian.
File/English	Work with program in English.
File/Reg file	Create registration file to take temporary code (license). Administrator privileges are required.
File/Exit	Exit program.
Options /Quality control module	Run quality control window and edit raw data.
File/Project information	Show downloaded project information.
Options /Mesh constructor	Run starting model setup dialog.
Options/Data editor	Run electrodes editor.
Options/Program setup	Run inversion parameters setup dialog.
Options/3D sections plot	Run geoelectrical sections 3D visualization window.
Options/Geological editor	Run window of geoelectrical section geological interpretation.
Options/ Inversion /Set	Run boundaries setting dialog. The program takes account they in



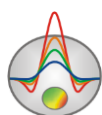
boundaries	inversion.
Options/Inversion/ Optimization	Optimization settings.
Options/Inversion/ Optimization/Display Process	Display inversion process.
Options/Inversion/ Optimization/Line search	Search Dampier optimal factor (partial optimization) for each iteration (it slows down the inversion process approximately 3-4 times).
Options/Inversion/ Optimization/Full	Complete optimization of the Dampier factor at each iteration (slow inversion process approximately 5-6 times).
Options/Inversion/ Optimization/Factor	Set reduction factor of the Dampier factor from iteration to iteration. This parameter is used when turning off optimization. It may lead to a divergence of the algorithm.
Options/Inversion/ Optimization/Length step	Optimization of increment length step to model parameters. It slows down inversion.
Options/Inversion/ Optimization/Automati c eta scaling	Automatic selection of the IP data normalization coefficient in the inversion.
Options/Inversion/Inve rt only visible graphs	Take into account only displayed data (in graphs mode) when inversion. Data that is not to be selected are disabled in the legend.
Options/Inversion/Not suppress	Not suppress P and C effects.
Options/Inversion/Supp ress P static shift. Max	Suppress P effect. Numeric parameter specifies maximum shift of curve segment in logarithmic form decades (suppression degree).
Options/Inversion/Supp ress C static shift. Max	Suppress C effect. Numeric parameter specifies maximum shift of curve segment in logarithmic form decades (suppression degree).
Options/Inversion/Supp ress border influence	Suppress border influence.
Options/Inversion/	Inversion in the "sliding window" mode. Can be used for long



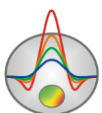
Long line inversion	profiles.
Options/Inversion/ Underwater options	Settings for working with underwater data.
Options/Inversion/ Underwater options/resistivity	Set water resistivity.
Options/Inversion/ Underwater options/invert	Turn on the option to select water resistivity.
Options/Inversion/ Underwater options/sublayers numbers 4	Set the number of partitions to the water layer.
Options/Inversion/ Average window 8:4	Set window to calculate model smoothness (when using Occam inversion)
Options/Inversion/ Diagonal flatness filter	Add diagonal smoothing.
L2 smoothness norm	Use L2 norm.
Data /Apparent resistivity	Display apparent resistivity values.
Data/Resistance	Display resistance values.
Data/Pseudo-section	Display observed and calculated data in pseudosection.
Data/Graphics-plot	Display observed and calculated data in graphics plan.
Data/Calculated data	Display calculated pseudosection in second part of window.
Data/Data misfit	Display misfit pseudosection in second part of window.
Data/Data weights	Display measurement weights pseudosection in second part of window.
Data/Iso-CC graphics	Run graphics plan feature, where each graphic corresponds to certain source position.
Data/Iso-psZ graphics	Run graphics plan feature, where each graphic corresponds to certain pseudo-depth (array geometric factor).



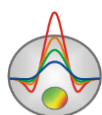
Options/Data/Iso-PP graphics	Run graphics plan feature, where each graphic corresponds to certain array line position.
Data/Display/Display every N point	Display every N point of pseudosection. This option is used in case of having more than 3000 observed values.
Data/ Display/	Display data for selected arrays (when measuring configuration has several arrays).
Options/Data/ Display/Error gates	Display measurements dispersion - weight between 0 and 1 (parameter is calculated by some instrumental systems and based on analysis). Weight can be changed by pressing the right or left mouse button while holding down the ALT key.
Options/Data/U/AB/M N norming	Specifies additional normalization for receiving and transmitting lines (when the option Data/Resistance is turned on).
Options/Data/G-Res plot	Display dependence resistivity from settings factor (used for borehole measurements)
Model /Block section	Display model as blocks.
Model/Smooth section	Display model in smooth interpolated graphic palette.
Model/Contour section	Display model as contour section.
Model/Resistivity	Display resistivity model.
Model/Sensitivity	Display sensitivity function model as contour section.
Options/Model/Model quality	Display model quality function (sensitivity normalized to model selection misfit).
Options/Model/DOI index	Display depth index as contour section (to calculate it is necessary to have two inversion cycle)
Options/Model/X=Z scale	Set vertical and horizontal scales equal.
Options/Model/ModelEditor toolbar	Display block model toolbar.
Options/ Topography /Topo coefficient	Set relief distortion coefficient with depth.
Options/Topography/Strike	Set an angle between profile and 2D structure strike.



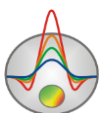
Options/Topography/ Import topography	Import profile topography data.
Options/Topography/ Remove topography	Remove topography data.
Options/Topography/ Restore topography	Restore topography data.
Options/Topography/ Edit topography	Edit profile topography data.
Options/Topography/ Smooth topography	Turn on relief points smooth interpolation mode.
Options/Topography/ Absolute coordinates	Display absolute altitude along profile, but not relative height.
Options/Topography/ X-factor	Increases (decreases) distance between electrodes in X times.
Options/Topography/ Reverse Line	Rotate profile for 180°.
Options/Topography/ Shift Line	Move profile for specified value (m).
Options/ Cutting /Cutting angle 30	Set cutting model angle as 30°.
Options/Cutting/No cutting	Do not cut model.
Options/Cutting/Cut by angle	Set an angle of model cutting in left and right borders.
Options/Cutting/Cut by sensitivity	Set an angle of model cutting by specified sensitivity degree.
Options/Cutting/Extend bottom	Extend bottom of modeling field.
Options/Cutting/ Pseudosection bounded	Limit model by horizontal pseudosection boundaries.
Options/ Borehole /Create	Add (edit) borehole data (lithological columns).



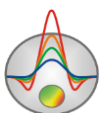
te/Edit borehole data	
Options/Borehole/Load borehole data	Open file with borehole data and lithological columns and model files (1D interpretation files).
Options/Borehole/Remove boreholes	Remove borehole data and lithological columns.
Options/Borehole/Set column width	Set lithological column width when displaying on section.
Options/ <i>Extra</i> /Model smooth/rough	Run dialog of section cells merge (roughing or smoothing section).
Options/Extra/Potential & sensitivity in model	Displaying settings of potential and sensitivity in model.
Options/Extra/Potential & sensitivity in model/ Potential isoline	Turn on mode of displaying potential isolines for each feed line position. Current measurement is chosen in electrodes editor table.
Options/Extra/Potential & sensitivity in model/ Sensitivity isolines	Turn on mode of displaying sensitivity isolines for each measurement. Current measurement is chosen in electrodes editor table.
Options/Extra/Potential & sensitivity in model/ Sensitivity contours	Turn on mode of displaying sensitivity contours for each measurement. Current measurement is chosen in electrodes editor table.
Options/Extra/Display Rho&Eta together	Display polarizability isolines over resistivity model or inversely.
Options/Extra/Extra isoline color settings	Run dialog of the second parameter isolines settings for previous option.
Options/Extra/Resistivity & IP summary plot	Run window with image of resistivity and polarizability model.
Options/Extra/Model & Data histograms	Show graphic of apparent and true model parameters distribution.
----- On load file ----- ----	To perform the following functions in the menu section Extra, it is necessary to activate them before data file opening.
Options/Extra/Open in	Open data file in the modeling mode.



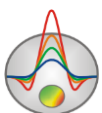
modeling mode	
Options/Extra//Include extended nodes	Adds additional nodes along the model edges.
Options/Extra/Convert AMN&MNB → AMNB	Convert data of direct and oncoming three-electrodes arrays to data of four-electrodes array.
Options/Extra/Convert AMN&MNB → AMNA	Convert data of direct and oncoming three-electrodes arrays to data of differential array (residual transformation, method of pure anomalies).
Options/Extra/Remote to infinite	Set remote electrode position in “infinity”.
Options/Extra/Edit remote position	Indicate (edit) “infinity” electrode position.
Options/Extra/Display electrode RMS	Display relative misfit value for each electrode.
Options/Extra/Remove data with big misfit	Remove data with misfit higher than set one.
Options/Extra/EM coupling removal	Suppress induction influence.
Options/ Import/Export / Import model\data	Import model\data from other Zond program.
Options/Import/Export/ Remove model\data	Remove imported model\data from project.
Options/Import/Export/ Save selection	Save selected in editor cells parameters.
Options/Import/Export/ Load selection	Open file with selected cells and paste from the current cursor position.
Options/Import/Export/ Extract 1d log	Save vertical resistivity (or polarizability) profile for specified horizontal coordinate.
Options/Import/Export/	Load to model vertical resistivity (or polarizability) profile and their




Load 1d log	limits at specified horizontal coordinate.
Options/Import/Export/ Background image	Import background image – graphic bmp file or SectionCor (*.sec) file.
Options/Import/Export/ Remove background	Remove background.
Options/Import/Export/ Load 1D model (ZondIP1D)	Load 1d model from file ZondIP1D format.
Options/Import/Export/ Import RAW data/COMx64, ERA- MULTIMAX	Import measurement data format as complex Mery/COMx64 and ERA-MULTIMAX
Options/Import/Export/ Import data from text/Excel	Import data from text or Excel file.
Options/Import/Export/ Export to Excel/Model	Export model cells parameters to Excel.
Options/Import/Export/ Export to Excel/Data levels	Export diversity raw data to Excel.
Options/Import/Export/ Export model to CAD (DXF)	Export model (contour section in vector presentation) DXF (Autocad) file.
Options/Import/Export/ Convert to VES (ZondIP1D)	Convert model to file ZondIP1D program for 1d interpretation.
Options/Import/Export/ Direct drawing in Surfer	Draw model (contour section) in Surfer program.
Options/ Graphics / Observed graphics	Run dialog of observed data graphics parameters settings.

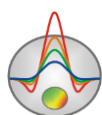


Options/Graphics/ Calculated graphics	Run dialog of calculated data graphics parameters settings.
Options/Graphics/ Smooth contours	Smooth model contours in corresponding displaying mode (Contour section).
Options/Graphics/ Copy pseudo colors to model	Use pseudosection colors when displaying model.
Options/Graphics/ Bitmap output settings	Run dialog of graphical image parameters when exporting settings.
Options/ MT data /Load MT data	Load MT data.
Options/MT data/Invert MT data	Mt data inversion.
Options/MT data/Show MT plot	Show MT data plot.
Options/MT data/With static shift	Take account static shift.
Options/MT data/Set weight of MT	Set weight of MT data.
Options/MT data/DET inversion	Determiner inversion.
Script /Start recording script	Start recording script.
Script/Stop recording script	Stop recording script
Script/Run packet processing	Run packet processing files.
Script/Save script file	Save script file.
Script/Load script file	Load script file.
Buffer /Model	Copy model to Buffer or paste from Buffer.
Buffer/Model	Display all buffer models in a single window.



When going into the polygonal modeling window (toolbar button  of the main window) the following options are available:

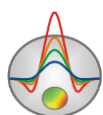
Modeling/Load background	Import background image from *.bmp file or SectionCor program document (*.sec).
Modeling/Show background	Display background.
Modeling/Remove background	Hide background.
Modeling/Get values from background	Set polygon parameters (resistivity, polarizability) values of corresponding block model cells (from results of preliminary inversion or modeling).
Modeling/Set values to background	Set polygon parameters (resistivity, polarizability) to corresponding block model cells.
Modeling/Remove all polygons	Delete all polygons.
Modeling/Save polygons	Save polygonal model as *.poly file.
Modeling/Load polygons	Load polygon model (*.poly file).
Modeling/Export to CAD	Export polygonal model as DXF (Autocad) file.
Modeling/ <i>GraviMagnetic</i> / Load new data	Load measurements of magnetic and (or) gravity field.
Modeling/ <i>GraviMagnetic</i> / Add new data	Add new data of magnetic and (or) gravity field measurements.
Modeling/ <i>GraviMagnetic</i> / Field settings	Set parameters of normal magnetic (gravity) field, survey parameters.
Modeling/ <i>GraviMagnetic</i> / Substract median grav	Substract median from the set values of the gravity field, it means to lead to the anomalous field values.
Modeling/ <i>GraviMagnetic</i> / Substract median mag	Substract median from the set values of the magnetic field, it means to lead to the anomalous field values.
Modeling/ <i>GraviMagnetic</i> / Inversion	Start polygonal inversion of gravity\magnetic data (parameters inversion for specified parameters).
Modeling/ <i>GraviMagnetic</i> / Display GM window	Open window of displaying gravity\magnetic field measurements.



Modeling/ <i>Self Potential</i> / Load data	Load data of self potential method.
Modeling/ <i>Self Potential</i> / Remove data	Remove data of self potential method.
Modeling/ <i>Self Potential</i> / Redox factor vs Z	Set function of redox potential changing with depth.
Modeling/ <i>Self Potential</i> / Use background resistivity	Use base model obtained by electrotomography data inversion.
Modeling/ <i>Self Potential</i> /Display SP window	Display SP window.
Modeling/Display color scale	Display color scale of resistivity\polarizability.
Modeling/Colors from color scale	Assign colors to polygons corresponding to a color scale (alternative to set colors manually).
Modeling/Exit from modeling mode	Exit from polygonal modeling mode.

“Hot” keys

Cursor pad/cursor in model editor	Change active cell of the model.
Delete/cursor in model editor	Clear active cell.
Insert/cursor in model editor	Insert current value to active cell.
F/cursor in model editor	Fix active cell value.
X/cursor in model editor	Use “magic wand” tool to select domain.
V/cursor in model editor	Delete selected.
Ctrl+C/cursor in model editor	When pressing the buttons, use mouse to move the selected area.
Up/down/cursor in model editor	Change current value.
Space	Calculate forward problem.




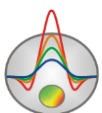
Status bar

Status bar is located in the lower part of program window and is divided into a few sections which contain different information:

Cursor and active cell coordinates.
Active cell parameters.
Model editor mode.
Process indicator.
Relative misfit.
Additional information. For example, number of observed values and model cells or calculation process status during inversion.

Starting model setup dialog

As soon as data file «*.Z2D» (or other file with supported format) is created, load it by pressing  button or using relevant menu item. After successful loading the start up model setup dialog appears. It allows specifying grid parameters, apparent resistivity, and polarizability of the medium. This model is also a starting model for inversion. This dialog can also be run using **Options/Mesh Constructor** in the main menu of the program (fig.1).



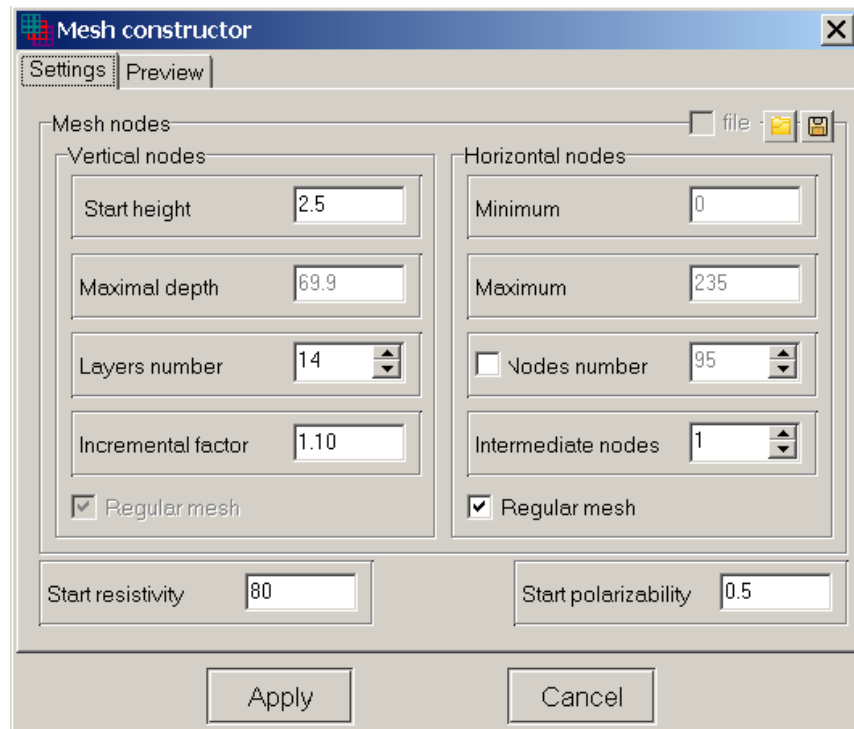


Fig. 1 **Mesh constructor** dialog

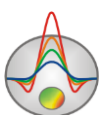
The **Vertical nodes** field contains options which set vertical grid parameters of the model. Program automatically selects these parameters using the following rules:

- Depth of bottom layer is equal to half of maximal pseudo-depth for used measurement system.
- Number of layers is equal to double quantity of array unique geometric factors for used measurement system but does not exceed 16.
- The thickness of the next layer is 1.1 times more than the previous one.

Start height – sets thickness of first layer. This value must be approximately equal to the width of the cell and necessary model resolution.

Maximal depth – sets depth of bottom layer. It should be remembered that the maximal depth value must not be too large because influence of geoelectrical section parameters decreases with depth.

Layers number – sets number of model's layers. Physically, it should not significantly exceed the number of unique array factors for the measurement system. Usually 12-14 layers are



enough for model description. It is not advisable to specify large values for this parameter because computation speed will decrease.

Incremental number – sets ratio between thicknesses of adjacent layers. This parameter value usually ranges from 1 to 2.

Horizontal nodes field contains options which set horizontal grid parameters.

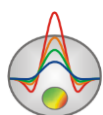
Minimum – shows (sets) minimal coordinate of the profile.

Maximum - shows (sets) maximum coordinate of the profile.

Intermediate nodes – sets number of complementary nodes between unique electrodes positions on the profile (0 - 4). It is advantageous for the accuracy of the forward problem solution, especially when using total potential calculation scheme.

Nodes number – if this function is ON mesh is created with regular step (on horizontal axis) from **Minimum** to **Maximum** field value. Number of nodes is specified in **Nodes number** field. This function should be used in case of irregular measurement grid. It should be considered that if this function is ON program uses Secondary potential calculation scheme which does not allow taking relief into consideration. It is advisable to use irregular mesh to work with non topography measurement systems, for example 2D VES data interpretation (Fig.2). Use the references <http://zond-geo.ru/zfiles/util/2dInversionofVES.exe> and <http://zond-geo.ru/zmovies/ert2ves2ertconvert.exe> to see a movie, that shows VES data 2D interpretation using ZondRes2d.

If this function is ON, number of cells considerably decreases but because of nonlinearity of potential function in case of current and potential electrodes getting in one cell the algorithm neglect this data.



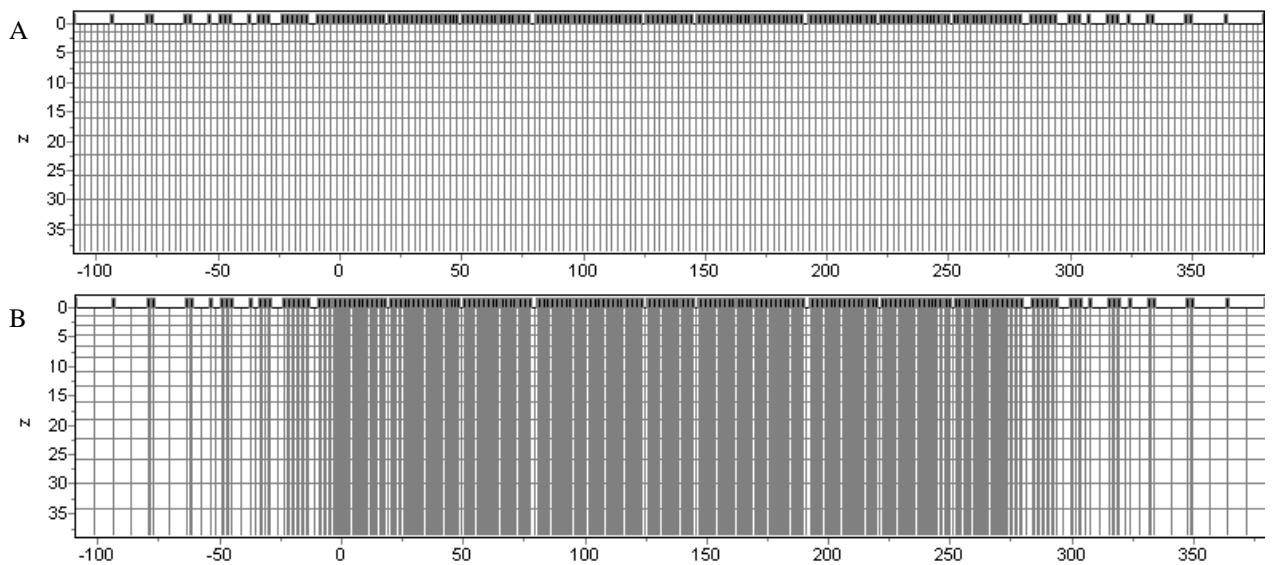


Fig. 2 Regular (A), irregular (B) model mesh



Regular mesh – starts horizontal mesh construction algorithm, and complementary nodes are selected from condition of split uniformity. This option should be used if distance between adjacent electrodes is very different (It is advantageous for the accuracy of forward and inverse problem solutions). If **Nodes number** function is ON, right click on **Regular mesh** panel to specify cells split step on X axis.

When working with borehole data **Regular mesh** options is enabled for both horizontally and vertically. This allows creating uniform split also in the vertical direction. If the option **In elec** is active in working with borehole data, it provides a coincidence mesh cells nodes and electrodes in borehole.

Start resistivity – sets resistivity of starting model.

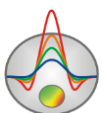
Start polarizability – sets polarizability of starting model.

Press **Apply** button after mesh setup is finished, and the program starts work mode.

Specified parameters can be saved and loaded (buttons  and  at the top right part of model parameters setup dialog).

Tab **Preview** allows displaying specified mesh.

Model editor functions can also be used to correct mesh: add or delete intermediate nodes, level cell height and width (see «[Model editor](#)» for more details).



The alternative of using data file is creating synthetic observation system that allows constructing various geological situations for the most widely-distributed electrodes arrays (see «Fig. 38 Models comparison with different cutting angles (**Options/Cutting/Cutting angle**): A - no cutting, B - 30° angle, C - 55 ° angle.

To view data apparent resistivity and model true resistivity distribution graph use **Options/Model/Histogram** dialog (fig. 39). Minimum and maximum of parameter color scale can be set by changing vertical lines positions.

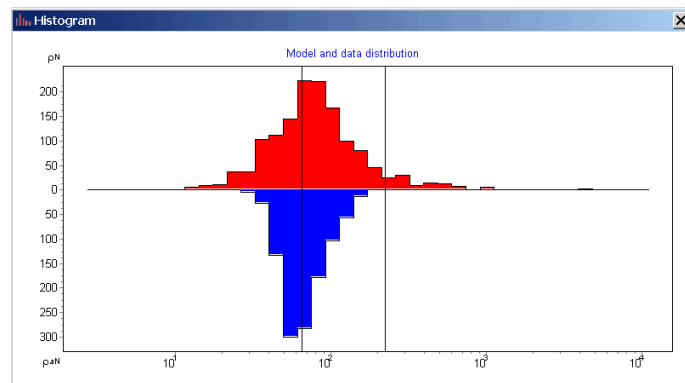


Fig. 39 **Histogram** dialog

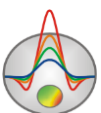
Modeling and working with model» for more details).

Main data file format

Program presents universal data format which consists of information about current and potential electrodes coordinates, relative relief elevation and measured values (normalized signal or apparent resistivity and apparent polarization).

ZONDRES2D data file format has *.Z2D extension.

*.Z2D files may have a different structure - be either data files (the structure of such file is described below), or project files (created by the program, when saving work results). The project file is binary, it can be read only by ZondRes2D. It contains all information which is used when working with the project - observed data, calculated data, models, inputted priori information, data of other methods etc.



For our purpose data file can be divided into 3 parts: 1) observed data, 2) topography data (if present), 3) model data (saved in file after inversion or modeling).

1st part of data file: Observed data

First line contains control keys which define what kind of data is contained in every column.

Following designations (keys) for electrode coordinates are accepted in «ZONDRES2D»:

Current **C1 C2 C1z C2z C1y C2y**

Potential **P1 P2 P1z P2z P1y P2y**

Y and Z coordinates can be specified if necessary. For example, in case of using plane-table measurement system, downhole source or marine measurements. [Sample file – sample_with_z_source](#). Distance along Y axis must not exceed 1/3 of maximum separation (distance between current and potential electrodes). [Sample file – sample_with_y](#). When specifying Z coordinates it should be remembered that positive values mean that electrode is below surface. Negative coordinates are only used in case of marine measurements (in this situation measurement surface is bottom).

For measured values:

Apparent resistivity **Ro_a**

Normalized signal modulus (potential to current strength ratio) **Res**

Normalized signal **SRes** (for example, for downhole source measurements, taking sign into account)

It is recommended to ALWAYS use Res or SRes values while creating data file to avoid mistakes if topography data is present.

Apparent polarizability **Eta_a**

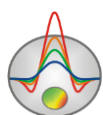
Apparent polarizability, measured in time domain and calculated with the following formula, can be used as Eta_a:

$$\text{Eta_a (\%)} = (\Delta U_{\text{IP}} / \Delta U_{\text{CUR}}) \cdot 100\%, \quad (7)$$

where ΔU_{CUR} – potential during current transmission, ΔU_{IP} – during pause. [Sample file – sample_with_ip](#). If charging is used, measured values should be divided by ten first.

In frequency domain apparent polarizability can be measured by phase displacement (φ_{IP}), which is converted to apparent polarizability using the following empiric formula.

$$\text{Eta_a (\%)} = - 2.5 \cdot \varphi_{\text{IP}} \text{ (degrees)} \quad (8)$$



Second and the following lines contain data corresponding to each measurement, written in the same sequence as control keys in the first line.

In case different (in number of electrodes) arrays are used in one profile * symbol should be written instead of missing coordinate value. [Sample-file – sample_with_array_combi](#).

Then (if necessary) horizontal mesh complementary nodes column is filled. Each new node coordinate is entered after *** symbol. Complementary nodes are usually added for model extension over last electrodes of cable and if there is sharp relief beyond cable. [Sample-file – sample_with_ext_nodes](#).

After file is read data normalization (if necessary) can be performed using the following formula which corresponds to apparent resistivity on direct current.

$$\rho_a = G \cdot \frac{|U|}{C}, \quad (9)$$

where G - array geometric factor, U – measured value, C – current strength value.

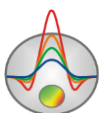
2nd part of data file: Topography data

If topography data is present, the next line contains word **topo** and is followed by list of coordinate and relief excess values. If relief is taken into consideration during interpretation, it is advantageous to use **res** values as initial data.

In describing relief profile observation X-coordinate can be specified in two ways - in the distance along the cable and in horizontal projections. Different methods of specifying relief correspond to the following additional keys.

topo - this key is used if coordinates of electrodes and topographic mapping are given in horizontal projections. [Sample-file – topo1](#).

topo~ adjustment to horizontal surface. Relief curve is approximated using least-squares method to line, and then is rotated together with all relief points so that it converges to horizontal axis (fig. 3). This method can be used if measurements are performed along slope with known absolute relief heights. [Sample-file – topo2a/topo2b](#).



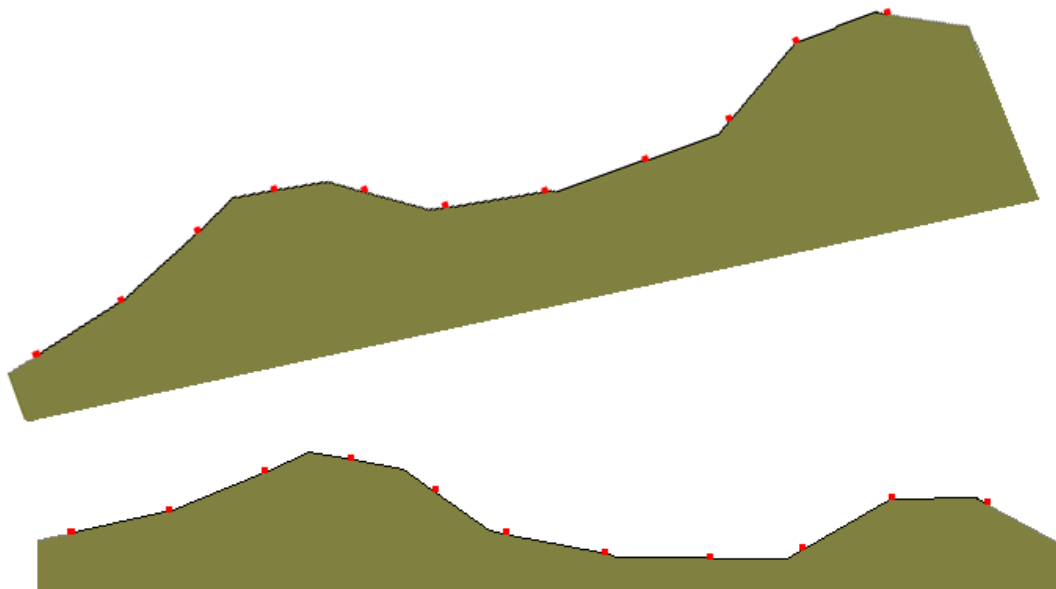


Fig. 3 Relief adjustment to horizontal surface

topo# specifies profile coordinates by lengths (“curve”). In this case distance along the cable but not X projection is equal to horizontal coordinate of electrodes (fig. 4). X coordinates of electrodes are recalculated from lengths to horizontal projections. The line that follows the one with the key must contain tie of one of the cable electrodes **P** to relief point. The second note is electrode coordinate (in lengths), the first one – corresponding to it X coordinate from the list of relief excesses. [Sample-file – topo3](#).

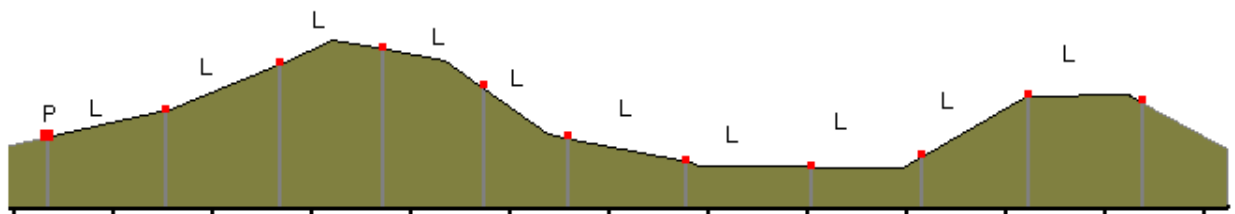
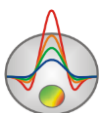


Fig. 4 Electrode positions with shown distance between them adjusted for relief changes (along the cable)

topo% - this key is necessary if **ro_a** values, calculated for projections, are set as initial data, that is distances between electrodes are recalculated from distances along the cable (L) to actual coordinates.



topo^ - this key is necessary if topography data and electrodes coordinates are specified in distances along the cable. [Sample-file – topo4](#).

Keys can be combined, for example, **topo~#**.

topow – this key is used if marine survey data interpretation is performed (on water surface or on the bottom). In this case bottom or bottom going to land (combined measurements) profile is used as relief coordinates (fig. 5). After space in the same line water surface level (relative to fixed below coordinates of profile bottom (in the same coordinate system as the relief)), water resistivity and number of complementary water layer splits (3-10) must be also specified. Main window function **Options/Advanced/Inverse procedure/Underwater options** can be used to change the former two parameters. [Sample-file – water1](#). Combined systems when measurements are performed on the bottom and on water surface can be specified. [Sample-file – water3](#). In order to do that vertical electrode coordinates relative to bottom profile level should be entered.

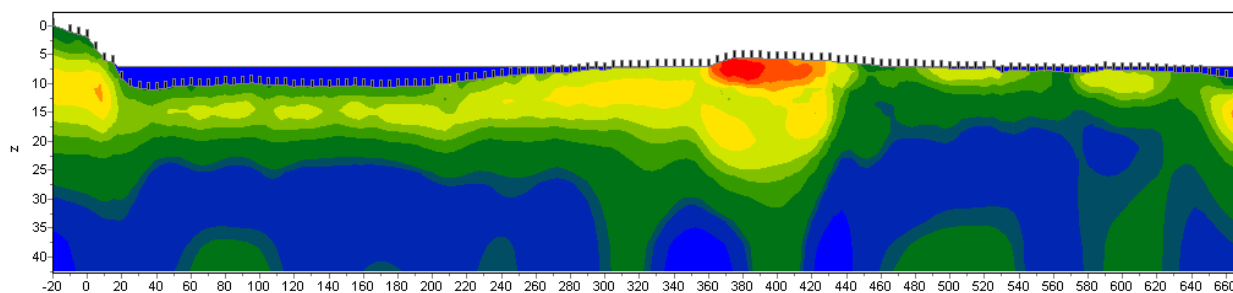


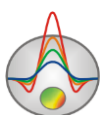
Fig. 5 Geoelectrical cross-section as a result of combined land and marine measurements

topo* – this key simplifies entering topography data if cable is located on water surface (mobile system). [Sample-file – water2](#).

It is also recommended to become familiar with additional options *described in* [«Specifics of work with topography data and marine measurements»](#).

3rd part of data file: Model data

Model description starts with key word **model**. The first line sequentially contains the following parameters: resistivity, minimum and maximum resistivity of color scale, model to relief distortion coefficient (**topo coefficient**).



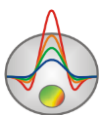
The second and the third line contain lists of horizontal and vertical coordinates of mesh nodes (n and m length).

The fourth line n lengthways specifies if electrode position corresponds to current node.

Following n-1 lines m-1 lengthways contain model resistivity matrix.

Following n-1 lines m-1 lengthways contain matrix of model cell fixing.

Then model polarizability is described. The first line sequentially contains the following parameters: resistivity, minimum and maximum polarizability of color scale. Following n-1 lines m-1 lengthways contain model polarizability matrix.



Input and edition of topographic information

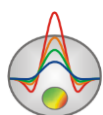
Topographical information includes a file shift along profile line relative to its beginning (placement shift), distance between electrodes, excesses along observation profile. Respective options are available in the menu Options/Topography.

To change step between electrodes use the menu item **Options/Topography/X factor**, where in the appearing dialog, you can specify step zoom factor between electrodes compared with the step specified in the source file. Option **Options/Topography/Reverse line** allows reversing profile, the option **Options/Topography/Shift line** allows setting placement shift in observations profile coordinates. If the program offers several data files at once, shift selection dialog is called automatically.

Accounting of profile shift from straight line, as well as data selection from different measurement profiles on one area is described in the part "[Working with planned electrode coordinates](#)".

There are two ways to input relief - through the information in the input data file (the part [«II data file part: Topographic data»](#)) or by using the option of topographic information import of the main menu **Options/Topography/Import topography**. The last option is to enter the topographic information, given in the form of distances along the spit and elevations. The values of the distances along the spit can be irregular (some electrodes elevation may be omitted), but the distance should be multiplied of the distance between electrodes.

This information is saved as a text file that contains two columns: the distance on the spit and elevation. Option **Options/Topography/Import topography** creates a table with information from the selected text file is loaded (Fig. 6). X and Alt must be selected in names of the corresponding columns (to do it make active the cell with column name, press the left button to appear a list of columns names). To end the topographic information input press the button



#Ind	None	Ind	None	None
1	None	1	-2.1	-7.723
2	oldX	2	-1.8	-7.952
3	newX	3	-1.5	-8.182
4	X	4	-1.2	-8.411
5	Alt	5	-0.9	-8.64
6	#Ind	6	-0.6	-8.87
7	Ind	7	0.3	-9.099
8	Ind	8	0.6	-9.329

Fig. 6 Window of topographic information inputting

If the topographic information is recorded in the data file, no further specifications to the data input stage is usually required. An exception is a situation when electrodes are arranged on the ground immediately in projection distance during measuring (it is convenient for large scale survey with a small step between electrodes). In this case, the distance between the electrodes on the spit is greater than the inter-electrode distance specified in the data file, and when you open it you should confirm that this is not an error (Fig. 7).

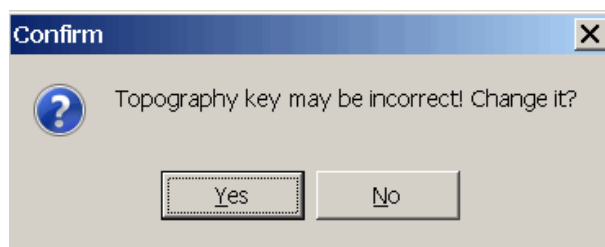
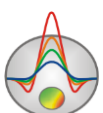


Fig. 7 Dialog of validation topography key

Selecting **No** will leave the situation unchanged, when you select **Yes**, the program converts topography data to the coordinate system distance on the spit - height.

Menu item **Options/Topography/Edit topography** calls a dialog that allows editing topographic information using the table.

Use the menu items **Options/Topography/Restore topography** and **Options/Topography/Remove topography** to recover and remove, respectively, topography data.



Settings of working with topography data is described in the part «[Working with topography data and marine measurements](#)».

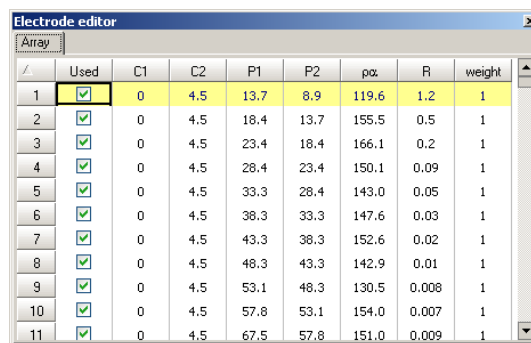
Electrodes editor

Electrodes editor serves for visualization of acquisition geometry and measured values. Window contains table that allows adjusting every measurement parameter (fig. 8). The table contains 9 columns:

ID	Measurement index
Used	Indicator specifying whether measurement is used in inversion or not.
C1	Position of the first current electrode on profile.
C2	Position of the second current electrode on profile.
P1	Position of the first potential electrode on profile.
P2	Position of the second potential electrode on profile.
ρ_a	Measured apparent resistivity value.
R	Measured resistivity (normalized signal) value.
Weight	Measurement weight.

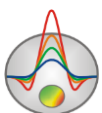
If electrode is absent its coordinate is replaced by symbol *.

The last three columns can be edited using keyboard if necessary. Absurd values of apparent resistivity and normalized signal should not be entered. Measurement weights range from 0 to 1. Table content sorting in current column can be run by clicking column heading.



	Used	C1	C2	P1	P2	ρ_a	R	weight
1	<input checked="" type="checkbox"/>	0	4.5	13.7	8.9	119.6	1.2	1
2	<input checked="" type="checkbox"/>	0	4.5	18.4	13.7	155.5	0.5	1
3	<input checked="" type="checkbox"/>	0	4.5	23.4	18.4	166.1	0.2	1
4	<input checked="" type="checkbox"/>	0	4.5	28.4	23.4	150.1	0.09	1
5	<input checked="" type="checkbox"/>	0	4.5	33.3	28.4	143.0	0.05	1
6	<input checked="" type="checkbox"/>	0	4.5	38.3	33.3	147.6	0.03	1
7	<input checked="" type="checkbox"/>	0	4.5	43.3	38.3	152.6	0.02	1
8	<input checked="" type="checkbox"/>	0	4.5	48.3	43.3	142.9	0.01	1
9	<input checked="" type="checkbox"/>	0	4.5	53.1	48.3	130.5	0.008	1
10	<input checked="" type="checkbox"/>	0	4.5	57.8	53.1	154.0	0.007	1
11	<input checked="" type="checkbox"/>	0	4.5	67.5	57.8	151.0	0.009	1

Fig. 8 **Electrodes editor** dialog window



While moving the cursor through the table, positions of active measurement electrodes are displayed on pseudosection or graphics plan.

Advanced/Distribution option can also be used to run electrodes editor in potential (**Potential distribution**) or sensitivity contour (**Sensitivity contour**) display mode. Analysis of potential or sensitivity distribution helps to understand electrotomography principle of operation (fig. 9).

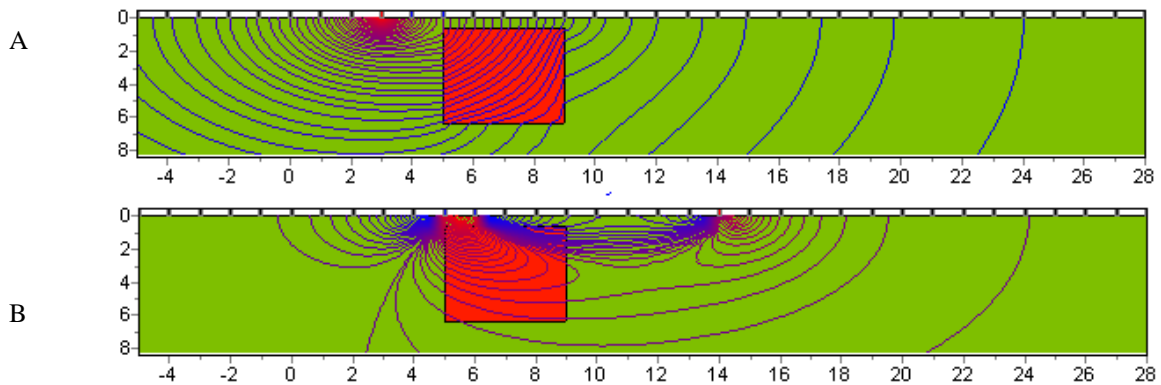


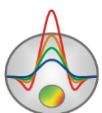
Fig. 9 Potential contours (A) and sensitivity contours (B) of pole-dipole array

While moving the cursor through the table potential and sensitivity contours change for current measurement value.

Working with planned electrode coordinates

During field electrotomographic researching electrodes are often not located directly on the profile line. This can happen if there are difficulties when splitting profiles, and, for example, when it is necessary to physically remove supply line from the receiving one.

There is special module in the program ZondRes2D for a correct accounting such situations. It runs automatically when you open the data file containing electrodes Y-coordinate (Fig. 10). The module allows specifying a single line of observations profile, excluding from consideration unnecessary electrodes (for example, too far locating from the profile), projecting measurement results on the line. As a result, the corresponding changes will be made in the model cell mesh.



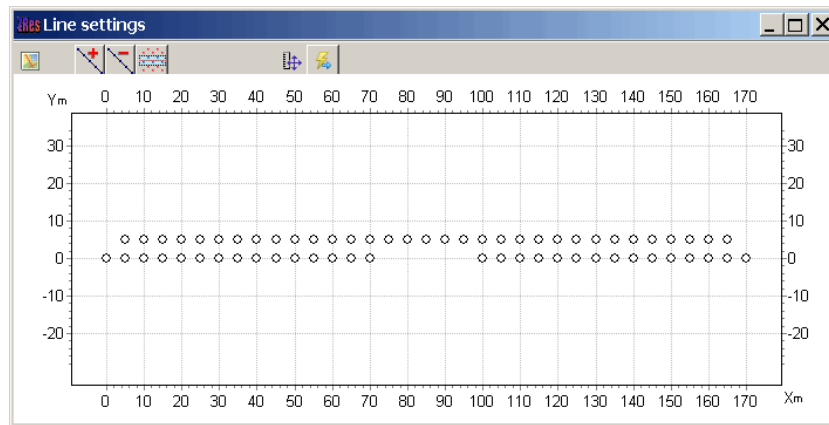








Fig. 10 Window of profile points planned position editing

In the window of accounting electrode position deviation from straight line points corresponding electrode positions in the coordinate system X - distance on the profile, Y - deviation from profile line in the perpendicular direction (as they are defined in the data file) are displayed. To change the position of any electrode right-click on this electrode. In appearing window you can change it. Use the toolbar button  to load a raster background - a map or a plan.

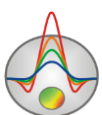
When the actual electrode position was set and edited it is necessary to hold the profile line - the line input mode is switched on (off) by pressing the button . Use the left mouse button to add points to the line, right - to complete creating line. To delete entered line press the button .

The next step is to select points (electrodes) which will be projected on the specified line. To highlight (or deselect) individual electrode left-click its point, the point is painted in blue (or back to white) color. To select all points press the toolbar button . After the electrode has been selected, its projected position in the profile is indicated by a green point. An example of working with window shown in Fig. 11.

To scale of plan displaying while working with the window press the button  on the toolbar.

After completion of the described operations press the button  to set model mesh and interpretation.

[Sample-file - sample_with_y.z2d.](#)



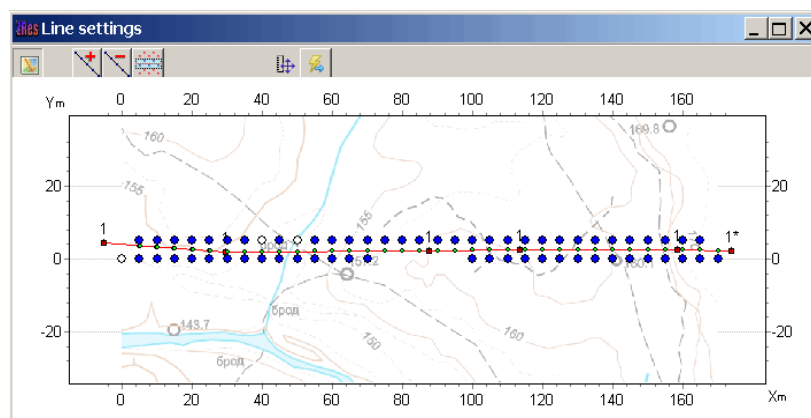


Fig. 11 Working in the window of editing profile points planned position

The process of working with the described module can be seen in the film <http://zond-geo.ru/zmovies/XYcoordZond2d.exe>.

An additional module purpose is to work with the data of area measurement. If electrodes location in the area is specified, the program allows you to get data of any user-specified profile.

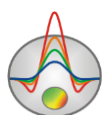
The process of working with the areal measurements shown in the film http://zond-geo.ru/zmovies/Work_with_areal_survey.exe.

Apparent parameters visualization

There is a possibility to visualize apparent parameters as pseudosection **Options/Data/Pseudo-section** and as graphics plan **Options/Data/Graphics-plot**. In resistivity mode parameter can be represented as normalized signal (**Option/Data/Resistance U/I**) or apparent resistivity (**Option/Data/Apparent resistivity**).

Graphics plot

Graphics plan is used for apparent resistivity values visualization as graphs along profile. Graphing method can be chosen in the main menu of the program: by certain pseudodepth **Options/Data/Graphics-plot/Iso-PsZ** with different array factor or by certain current circuit position **Options/Data/Graphics-plot/Iso-Pole** (fig. 12).



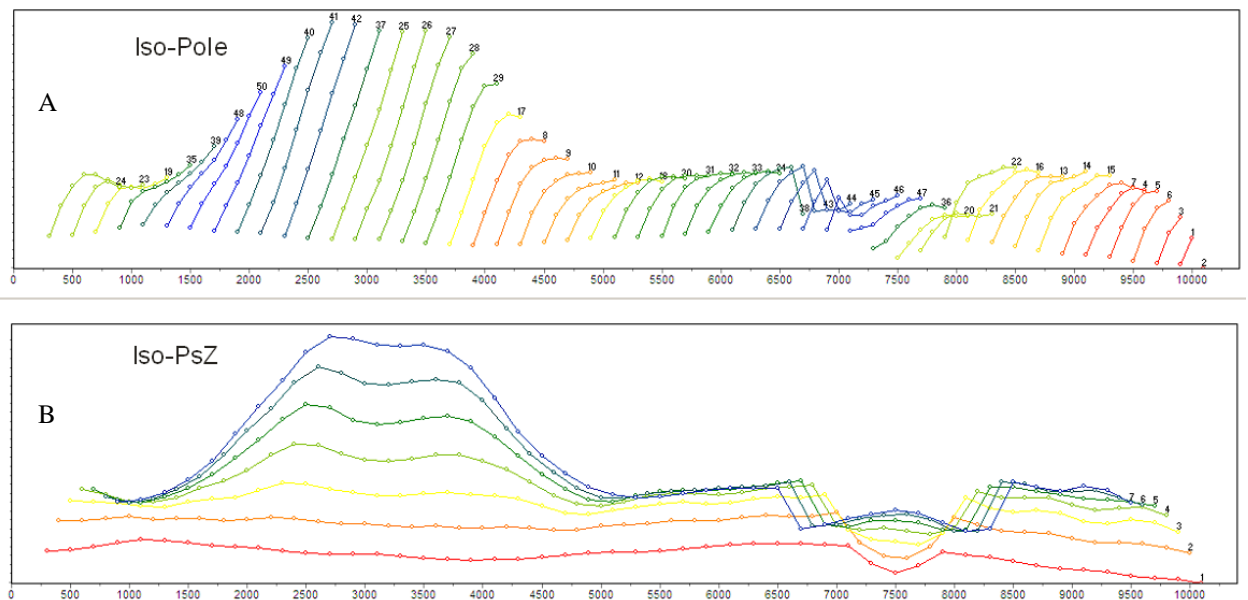


Fig. 12 A. Graphics plan created by certain current circuit position (**Iso-Pole**),
 B. Graphics plan created by certain pseudodepth (**Iso-PsZ**)

Mouse clicks are used to work with graphics plan.

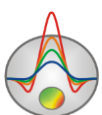
Zooming in or dragging some part is performed with pressed button (“rubber rectangular” tool). To zoom in a segment move mouse cursor down and to the right with left button pressed (Fig. 16 A). To return to primary zoom do the same but with mouse cursor moving up and to the left (Fig. 16 B). To scroll graphic move the mouse while holding down the right mouse button.

The following actions are performed by pressing left mouse button on graphic point: all other graphics are hidden and electrodes positions for active point are shown (while the button is pressed). To edit measured values move plot point with right button pressed.

To run graphics plan setup dialog use **Options/Observed graphics Calculated graphics** in the main menu (*see* Specifics of work with borehole data measurements)

ZondRes2D allows to model and perform the inversion of borehole measurements. These include inter-borehole measurements, borehole-surface measurements, and their combinations. It supports both vertical and deviated boreholes.

The measurement configuration is described in the input data file of the ZondRes2D format (more details in the part «*Main data file format*»). When describing electrode measurements located in boreholes, as in the case of terrestrial measurements, planning altitude coordinates are indicated. There is no opportunity to create measurement configuration



containing electrodes in borehole in the program. The corresponding data file is created in text or tabular editor ([Sample-file – sample_with_z_source](#)).

Modeling and inversion of borehole and borehole-surface data is the same as for ground-based measurements. The difference is in the method of apparent resistivity displaying - unlike conventional methods for ground-based measurements dependence of the apparent resistivity from the array factor are built here (Fig. 77). Measured and calculated data can also be viewed in a separate window via the option **Options/Data/G-Res plot**.

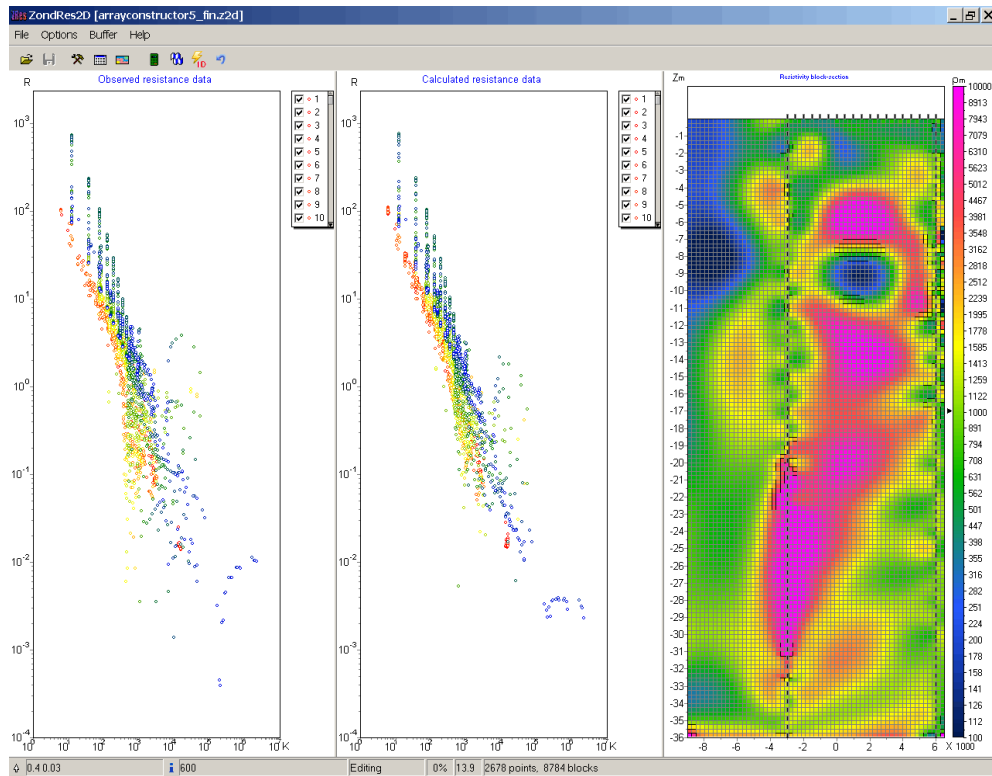
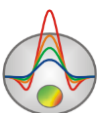


Fig. 77 Main program window when working with borehole measurements

IP parameters inversion

ZondRes2D allows you to work not only with integral polarizability values, but also with IP process parameters. In practice, appropriate investigations are carried out in one of two modes - time or frequency. When working in the time domain, measurement result for each observation point is a IP decay curve - voltage dependence (sometimes normalized by width of the measurement time window) from time on a fixed set of delay times ([sample-file – timedomain.zd2](#)). Dependences of resistivity and phases shift from frequency for a fixed



frequencies set ([sample-file - frequencydomain.z2d](#)) are input data for each observation point, when working in the frequency domain.

For both modes the program allows visualizing the source data, carrying out their smoothing and performing inversion, determining for all observation points integral IP value for each delay time (frequency) or the Cole-Cole model parameters.

The input data file format for IP parameters inversion (extension *.z2d): the first line contains the times delay lists for time and frequencies for frequency measurement mode after the word `time_ # chann`. The following is a table whose rows correspond to the points of observation, and the columns - to electrodes coordinates (similar to The main data file format), to the Res ($\Delta U/I$) value and to the measurement values for the respective delay times (column headings - `ipi1, ipi2` etc.) and frequencies (the column headings - `mod1, mod2`, etc. for module, `pha1, pha2` etc. for phase).

An example of the measurement time mode data file beginning:

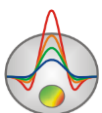
time_#chann	0.05	0.065	0.085	0.115	0.155					
C1	C2	P1	P2	res	ipi1	ipi2	ipi3	ipi4	ipi5	
0	*	5	10	0.001	0.279	0.260	0.242	0.218	0.194	
5	*	10	15	0.001	0.306	0.286	0.266	0.241	0.215	
10	*	15	20	0.002	0.338	0.317	0.295	0.268	0.239	

An example of the frequency measurement mode data file beginning:




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

To work with IP parameters use the menu item **Time lapse/domain** of the main menu (menu appears when you load the appropriate data file). To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.

Option **Time lapse/domain/Channels** runs the channel setting table. It is possible to edit the values of the time delay or the measurement frequencies and headers for specified values



(will be displayed in the drop-down list on the toolbar of the main window). For the some options operation it is necessary to times/frequencies were inputted previously.

Option **Time lapse/domain/Display channel plot** runs the window (Fig. 78), showing the decay curves (frequency curves) for each measurement point. To move to another measurement use the mouse (on pseudosection or graph) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point displays. A separate point of the displaying curve can be edited using the menu items. They appears, when you press the left mouse button on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), to return to the previously position (**Origin point**). The toolbar button  allows you to smooth the entire curve.

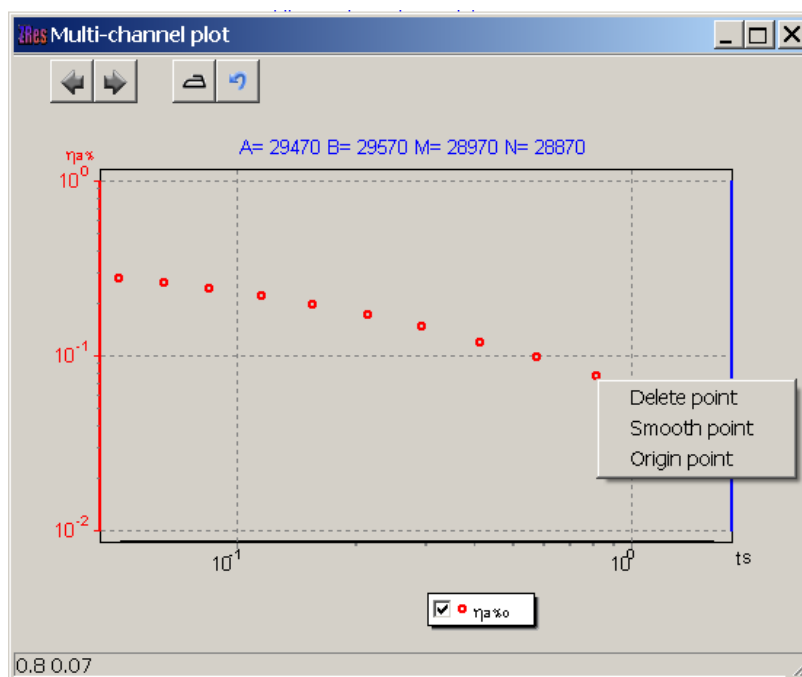
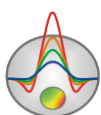


Fig. 78 The curve editing window for a single measurement point

When working in the time mode after editing curves it is necessary to use the option **Time lapse/domain/Recalculate chargeability**, which recounts an integral IP parameter accounting data changes.

The option **Time lapse/domain/Time lapse inversion** runs IP data serial inversion mode for each implementation (in this case, the standard calculation scheme, as well as for the integral IP parameter). If the option **Time lapse/domain/Next from previous** is switched ON, as an



initial approximation for each next iteration the results of previous one is used. If this option is switched OFF, as an initial approximation for each next iteration the current implementation model is used. These models are displayed in the model editor. To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.

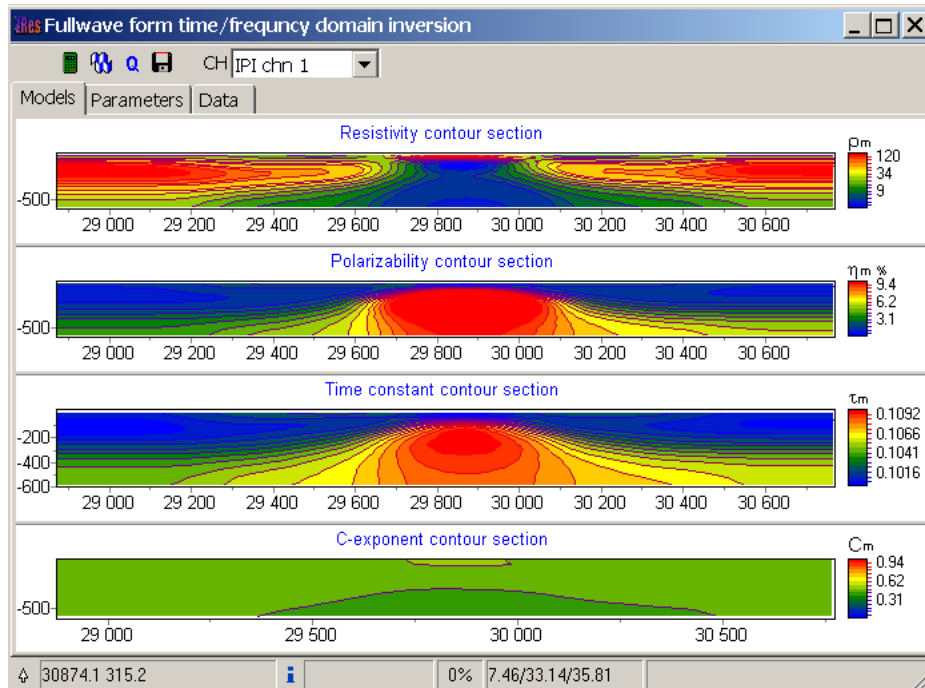
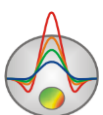


Fig. 79 The main window of the IP parameters module inversion

It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Window IP parameters full-wave inversion (Fig. 79) is run via the menu item **Time lapse/domain/ Fullwave form inversion**. The window contains three tabs. Tab **Models** displays resistivity, polarizability and Cole-Cole parameters (τ , and c), which are obtained as model inversion results.



Tab **Parameters** allows you to select parameters for which you want to carry out inversion and specify (fix) ranges of these parameters (Fig. 80). **ReBuild** button allows changing the each parameter values to the value from the table (value). If instead of the number symbol "*" are specified, the parameter value on the section will not be replaced.

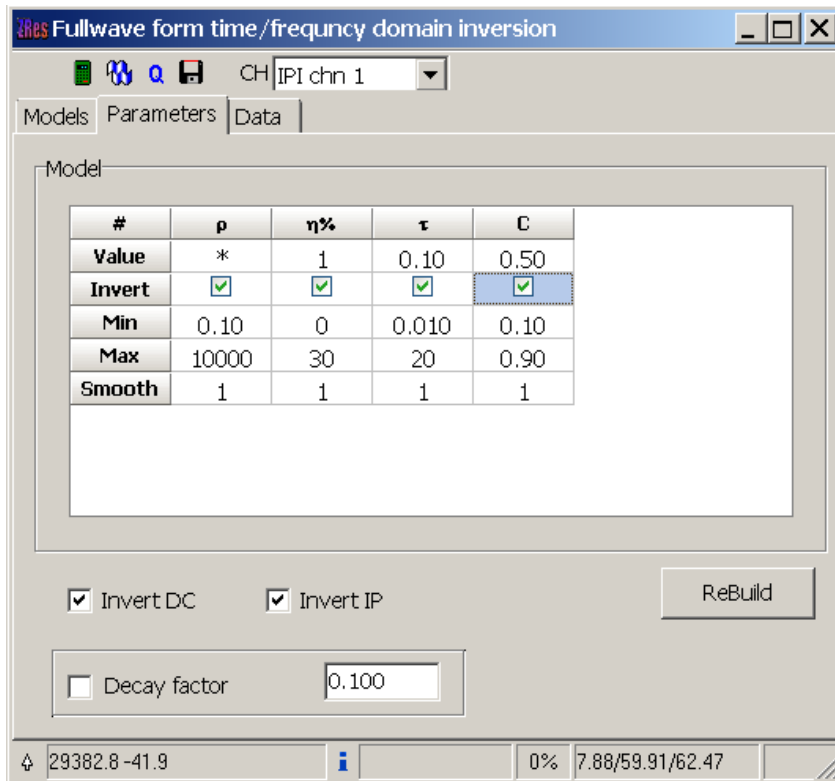


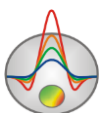


Fig. 80 Window for selecting parameters for curves IP inversion

Option **Module** (available when working in the frequency mode) indicates whether or not to select the signal module, or use phase only. As a rule, used in the frequency range module changes very little, and unlike the phase, has little effect on the inversion results.

To start inversion, press the button . This inversion occurs simultaneously on all selected parameters. Button  provides an alternative method in which the polarizability inversion occurs independently for each time (frequency) by time-lapse inversion and Cole-Cole model parameters are recovering based on obtained "real" curves.

Tab **Data** displays observed and calculated polarizability model. The drop-down list in the toolbar (available when working with all the tabs) allows switching between sections for different decay times (frequencies).



Dialog **Time lapse/domain/ Digital filter** contains temporary filter settings for the Fourier transform to the time domain, and allows you to set the length of the current impulse. *Video dedicated to the IP data inversion in the time and frequency domain, can be downloaded from <http://zond-geo.ru/zmovies/TDvsFD.exe>.*

Geoelectric monitoring data inversion

Monitoring involves measurements with the same parameters at different time. The program recognizes file with the *.z2d extension as an inputting monitoring file. The file has following structure: the first four columns contain electrodes coordinates (similar to Main data file format); next - columns, entitled res1, res2, etc. that contain Res ($\Delta U/I$) values for the first, second, etc. measurement cycles.

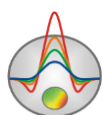
The following example shows the beginning of a monitoring data file by three-electrode array ([sample-file – monitoringERT.z2d](#)):

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




Working with monitoring data, in general, is similar to the work with multifrequency IP data or temporary IP containing information about decay curve. Only instead of the polarizability values at a certain frequency or time is considered the resistivity values at a certain time.

Use the menu item **Time lapse/domain** to work with the monitoring (menu appears when you load the appropriate data file). Switching between the data and the models corresponding to different measurements times is carried out using the drop-down list on the control panel of the main window.

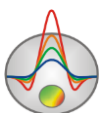
Option **Time lapse/domain/ Channels** runs the table in which you can enter information about each measurements implementation. An information about measurement time is inputted



at the **time, s** column. Arbitrary text information, entered in the **name** column, will displayed at the drop-down list on the main window control panel (when switching between implementations).

Option **Time lapse/domain/ Display channel plot** runs the window that displays the Res ($\Delta U/I$) values for each measurement point, depending on the time (Fig. 81). To move to another measurement use the mouse (on pseudosections or graphs) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point are displayed. A separate point of the displaying curve can be edited using appearing menu items when you press the left mouse button menu on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), returned to the previous position (**Origin point**). The toolbar button  allows you to smooth the entire curve.

Option **Time lapse/domain/ Time lapse inversion** runs consistent inversion mode of monitoring data for each implementation. If the option **Time lapse/domain/ Next from previous** is switched ON, results of previous iteration are used as an initial approximation for each next one. If this option is switched OFF, the current model in this implementation is used as an initial approximation for each next iteration. Obtained models are displayed in the model editor. To switch between models corresponding to different times use the drop-down list on the control panel of the main window.



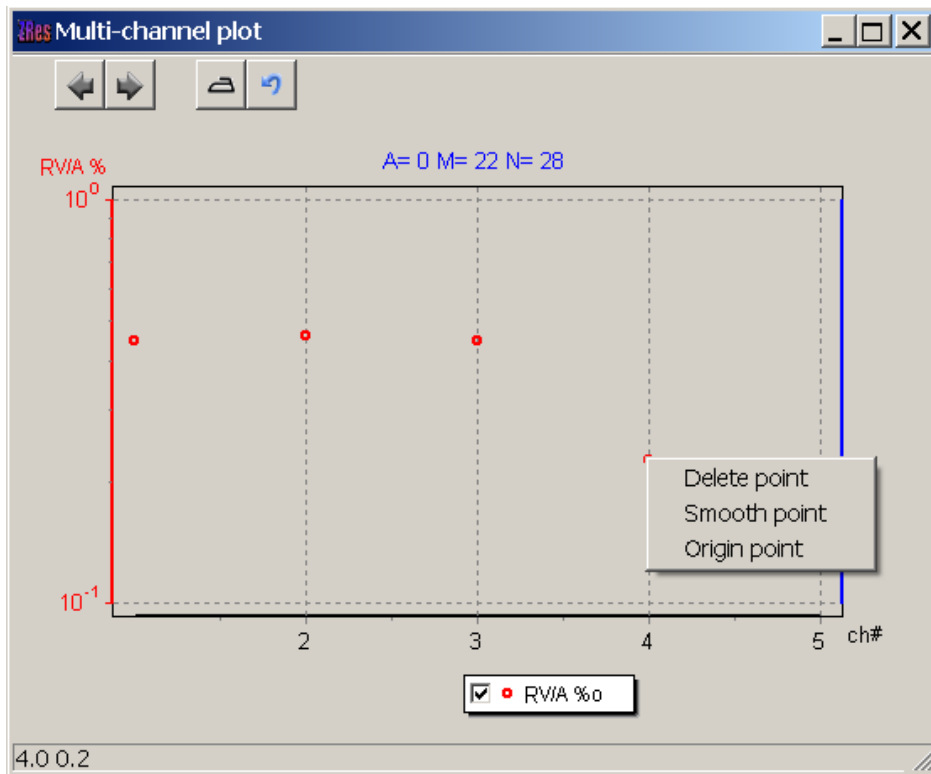
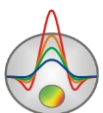


Fig. 81 Window of resistivity changes depending on the time for a single measurement point

It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Sample - video - <http://zond-geo.ru/zmovies/monitoringERT.exe>.



Magnetotelluric data inversion

If you have magneto-telluric data measurement, they can be used in inversion accounting data of resistivity method on DC. To work with magnetotelluric data measurements use the menu item **Options/MT Data**.

The magnetotelluric data inputting file is a file with *.m2d format. Its structure is described in the section Magnetotelluric data import.

The menu items **Load MT data** and **Show MT plot** allow, respectively, loading and visualizing data of magnetotelluric measurements. Selecting **With static shift** specifies whether static shift (normalization) MTS curves. To specify MT-measurements weight use the menu item **Set weights of MT**.

If you select the menu item **Invert MT data**, the program will carry out joint inversion of MT data and the resistivity method.

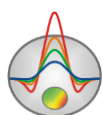
Option **DET inversion** allows carrying out inversion simultaneously both longitudinal (TE), and transverse (TM) impedance.

Joint data inversion of electrotomography and PMT is shown in the movie http://zond-geo.ru/zmovies/MT_ERT.exe.

Script-files (batch processing)

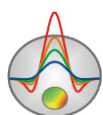
When you need to repeat the same action, they can be written to a script file. When it started, the program will perform the action automatically. For example, you can record the process of opening data file, setting the starting model, the inversion parameters, starting the inversion process and exporting results. If it is necessary to process multiple profiles data by this scheme, run the script and select all data files, for each of them the corresponding actions sequence is automatically will be applied. This tool allows you to save time when the same operations repeat and allows the user to "standardize" the process or individual working process steps with data.

To start recording a script file select the menu item **Script/Start recording script** before performing operations that need to be remembered. If script recording is currently active, the tick will be displayed on the left of the specified menu item. To stop script recording, use the menu



item **Script/Stop recording script**, then it is necessary to save the script file (**Script/Save script file**).

To run the script file, it must be loaded **Script/Load script file** and using the menu item **Script/Run packet processing** select the data file for processing. When you save a file processing results in automatic mode, the results file name - the source file name with underscore addition.

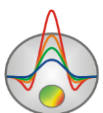


Appendix 1: Graphics set editor *for details*). To run graphic's editor right-click on graphic with SHIFT button pressed (*is described in [Appendix 2: Graphics editor](#)*) Right click on necessary axis with SHIFT button pressed to run axis editor (*see [Appendix 5: Axes editor](#)*). Right click on legend to the right of the graph with SHIFT button pressed to run legend editor (*see Specifics of work with borehole data measurements*

ZondRes2D allows to model and perform the inversion of borehole measurements. These include inter-borehole measurements, borehole-surface measurements, and their combinations. It supports both vertical and deviated boreholes.

The measurement configuration is described in the input data file of the ZondRes2D format (more details in the part «*Main data file format*»). When describing electrode measurements located in boreholes, as in the case of terrestrial measurements, planning altitude coordinates are indicated. There is no opportunity to create measurement configuration containing electrodes in borehole in the program. The corresponding data file is created in text or tabular editor ([Sample-file – sample_with_z_source](#)).

Modeling and inversion of borehole and borehole-surface data is the same as for ground-based measurements. The difference is in the method of apparent resistivity displaying - unlike conventional methods for ground-based measurements dependence of the apparent resistivity from the array factor are built here (Fig. 77). Measured and calculated data can also be viewed in a separate window via the option **Options/Data/G-Res plot**.



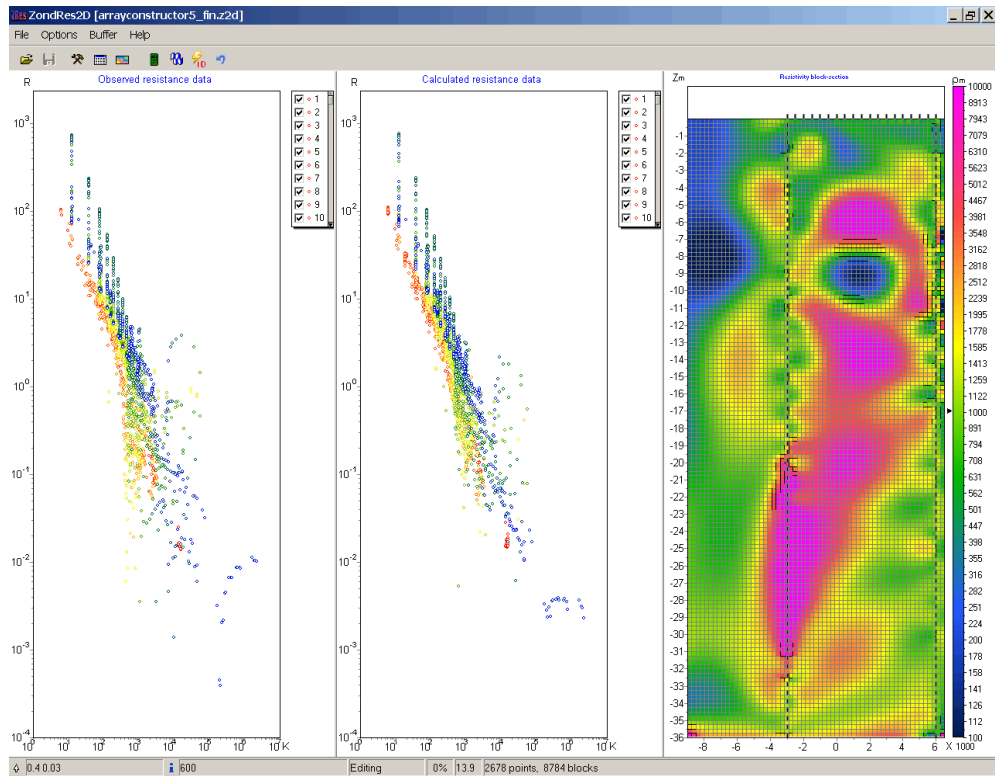


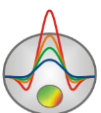
Fig. 77 Main program window when working with borehole measurements

IP parameters inversion

ZondRes2D allows you to work not only with integral polarizability values, but also with IP process parameters. In practice, appropriate investigations are carried out in one of two modes - time or frequency. When working in the time domain, measurement result for each observation point is a IP decay curve - voltage dependence (sometimes normalized by width of the measurement time window) from time on a fixed set of delay times ([sample-file – timedomain.zd2](#)). Dependences of resistivity and phases shift from frequency for a fixed frequencies set ([sample-file - frequencydomain.z2d](#)) are input data for each observation point, when working in the frequency domain.

For both modes the program allows visualizing the source data, carrying out their smoothing and performing inversion, determining for all observation points integral IP value for each delay time (frequency) or the Cole-Cole model parameters.

The input data file format for IP parameters inversion (extension *.z2d): the first line contains the times delay lists for time and frequencies for frequency measurement mode after the



word time_ # chann. The following is a table whose rows correspond to the points of observation, and the columns - to electrodes coordinates (similar to The main data file format), to the Res ($\Delta U/I$) value and to the measurement values for the respective delay times (column headings - ipi1, ipi2 etc.) and frequencies (the column headings - mod1, mod2, etc. for module, pha1, pha2 etc. for phase).

An example of the measurement time mode data file beginning:



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

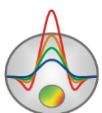
An example of the frequency measurement mode data file beginning:


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

To work with IP parameters use the menu item **Time lapse/domain** of the main menu (menu appears when you load the appropriate data file). To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.

Option **Time lapse/domain/Channels** runs the channel setting table. It is possible to edit the values of the time delay or the measurement frequencies and headers for specified values (will be displayed in the drop-down list on the toolbar of the main window). For the some options operation it is necessary to times/frequencies were inputted previously.

Option **Time lapse/domain/Display channel plot** runs the window (Fig. 78), showing the decay curves (frequency curves) for each measurement point. To move to another measurement use the mouse (on pseudosection or graph) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point displays. A separate point of the displaying curve can be edited using the menu



items. They appear, when you press the left mouse button on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), to return to the previously position (**Origin point**). The toolbar button  allows you to smooth the entire curve.

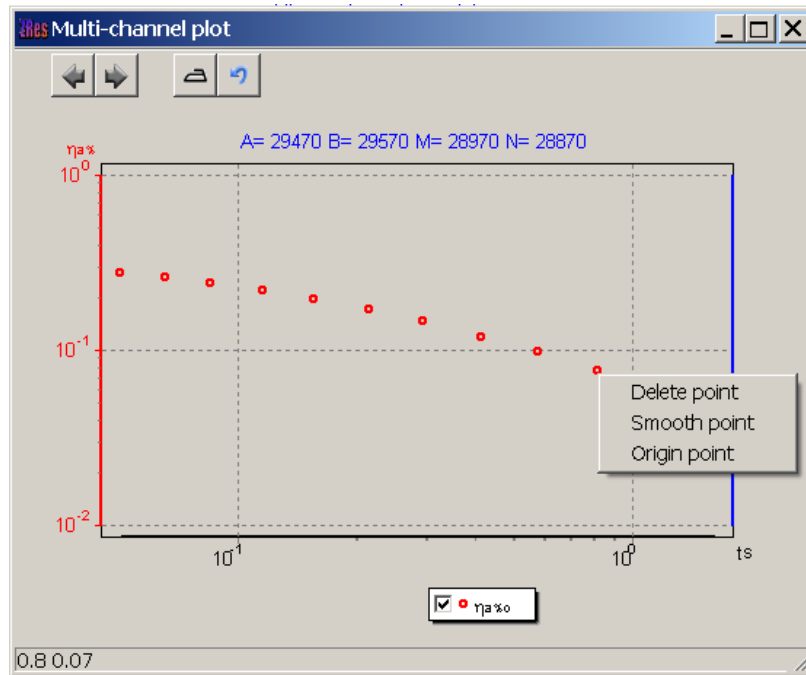
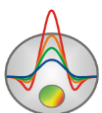


Fig. 78 The curve editing window for a single measurement point

When working in the time mode after editing curves it is necessary to use the option **Time lapse/domain/Recalculate chargeability**, which recounts an integral IP parameter accounting data changes.

The option **Time lapse/domain/Time lapse inversion** runs IP data serial inversion mode for each implementation (in this case, the standard calculation scheme, as well as for the integral IP parameter). If the option **Time lapse/domain/Next from previous** is switched ON, as an initial approximation for each next iteration the results of previous one is used. If this option is switched OFF, as an initial approximation for each next iteration the current implementation model is used. These models are displayed in the model editor. To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.



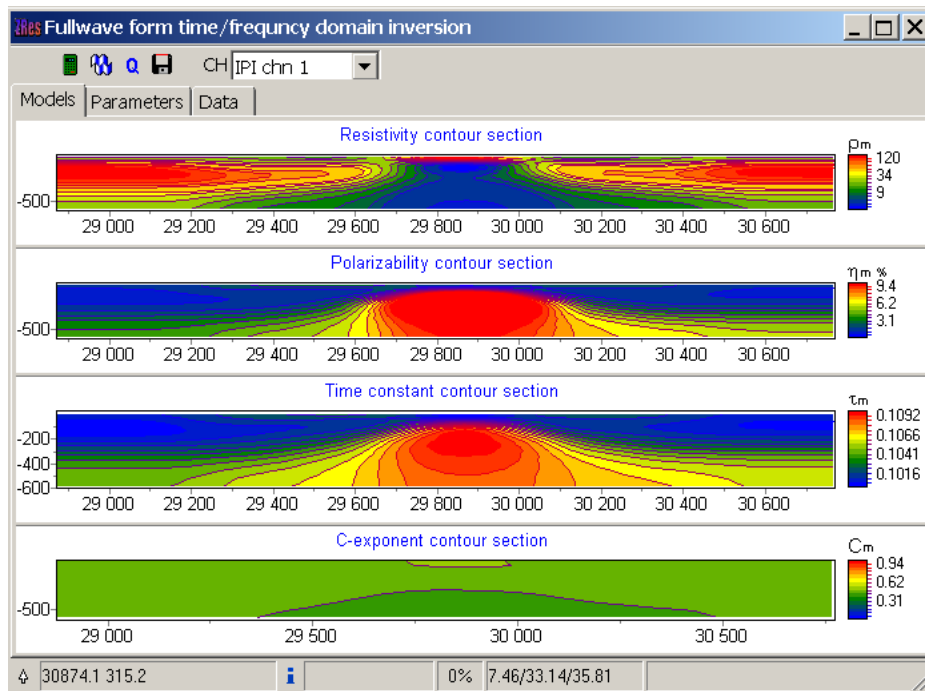


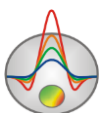
Fig. 79 The main window of the IP parameters module inversion

It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Window IP parameters full-wave inversion (Fig. 79) is run via the menu item **Time lapse/domain/ Fullwave form inversion**. The window contains three tabs. Tab **Models** displays resistivity, polarizability and Cole-Cole parameters (τ , and c), which are obtained as model inversion results.

Tab **Parameters** allows you to select parameters for which you want to carry out inversion and specify (fix) ranges of these parameters (Fig. 80). **ReBuild** button allows changing the each parameter values to the value from the table (value). If instead of the number symbol "*" are specified, the parameter value on the section will not be replaced.



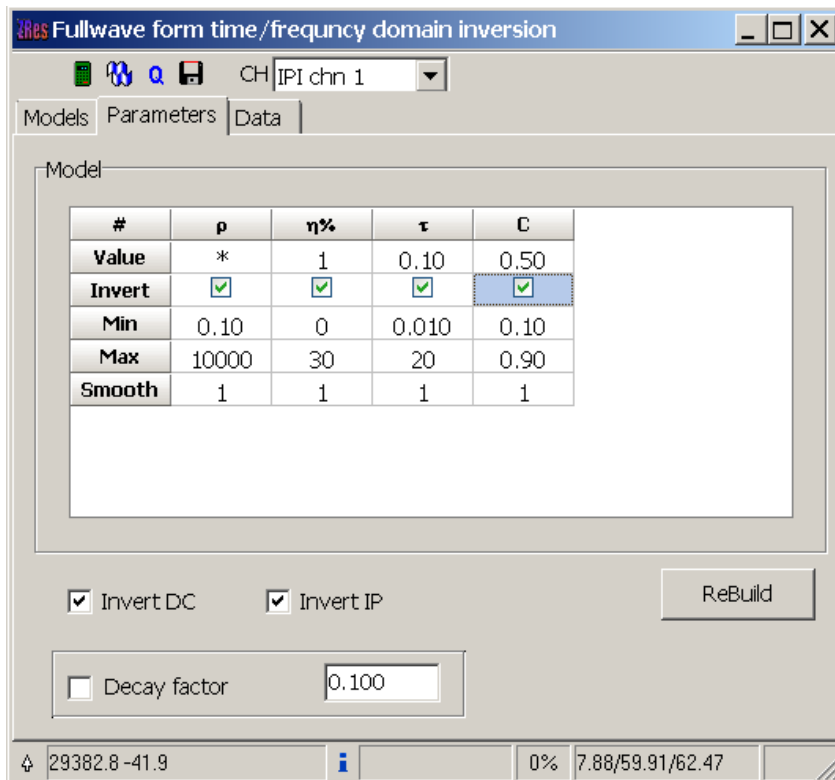


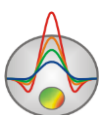
Fig. 80 Window for selecting parameters for curves IP inversion

Option **Module** (available when working in the frequency mode) indicates whether or not to select the signal module, or use phase only. As a rule, used in the frequency range module changes very little, and unlike the phase, has little effect on the inversion results.

To start inversion, press the button . This inversion occurs simultaneously on all selected parameters. Button provides an alternative method in which the polarizability inversion occurs independently for each time (frequency) by time-lapse inversion and Cole-Cole model parameters are recovering based on obtained "real" curves.

Tab **Data** displays observed and calculated polarizability model. The drop-down list in the toolbar (available when working with all the tabs) allows switching between sections for different decay times (frequencies).

Dialog **Time lapse/domain/ Digital filter** contains temporary filter settings for the Fourier transform to the time domain, and allows you to set the length of the current impulse. *Video dedicated to the IP data inversion in the time and frequency domain, can be downloaded from <http://zond-geo.ru/zmovies/TDvsFD.exe>.*



Geoelectric monitoring data inversion

Monitoring involves measurements with the same parameters at different time. The program recognizes file with the *.z2d extension as an inputting monitoring file. The file has following structure: the first four columns contain electrodes coordinates (similar to Main data file format); next - columns, entitled res1, res2, etc. that contain Res ($\Delta U/I$) values for the first, second, etc. measurement cycles.

The following example shows the beginning of a monitoring data file by three-electrode array ([sample-file – monitoringERT.z2d](#)):

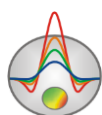
C1	C2	P1	P2	res1	res2	res3	res4	res5
0	*	2	4	5.57	5.57	5.57	5.60	5.63
0	*	4	6	1.55	1.55	1.56	1.59	1.62
0	*	6	8	0.84	0.84	0.85	0.88	0.91
0	*	8	10	0.48	0.48	0.49	0.51	0.52
0	*	10	12	0.44	0.44	0.45	0.49	0.46
0	*	12	14	0.25	0.26	0.26	0.29	0.24




Working with monitoring data, in general, is similar to the work with multifrequency IP data or temporary IP containing information about decay curve. Only instead of the polarizability values at a certain frequency or time is considered the resistivity values at a certain time.

Use the menu item **Time lapse/domain** to work with the monitoring (menu appears when you load the appropriate data file). Switching between the data and the models corresponding to different measurements times is carried out using the drop-down list on the control panel of the main window.

Option **Time lapse/domain/ Channels** runs the table in which you can enter information about each measurements implementation. An information about measurement time is inputted at the **time, s** column. Arbitrary text information, entered in the **name** column, will displayed at the drop-down list on the main window control panel (when switching between implementations).

Option **Time lapse/domain/ Display channel plot** runs the window that displays the Res ($\Delta U/I$) values for each measurement point, depending on the time (Fig. 81). To move to another



measurement use the mouse (on pseudosections or graphs) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point are displayed. A separate point of the displaying curve can be edited using appearing menu items when you press the left mouse button menu on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), returned to the previous position (**Origin point**). The toolbar button  allows you to smooth the entire curve.

Option **Time lapse/domain/ Time lapse inversion** runs consistent inversion mode of monitoring data for each implementation. If the option **Time lapse/domain/ Next from previous** is switched ON, results of previous iteration are used as an initial approximation for each next one. If this option is switched OFF, the current model in this implementation is used as an initial approximation for each next iteration. Obtained models are displayed in the model editor. To switch between models corresponding to different times use the drop-down list on the control panel of the main window.

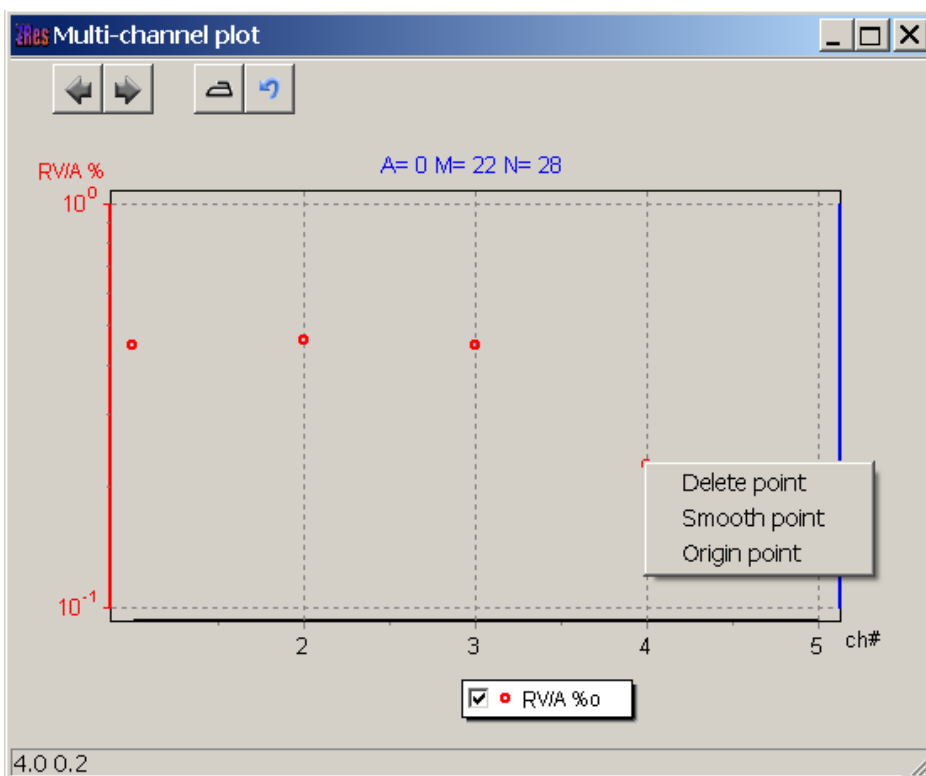
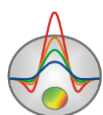


Fig. 81 Window of resistivity changes depending on the time for a single measurement point



It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Sample - video - <http://zond-geo.ru/zmovies/monitoringERT.exe>.

Magnetotelluric data inversion

If you have magneto-telluric data measurement, they can be used in inversion accounting data of resistivity method on DC. To work with magnetotelluric data measurements use the menu item **Options/MT Data**.

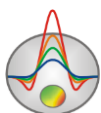
The magnetotelluric data inputting file is a file with *.m2d format. Its structure is described in the section Magnetotelluric data import.

The menu items **Load MT data** and **Show MT plot** allow, respectively, loading and visualizing data of magnetotelluric measurements. Selecting **With static shift** specifies whether static shift (normalization) MTS curves. To specify MT-measurements weight use the menu item **Set weights of MT**.

If you select the menu item **Invert MT data**, the program will carry out joint inversion of MT data and the resistivity method.

Option **DET inversion** allows carrying out inversion simultaneously both longitudinal (TE), and transverse (TM) impedance.

Joint data inversion of electrotomography and PMT is shown in the movie http://zond-geo.ru/zmovies/MT_ERT.exe.

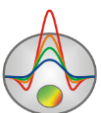


Script-files (batch processing)

When you need to repeat the same action, they can be written to a script file. When it started, the program will perform the action automatically. For example, you can record the process of opening data file, setting the starting model, the inversion parameters, starting the inversion process and exporting results. If it is necessary to process multiple profiles data by this scheme, run the script and select all data files, for each of them the corresponding actions sequence is automatically will be applied. This tool allows you to save time when the same operations repeat and allows the user to "standardize" the process or individual working process steps with data.

To start recording a script file select the menu item **Script/Start recording script** before performing operations that need to be remembered. If script recording is currently active, the tick will be displayed on the left of the specified menu item. To stop script recording, use the menu item **Script/Stop recording script**, then it is necessary to save the script file (**Script/Save script file**).

To run the script file, it must be loaded **Script/Load script file** and using the menu item **Script/Run packet processing** select the data file for processing. When you save a file processing results in automatic mode, the results file name - the source file name with underscore addition.



Appendix 1: Graphics set editor editor).

Click on legend with SHIFT button pressed to select one graph and delete all others. Second click performs inverse operation.

Use mouse wheel to scroll graphs. In order to do that select a few adjacent graphs (on the legend) and scroll mouse wheel with mouse cursor in the centre of the legend. Indices of active graphs will change. Right click on plot point to select value in the table.

Graphics mode allows rejecting some values that will be assigned 0 weights. To reject certain value left click on plot point with ALT button pressed. Right click with ALT button pressed on graphic to set 0 weights to all points of the graph.

If the values of the measurement weights are given in the input data file, you can display an appropriate range of the resistivity values in the graphs (CI) using the option **Options/Data/Display/Error gates**. Confidence intervals (weights) can be edited in the graph mode by pressing the right or left mouse button while holding down the ALT button.

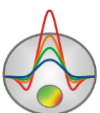
Pseudosection

In the first approximation pseudosection visualizes parameter distribution with depth along profile.

Contour graph is created in profile coordinate to pseudodepth axes. Color scale correlates value to color.

Double click next to object axes runs context menu which contains the following options:

Log data scale	Use logarithmic scale on color bar.
Smooth mode	Use smooth interpolation palette/contour section.
Display grid point	Display measurement point ticks.
Display ColorBar	Display color scale.
Setup	Run pseudosection parameters setup dialog.
Print preview	Print pseudosection.
Save picture	Save pseudosection in graphic file *. emf.
Save XYZ file	Save pseudosection in data-file.
Default	Set pseudosection parameters on default.



Pseudosection parameters setup dialog **Setup** is described in [Appendix 2](#): Graphics editor. Right click on necessary axis with SHIFT button pressed to run axis editor (see [Appendix 5](#): Axes editor). Right click on necessary point with SHIFT button pressed to run pseudosection point editor (see [Appendix 7](#): Pseudosection points editor).

Pseudosection plotting method differs for various arrays. Pseudosection point position (is shown on Fig. 34 in the part «[Creating survey layout in ZondRes2D dialog](#)»):

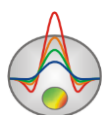
1. for Wenner array corresponds to array center on X axis and to $\frac{1}{4}$ of separation between the last electrode and array center on Y axis;
2. for Pole-Dipole array - center between potential electrodes on X axis and separation between current electrode and center of potential electrodes on Y axis;
3. for Dipole-Dipole array - array center on X axis and $\frac{1}{2}$ of distance between dipole centers (for current and potential electrodes) on Y axis;
4. for Gradient array – center between potential electrodes on X axis and $\frac{1}{2}$ of separation between nearest current electrode and center of potential electrodes;
5. for Pole-Pole array – array center on X and Y axes;
6. for VES array – array center on X axis and $\frac{1}{4}$ of distance between the last electrodes on Y axis.

If data is displayed as pseudosection, use **Options/Data/ Display/** menu option to view certain array data. In case of having more than 3000 observed values **Display every N point** option can be used to subsample pseudosection points.

Value weights (**Options/Data/Data Weights**) and relative misfit of every measurement after inversion process finishing (**Options/Data/Data Misfit**) can be displayed as pseudosection in the middle graphic section of the program.

Additional preparation of data for inversion

In general, preparing data for inversion includes the following procedures set (if it is necessary): field data quality assessment, removing low-quality measurements, weight



measurement assignment, depending on their quality, the combination of several arrays in one profile, data filtering, input topographical information.

Estimation of filed electrotomography data can be performed on two levels: analysis of standard statistics received during signal processing, analysis of measurements results distribution (voltage, current values). The first level of estimation is usually carried out in the programs, working with array complexes. The second level is there as a quality control and data processing tools in ZondRes2D program.

Electrotomography uses much more information in comparison with classic profiling and sounding. For this reason it is acceptable to use data that contains some rejected values for inversion.

What matters most is not the percentage of errors, but the distribution of this error in the section. There are two cases: uncorrelating noise (separate spikes), the overall noise high level and correlating noise in any section part.

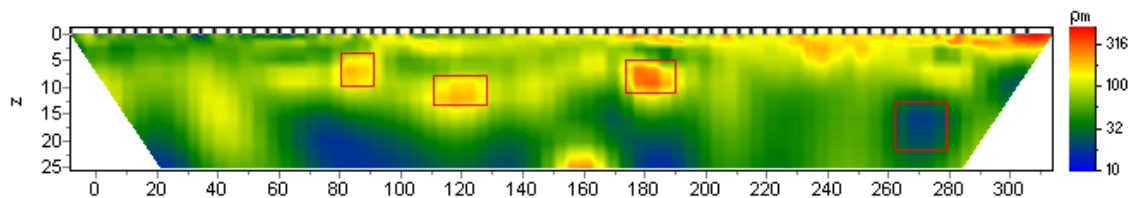
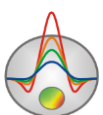


Fig. 13 The section of the electrical resistivity with false anomalies due to the inversion of low-quality field data

In the first case (uncorrelated noise, spikes) low-quality measurements presence can have no effect on interpretation results, but sometimes it causes significant model distortion. It was repeatedly noticed that small amount of poor data led to formation of false anomalies as “compensated dipoles” system (Fig. 13). It results in interchanging of local anomalies of relatively low and high resistivity. In this case using adaptive robust systems is not helpful, only total deletion of poor data from inversion is effective.

The second case (noisy) requires data filtering. This will significantly reduce the noise level, but may lead to a reduced resolution.

The third case (correlating noise in any section area) is more complex problem because for data processing it is necessary to understand the cause of poor data quality in this area. For example, this reason may be poor quality of some electrodes earthing, systematic external noisy

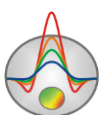
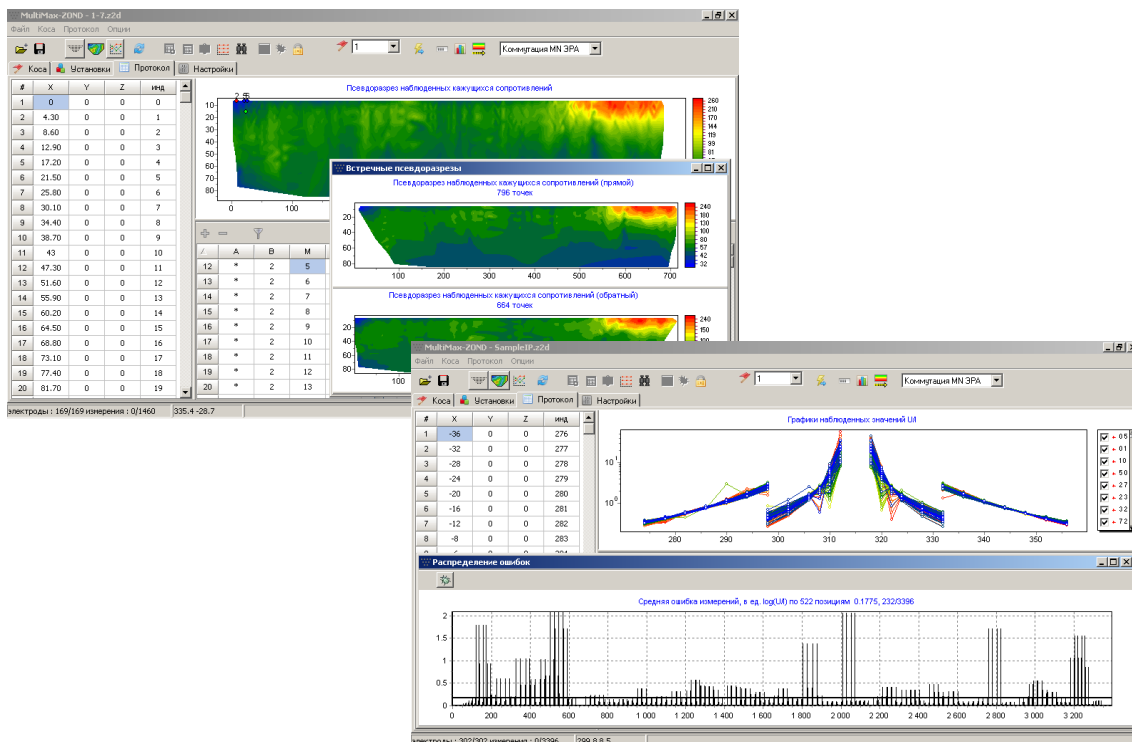


signal, low signal level, etc. Thus, we should at first, as far as possible, identify the cause of poor quality, and then find a way to remove it. To solve this problem there are variety analysis tools for source data in the program.

In preparing data for inversion it is necessary to remember that the worse data quality, the less reliable the inversion results will be. Accordingly, if for any section area quality is poor or data are not available, the inversion result of this area may not have anything to do with reality.

In ZondRes2D preparing data for inversion is carried out using a special tool of quality control and editing data **Options/Quality control module**, the electrodes editor **Options/Data editor** (is described in "[Electrodes editor](#)") and graphics plot of apparent resistivity **Options/Data/Graphics plot** (described in "[Graphics pl](#)").

An alternative possibility for analysis and editing of electrotomography data provides **ZondProtocol** program of Zond package. The program solves the following problems: specifying measurement protocols(automatic and interactive), visualization and analysis of measurement results, separate displaying arrays, data conversion in various arrays, import and export, preparing data for inversion. Due to its usability and compatibility for **ZondRes2d** program it allows receiving good results right on the profile. (Fig. 14).



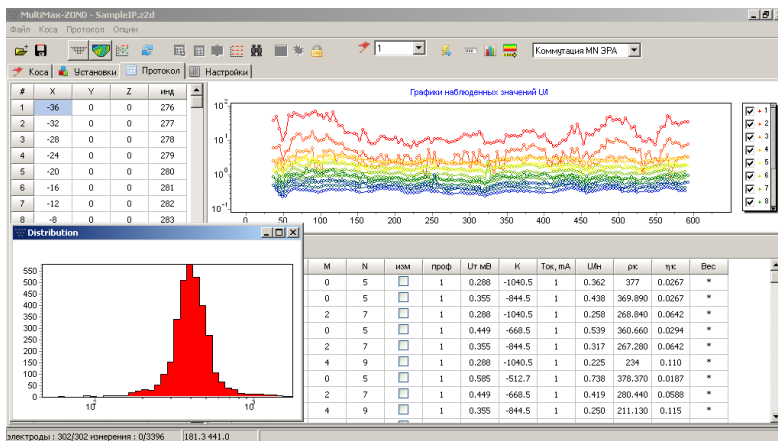


Fig. 14 ZondProtocol working windows

Module of data quality control

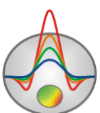
For quality control and data editing a special module is available in the ZondRes2D program. To run it, select in the menu **Options/Quality control module**. The module allows performing the following operations with data:

1. Visualize measurement results as of graphs of various parameters and as different arrays data pseudosections;
2. If it is possible, estimate data quality (determine measurement weights) based on the principle of reciprocity;
3. Delete the data directly on the graphs of displayed parameters;
4. Edit data manually and automatically;
5. Perform data smoothing.

The ability to group data by arrays, signal levels, different geometric parameters, measurement weights with the options of manual and automatic editing allow realizing entire cycle of data quality control while preparing data for inversion.

Window of quality control module contains tools and three tabs – graphs of displayed parameters (**Data graphics**, Fig. 15), selecting parameters to display (**Settings**) and pseudosections (**Pseudo**).

Tab of selection displaying parameters allows you to select an array with data that will be produced work (**Electrode array**), to specify parameter of data grouping for the selected array



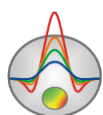
(by diversion, position of supply and receiving electrodes, etc., **Graphyc Type**), and displayed value - normalized signal, the apparent resistivity or apparent polarizability.

Graphs of the selected parameters are displayed on the **Data graphics** tab. You can work with data in an interactive mode using the toolbar. Measurements are displayed by points on the graphs. The measurement corresponding to electrodes position is displayed at the bottom of the graphs window when choosing measurement point by left mouse button (Fig. 8). The parameter values corresponding to the selected measurement are displayed when moving mouse to the corresponding graph point.



Fig. 15 Module of quality control - window of displaying parameters graphs

Zooming in or moving part is performed with left button pressed (the tool - "rubber rectangle"). To select an area, which you want to zoom in, move the cursor down and to the right, with the left button pressed (Fig. 16A). To return to the original scale, do the same actions, but the mouse is moving up and to the left (Fig. 16B). To move (scroll) graphic, move the mouse with the pressed right mouse button.



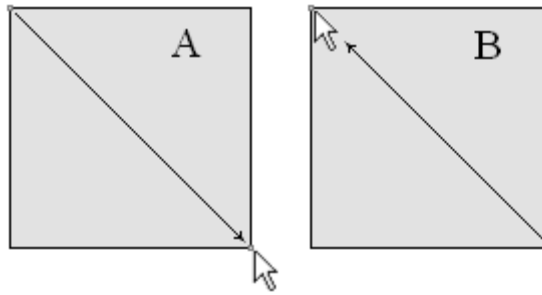













Fig. 16 The direction of mouse movement when zooming

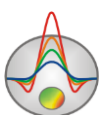
To shortly display only one graph left-click on the graph line - while other graphics will be invisible. To display one or more selected graphs use graph list in the right window part. If graphs are active (inactive), press the Shift key to select one graph and make all other graphs inactive (active).


Data quality can be controlled in three ways - based on measurement weights (estimated during measuring or calculated using reciprocity principle), visual by graphs smoothness and based on a comparison of source data and the direct problem solution from inversion model.


Displaying of measurement weights regulated by toolbar buttons: when pressing the button , weights are not displayed, pressing the button  weights (determined based on the reciprocity principle) are displayed, when the button  is pressed - weights specified in the data file (in this and the previous case, the green color of graph points corresponds to a good quality measurement, yellow - average, red - bad).


Displaying of calculated graphs for inversion model (drawn by a dotted line) is regulated by the button . We can assume that source data points, which are not coincide with calculated, have of low quality - they make sense to be removed or edited, and then carry out the inversion again.


To remove low-quality data highlight corresponding points when the button  is pressed (points selection mode). To highlight points right-click or each separately (must be active the instrument ) or on the graph plot area (tools  or ). Selected points on the graphs are displayed by crosses. To cancel the selection use the tool , similar to described one above. Work with pressing the button  makes the selected points unselected and vice versa. To hide/show the selected points press the button .




When pressing the button , it is possible to edit measurements - by left mouse button, "drag" the point up or down to the desired value.

To set graphs displaying parameters use the button (parameters of lines and points - the thickness, color, etc.). Use the button  to fix the axes change limits to they have not changed automatically in the process of working with data.

The tool  allows smoothing data. Each press of the button increases the degree of smoothing.



Click the button  to update pseudosection (tab **Pseudo**), it does not happen automatically.

Changes, which are made to the data while editing in the QC module, are not saved by default.

To save the changes and move to the inversion press the button .

Examples of working with data quality control module are shown in the movies http://zond-geo.ru/zmovies/ert_qc_denoising.exe and <http://zond-geo.ru/zmovies/editsmoothert.exe>.

Data inversion

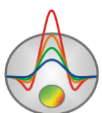
After loading the data file, setting the starting model, data editing and input topographical information it is necessary to select the inversion type and set inversion parameters. Also, if it is necessary, it is possible to edit and change of the starting model resistivity values (described in the “[Modeling and working with model](#)”). The program supports two basic modes of inversion: in the block model (if this option  is turned on), and in layered medium model (if this option  is turned on). To run inversion parameters setup dialog click the button or use menu item **Option/Program setup**.

Inversion parameters setup dialog

This dialog serves for specifying parameters connected with forward and inverse problem solutions.

Default – set parameters default values.

The main inversion parameters set in the tab **General** (fig. 17).



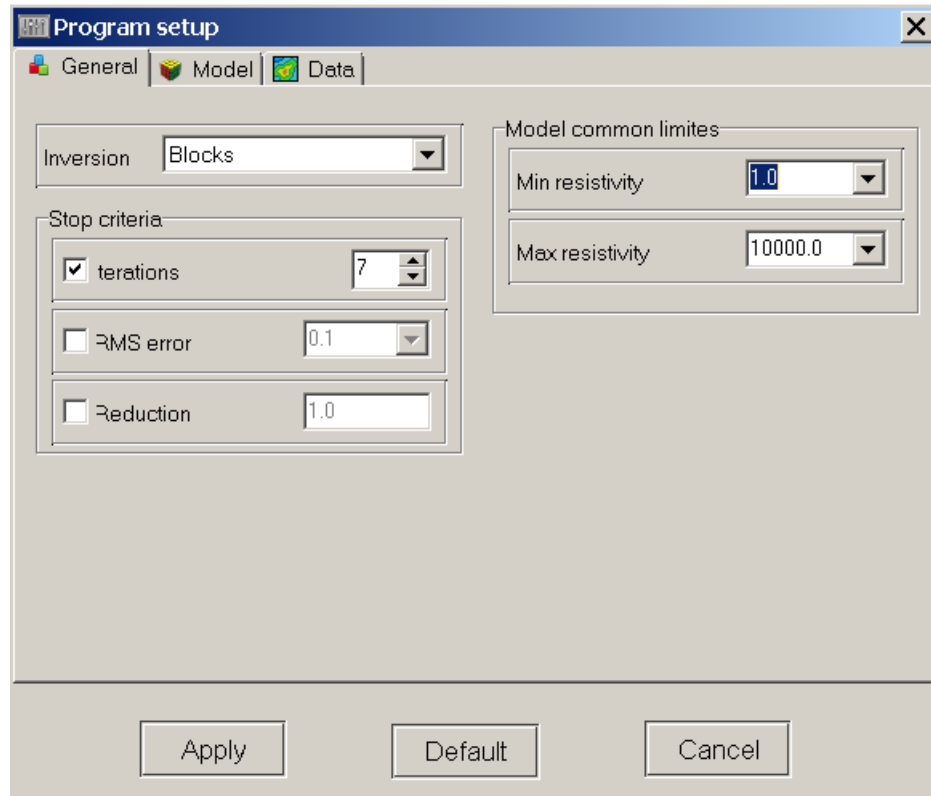


Fig. 17 **Program setup** dialog window, **Inversion** tab

Inversion option defines algorithm that is used for inverse problem solution.

Let us consider inversion algorithms by example of subsurface model that consists of several blocks (fig. 18).

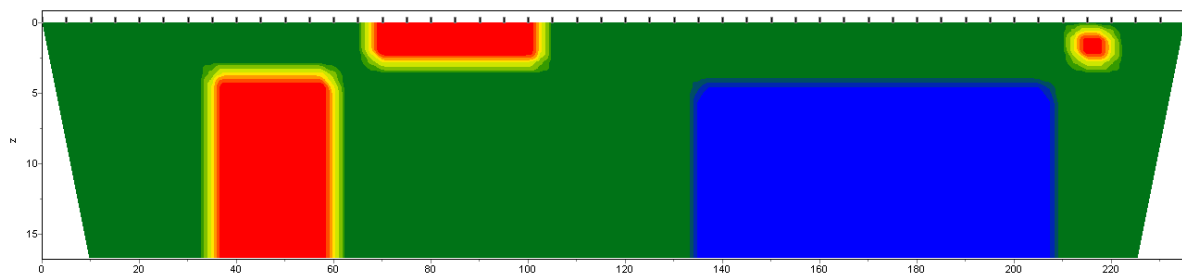
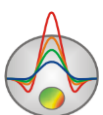


Fig. 18 Test subsurface model

For algorithm testing theoretical response should be calculated and 5 percent Gaussian noise superimposed.



Smoothness constrained is inversion by least-square method with use of smoothing operator. As a result of this algorithm smooth (without sharp boundaries) and stable parameter distribution is received (fig. 19).

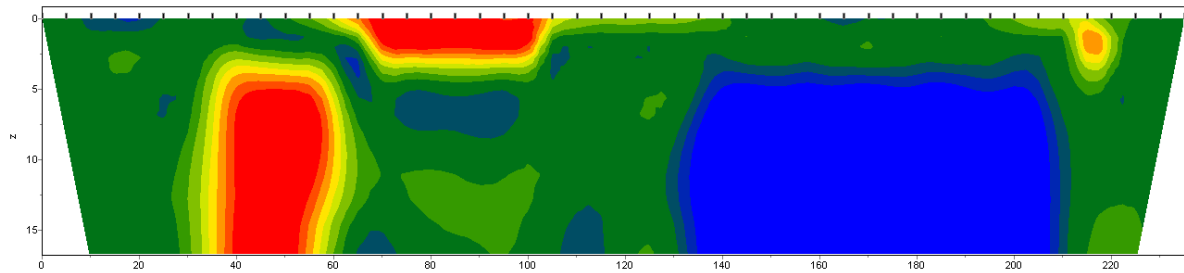


Fig. 19 Regenerated model as a result of *Smoothness constrained* inversion

Matrix equation for this kind of inversion is the following:

$$(A^T W^T W A + \mu C^T C) \Delta m = A^T W^T \Delta f \quad (10)$$

Judging by the equation it can be said that model contrast is not minimized during inversion. Current algorithm allows receiving minimum misfit values. In the majority of cases it is recommended to use it for initial stages of interpretation.

Occam is inversion by least-square method with use of smoothing operator and additional contrast minimization [Constable, 1987]. As a result of this algorithm the smoothest parameter distribution is received (fig. 20).

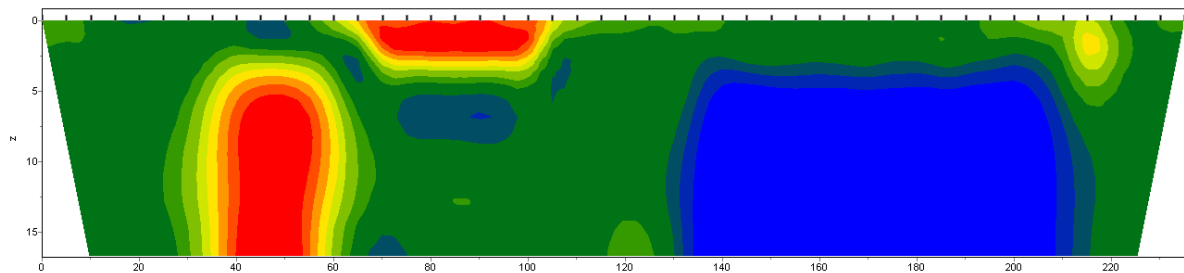
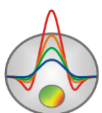


Fig. 20 Regenerated model as a result of *Occam* inversion

Matrix equation for this kind of inversion is the following:

$$(A^T W^T W A + \mu C^T C) \Delta m = A^T W^T \Delta f - \mu C^T C m \quad (11)$$



Degree of smoothness of received model is in direct proportion to **Smoothness factor** value. It should be noted that high values of this parameter can lead to misfit increase.

Marquardt – classic inversion algorithm by least-square method with regularization by damping parameter (**Ridge regression**) [Marquardt, 1963]. In case of little quantity of section parameters this algorithm allows receiving contrast subsurface model (fig. 21).

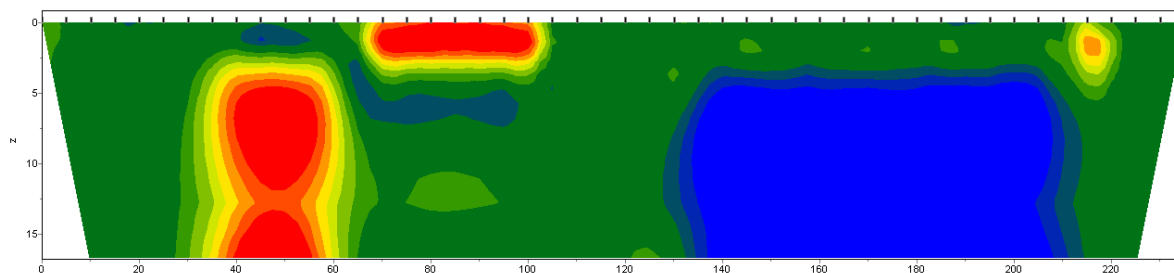


Fig. 21 Regenerated model as a result of *Marquardt* inversion

Matrix equation for this kind of inversion is the following:

$$(A^T W^T W A + \mu I) \Delta m = A^T W^T \Delta f \quad (12)$$

Unwise usage of this inversion method modification can lead to receiving unstable results and increasing of RMS deviation, that is algorithm discrepancy.

The best option is to use *Marquardt* method as specializing (for misfit decrease) after *Smoothness constrained* or *Occam* inversion is performed.

Focused is inversion by least-square method with use of smoothing operator and additional contrast focusing [Portniaguine, 2000]. As a result of this algorithm piecewise smooth parameters distribution (that is model which consists of blocks with constant resistivity) can be received (fig. 22).

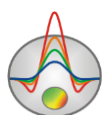
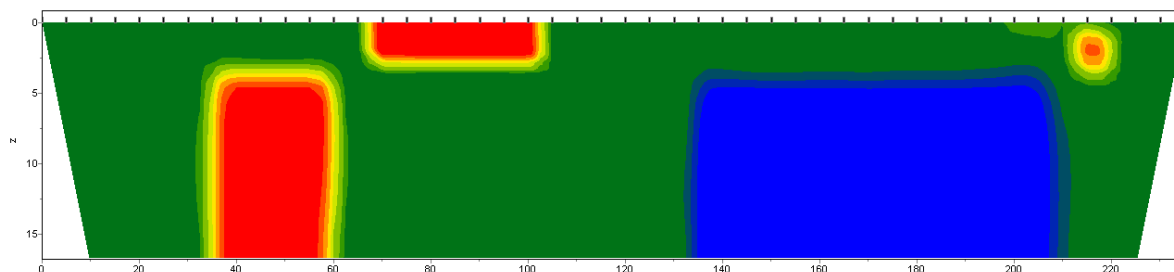


Fig. 22 Regenerated model as a result of **Focused** inversion

Matrix equation for this kind of inversion is the following:

$$\left(A^T W^T W A + \mu C^T R C\right) \Delta m = A^T W^T \Delta f - \mu C^T R C m \quad (13)$$

In case of using this type of inversion threshold contrast parameter **Threshold** (it can be set in the tab **Model**) should be carefully selected. This parameter defines threshold contrast value for adjacent cells, if it is reached adjacent cells parameters are not averaged (it is considered that there is a boundary between these cells). Dependence of averaging degree (or weight) of two adjacent cells R_i on contrast threshold e and contrast between these cells r_i is the following:

$$R_i = \frac{e^2}{e^2 + r_i^2}. \quad (14)$$

Blocks – fits parameters for certain domains which differ in resistivity. Domains with equal resistivity consider as single blocks (fig. 23).

Matrix equation for this kind of inversion is the same as for **Marquardt** algorithm:

$$\left(A^T W^T W A + \mu I\right) \Delta m = A^T W^T \Delta f \quad (15)$$

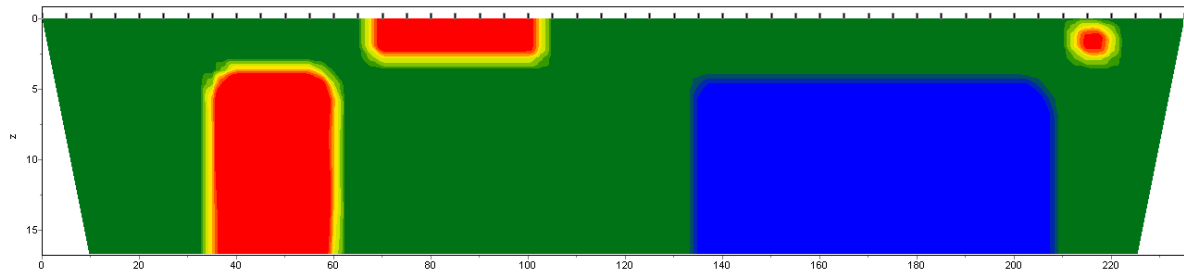
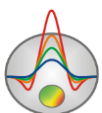


Fig. 23 Regenerated model as a result of **Blocks** inversion

It is recommended to use this algorithm for more precise definition of already received results of preceding methods (**Focused** is the best option) after merging cells in necessary blocks using **Cell summarization** function. Model editor should be used to select certain blocks



manually: certain domains should be assigned with different parameters. Certain domains will be highlighted by boundary while working with this dialog window.

ProfileR – use the program ProfileR A. Binley for inversion

(<http://www.es.lancs.ac.uk/people/amb/Freeware/Profiler/Profiler.htm>).

Stop criteria field contains inversion stopping criterion.

Iterations – if this function is ON inversion process stops as soon as specified iteration number is reached.

RMS error – if this function is ON inversion process stops as soon as specified RMS level is reached.

Reduction – if this function is ON inversion process stops as soon as RMS error increases repeatedly for two sequential iterations (on selected value, in percents).

Field **Model common limits (Min resistivity, Max resistivity)** – set model parameters limits of variation for inversion.

Tab **Model** (fig. 24) contains additional inversion settings

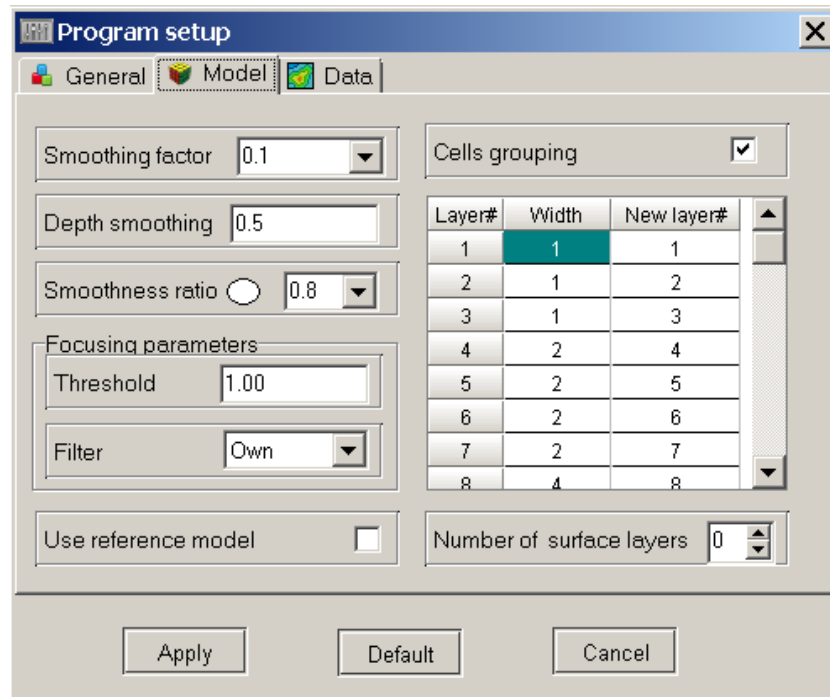
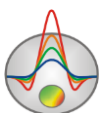


Fig. 24 Inversion parameters setup dialog. Tab Model

Smoothing factor sets dependence of measurement misfit minimization on model misfit. In case of noisy environment or in order to receive smoother and more stable parameters



distribution quite high smoothing parameter value is chosen: 0.5 – 2.0; 0.005 – 0.1 values are used for high quality data. High smoothing parameter values mostly lead to high data misfits (fig. 25). Smoothing factor is used in *Occam* and *Focused* inversion algorithms.

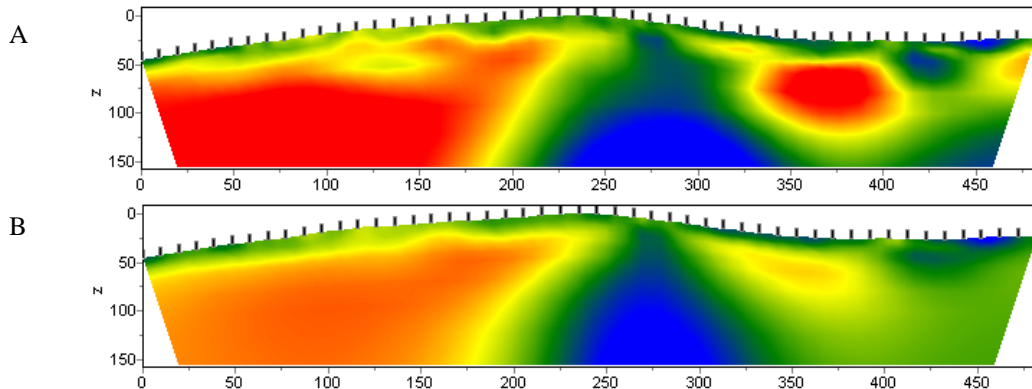


Fig. 25 Geoelectric models as a result of *Occam* inversion with **smoothing factor**: 0.01 (A) and 1.0 (B).

Resultant misfit for case A – 4.5 %, case B – 6 %.

Robust weighting scheme – this option should be turned on if there are individual high deviations caused by systematic measurement errors. It is possible that current algorithm will not give positive results if amount of rejected data is comparable to amount of high quality data.

Smoothness ratio – specifies smoothness ratio in horizontal and vertical direction. Set this parameter larger than 1 for vertically-layered subsurface and smaller than 1 for horizontally stratified earth. Usually this parameter value ranges from 0.2 to 1 (fig. 26).

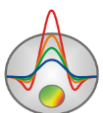
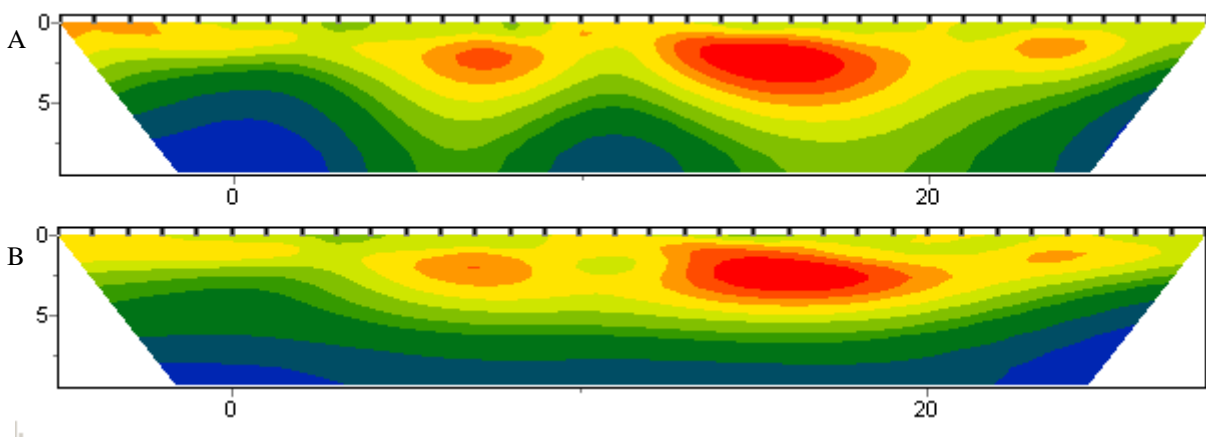


Fig. 26 Geoelectric models as a result of “smooth” inversion with **Smoothness ratio** parameter:
1 (A) and 0.3 (B)

Focusing parameters field

Threshold – defines threshold contrast value for adjacent cells, and if it is reached adjacent cells parameters are not averaged (it is considered that there is a boundary between these cells). This parameter value is chosen empirically (0.001-1) (Fig. 20). Small parameter value can cause algorithm discrepancy (in this case it needs increasing). Large value leads to receiving smooth distribution.

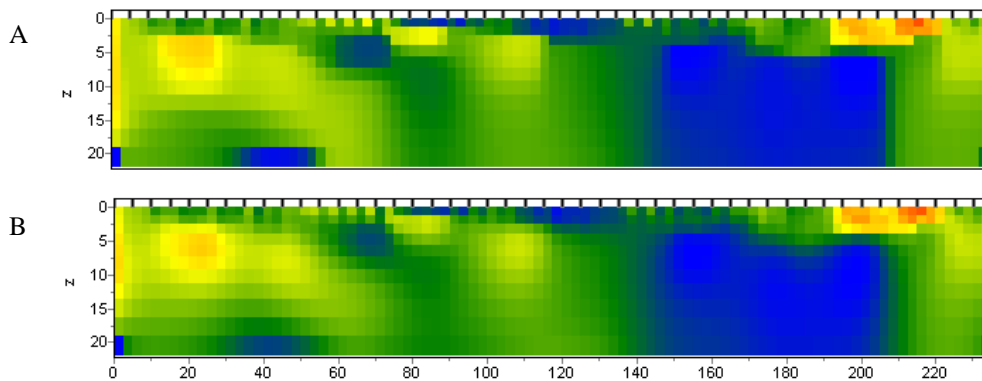
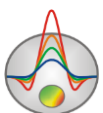


Fig. 27 Geoelectric models as a result of **Focused** inversion with **Threshold** parameter:
0.01 (A) and 0.1 (B)

Sharpness – defines ratio between minimization of causative body volume (0) and construction of piecewise-smooth distribution (1) (fig. 28). Value of this parameter is chosen empirically (0.7).



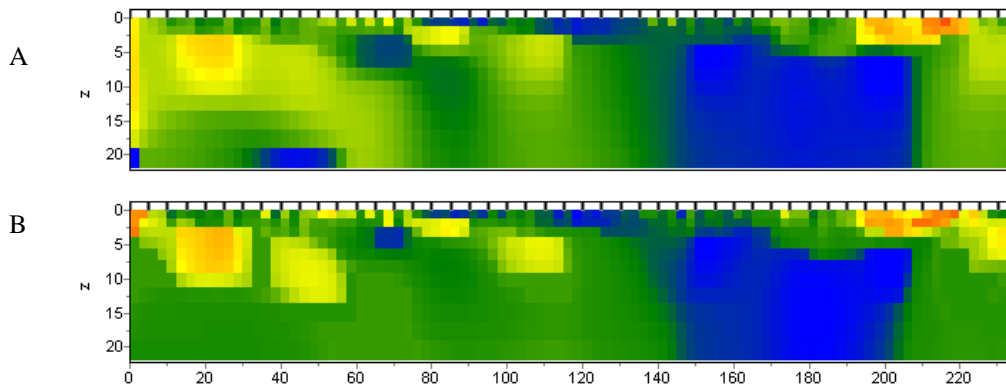
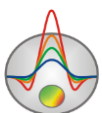


Fig. 28 Geoelectric models as a result of *Focused* inversion with **Sharpness** parameter: 0.8 (A) and 0.2 (B)

Filter – defines mechanism of focusing filter construction. If **Own** filed is selected, filter is constructed using current parameters (in current interpretation mode). If **Other** is selected, focusing filter based on other medium parameter is used. For example, resistivity model with boundaries which correspond to our idea about medium parameters is received. In order to create polarizability model with the same boundaries go to induced polarization method in data interpretation mode, and choose **Other** and **Threshold** value that corresponds to expected polarizability section specifics.

Use reference model – use starting model as a priori. If this function is ON resultant model will not differ much from the starting one (fig. 29). It is possible to use starting model for inversion using *Occam* and *Focused* algorithms. Discrepancy between starting and final model can be regulated by changing ratio between minimization value misfit and model misfit. It means that the lower smoothing parameter value is the more significant discrepancy between starting and resultant model may be.



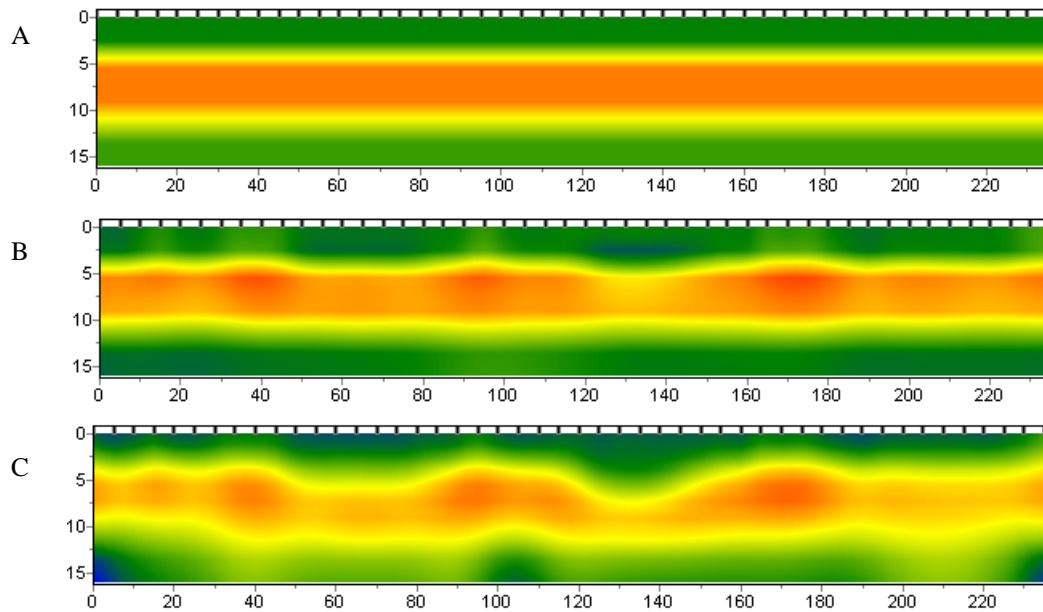
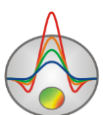


Fig. 29 Starting geoelectric model (A), geoelectrical model as a result of *Occam* inversion with use of starting model (B) and without it (C)

Cell grouping – use this option in the majority of models. It activates table that allows merging adjacent cells and receiving less determinate parameters for inversion. If this option is used number of cells for forward solution remains the same but number of cells for inversion decreases. Ideally, number of determinate parameters should be close to amount of data.

The table contains three columns. Number of initial model layer is specified in the first column (**Layer**). Layer number of inversion mesh is set in the third column (**New layer**). Number of cells (in horizontal direction) that contain in every cell of inversion mesh for current layer is set in the second column (**Width**). Inversion mesh is displayed in model editor during setup. Double left click on cell in **Width** column to merge cells in horizontal direction for current layer, right click – run dialog, in which it is necessary to specify number of merging layers. It will be applied for the current and all underlying layers. Double left click on cell in **New layer** column to merge cells in vertical direction for current layer, right click – for current and all underlayers.

There are three examples of inversion meshes below: in the first picture inversion mesh corresponds to model one (fig. 30 A); in fig. 30 B cells starting from the second layer merge by two; in the third picture cells starting from the second layer merge by four (fig. 30 C).



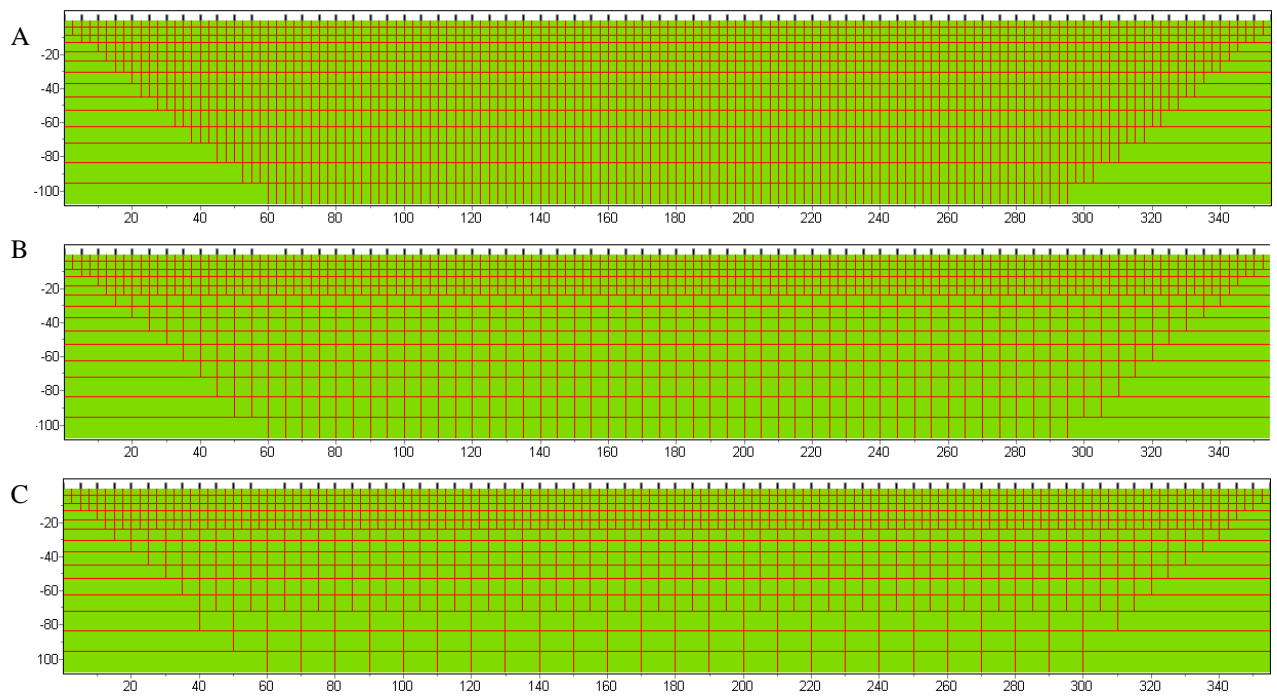
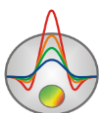


Fig. 30 Examples of inversion meshes

The similar functions are run by the option **Options/Cutting**. If the **Cell grouping** is created to merge cells in the "middle" profile part, then **Options/Cutting angle** allows doing this for the edge section fields.

Number of surface layers (0-2) – sets number of layers in which strong variation of parameters is possible. Use 1 or 2 layers if near-surface section is very heterogeneous and it has negative effect on the whole section. If this option is used the majority of heterogeneities are concentrated in hear-surface layer and bottom part of the section is smooth after inversion.

The third tab **Data** serves for setting additional inversion parameters, associated with the way of working with data.



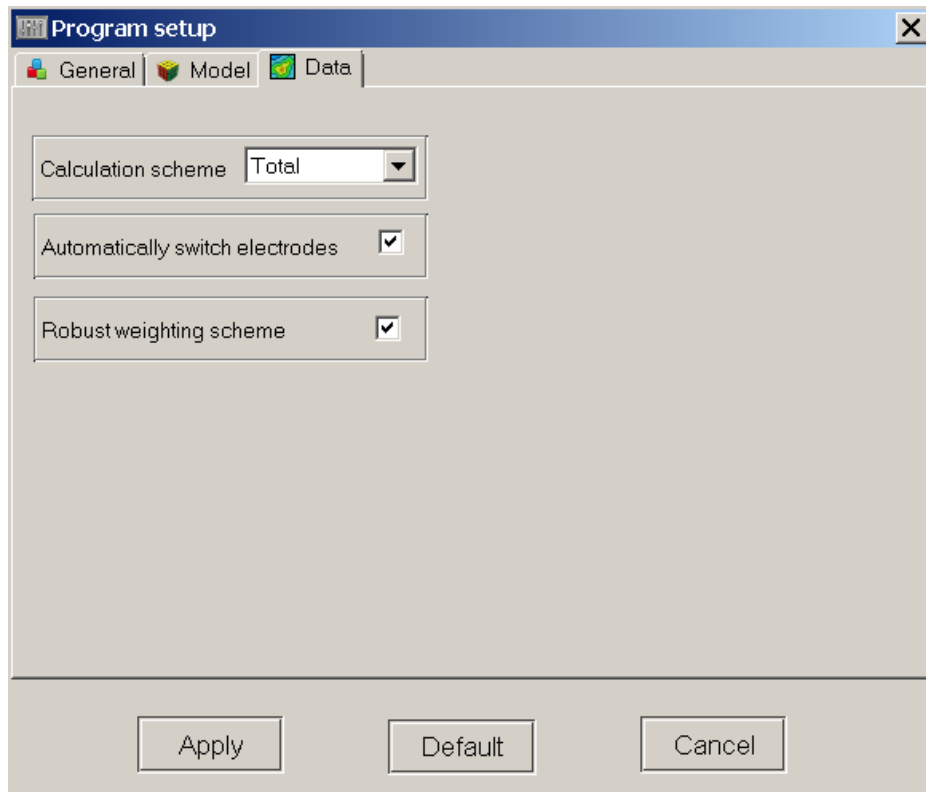


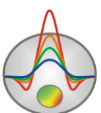
Fig. 31 **Program setup** dialog window. Tab Data

Calculation scheme (Secondary, Total) defines algorithm of electric potential calculation. Secondary potential is calculated if **Secondary** is chosen. This method is slower and does not allow taking surface topography into consideration but gives quite exact results if fairly crude mesh is used. More detailed mesh (1 or 2 nodes between adjacent electrodes) and external model boundary shift further is necessary for total potential calculation (**Total**). It is due to low accuracy of potential determination near current electrode. Secondary mode should be used when grid is irregular, for example, when you work with VES data.


Automatically switch electrodes – if this function is ON current electrodes are automatically changed over in order to receive positive values of array factor.


Robust weighting scheme – this option should be turned on if there are individual high deviations caused by systematic measurement errors. If number of waste data is comparable with quality measurements the algorithm may give no positive results.

Before inversion sometimes it is necessary to input a priori information. It can be starting model based on geological data, data of other geophysical methods or result of previous



inversion. The ability of inputting a priori information in ZondRes2D program are described in the part [Incorporating a priori information in the inversion](#).

After setting the parameters and inputting a priori information inversion process is started by pressing the button  on the toolbar. Inversion process is displayed in the status bar. For more information about inversion process use the menu item **Options/Inversion/Optimization/Display process**. During inversion changing of geoelectric section occurs in the model window.

Option  on the main program window toolbar allows making one-dimensional inversion.

A large inversion parameters number allows you to receive a plurality of equivalent models. When interpreting, they should be compared at least to select the most real model based on geology.

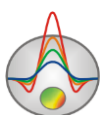
A handy tool for working with multiple models in a single project is the function Buffer of main menu. It provides access to the models buffer, in which it is possible to copy and from which paste model. The button **Buffer/Open** allows you view in a single window all created models. That is useful to compare them. More details of working with models buffer is described in [Working with multiple models in a single project](#).

Incorporating a priori information in the inversion




Inversion can be carried out without a priori information (pure transformation) and taking into account prior information (in this case interpretation element is added to the transformation). The incorporation of a priori information in the inversion is most often performed in two ways: by specifying the starting model (hypothetical choice of resistivity section) or by the tear boundaries position.

By default, the starting model for the inversion is the current ZondRes2D model, which is displayed in the model editor. It will be changed in the inversion. Model obtained in the inversion depends from the model type of starting model. The similarity degree is determined by the parameter Smoothing factor and by option Use reference. The higher the Smoothing factor value, the less the result will be different from the prior model.

Geoelectric model obtained in the one-dimensional data interpretation may be used as a priori data in inversion. This is especially valuable in the case of high vertically model contrast:



1D interpretation in this case works much better, allowing more effective vertically "squeeze" section. An example of inversion with reference model, obtained by 1D interpretation, is shown in <http://zond-geo.ru/zmovies/referencemodel.exe>.

Contrasting boundaries position, obtained by borehole and other geophysical methods is set by using the toolbar, called by the menu item **Options/Inversion/Set boundaries**. Boundaries editing and accounting in the inversion is made when the button  is pressed. To add the boundary points left-click mouse button while pressing the button . To end adding boundary points right-click. Then it is possible to start adding the next boundary or starting inversion. When defining boundaries it is recommended that they, whenever possible, do not cross cell edges. To remove the boundary right-click with the button  being pressed. Boundaries can be saved and loaded.

Example of models obtained by inversion without and with taking into account boundaries shown in Fig. 32.

Movies showing the process of taking a priori boundaries in inversion are available on page <http://zond-geo.ru/examples/video-lessons/> (<http://zond-geo.ru/zmovies/Boundaries.exe> and <http://zond-geo.ru/zmovies/BoundaryInversion.exe>).

Program allows displaying a priori information on the section, which significantly facilitates the creation of a resistivity reference model or boundaries setting. A priori information displaying process is described in "[Additional information visualization](#)".

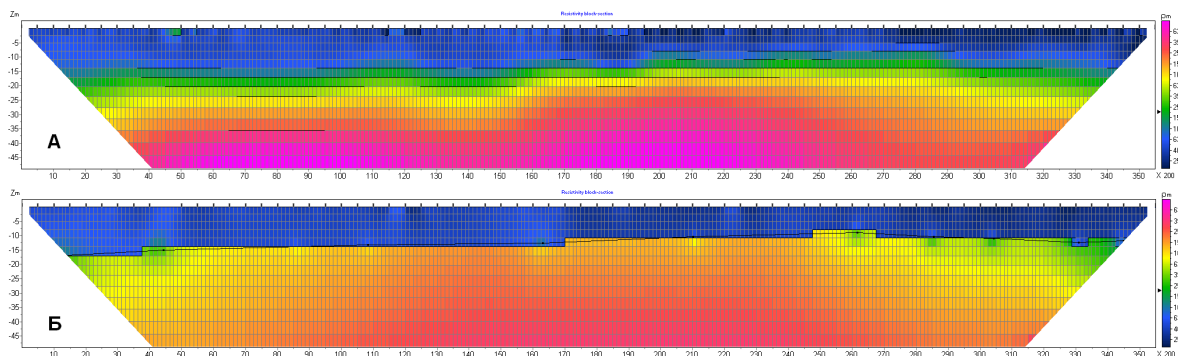
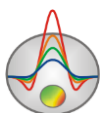



Fig. 32 A - model without inputted boundaries. B – Model with inputted boundary in the inversion



Inversion in layered medium

Option  of the main program window toolbar allows making inversion in arbitrarily-layered medium model. In this mode, user determines layers number in the resulting model and sets parameters that determine the rate of boundaries position changing and defined layers properties.

To run starting model setup, use option **Options/Mesh constructor**, as in the grid mode (Fig. 33).

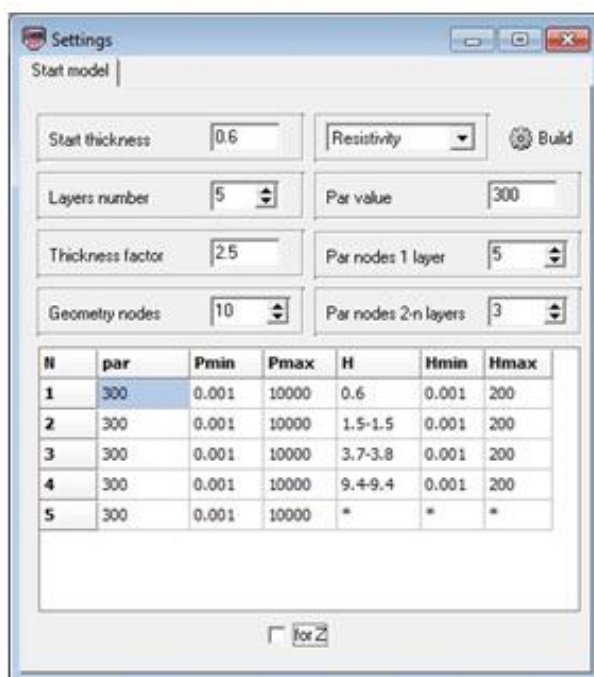
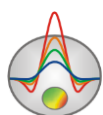


Fig. 33 Boundaries settings dialog in the mode of working in layered model

Start thickness parameter determines the first layer thickness, **Layers number** - the number of layers, **Thickness factor** - magnification factor of the underlying layer thickness. **Geometry nodes** specifies the number of points, which are contained boundary. During the inversion program will change the boundary form due to the vertical shift of these points. Accordingly, the more points, the more "smooth" boundary will be available as a result of inversion.




Par value defines the resistivity value of the first layer. Parameter **Par nodes 1 layer** defines the boundary nodes (points) number, in which selecting parameter (resistivity) can be changed. If set to one, the resistivity will be the same for all layers. If the value is two, the resistivity will vary linearly from the left layer boundary to the right. The same parameter for other layers is set in the field **Par nodes 2-n layers**. To apply described above parameters changes to the model, press the button .

Table of starting model setup window is used to display and modify parameters for each layer separately. Activate option **for Z** to jump thickness values to the depth values.

Set boundaries nodes positions can be edited manually by moving them with the left mouse button pressed in the model editor of the main program window.

Example of inversion results in the layered model is shown in Fig. 34 and Fig. 35.

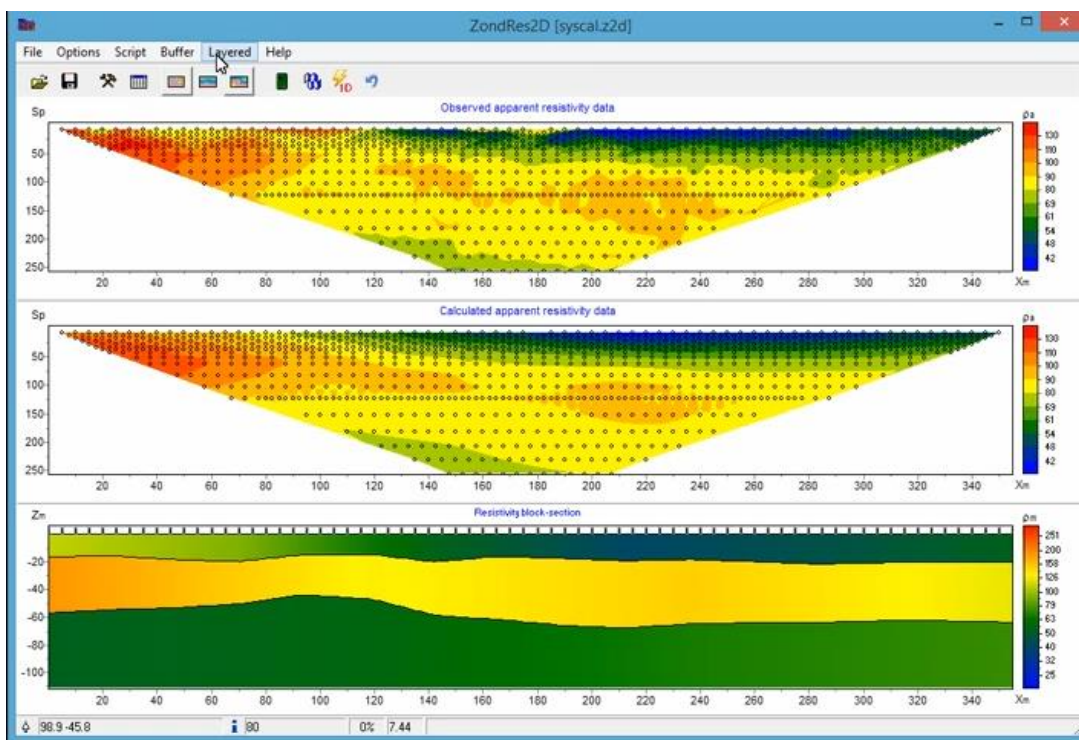
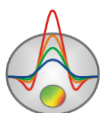


Fig. 34 Inversion results in the layered model



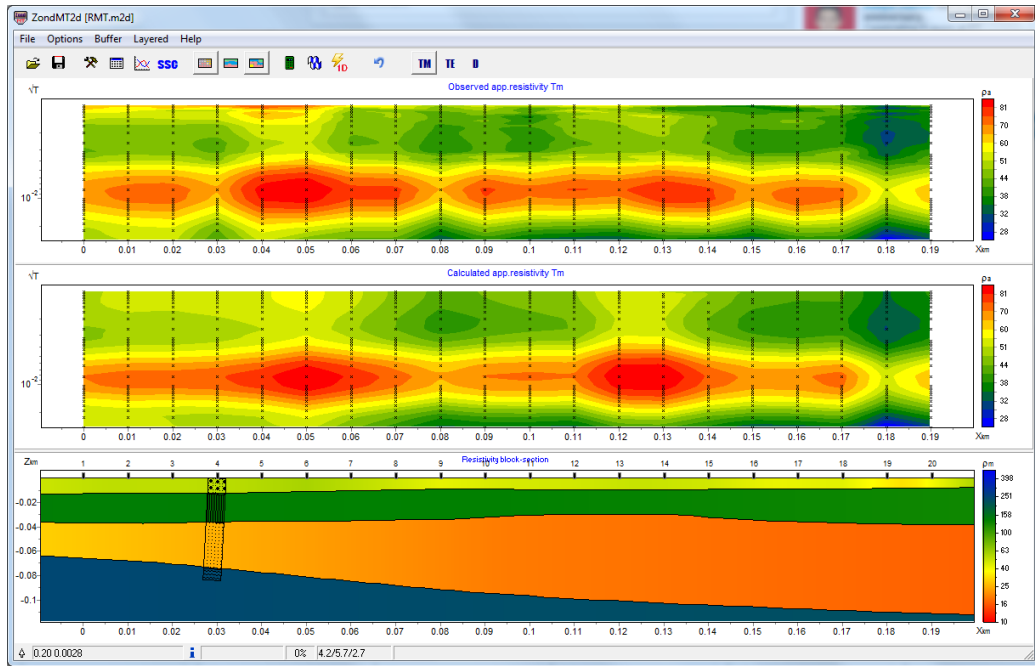


Fig. 35 Inversion results in the layered model

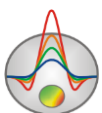
Misfit estimation as a result of inversion

Rapid assessment of inversion results can be given by relative misfit value in the program status bar. As a rule, the "average" data the value must not exceed 5%.

Convergence in each measurement between observed and calculated values can be estimated by displaying pseudosection of relative misfits using the option **Options/Data/Data Misfit** (fig. 36).

Misfit of each electrode can be viewed using the option **Options/Extra/Display electrode RMS** (fig. 36B).

Based on misfit estimation repeated data rejection can be carried out to eliminate low-quality, poorly-chosen data, leading to the false anomalies appearance due to inversion. To exclude data by use the option **Options/Extra/Remove data with big misfit**. Option **Options/Extra/Get synthetic for big misfit** allows replacing the values to the calculated values (resulting from the solving forward problem for the current model).



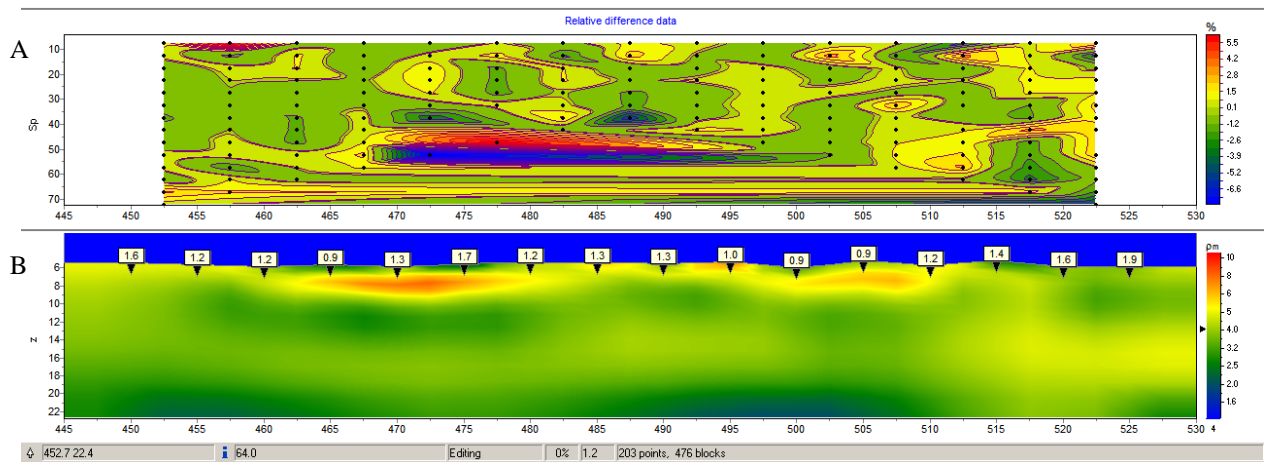



Fig. 36 A. Pseudosections of relative misfits,
B. Model with displaying each electrode misfits.

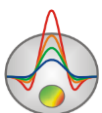
A useful option may be the option **Options/Inversion/Invert only visible graphs**, allowing to highlight data that should not be taken into account in the inversion. To do this, it is necessary to go to the displaying data as graphs mode and using the legend to switch off those that we do not want to include in the inversion. After selecting the option, program will not take into account disabled graphs in the inversion.

Additional options and data for inversion

The program also supports the inversion of other parameters (data of polarizability, monitoring (dependence resistivity from time), gravity and magnetic measurements (polygonal inversion), magnetotelluric methods data), not only inversion of resistivity measurement along profile.

If there is polarizability data on buttons ρ and η appear on the toolbar and allow you to switch between the modes of working with resistivity and polarizability data. In the latter case, pressing the button  start the data inversion of integral polarizability accounting the current resistivity section. If there are full waveform data program may perform the polarizability inversion for the entire curve, as result the true full waveform parameters will be determined by Cole-Cole model. Working with the IP data is described in Section [IP parameters inversion](#).

If you work with a large volume data, you can use the option **Options/Inversion/Long line inversion** to speed up the process of inversion and save memory. With this option, the inversion is made in the "sliding window", the width of which is defined by the user



interactively.

The data inversion of geoelectric monitoring is done via the menu item **Time lapse/domain/Time lapse inversion** (appears in the main menu when loading monitoring data). Working with monitoring data is described in the part [Geoelectric monitoring data inversion](#).

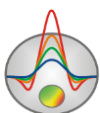
The polygonal inversion of gravi-magnetic data is available for the joint data interpretation of electric and gravi-magnetic methods. The polygonal model is created by user based on research results by resistivity method, and polygon properties are selected by the program using inversion (see [Working with gravity and magnetic data measurements](#)).

If there is measurement of magnetotelluric methods on the profile their inversion can be produced, taking into account resistivity method on the direct current (**Options/MT Data**, see [Magnetotelluric data inversion](#)).

If it is necessary, use the same working scheme with data for several files (profiles), you can use the batch processing function (the **Script** menu). The actions sequence is recorded in the script file, and applies to all the selected data files. Details of the program automation process is described in the part [Script-files \(batch processing options\)](#).

Model visualization modes and parameters

Model can be displayed as cells **Options/Model/Block-section** (fig. 34 A), in smooth interpolated palette **Options/Model/Smooth-section** (fig. 34 B) and as contour section **Options/Model/Contour-section** (fig. 37 B).



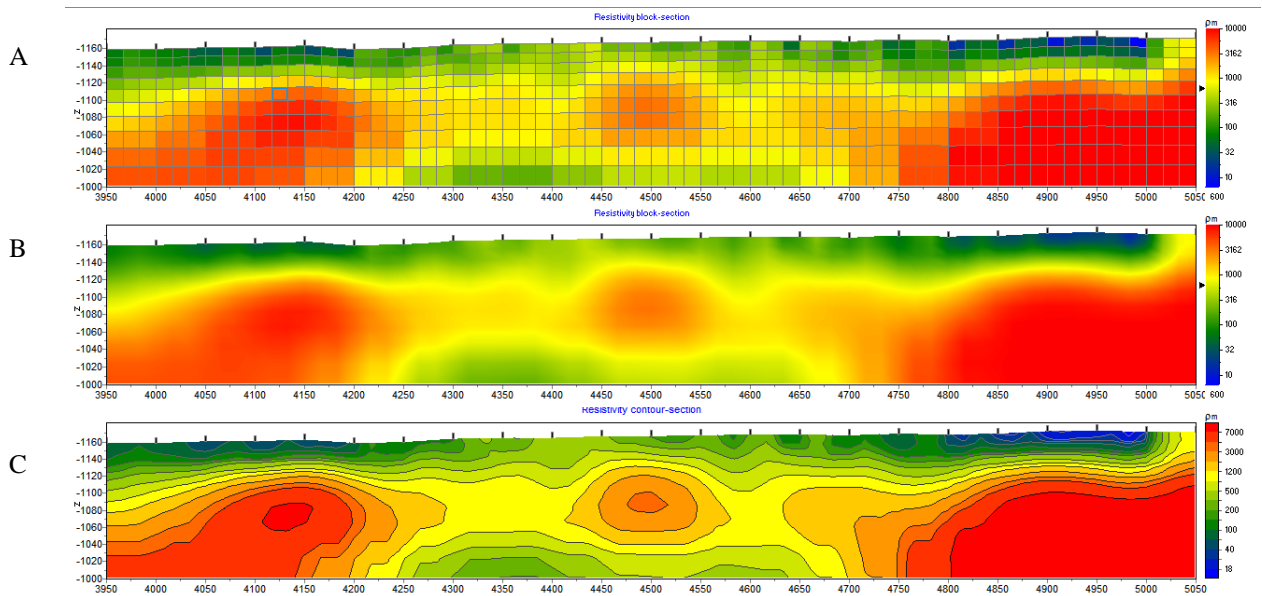
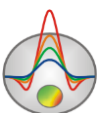


Fig. 37 Model display options:

Block-section (A), Smooth-section (B), Contour-section (C)

Double click in different domains of model editor to run context menu with the following options:

Top domain	Display model mesh	Display model mesh
	Display objects border	Display objects border
	Display color bar	Display color bar
	Setup	Run model parameters setup dialog
	Zoom&Scroll	Turn on zoom and scroll mode
	Print preview	Print model
Color bar	Set minimum	Set minimum value of color bar
	Set maximum	Set maximum value of color bar
	Set incremental factor	Set minimum and maximum value of color bar relative to value of host medium
	Automatic	Automatically select minimum and maximum value of color bar
	Log scale	Set logarithmic scale on color bar



	Set halfspace value	Set halfspace value of host medium
	Set cursor value	Set current value of the parameter
	Colors as histogram	Set colors based on the distribution model cells resistivity.
Section area	Context menu of working with model	Described in the part « Working with a block model »
Axes area	Model editor	Described in the part « Model editor »

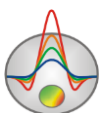
When moving the mouse cursor on the windows created during the working with the program coordinates corresponding to its own window axis displays in the left status bar section of the main window.

Model parameters setup dialog is described in [Appendix 6](#): Dialog of model visualization parameters(for **Block-section** and **Smooth-section** modes) and in [Appendix 4](#): Pseudosection (contour section) parameters setup dialog (for **Contour-section** mode).

Optional features depend on selected model display mode. It is more convenient to perform numerical modeling in **Block-section** mode. Total sensitivity distribution is displayed in **Contour-section** mode (use **Option/Model/Sensitivity** option). Potential and sensitivity distribution is displayed in **Block-section** mode or in interpolated palette in **Smooth-section** mode (see «[Additional features of model](#) visualization» for details).

Border parts of the model are usually characterized by worse resolution. Very often these domains contain numerous false anomalies. **Options/Model/Cutting** option allows hiding model borders by specifying cutting angle (in degrees) **Cut by angle** or sensitivity level (**Cut by sensitivity**).

The option **Options/Cutting/Expand bottom** switches ON or OFF stretching of lower model cells.



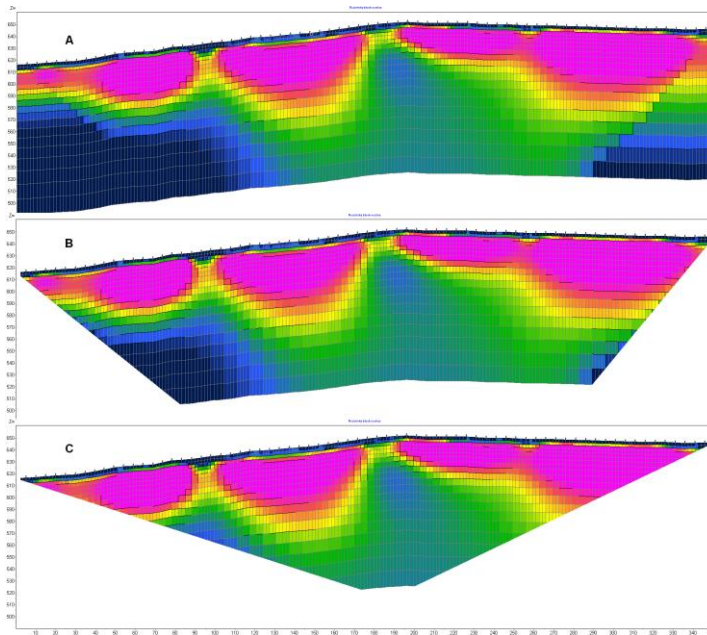


Fig. 38 Models comparison with different cutting angles (**Options/Cutting/Cutting angle**): A - no cutting, B - 30° angle, C - 55 ° angle.

To view data apparent resistivity and model true resistivity distribution graph use **Options/Model/Histogram** dialog (fig. 39). Minimum and maximum of parameter color scale can be set by changing vertical lines positions.

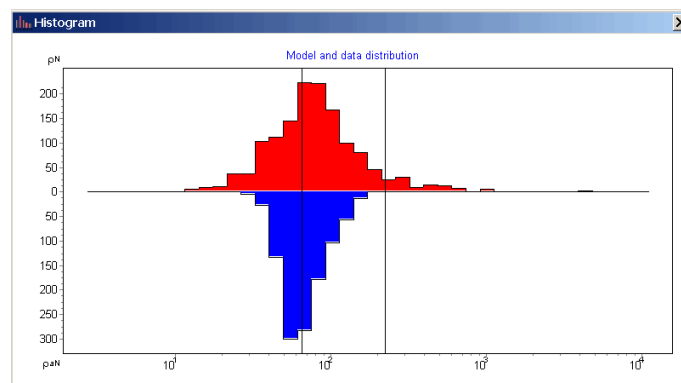
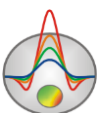


Fig. 39 **Histogram** dialog

Modeling and working with model

Modelling is an important process that precedes the field work. It allows determining feasibility to solve a geological problem with geophysical methods, as well as to select the



optimal measurement system parameters for it, to assess the signal level and the resolution of the selected survey layout. Using a priori information the interpreter can model different geological scenarios, planning geophysical work.

The program ZondRes2D allows modelling numerous practical applications of resistivity and induced polarization methods in two dimensions, taking into account the profile relief (any applying practically frequency ranges, an arbitrary profile step, block structure arbitrary model setting, polygonal models creation).

Parameters of the survey layout and environment model are the necessary input data for modelling. ZondRes2D allows creating survey layout and environment models in the program as well as use the survey layout saved using other formats and create geoelectric models based on other Zond programs and geological sections, defined as a background.

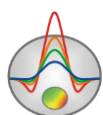
Survey layout definition

Observation (measurement) system or **survey layout** can be set in two ways: create it directly in the program or load from the data (measurements) file.

Creating survey layout in ZondRes2D dialog

To go to the dialog contains options set for choosing electrotomographic survey layout parameters, use the option **File/Create synthetic survey** of the main menu.

Using this dialog you can create measurement configuration using any array from the most common (drop-down list **Electrodes array**, fig. 40, 41.): Two-electrode (Pole-Pole, AM), three-electrode (Pole-Dipole, A-MN), dipole (Dipole-Dipole, AB-MN), Wenner (Wenner, AMNB), Schlumberger (Schlumberger, A-MN-B) and gradient (Gradient, A-MN-B).



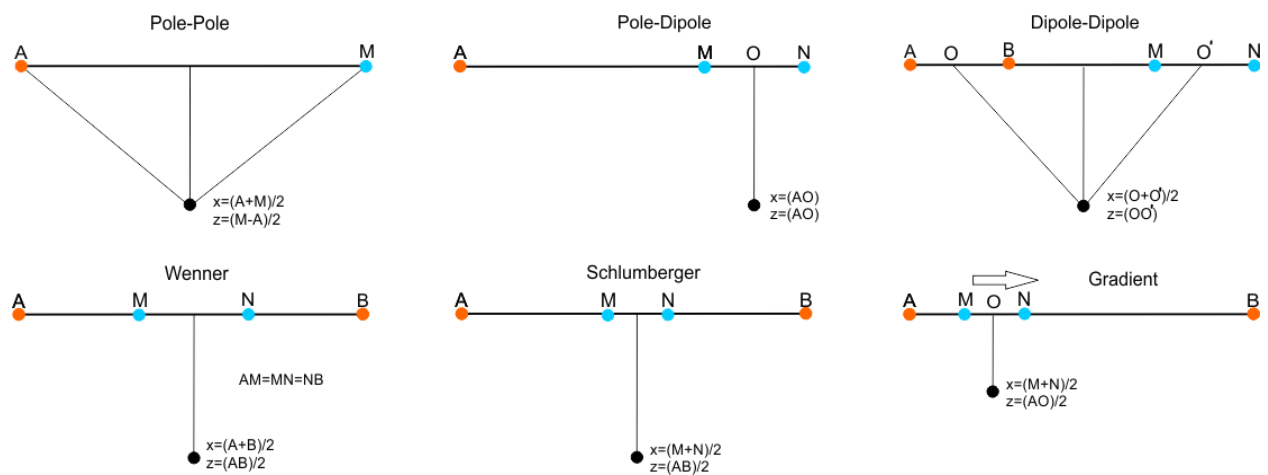
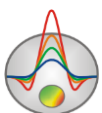


Fig. 40 The types of the most common arrays

The list **Sub array type** allows selecting a array subtype defined above (arrangement order of the electrodes). For example, for a three-electrode array we need a direct array (Forward, A-MN), a counter (Reverse, MN-A) or their combination (A-MN & MN-A).

The main numerical parameters for creating survey layout are set in the fields group **Array's geometry**: step between electrodes in meters (**Minimal separation, m**), the number of electrodes in single arrangement (**Electrodes number**), the shifts number on the profile (**Number of shifts**), shift between neighboring arrays (electrodes number, **Cable shift, n**).

Tab **Electrodes array worksheet** (fig. 42) allows you to specify additional observational geometry features. In fact it is a complete tool for creating measurement protocol of selected array. In the table at the tab bottom column **AO** contains all the possible shift values (defined by the formulas fig. 40), columns **n 3 a** - the length of receiving line (the tick determines selecting and deselecting), the columns **step [K]** - horizontally measurements frequency with selected receiver line length (for example, step [K] = 3 means that measurements will be conducted on every third potential point).



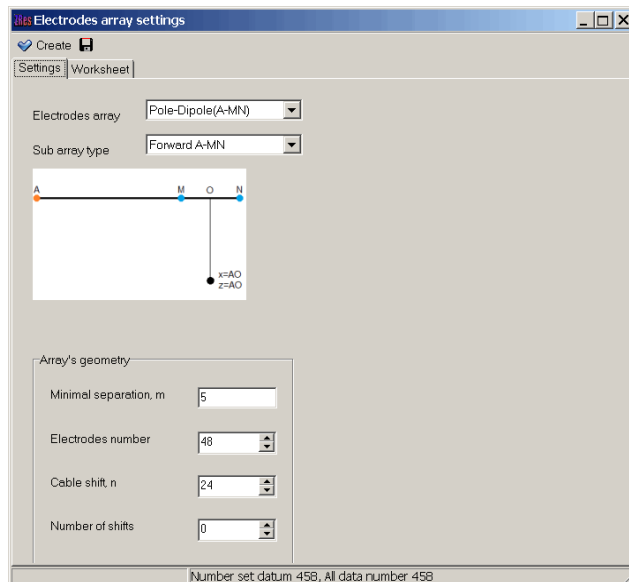


Fig. 41 Electrodes array settings dialog window (the array type and the main geometry features selection)

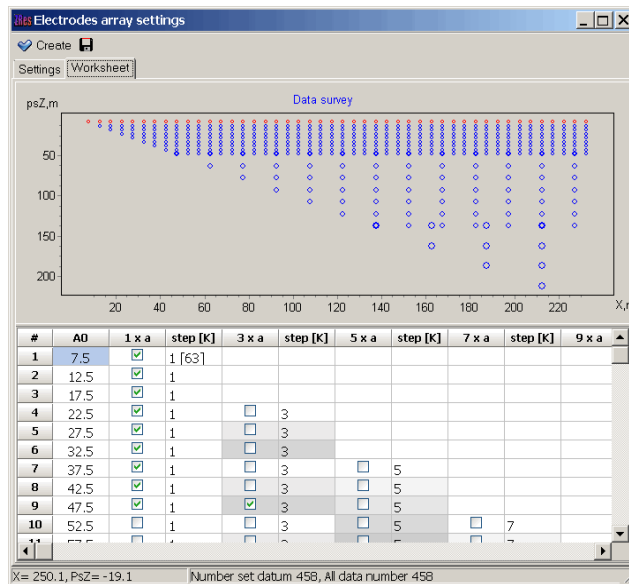

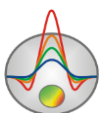
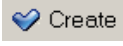


Fig. 42 Electrodes array worksheet dialog box (additional geometry features)

Figure Data survey at the upper tab part Electrodes array worksheet contains a graphic representation of the created survey layout.

The survey layout can be saved as a text file or ABEM protocol format (*.xml) and MAE instrument (*.sem) using the button  in the upper left corner of the dialog box.



After setting the parameters and clicking the button , dialog of grid settings **Mesh constructor** appears (more details in [«Starting model setup dialog»](#)). Specify host medium resistivity (and polarizability) in **Start resistivity** field (and in **Start polarizability** field). After pressing **Apply** button in main window panel functional buttons to work with data activate, and brief information about data and model appears in the right section of status panel.


It is possible to change the survey layout, while saving the model unchanged, to do it use the option **File/Change electrodes array** of the main menu, which returns to the dialogue of creating survey layout.

Loading survey from data file

An alternative to creating survey layout directly in ZondRes2D is its loading from a data file with various supporting formats. To do this, simply open the data file via the main menu **File/Open file**. This option allows modelling for the survey layout, which is used in field research, as well as set the measurement system with all sorts of settings.

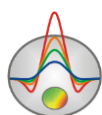
For the further array changing before opening the measurements file it is necessary to go to the modelling mode: **Options/Extra/Open in modelling mode**. Without this the array changing is impossible, the menu **File/Change Electrodes array** will be inactive.

Model building in ZondRes2D

Geoelectric medium creation produced in the model editor - lower section of the graphical window. The program ZondRes2D has two modes to create a model: block (**Options/Model/Block-section**) and polygon (toolbar button ). The block mode is performed for directly model cells editing (the width or thickness of the selected row of cells and cell properties). Polygon mode is used to set the environment consisting of a set of related or unrelated to each other bodies.

When working in block mode, it is also possible to edit some grid parameters created in the dialog **Options/Mesh constructor**. They are described in the [«Model editor»](#).

It should be noted that any mode of creating models has developed work settings with a priori information: loading of borehole and logging data, background as a raster file, models from other ZondRes2D programs. These tools are described in the [«Data import and export»](#).



Model editor


Double click on vertical and horizontal axes and right click in model edit region to run context menu that contains options to edit mesh created in **Mesh constructor** dialog.

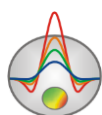
Vertical axis	Log scale	Set logarithmic scale on vertical axis
	Set maximum	Set bottom layer depth value
	Redivide	Set equal thickness for all model layers (in current scale)
	Thick mesh	Delete every second node of vertical grid
	Thin mesh	Add intermediate nodes in vertical grid
Horizontal axis	Redivide	Set equal width of cells situated between unique positions of electrodes
	Thick mesh	Delete every second node of horizontal grid (if there is no electrode in this node)
	Thin mesh	Add intermediate nodes in horizontal grid










Working with a block model

Block model editor is used to change the parameters of individual model cells using a mouse. Working with model cells is like editing of raster image in graphics editors. When moving the cursor in the model, the status bar at the bottom of the main window displays the coordinates and parameters of the active cell in which the cursor is located. The currently active cell is surrounded by a rectangle - the cursor. The selected or fixed cell is hatched with white or black points.

To work with the block model, there are two tools sets in the program: the context menu (right-click in the editing model), external toolbar (**Options/Model/ModelEditor toolbar**) and color scale. The functions of context menu and external toolbar largely duplicate each other:

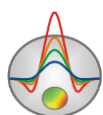
<i>Context menu</i>	<i>Model tools</i>	<i>Option</i>
		Zooming, scaling of a model



Display cell setup		Run cell parameters setup dialog.
Cell to cursor value		Use active cell parameter as current value
Edit mode		Run Edit mode
Selection\Free form selection		Highlight set of cells within edit region using mouse. Field is limited by user.
Selection\Rectangular selection		Highlight set of cells within edit region using mouse. Field has rectangular form.
		Highlight set of cells within edit region using mouse. Field has elliptic form.
Selection\Magic wand		Highlight set of cells within edit region using mouse. Active cell and adjacent cells whose parameters are close to its parameter are highlighted. Proximity is specified in model parameters setup dialog.
Selection\Remove selection		Delete selected
Mesh options\add column /row		Add new vertical or horizontal boundary. Click in necessary point to create new boundary there.
Mesh options\remove column /row		Delete selected vertical or horizontal axis
Mesh options\resize column /row		Change row or column thickness using mouse
Clear model		Clear current model parameters
Clear parameters		Clear resistivity values of the current model


The color scale, linking the color value with the value of the resistivity is on the right of the model editing area. To select the current value left-click on the scale, and the value is displayed below of the color scale. To change the cell parameter value left-click on the cell to change its parameter to the current one.

The program has several modes of selecting cells to create a model quickly: by a rectangle, by an ellipse, by free form and by for a specific parameter value. To the options use



the context menu by pressing the right mouse button in the editing model area, or as a separate remote toolbar (**Options/Model/ModelEditor toolbar**). If a certain section area is highlighted, then press the left mouse button anywhere in the selection area to set for all cells in the field the current parameter value.

For example, to set certain resistivity value to rectangular area section, select this area section using the tools described above, the left-click on the color bar (select the required resistivity), and left-click in any point of the selection area, which then gets assigned the selected resistivity value.

Zoom-in or moving of a section part is carried out in the mode **Zoom&Scroll** (is turned on in the context menu of model parameters setup by right-clicking near the Resistivity block-section in the model window) or after pressing the button  on the panel Model tools. To select the area to zoom in, the cursor moves down and to the right with the pressed left button. To return to the original scale, the same actions, but the mouse is moving up and to the left.

To increase the cell parameter left-click the while holding SHIFT on the cell or cells group. Right-click on the cell while holding SHIFT to decrease the cell or cells group parameter. Percentage of parameter value changing sets in the dialog of model parameters setup. If the active cell is selected, then all the above described operations are applied to the all selection.

Clicking the mouse while pressing CTRL allows moving selected cells set within the editing area using the mouse. When moving the selection with pressing the left mouse button the contents of the selected cells are copied to the new location. When moving the selection with pressing the right mouse button the contents of the selected cells is cut and copied to the new location.

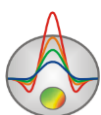
It is also possible to set the parameter value to the selected cells using the cell parameters setup dialog **Cell setup** (fig. 43).

Dialog is intended for selection or choosing the cell parameters.

Value - sets cell parameter value.

Fixed - fixes or frees cell parameter.

Min value, Max value – sets cell parameter size of changing.



The selected area is saved when going from the resistivity mode to the polarizability mode.

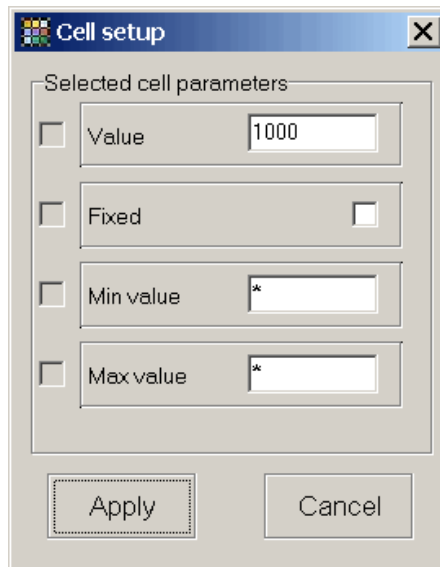

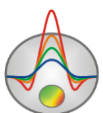


Fig. 43 Dialog window **Cell setup**

To calculate the response of apparent parameters from a given geoelectric model (to solve the forward problem), press the button  on the toolbar.

To restore the original model, that is, to solve the inverse problem for the calculated from the given model data, save the theoretical signals with filter **Zond calculated data**, and then open as observed. To compare the restored and original models use the option **Options/Import/Export/Import model/data** (



Data import and *export part*), before it save the original model by selecting the filter **Zond model with calculated**. A simpler way to compare models is using the tool **Buffer**, it allows working with several models at the same time («*Working with multiple models in a single project*» part).

Cell summarization dialog

Cell summarization (Options/Advanced) option serves to smooth or roughen (divide it into blocks) current model. Block model can be used for inversion of **Blocks** type. Selection of parameters for each block is performed then. It is better to use focusing inversion before roughing into blocks.

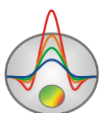
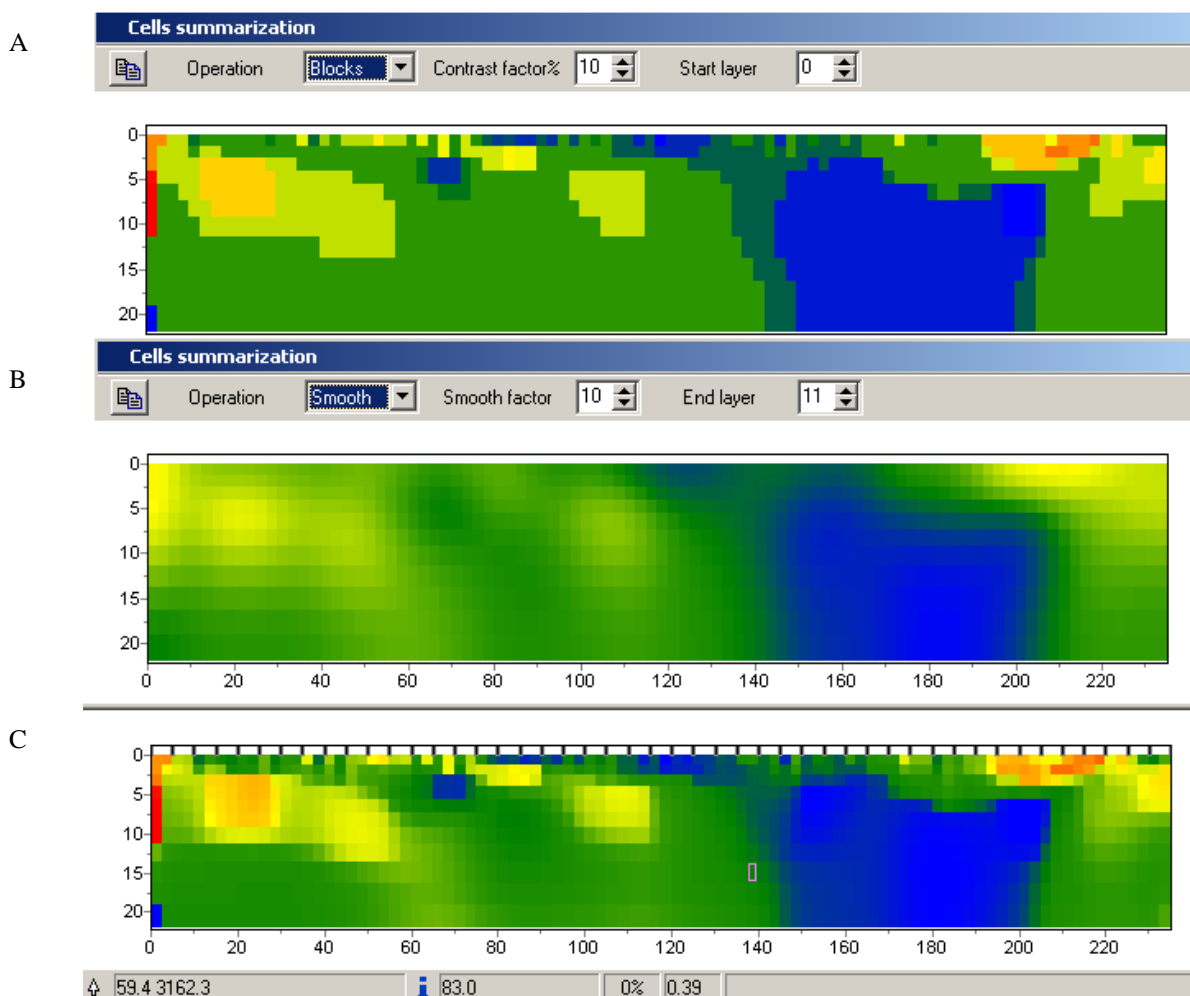




Fig. 44 Example of smoothing (B) and dividing into blocks (A) geoelectric model (C) when **Cell summarization** dialog is used

In **Blocks** mode cells with similar parameters in constant value domain merge depending on **Contrast factor**. **Start layers** option sets layer number starting from which this operation is performed.

In **Smooth** mode cell parameters are averaged depending on **Smooth factor**. **End layers** option sets layer number in which this operation stops.



Button  copies received model to model editor.

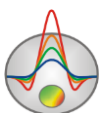
Polygonal modeling










Instead of setting model using blocks, it is more practical to define geological bodies, which have their resistivity values. Polygonal modelling mode provides this functionality within ZondRes2D. To run it, press the button , after that the toolbar for creating a polygonal model appears on the top left window corner. If polygonal model is created in the current project, it will be displayed in the model window.

Polygonal models creation in ZondRes2D is similar to the process of creating closed polygons in vector graphic editors.

Polygonal modelling mode allows describing individual polygons (bodies) in a homogeneous environment and the system of connected to each other polygons (bodies). To create a polygon and edit it select the tool from the polygonal modelling toolbar:

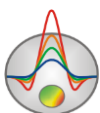
<i>Tool</i>	<i>Option</i>
	Create polygon. Left-click to add a new node to a polyline – polygon boundary. Description of the polygon boundaries is completed by right-clicking, after that the polyline closes automatically.
	Remove polygon. After selecting the tool press the right mouse button on the polygon to delete it.



	<p>Create polygon adjacent to the existing one or to modelling boundary. After selecting this tool, it is necessary to specify boundary, not adjacent to the existing polygon. The first and the last point of defined boundary must belong to either the boundary of the adjoining body or the boundary of the modelling area. To finish boundary setting press the right mouse button. Adjoining boundary program will select automatically or ask to select by dialog.</p>
	<p>Separation of adjacent polygons. If using the previous tool, a model with adjacent polygons was created, this tool allows dividing them, to be able to change the boundaries, move or delete polygon. After selecting the tool select polygon (which it is necessary to separate) by left mouse button, (single click anywhere in the polygon, with its boundaries change color). Right-click to end the separation process.</p>
	<p>Divide polygon by a straight line (to create a single polygon from two). Left-click to set the first point and right-click to set the second line point. Both points must be on the boundary of dividing polygon.</p>
	<p>Move polygon. Select polygon by pressing the left mouse button. When moving the mouse polygon moves. Polygon position is fixed by right-clicking.</p>
	<p>Move a part of polygon</p>
	<p>Add node. Right-click on the boundary point where it is necessary to add a node.</p>
	<p>Remove node. Right-click on the node, which it is necessary to delete a node.</p>
	<p>Move node. Left-click to select a node, to move - move the mouse, to end moving – right-click.</p>
	<p>It allows zooming in models.</p>

When using polygonal modelling tools, remember that all operations are completed by pressing the right mouse button.

To change the resistivity, polarizability of polygon and visualization properties double-click the left mouse button at any point inside it. Dialog window **Body parameters** appears (Fig. 45).



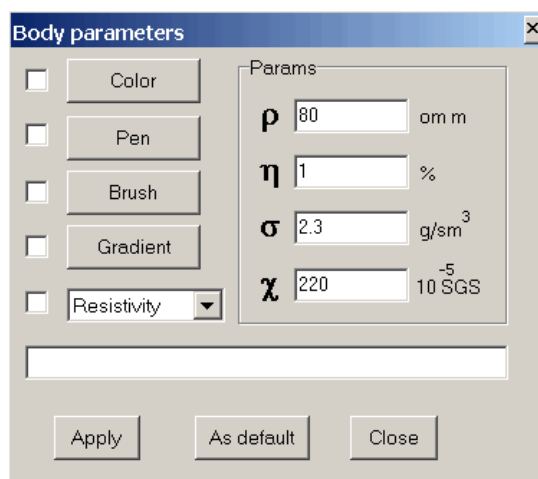

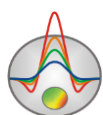


Fig. 45 Dialog window **Body parameters**

Resistivity, polarizability, density and magnetic susceptibility values are set in the fields ρ , η , σ , χ . Buttons **Color**, **Pen**, **Brush**, **Gradient** run dialogs of polygon color settings (different from model color bar), polygon boundaries colors, type of hatching and gradient method. The appearing list contains the polygon signature – resistivity, polarizability values or any text that can be entered in the below text field.

The association between the polygonal model resistivity values and block model is used by the menu **Modelling/Get values from background and Modelling/Set values to background**. The first option sets block model resistivities to polygons (from the results of preliminary inversion or modelling), the second one - the polygon resistivity to block model. The transition between the block/polygonal modes is done by pressing the button  on the main toolbar.

An example of model created by using polygonal modelling tools is presented in Fig. 46.



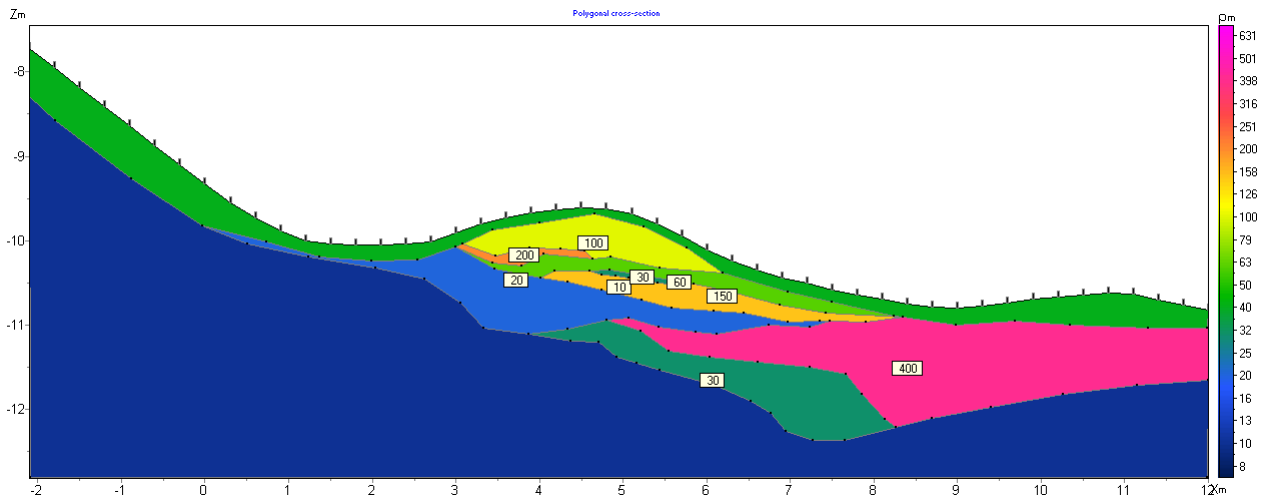




Fig. 46 An example of a polygonal model created in ZondRes2D

Polygonal models can be saved and loaded by the option **Save polygons** and **Load polygons** of menu **Modelling** main toolbar. Use the option **Export to CAD** to export polygon model to Autocad dxf file.

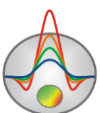
Click the button  on the toolbar or press Spacebar to solve the forward problem with created model.

To improve the calculation accuracy of the forward problem it is recommended to set a finer grid. When creating a polygonal model, you can go to the block mode (by using the button ) and use the option **Thin mesh** in the each axis properties, and then return to the polygonal modelling mode, and click **Modelling/Set values to background**. After that, the calculation of the forward problem will be done already using the finer grid.

Polygonal modelling process is shown in movies <http://zond-geo.ru/zmovies/polygonalmodeling2d.exe> and <http://zond-geo.ru/zmovies/polygonalmodel.exe>.

Working with gravity and magnetic data measurements

There is possibility of electrical exploration interconnecting with measuring data of gravitational and (or) magnetic field in the polygon modeling mode (realize polygon modeling and data inversion). The option is available in the polygonal modeling mode via the menu **Modeling/Gravimagnetic**, if you import corresponding data (more details in «[Gravity and magnetic data measurements import for interpretation](#)»).



To set manually gravity/magnetic properties bodies of polygonal model for each block use the dialog box **Body parameters**, or automatically (hold polygonal inversion) using the menu **Modeling/Gravimagnetic/Inversion**. Graphs of observed and calculated fields appear in the gravity and magnetic data window. Further, changing polygons parameters (geometry and properties), you can manually interactively build a polygonal model, corresponding to several geophysical methods.

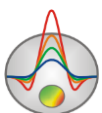
Complex electrotomography and magnetic data interpretation are shown in the film <http://zond-geo.ru/zmovies/ert&magnetic.exe>.

Working with multiple models in a single project

It is often necessary to store several models and to compare them in a single project. For example, when determining the optimum inversion parameters it is more convenient to not create a separate project for each parameters set, but rather store all inversion results obtained in the same project and be able to compare them in a single window simultaneously. Also in the modelling mode when calculating the forward problem from several related models it is convenient to store and compare them in a single project.

Described functions are implemented by option **Buffer** of the main menu. Buttons **Model 1 - Model 5** correspond to the five buffer models that can be stored in a single project.

To record the current model to the buffer press one of the buttons corresponding to the buffer models. If the selected buffer model is empty, the current model will be written into it. Appearing dialog allows inputting the buffer model name, which will be displayed on the corresponding button in the **Buffer** list and as the title – when displaying model.



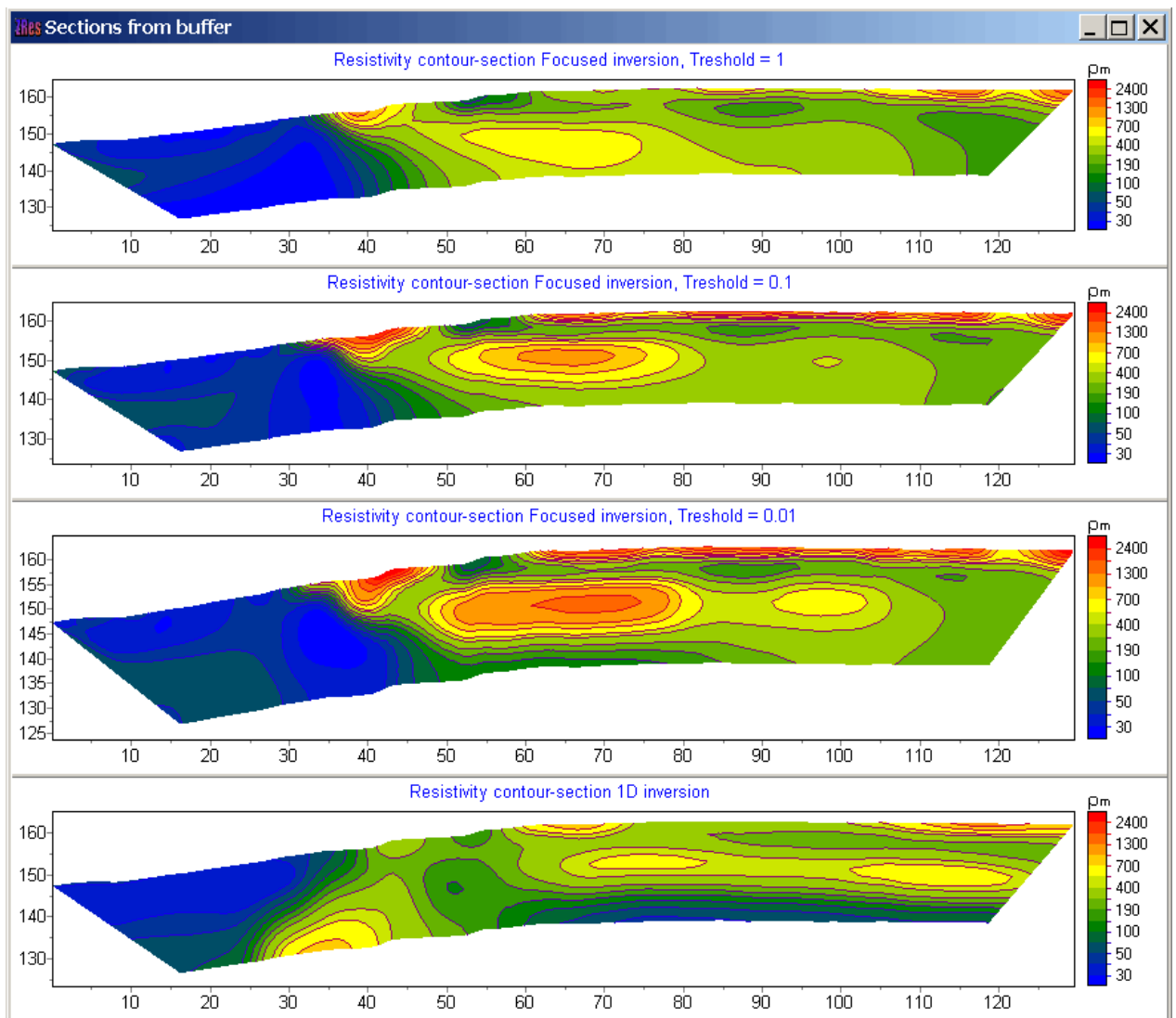
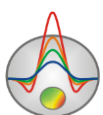


Fig. 47 Window with several models from **Buffer/Open**. Example of inversion of the same data using different parameters.

After the first buffer model was set, one can clear the current model and create the next one and write it in the following buffer model.


If the selected buffer model is not empty, the program will ask open this buffer model (**From Buffer**), or write the current in its place (**To buffer**). When selecting **From Buffer** model from buffer will be put in current active model place to the section editor.

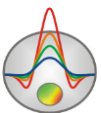
Button **Buffer/Open** allows displaying all created models in a single window to compare them (Fig. 47).



Saving interpretation results

Profile interpretation result is hold in «ZONDRES2D» file format (extension *.z2d) (*see* «[Main data file format](#)» *for details*). Field data, relative measurement weights, and current subsurface model is saved in this file. Data from the file is used for further load and subsurface model creation.

Use  button in the tool bar or corresponding menu option **File/Save file** to save interpretation result. This dialog also allows choosing file format to save observed (**Observed**) and calculated for current model (**Calculated**) apparent resistivity and polarizability values and images (**Model, WorkSheet**) in *.BMP format in necessary scale. Image scale can be set using the dialog **Picture settings** of the menu **Options/Graphics/Bitmap output settings** (*described in* «



Data import and export»).

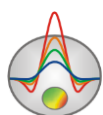
Zond project data	Save observed data and current subsurface model
Zond calculated data	Save calculated data
Zond observed data	Save observed data
ProfileR observed data	Save observed data in ProfileR format
ProfileR calculated data	Save calculated data in ProfileR format
Res2dInv observed data	Save observed data in Res2dInv format
Res2dInv calculated data	Save calculated data in Res2dInv format
Worksheet	Save three graphic section of the window in BMP format
Model	Save bottom graphic section of the window in BMP format. Use Picture settings dialog to adjust image scale.
Program configuration	Save inversion parameters of the program
Zond model with calculated	Save calculated data and current subsurface model
Grid file	Save model data for the Surfer program: values in the text file *.dat (resistivity and logarithmic resistivity), model boundaries (file *.bln) and color scale (file *.lvl).
Section file	Save current model in SectionCorrector format

When saving the project a file with extension *.mod2d is automatically created, which contains the model description. This file is used when importing models to other projects ZondRes2d (**Options/Import/Export/Import model/data**) or other Zond programs.

Project information

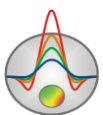
Use **File/Project information** option in the main menu to view current project information. The following information is displayed sequentially in the dialog:

- Number of unique current circuit or electrode positions;
- Minimum/ maximum separation;
- Number of measurements;



- Percentage of rejected data (compared to total quantity);
- Number of unique separations;
- Number of topography measurements;
- Number of cells used for forward problem solution;
- Number of cells used for inverse problem solution.

User can also input additional information about the project (work organization name, work area, executor, etc.) in free fields.



Data import and export

The most effective methods of data interpretation quality improving is interconnecting of geophysical exploration methods and the account of a priori information. ZondRes2D program has advanced functions to solve this problem. They are the possibility of gravity and (or) magnetic data inversion and interactive interpretation, additional information visualization in the program: the borehole columns, logging data, graphs of observed geophysical fields and any other graphs, models from to other methods, obtained in the Zond programs, as well as any information, defined as a bitmap - geophysical or geological sections.

The program also has advanced tools for data import and export to work with the apparent resistivity, geoelectric models.

Additional information visualization

There are several modes to visualize a priori information in the program. Using the option **Options/Import/Export** it is possible to load geological and geophysical information: lithological columns, logging data, profile measurements as graphs, models from projects of other Zond programs, a graphic image as background for the geoelectric section (for example, geological or seismic sections).

To load logging measurements or lithological columns into the model window use the option **Options/Borehole/Load borehole data** (fig. 48). Working with the lithological columns and logging data is described in «[Logging and lithology data file format](#)». *Sample-file – sample with_bhdata.*

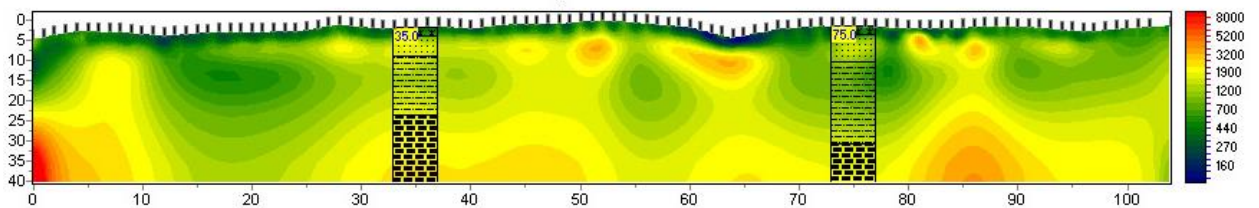
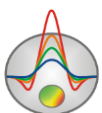


Fig. 48 Resistivity model with lithological columns

Option **Options/Import/Export/Import model/data** allows loading environmental models from other ZOND programs projects to separate windows (Fig. 49). The option can be useful in comparing the interpretation results on adjacent profiles or complex interpretation of the various methods.



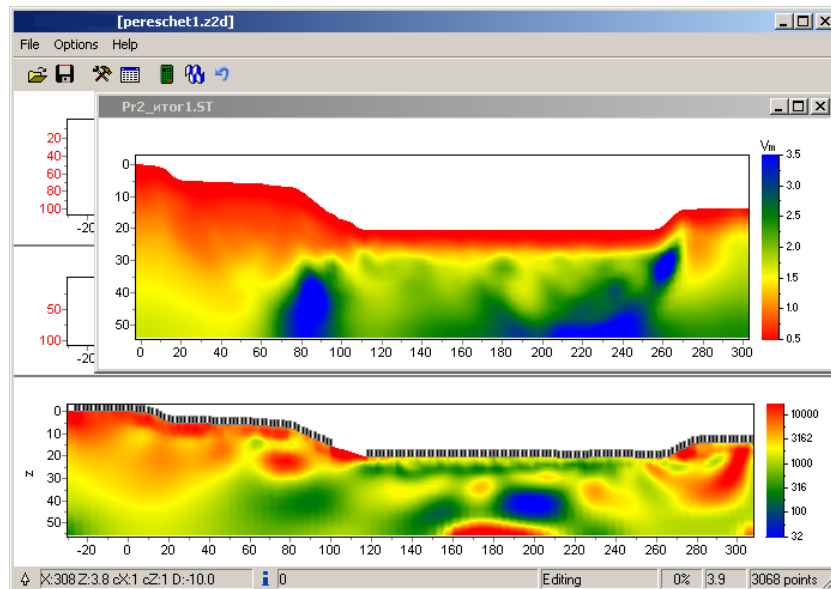


Fig. 49 Working window with importing high-speed model

During the cursor movement in the model editor it will be displayed in all other importing sections, according to the size of the current cell when displaying model as blocks (Fig. 50).

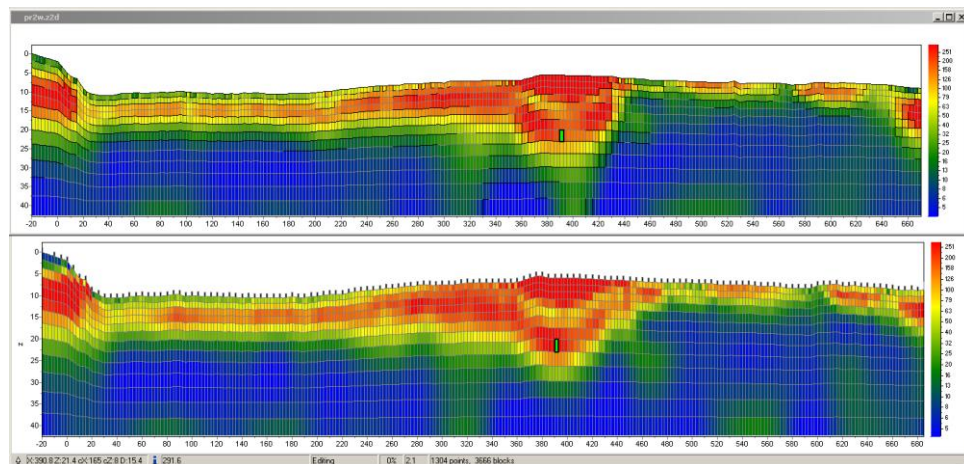
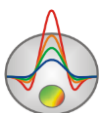


Fig. 50 Models of electrical resistivity, working, and importing as blocks with illuminated cursor

When importing two-column file with *.dat extension, graph associated with the right axis appears in the window with the calculated data (in the **graphics-plot** mode) (Fig. 51). [Sample-file – sample_with_exported_graphic](#). Horizontal coordinates of measurement point on the profile set in the first column of the *.dat file, in the second - measured values. Thus it is possible to display the graphs of any physical quantities changing along a profile.



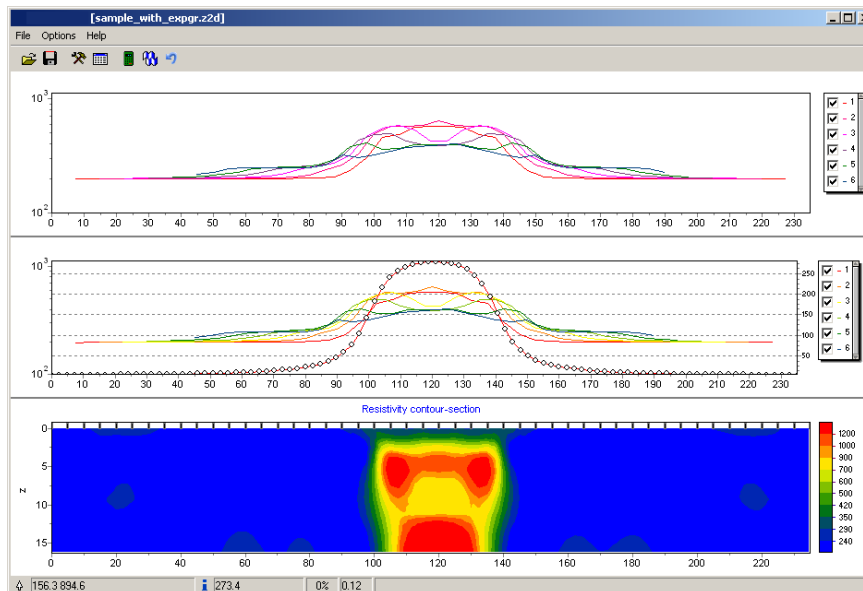


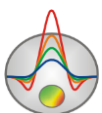
Fig. 51 Working window with importing graphics of anomalous magnetic field

When right-clicking with the SHIFT button pressed on the axis or the graph, a pop-up window appears and allows running object parameters setup dialog. To save or load a model fragment, use the option **Save/Load selection**. To save the fragment, when turning on the mode of displaying model as blocks (**Blocks-section**), using the section **Selection** options highlight fragment, and click **Save selection** (**Selection** options are described in the [«Model editor»](#)).

To load model fragment it is necessary, select a small area of the current model. The upper-left edge will be considered as the place from which will be embedded fragment. Run option **Load selection** and select the file name. If the selection is not available, then the fragment is inserted from the top left edge of the model.

To save or load parameter vertical profile for the given horizontal coordinate use the option **Extract 1d log/Load 1d log**. When saving the vertical profile it is necessary to specify X coordinate in the dialog box. When loading the vertical profile it is required to specify the range of the X axis. This option can be used, for example, for logging data inputting, or when intersections profiles places researching.

If there is a priori information, you can use it (as a background under the model editor) with the option **Options/Import/Export/Background image**. This may be, for example, geological, electrical or seismic section of adjacent profile. There are two background formats in



the program: the graphic *.bmp file and the *.sec file, containing graphic information and the image binding information.

After selecting *.bmp file the dialog of image coordinates settings (Fig. 52), in which the image boundaries coordinates in the section coordinate system specifies.

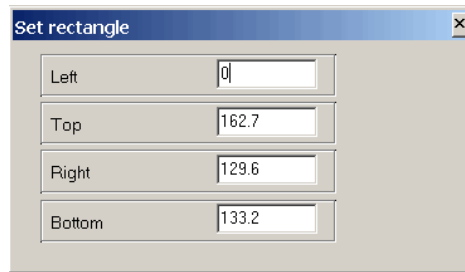


Fig. 52 Model with background in **Block-section** mode

To make the image transparent in the model settings dialog (right-click next to Resistivity block-section in the model window, the option Setup) the half-space transparency mode should be selected.

File * .sec ([Example in the directory – sample_with_exported_graphic](#)) has the following structure:

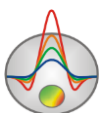
1st row - image file name;

2nd row - four coordinates of the upper left and lower right corner of the image X1 Y1 X2 Y2 are specified consistently through the SPACE.

sect.emf

0 0 152.4 53.3

The **Blocks section** mode displays those cells whose values are different from the host medium. Thus, it is possible to model anomalous objects on top of the background (Fig. 53).



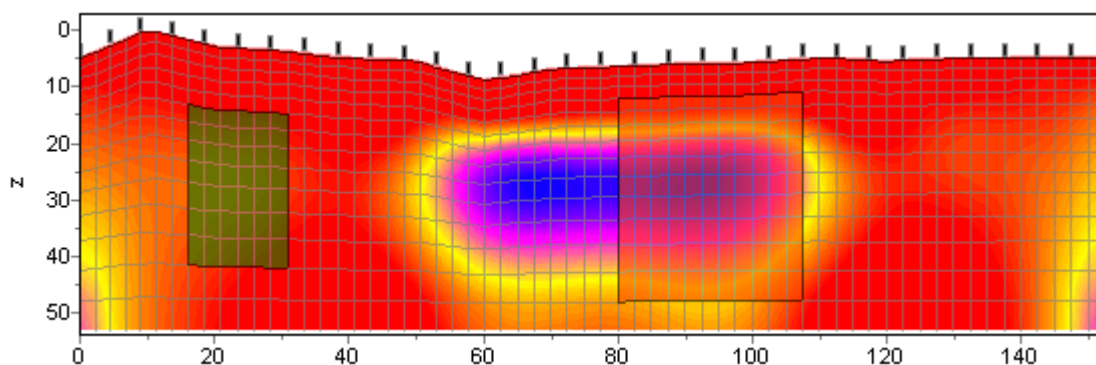


Fig. 53 Model in the mode Block-section with background

In the **Smooth section** mode it is possible to see the features of the two sections at the same time, because colours of background and current model will be mixed (Fig. 54).

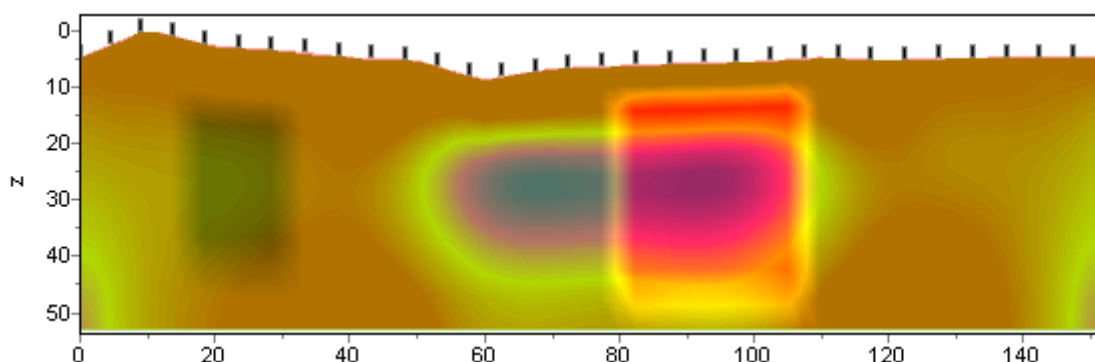
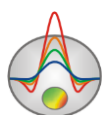


Fig. 54 Model in the mode Smooth-section with background

Using background allows taking into account the results of other profile geophysical research methods (e.g. seismic) and geological sections of the profile when carrying out the interpretation (setting boundaries on inversion, the creation of a priori models). Figure 55 shows an example of working with background when using geoelectric model in the mode Smooth-section.

The film is dedicated to working with background-file in ZondRes2D is available on the page <http://zond-geo.ru/examples/video-lessons/>



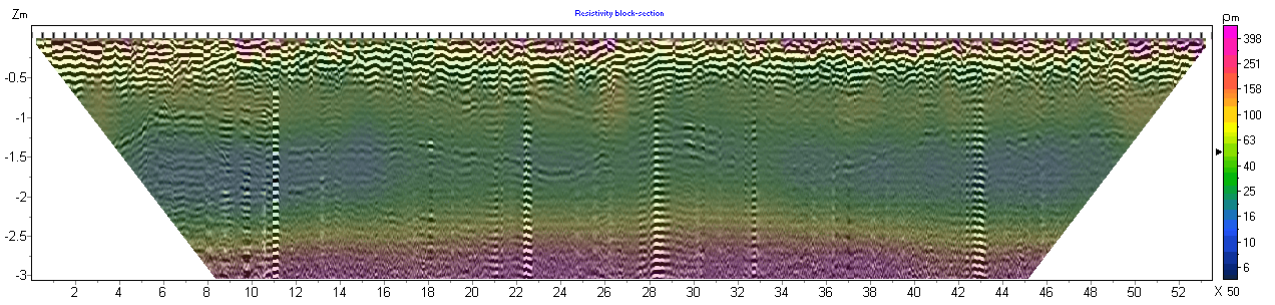


Fig. 55 Model in the mode Smooth-section with background – georadar data


In the "Interpretation results saving" describes the various possibilities of data and resulting model export. The data file can be exported to the most popular program formats: RES2DINV (Geotomo Software, M.H. Loke), SENSINV2D (Geotomographie, T. Fleschner), ABEM data and ProfileR (A. Binley). For further geological interpretation and reporting graphs it is possible to save the current model as *.dat file. The model can be saved as a bitmap specified resolution and size using **Picture settings** dialog.

Text (tabular) data files – import and export

ZondRes2D program has advanced tools for import from text (tabular) files and export to the text (tabular) files. It is possible to import measurement data from a text or tabular file and to export measurement data and current geoelectric model.

Data import from text (tabular) file

Import data from text file with *.txt extension or tabular Excel file is carried out by using the menu item **Options/Import/Export/Import data from text/excel**. After selecting the file, a dialog of data import appears. Use the **Start** and **End** buttons at the top of the dialog box, to specify respectively, the first line, from which data import are made, and the last line, by which import is completed.

Each cell of the top table row of the dialog box contains a drop-down parameters list. For each importing column, it is necessary to select the appropriate parameter. To finish working with the data import window press the button  **OK**.

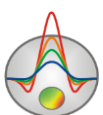


Fig. 56 Gravity and magnetic measurements loading dialog

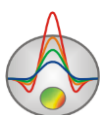
Observed data export to Excel tabular file

To export observed data to a tabular Excel file use the menu item **Options/Import/Export to Excel/Data levels**. Data are grouped by arrays coefficients. Data for each of them are described in three lines: the line starting with «Level (K, 1/m)», contains the array factor value; the line starting with «X, m», describes the profile distance for each export point (inputting points coordinates); the line starting with «app res», contains apparent resistivity values corresponding to the specified array factor and recording points.

Geoelectric model export to Excel tabular file

Export geoelectric model to Excel tabular file is carried out by using option **Options/Import/Export to Excel/Model**. In the creating Excel file geoelectric model is written as a matrix whose rows correspond to the horizontal coordinates of cells and columns - vertical (Fig. 57).

The left column contains the vertically block levels (centres) value; row starting with the «X» cell - the horizontal coordinates of the cell centres; row starting with the «Elev» cell - the relief value at the corresponding points.




	A	B	C	D	E	F	G	H
1								
2		X	0	0.5	1	1.5	2	2.5
3		Elev	0.738	0.756	0.775	0.794	0.812	0.83
4	Z1	0.5	10000	2586.5	4058.9	4494	2167.4	5497.6
5	Z2	1.05	10000	10000	10000	10000	10000	10000
6	Z3	1.655	10000	10000	10000	10000	10000	10000
7	Z4	2.321	10000	10000	10000	10000	10000	10000
8	Z5	3.053	10000	10000	10000	10000	10000	10000
9	Z6	3.858	10000	10000	10000	10000	10000	10000
10	Z7	4.744	5204.4	5204.4	5204.4	5204.4	5204.4	5204.4
11	Z8	5.718	1503.7	1503.7	1503.7	1503.7	1503.7	1503.7
12	Z9	6.79	637.2	637.2	637.2	637.2	637.2	637.2
13	Z10	7.969	306.01	306.01	306.01	306.01	306.01	306.01
14	Z11	9.266	141.31	141.31	141.31	141.31	141.31	141.31
15	Z12	10.692	14.348	14.348	14.348	14.348	14.348	14.348
16	Z13	12.261	4.13	4.13	4.13	4.13	4.13	4.13
17	Z14	13.987	3566.2	3566.2	3566.2	3566.2	3566.2	3566.2
18								

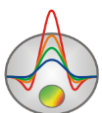
Fig. 57 The example of exporting model Excel-file

Gravity and magnetic data measurements import for interpretation

ZondRes2D allows not only to visualize measurements of magnetic and gravitational fields, but also to make polygonal modeling and inversion of the data (to determine the appropriate properties of the specified polygons). This program option is intended for interconnecting these methods with electrical exploration data. Therefore work with the appropriate menu items occurs after polygonal geoelectric model creation (*more details in «[Polygonal modeling](#)»*). It is possible to work with vertical component of the gravity field – gravity and its vertical gradient, for magnetic field it is possible to work with full vector measurements and vertical gradient.

Gravity/magnetic measurements data are loaded into the program from a text file that contains columns with the distance on the observation profile and field value. To import the data go to the polygon modeling mode (main toolbar button ) and use the menu item **Modeling/Gravimagnetic/Load new data**. If the measurement data are contained in different files (for example, in one - a magnetic field measurement data, in other - gravity), to add the data item use the menu item **Modeling/Gravimagnetic/Add new data**.

After the file selection import data window appears (Fig. 58). Use the drop-down list in the table upper rows cells to specify what data and in which column contains. The top line



specifies the columns that contain the distance on the profile and the data type - gravity and (or) magnetic. The second line contains the measurement units. For the gravity field - milligals or microgals, for the magnetic - nanoteslas. **Start** and **End** buttons at the upper window area allow you to specify the file lines to start and finish the data import.

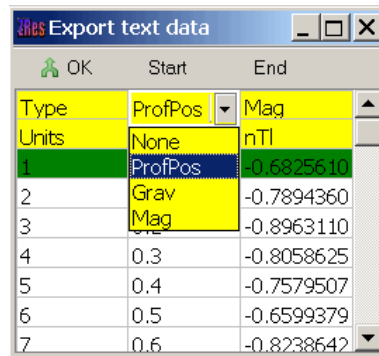


Fig. 58 Dialog of gravity and magnetic measurements loading

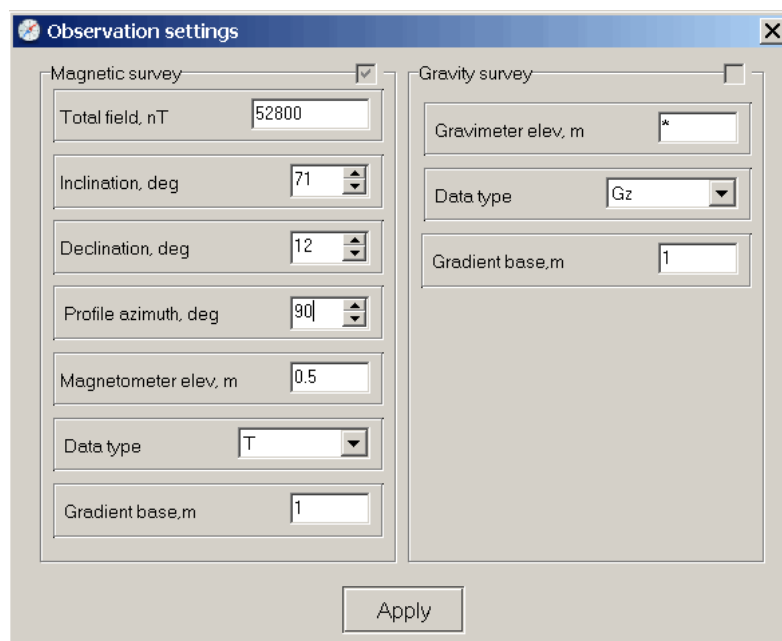
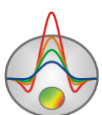


Fig. 59 **Observation settings** dialog box

The applying changes (button OK) leads to appearing of measurement parameters and the normal field setting dialog (Fig. 59). The same dialogue can be at any time run via the menu item **Modeling/Gravimagnetic/Field settings**.

For the gravity field the corresponding parameters are:



Gravimeter elev, m	Gravimeter height, m
Data type	Data type - vertical component of the gravity field and its vertical gradient
Gradient base, m	In case of vertical gradient data – measurement base, m

For the magnetic field the corresponding parameters are:

Total field, nT	Normal field amplitude, nT
Inclination, deg	Magnetic inclination, degree
Declination, deg	Magnetic declination, degree
Profile azimuth, deg	Profile azimuth, degree (measured from the North in a clockwise direction)
Magnetometer elev, m	Magnetometer elevation, m
Data type	Data type – full vector or field vertical gradient
Gradient base,m	In case of vertical gradient – measurement base, m (distance between sensors)

Closing settings dialog run appearance of gravity and magnetic data window (Fig. 60). The window can be closed and opened again using the menu item **Modeling/Gravimagnetic/Display GM window**.

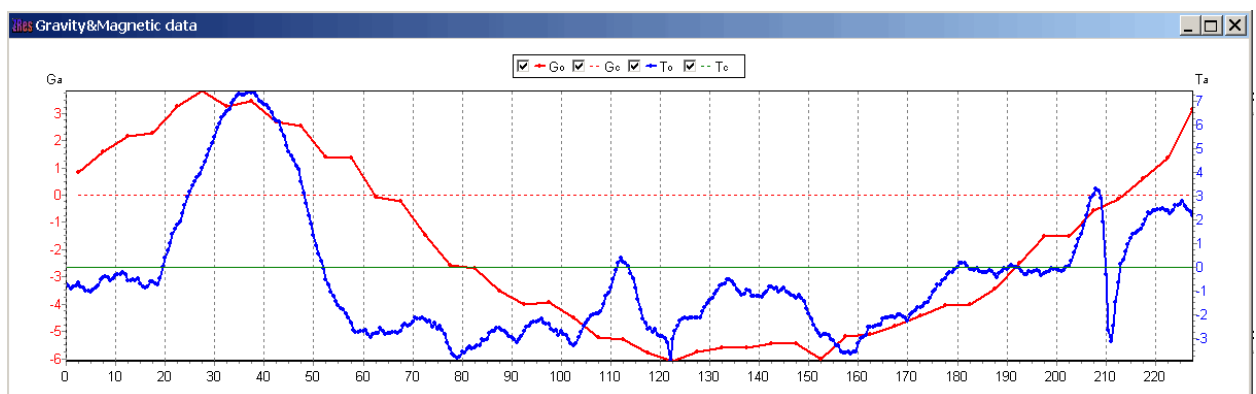
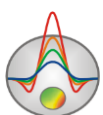


Fig. 60 Gravity and magnetic data dialog box



Options **Modeling/Gravimagnetic/Subtract median grav** and **Subtract median mag** allow subtract the median value from the observed field, it means to lead to the values of the anomalous field.

After creating a polygonal model in the model editor, there are two work ways: field modeling and inversion. When modeling (to run it, double-click on the gravity and magnetic data window), the program solves forward problem for gravity/magnetic field based on specified gravity/magnetic properties of each polygon (the dialogue is on Fig.45). Created polygons gravity/magnetic properties are determined when inversion of gravity/magnetic measurements (**Modeling/Gravimagnetic/Inversion**).


The process of polygonal model selection corresponding to observed data, described in the part «[Working with gravity and magnetic data measurements](#)».

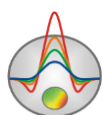
Data of self potential method import

ZondRes2D allows you to work with data of self-potential method by analogy with gravity and magnetic field. The program has opportunity to solve self-potential method forward problem for polygonal models surface bodies polarization based on their resistivity and specified law of redox potential variation with depth. Options for working with data of self-potential method are intended mainly to solve ore problems and are not suitable to study filtration potentials.

Polygonal model, as in the case of gravity and magnetic data, is built in polygonal modeling mode, based on geoelectric section.

Input file of self potential method consists of four columns: the recording point (Profile coordinate to which the measurement is related during observation plotting, m), M electrode coordinate (m), N electrode coordinate (m), field value (mV). Thus, the program works with data of self potential method, obtained both by the potential method, and by the gradient method. File import is realized in polygon mode using the menu item **Modeling/Self potential/Load data**.

The menu item **Modeling/Self potential/Display menu SP window** runs the self-potential method graphs displaying window - observed and calculated. Calculated graph is displayed after changing parameters of the redox potential variation with depth (**Modeling/Self Potential/Redox factor vs Z**, Fig. 61) and solving the forward problem (press the space bar or the button ).



The law of redox factor variation with depth (linear or exponential) and the corresponding coefficients are redox potential variable parameters.

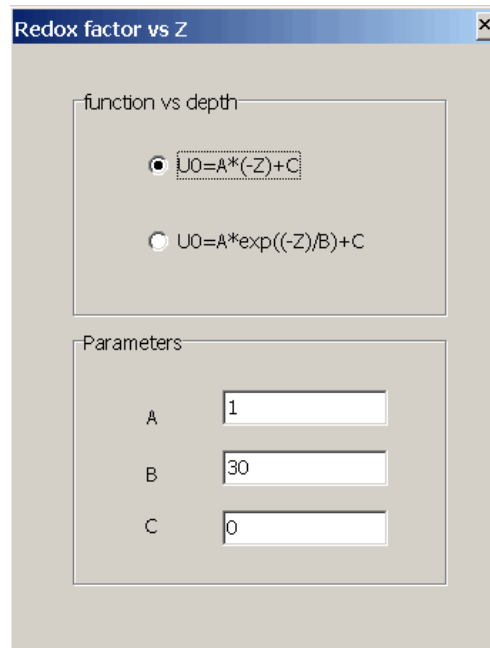


Fig. 61 Gravity and magnetic data dialog box

An example of a model of self-potential method field and calculated graphs shown in Fig. 62.

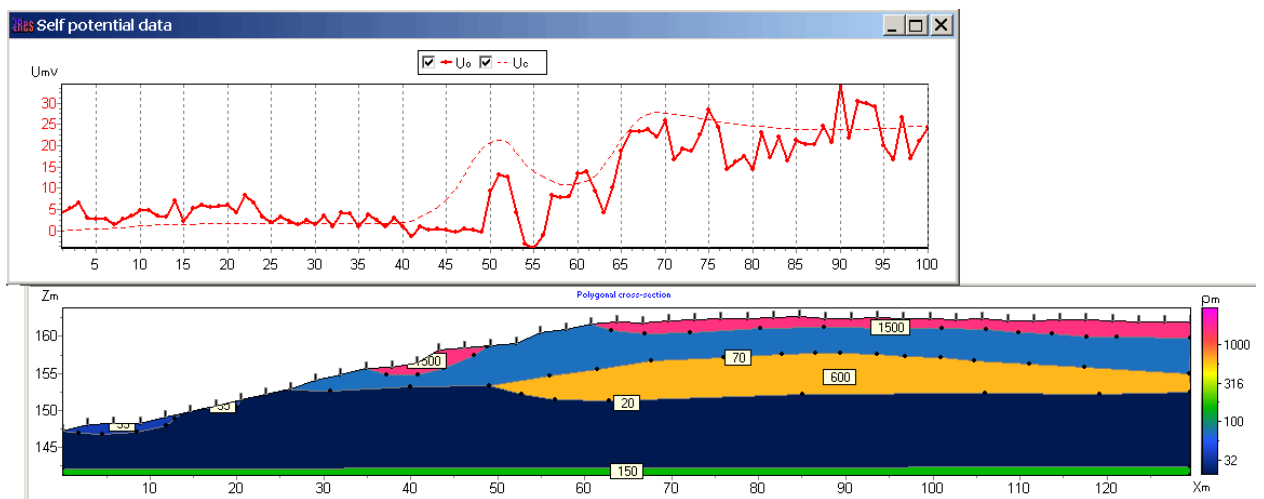
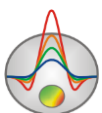


Fig. 62 The self-potential method data and resistivity polygonal model



Magnetotelluric data import

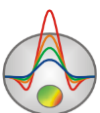
Magnetotelluric data can be imported into the project via the menu item **Options/MT data/Load MT data**, opening file dialog with *.m2d format.

The *.m2d file structure is, in general, similar to the *.z1d file structure, containing VES data: The first line contains the period enumeration (c). The data description for each measurement point is following. Before and after the description of each point put a line containing the brace. The next line after the brace - point name. After this is a line containing topographical information - profile distance (km), height (km) and rectangular coordinates of observation point.

Lines with data description are followed after the topography line. Each line corresponds to its data type, and begins with a key: ro_a_tm (apparent resistivity for the TM mode), ro_a_te (apparent resistivity for the TE mode), phi_tm (phase for the TM mode), phi_te (phase for the TE mode). Parameter values for each period are written after the key in accordance to description in the periods line (the first file line). Description of data measurement points is completed by a blank line with the brace.

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

```
0.0001 0.0001166 0.0001359 0.0001585 0.0001848
{
Point1name
0 0.788 5489.915738 0
ro_a_tm 897.408 992.547 1097.806 1214.112 1342.435
ro_a_te 531.065 572.484 616.483 662.444 710.411
phi_tm 15.846 15.951 16.07 16.212 16.363
phi_te 23.904 24.605 25.3 2 5.971 26.641
}
{
Point2name
0.103680119 0.76 5489.943395 0
ro_a_tm 411.023 458.506 511.063 569.166 632.274
ro_a_te 145.856 165.762 188.024 212.573 239.514
phi_tm 15.756 15.741 15.738 15.775 15.854
```



```

phi_te  6.53      8.194      9.846      11.47      13.058
}


```

After reading the data file the dialog **Enter shift of stations** is called. It allows you to enter the offset in the profile (in km) for all observation points.

Using option **Options/MT data/Show MT plot** you can go in the observed and calculated MT data visualization window.

Exporting image setup dialog

Dialog **Picture settings** (Fig. 63) allows adjusting the vertical scale **Vertical scale** (in meters per centimetre), horizontal scale **Horizontal scale** (in meters per centimetre), the resolution of the exporting image **Print resolution** (in DPI) and font size **Font size**.

These settings apply to model (Model) saved in BMP format , if the option **Automatic** is turned off. Otherwise, the image is saved in the same form as displayed in the screen.

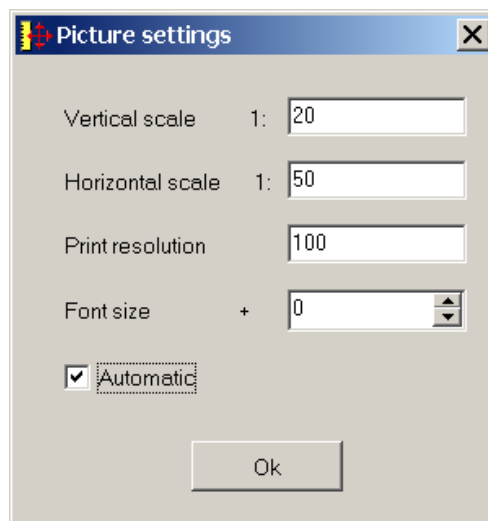
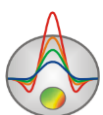


Fig. 63 **Picture settings** dialog window

Logging and lithology data file format

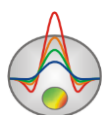
Logging data and lithologic columns are hold in certain file formats. First type of files has txt extension; these files contain logging and lithology data. The following structure is used to create logging data file:



First column contains measure point depth (from surface), second column contains borehole log measurements. Third and fourth columns are filled with zeroes.

Logging data sample-file is given below (fig. 64):

0.5	118.3035394	0	0
1	126.9002384	0	0
1.5	123.4170888	0	0
2	116.1519574	0	0
2.5	117.240884	0	0
3	111.9424174	0	0
3.5	142.0405875	0	0
4	125.3686538	0	0
4.5	521.0730567	0	0
5	735.5232592	0	0
5.5	707.7315998	0	0
6	706.3561614	0	0
6.5	725.9945623	0	0
7	722.433627	0	0
7.5	717.0991126	0	0
8	716.9836552	0	0
8.5	725.5024012	0	0
9	722.3551713	0	0
9.5	731.5717173	0	0
10	723.5097884	0	0
10.5	726.8844987	0	0
11	725.962034	0	0
11.5	743.2485878	0	0
12	726.4061156	0	0
12.5	734.399887	0	0
13	727.9166309	0	0
13.5	116.1921851	0	0
14	517.9613065	0	0
14.5	125.3706264	0	0
15	111.2952478	0	0
15.5	131.911879	0	0
16	107.9217309	0	0
16.5	114.9327361	0	0
17	134.0939196	0	0
17.5	138.4457143	0	0
18	129.1165104	0	0



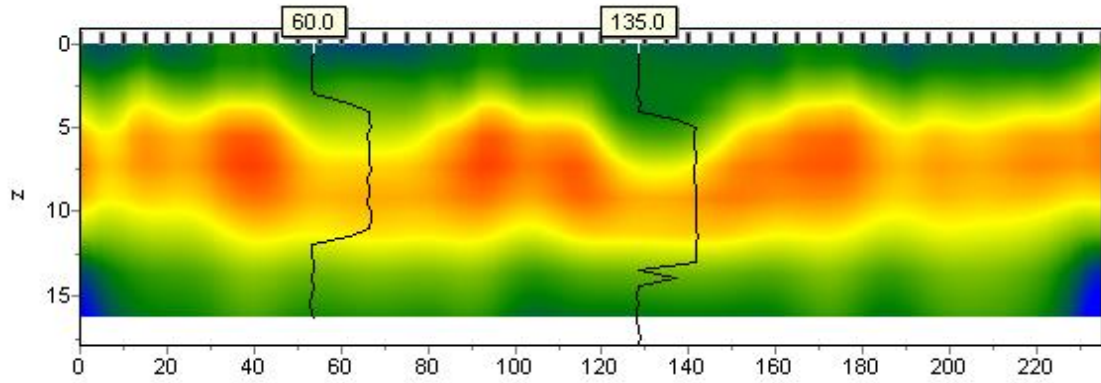


Fig. 64 Model with plotted borehole logs

The following structure is used to create lithology data file:

First column contains lithologic layer depth (from surface). Second column is filled with zeroes. Third column defines layer color for visualization, forth – type of pattern. An example model with plotted lithologic columns is shown on fig. 65.

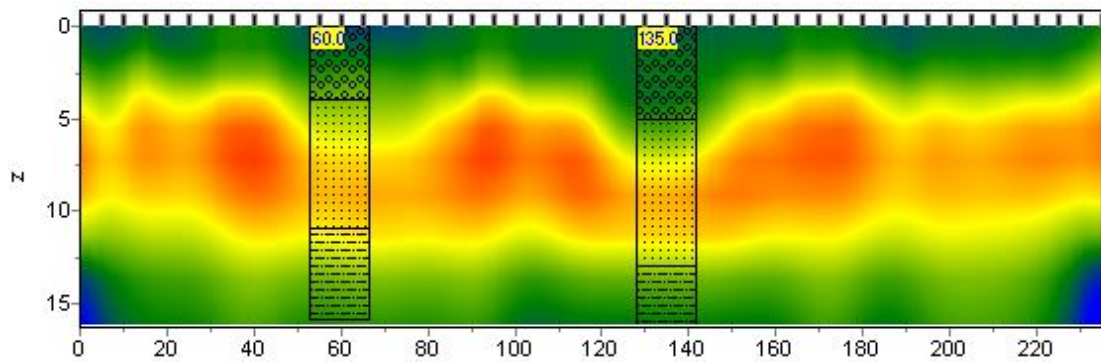


Fig. 65 Model with plotted lithologic columns

First 23 patterns for lithologic column creation are given below (fig. 66).

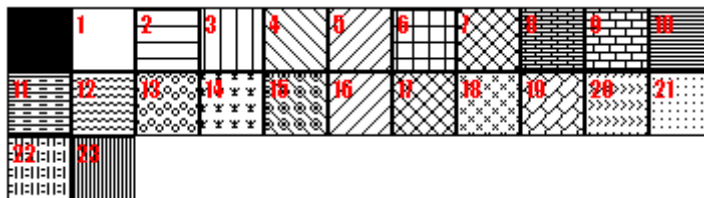
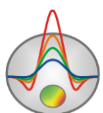


Fig. 66 Types of lithologic pattern



Lithologic data sample-file is given below:

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

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

2280.txt First line – logging or lithology data file
brhl2280 Second line – Borehole name (is displayed on borehole)
18 2 2 1 0 1 0 0 Third line contains control parameters

Data record 18 – borehole coordinate on profile.

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

2 – type of data visualization 0 - 3.

0 - logging data (as graph); [Sample-file - carot1.crt](#)

1 - logging data (interpolated color column), section color scale is used for visualization; [Sample-file - carot2.crt](#)

2 - lithologic column; [Sample-file - strati.crt](#)

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

1 – Logging data normalization parameter 0 - 2.

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

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

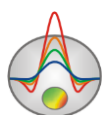
0 - Logging method index (if different logging methods are displayed indices of all methods should be specified) 0 – n-1, where n – number of methods.

1 – Plot color.

0 – Data scale is logarithmic 0 or linear 1.

0 – Vertical borehole shift relative to the earth's surface.





3246.txt Description of the following borehole on profile



brhl3246

102 2 2 1 0 1 0 0

To create lithology data files it is recommended to use the module called by a main menu item **Options/Borehole/Create/Edit borehole data** (Fig. 67). The module allows creating borehole lithological column along the observation profile, save files with created lithologic information, load and edit previously saved files, and perform these operations with the logging data.

To create (delete) boreholes press the buttons  and . Text field, which indicates the borehole position in the profile coordinates is (465m on fig. 67). To jump between boreholes to edit them use the buttons  .

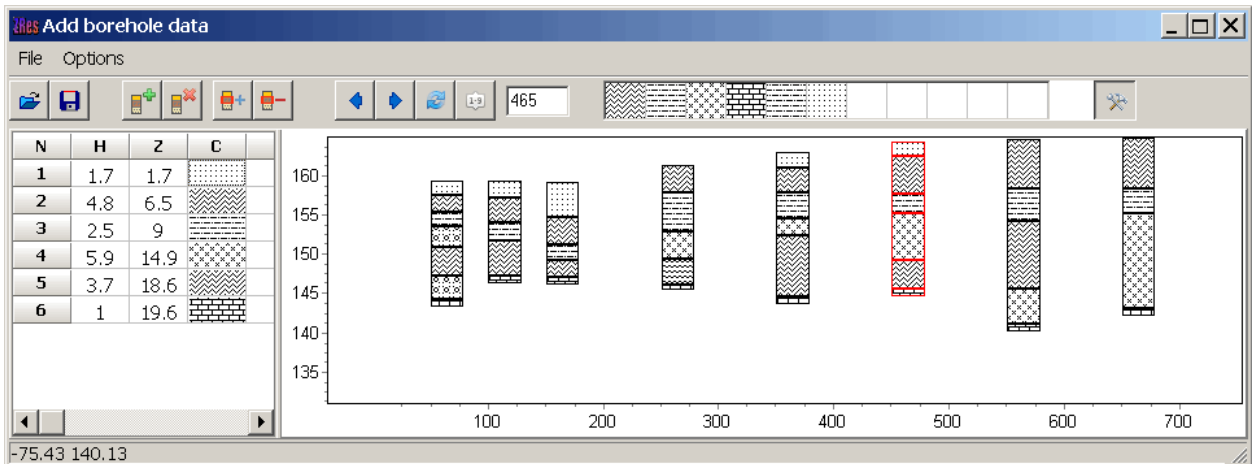


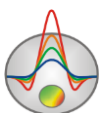


Fig. 67 Working window of borehole data setting module


After jumping to editing borehole (borehole, planned position which is specified in the field) using the buttons  and  new layers can be added or removed while editing. Created layers thickness are specified in the left column of the table (**H**). In the right column (**Z**) is automatically displayed layer bottom depth from the surface at each observation point.

Column **C** contains the image of the layer hatching. Choosing a hatch is done in the window called up by clicking on the appropriate column C cell.

As a rule, boreholes, which locate along the profile, contain common lithological layers. Therefore, for comfortable setting hatching to each time not to look right, it is possible to moving the selected hatching to the hatch panel at the window top (to the right of indicating the



borehole position), and then taking the hatches for a layer from toolbar. If select a cell with hatch in C column and right-click on panel cell, the hatch is copied to the panel cell. By left-clicking on the panel cell, the hatch from panel is copied to the table cell.

The column width (in percents from the total section length) is set in the menu Set borehole width, called by pressing the button . This option is available from the main window (Options/Borehole/Set column width).

Toolbar buttons of lithology data setting mode are duplicated in the menu **Options**. The function **Remove background** is at same place. It allows removing background - resistivity model when setting borehole data.

When creating or editing borehole data using described module the result is saved by program to a file of internal format with the *.bth extension.


Working process with borehole data creation mode is shown on the movie <http://zond-geo.ru/zmovies/addBH2d.exe>.

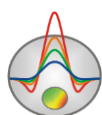
Additional features of model visualization


The program operates with the following types of images:

Two parameters simultaneously – resistivity and polarizability. Contours of one parameter overlay contours of another (in **Contour-section** mode) (fig. 68 A). To run this option use **Options/Extra/Display Rho&Etha together** in the main menu. Use **Options/Extra/Extra isolines color settings** dialog to adjust isolines parameters of second parameter contours.

Potential and sensitivity distribution analysis gives better understanding of electrotomography principle of operation.

Potential distribution – potential contours are displayed over section for any source (fig. 68 B). Source position is selected in electrodes editor. This option works in Block and Smooth section modes. Use **Options/Extra/Potential&sensitivity in model/Potential isolines** in the main menu to run it. Press  button in electrodes editor to activate contour parameters setup dialog. This dialog is similar to pseudosection parameters setup dialog ([Appendix 4: Pseudosection \(contour section\) parameters setup dialog](#)).



Sensitivity function distribution – sensitivity contours are displayed over section for current measurement (fig. 68 C). Measurement is selected in electrodes editor table. This option works in Block and Smooth section modes. Use **Options/Extra/ Potential&sensitivity in model/Sensitivity isolines** in the main menu to run it. Press  button in electrodes editor (**Options/Data editor**) to activate contour parameters setup dialog. This dialog is similar to pseudosection parameters setup dialog ([Appendix 4: Pseudosection \(contour section\) parameters setup dialog](#)).

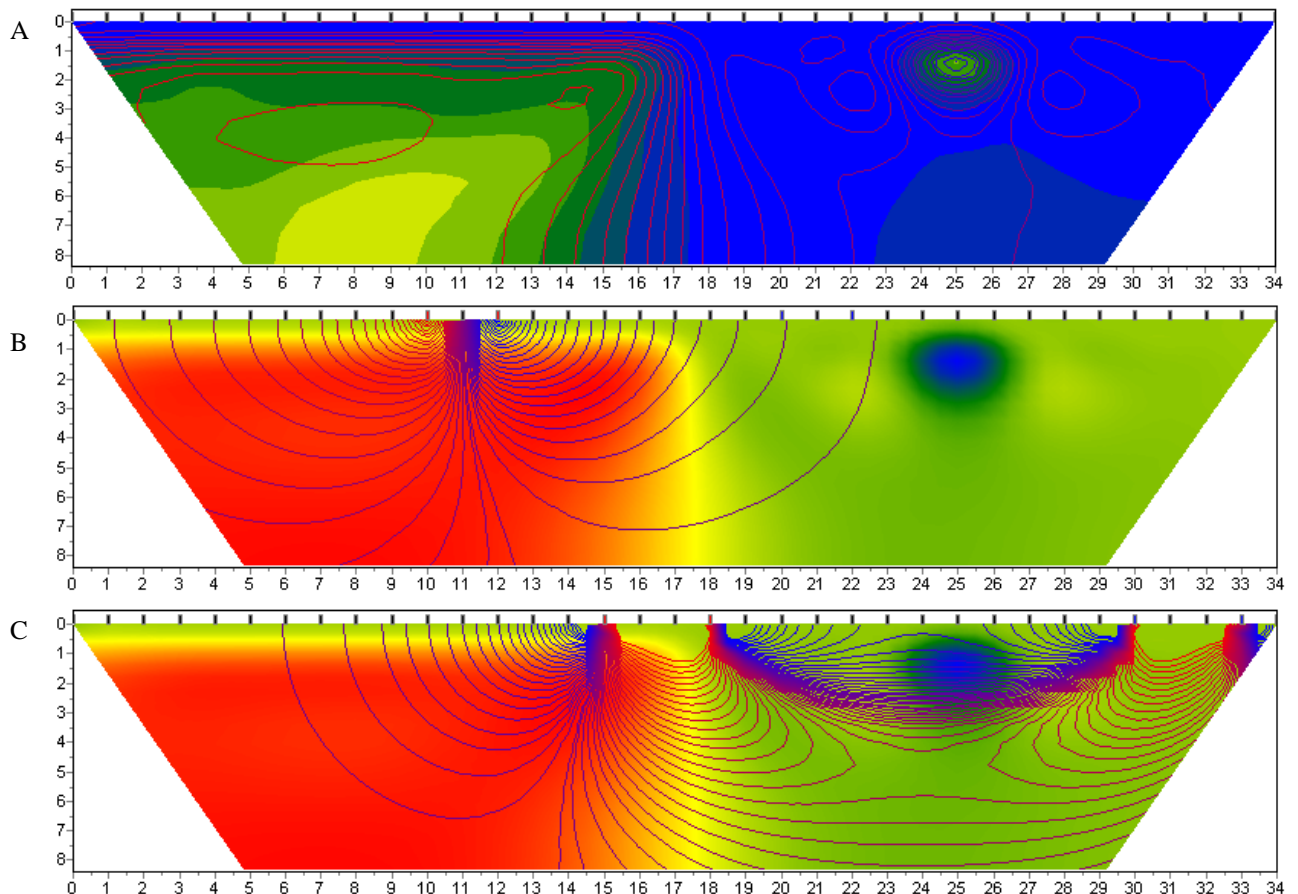
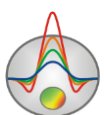


Fig. 68 A. Resistivity section with plotted polarizability contours. B. Resistivity section with plotted potential function contours. C. Resistivity section with plotted sensitivity function contours.



Window of geological-geophysical model builder

To create the geological-geophysical model (to carry out geological interpretation) use the editor of the geological model Geological editor, run it by using the menu **Options/Geological editor**. The editor allows interactively creating a geological model based on the current project model, borehole data, data from other Zond programs, a priori and raster information, print obtained sections in a given scale, save and export the interpretation results.

When running the geological model editor, it displays the current project model (Fig. 69). To load a priori information in *.sec format use the menu item **File/Open section**, to load borehole data – use **Options/Load borehole data**.

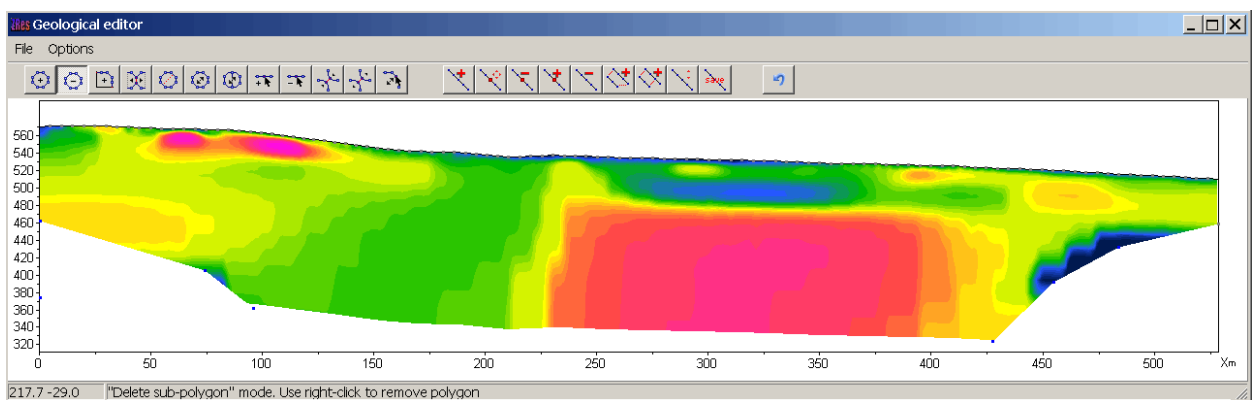






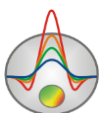






Fig. 69 Model editor window before starting work

To carry out geological interpretation on the toolbar, there are two button sets: one for creating polygonal bodies (with the possibility of editing hatches, fills, boundary style, etc.), and the other - to draw individual lines on the section. They can be used to indicate certain elements of the geological structure (for example, tectonic discontinuities) or generally improve the visualization of the section interpretation.

Working with tools for creating bodies is the same of polygonal modelling tools (more details in Section «[Polygonal modeling](#)»). Tools of creating lines is, in principle, similar in their functions:

	Create line		Move node
	Remove line		Close polyline
	Move line		Close polyline



	Add node		Save changes
	Remove node		Override

Example of created geoelectric section and created on the basis of its geological model is shown in Fig. 56. The menu item **File/Remove picture** allows removing background - the parameter section, based on which the interpretation is carried out (Fig. 71).

Using the menu item **File/Save section**, it is possible to save the results in *.sec format to work in other Zond programs or as a raster image. Menu Item **Options/Output settings** allows setting image scale to export or print (**File/Print preview**).

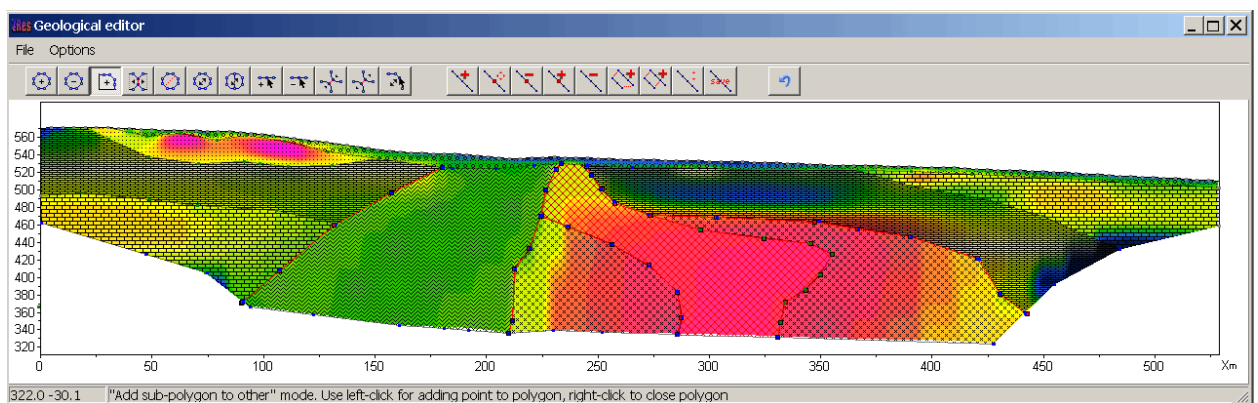


Fig. 70 Model editor window: the resistivity section and geological interpretation

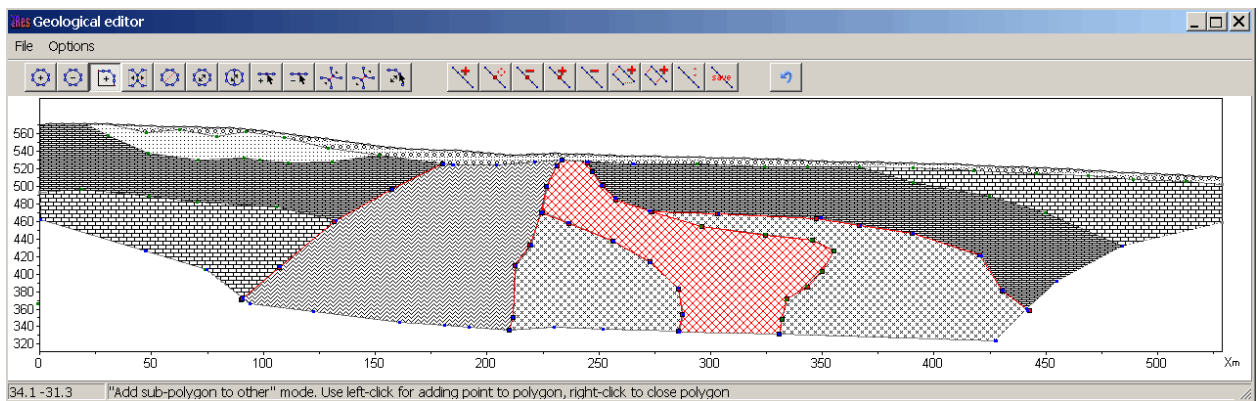
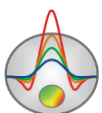


Fig. 71 Model editor window: geological-geophysical section based on the interpretation results

Geoelectric models volumetric visualization using several profiles



If several closely spaced profiles were measured, it is advisable to carry out their joint interpretation. It enables investigation of structures along strike and also facilitates the interpretation of each profile separately, simplifying the validation of the most reliable features of the model.

To run joint visualization of geoelectric models and polarizability models obtained by several profiles use the 3D section viewer module. Run it from the menu item **Options/3D sections plot**. It allows displaying the obtained models in the extent (accounting relief), and also building the selected parameter distributions in plan for a user-specified depth or absolute height.

Volume visualization window consists of three tabs - Lines (to set profile coordinates), 3D View (to display model), Options (image settings), and the toolbar (provides access to additional parameters and the ability to load, save and export the created three-dimensional model).

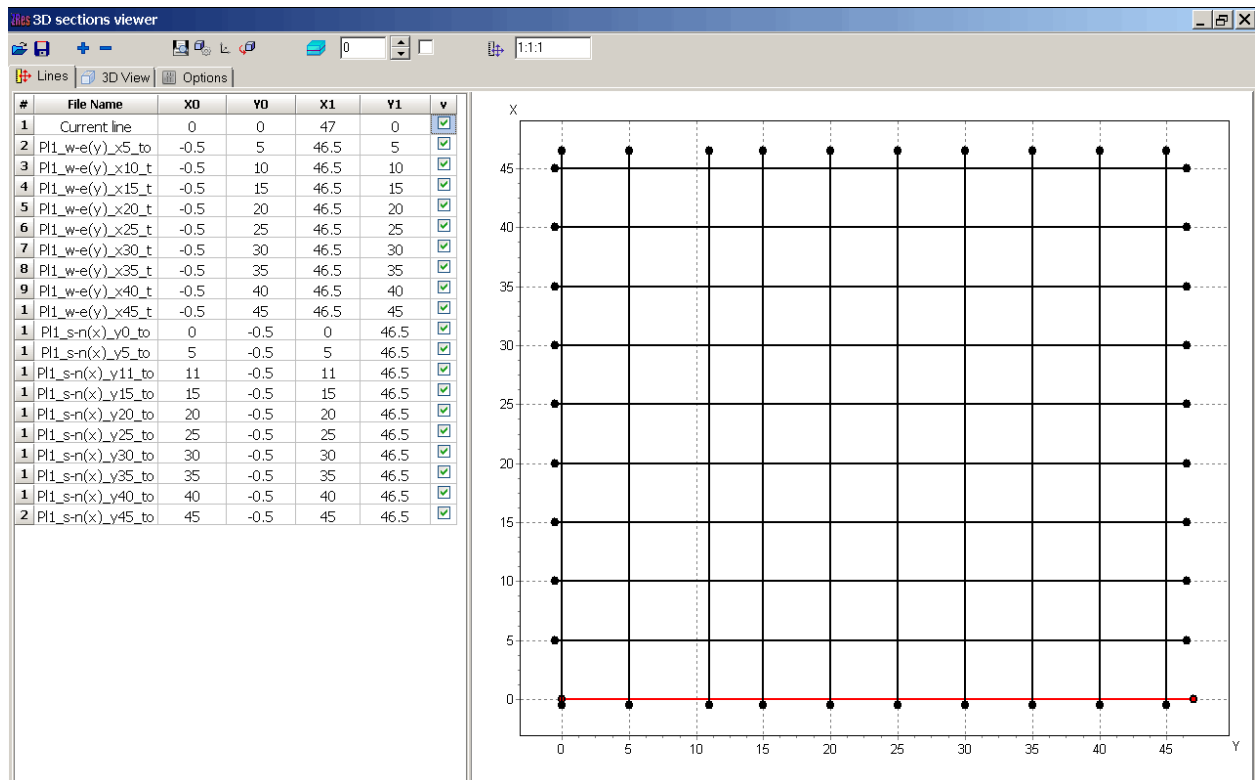
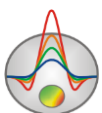


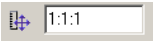



Fig. 72 Tab Lines of the 3D section viewer.

To add profiles and set coordinates use the tab Lines (Fig. 72). Each profile corresponds to a row in the table. To add a profile to the table, right-click on an empty cell of the first column



and select the desired file. For volume visualization program uses the file with extension *.mod2d, which is created automatically when saving the main project in the format ZondRes2D. To add an empty row to the table, or remove unnecessary file from it, use the buttons  and  on the toolbar. Table columns X0, Y0, X1, Y1 contain rectangular coordinates of the beginning and end of the corresponding profile. It should be noted that it permits arbitrary mutual profiles orientations - they can be parallel, non-parallel, intersecting or non-intersecting. Profiles plan is displayed as coordinates are being input in the right part of tab Lines. The last table column allows disabling visibility of some profiles.

Tab Options allows setting visualization parameters - color scale and limits of axes. The ratio of the scale can also be set using the field  on the toolbar. The field contains also scales of displaying of each axis. Press the button  to go to the maximum scale for an area bounded by axes.

The volume model is displayed in the tab 3D View (Fig. 73).

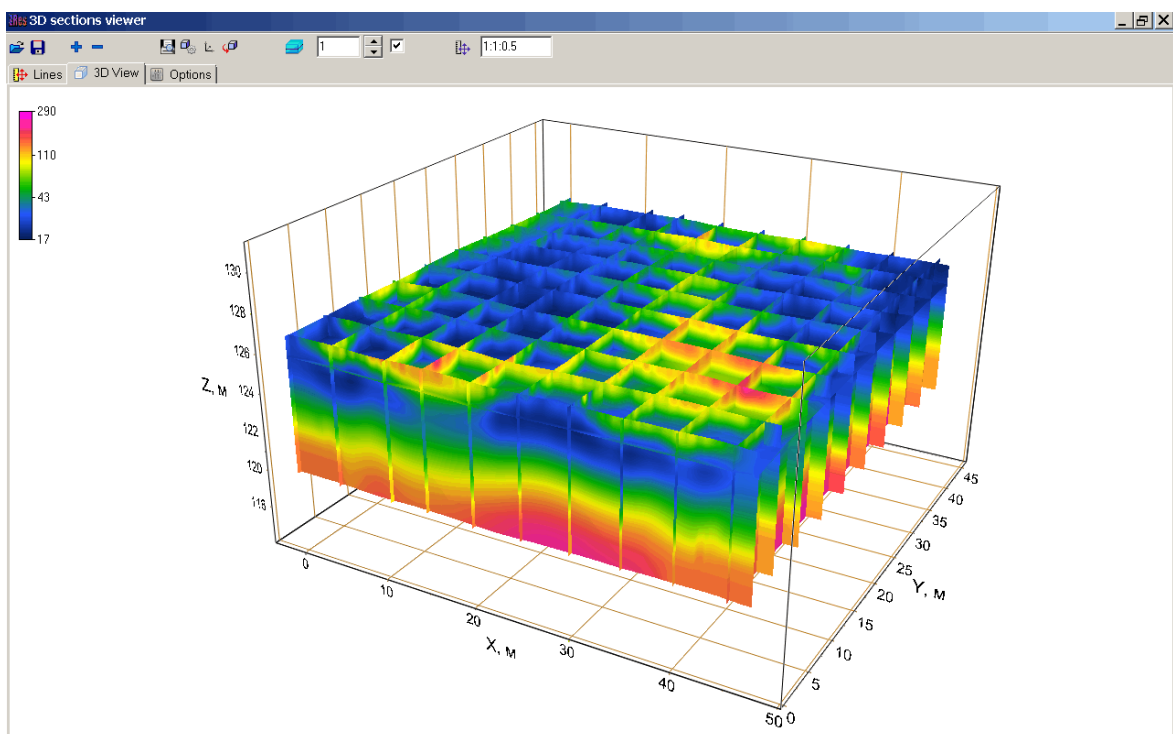
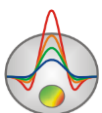






Fig. 73 Tab 3D View of 3D section viewer window. Volume visualization



Axis setting (individually) is carried out by dialogs ran with toolbar button . The corresponding dialog allows changing the axis label properties, set the plot grid, set an axis header, etc.

Access to settings related to the three-dimensional model visualization (projection types, scales, rotation, translate the whole model in screen coordinates, etc.) is done by pressing the button .

Working with the created model is also possible using the mouse. When pressing left button, it is possible to rotate the model, using the mouse wheel zoom is reached. When pressing the button  the model automatically rotates around its geometric centre.

A useful option is to create a map of parameter distribution. To access the corresponding option use the toolbar tools set . If checkbox is inactive, absolute altitude is input for which the map will be created. If the tick is set, the depth from the surface should be input in the numerical box. An example of mapping for the model is shown in Fig. 73 and Fig. 74.

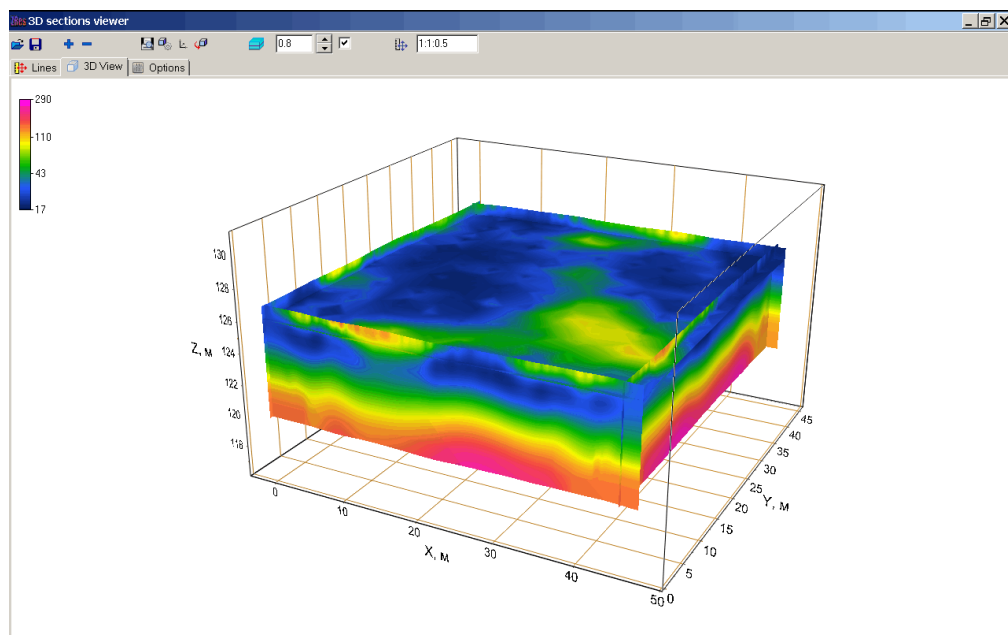

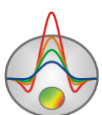




Fig. 74 Tab View 3D of window 3D section viewer. Resistivity map at a given depth from the surface

The results of working with the module 3D section viewer can be saved, exported and printed. To save and export use toolbar button .



program will create a file with the extension *.prj, which will contain absolute paths to files - profiles and their coordinates. Further when opening this file using the button , the program reads the data and re-builds a three-dimensional model. If in the saving dialog XY plane is selected, the program will create *.dat text file, containing the data for parameter map at the selected depth. This file can be further used in other programs, such as Surfer. When saving in Voxler 3d grid format, program creates *.dat text file, containing data for all model.

To print model window press the button  on the toolbar.

Working with 3D visualization of profile set is shown on the movie <http://zond-geo.ru/zmovies/mline2dvisualiz.exe>.

Specifics of work with topography data and marine measurements

It is important to specify model with due account to measurement surface relief [Dahlin, Loke, Oldenburg]. There are different keys in the program for fast and proper relief data entry (see «[2nd part of data file: Topography data](#)» for details). Option for topography smoothing (**Options/Advanced/Smooth topography**) is recommended in case of rough terrain (fig. 75). This option should be turned on before opening data file.

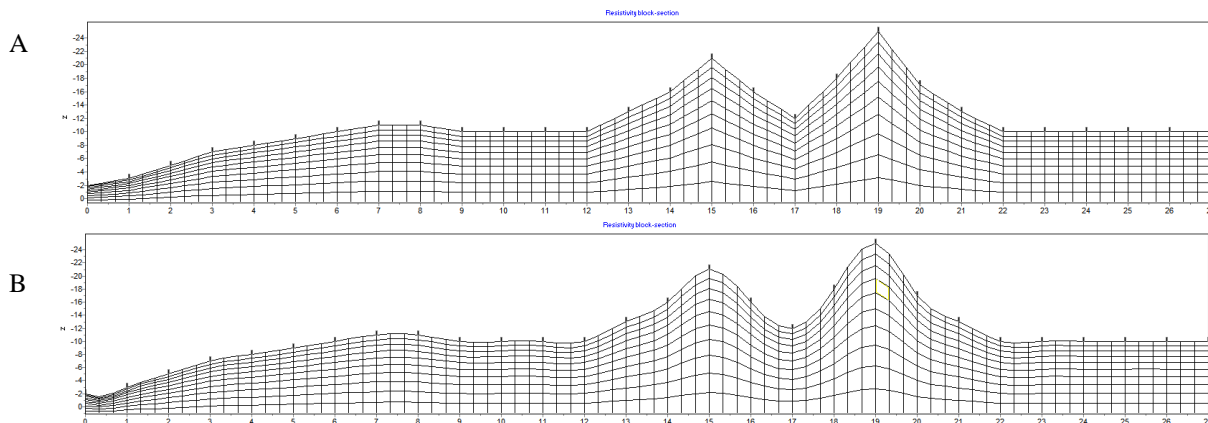
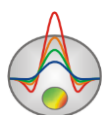


Fig. 75 Model mesh without smoothing (A) and with it (B)

In order to take into account relief beyond model domain complementary nodes should be added on model borders (**Options/Extra/Include extended nodes**). This option should be turned



on before opening file. Complementary nodes can be also added in the file before topography description, after *** key.

Factor of relief distortion with depth (0-5) can be set in the menu **Option/Topography/Topo coefficient**. 0 - relief of each following layer repeat the previous one. 1 - relief flattens with depth, the last layer - flat (fig. 76). Distorted depth is calculated by the following formula:

$$z^*(x, z) = Topo(x) + z \cdot \left(1 + \frac{\max(Topo) - Topo(x)}{\max(z)} \cdot Tcoeff \right), \quad (16)$$

where Topo – relief excess, z – depth from the surface.

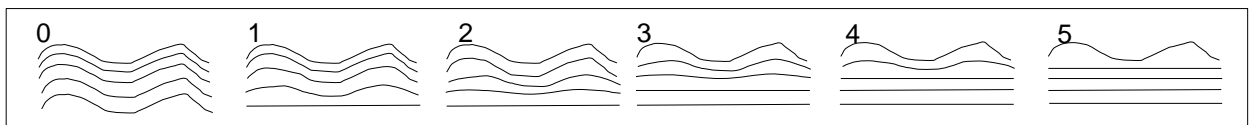


Fig. 76 Model layers distortion using the parameter **Topo coefficient** from 1 to 5

It is useful sometimes to switch OFF relief. To do it use the option **Options/Topography/Remove topography**. To restore relief use the option **Options/Topography/Restore topography**.

To edit topography data, use the option **Options/Topography/Edit topography**.

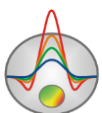
On default relief excesses are plotted relative to zero but if **Options/Topography/Absolute coordinates** option is used relief excesses specified in file are displayed.

Option **Absolute coordinates** should be turned on to save **Grid file** with actual excesses in outbound file.

It is desirable to have water resistivity values for marine survey data interpretation. Use selection option **Options/Inversion/Underwater options/ Invert** if water resistivity data is absent.

Water resistivity can be specified using **Options/Inversion/Underwater options/ Resistivity** option.

Use **Options/Inversion/Underwater options/ Sublayers number** option to specify number of water layer splits. It depends on water layer thickness.



Specifics of work with borehole data measurements

ZondRes2D allows to model and perform the inversion of borehole measurements. These include inter-borehole measurements, borehole-surface measurements, and their combinations. It supports both vertical and deviated boreholes.

The measurement configuration is described in the input data file of the ZondRes2D format (more details in the part «[Main data file format](#)»). When describing electrode measurements located in boreholes, as in the case of terrestrial measurements, planning altitude coordinates are indicated. There is no opportunity to create measurement configuration containing electrodes in borehole in the program. The corresponding data file is created in text or tabular editor ([Sample-file – sample_with_z_source](#)).

Modeling and inversion of borehole and borehole-surface data is the same as for ground-based measurements. The difference is in the method of apparent resistivity displaying - unlike conventional methods for ground-based measurements dependence of the apparent resistivity from the array factor are built here (Fig. 77). Measured and calculated data can also be viewed in a separate window via the option **Options/Data/G-Res plot**.

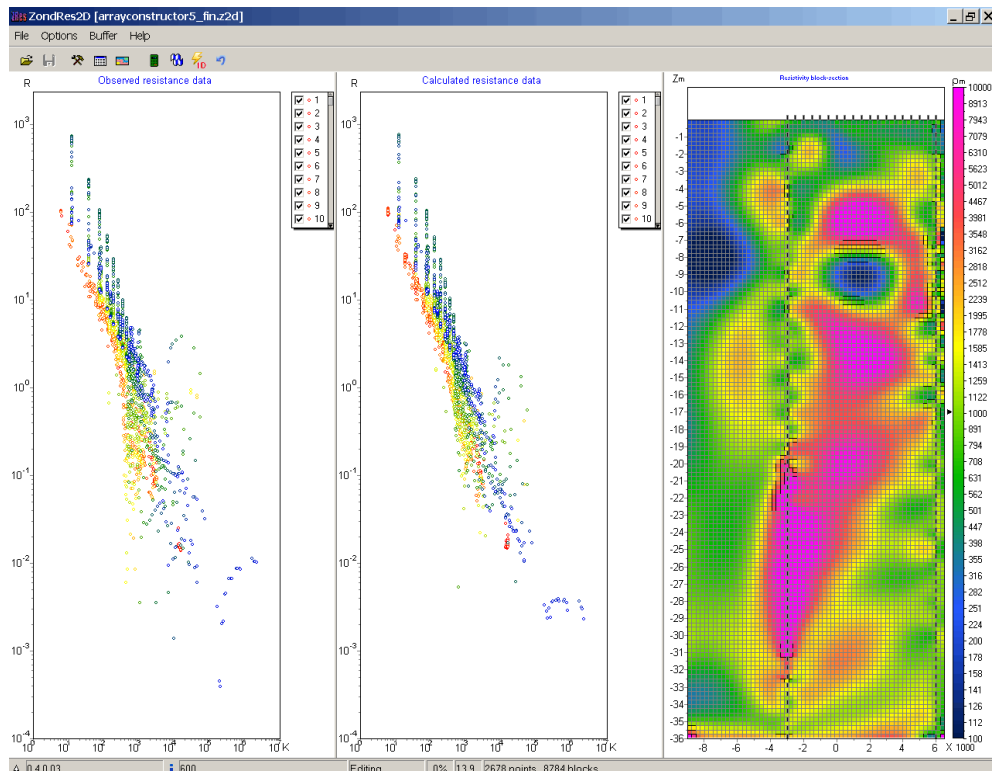
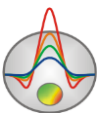


Fig. 77 Main program window when working with borehole measurements



IP parameters inversion

ZondRes2D allows you to work not only with integral polarizability values, but also with IP process parameters. In practice, appropriate investigations are carried out in one of two modes - time or frequency. When working in the time domain, measurement result for each observation point is a IP decay curve - voltage dependence (sometimes normalized by width of the measurement time window) from time on a fixed set of delay times ([sample-file – timedomain.zd2](#)). Dependences of resistivity and phases shift from frequency for a fixed frequencies set ([sample-file - frequencydomain.z2d](#)) are input data for each observation point, when working in the frequency domain.

For both modes the program allows visualizing the source data, carrying out their smoothing and performing inversion, determining for all observation points integral IP value for each delay time (frequency) or the Cole-Cole model parameters.

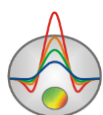
The input data file format for IP parameters inversion (extension *.z2d): the first line contains the times delay lists for time and frequencies for frequency measurement mode after the word time_ # chann. The following is a table whose rows correspond to the points of observation, and the columns - to electrodes coordinates (similar to [The main data file format](#)), to the Res ($\Delta U/I$) value and to the measurement values for the respective delay times (column headings - ipi1, ipi2 etc.) and frequencies (the column headings - mod1, mod2, etc. for module, pha1, pha2 etc. for phase).

An example of the measurement time mode data file beginning:

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

An example of the frequency measurement mode data file beginning:




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

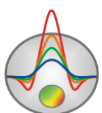


5	*	10	15	0.001	0.001	0.001	0.001	-0.005	-0.007	-0.012
10	*	15	20	0.002	0.002	0.002	0.002	-0.005	-0.009	-0.014

To work with IP parameters use the menu item **Time lapse/domain** of the main menu (menu appears when you load the appropriate data file). To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.

Option **Time lapse/domain/Channels** runs the channel setting table. It is possible to edit the values of the time delay or the measurement frequencies and headers for specified values (will be displayed in the drop-down list on the toolbar of the main window). For the some options operation it is necessary to times/frequencies were inputted previously.

Option **Time lapse/domain/Display channel plot** runs the window (Fig. 78), showing the decay curves (frequency curves) for each measurement point. To move to another measurement use the mouse (on pseudosection or graph) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point displays. A separate point of the displaying curve can be edited using the menu items. They appears, when you press the left mouse button on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), to return to the previously position (**Origin point**). The toolbar button  allows you to smooth the entire curve.



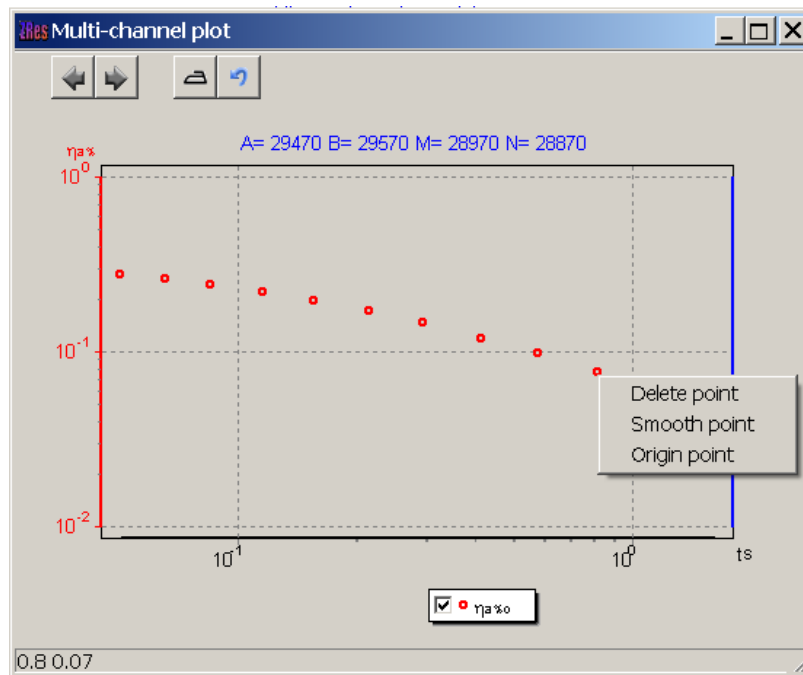
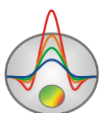


Fig. 78 The curve editing window for a single measurement point

When working in the time mode after editing curves it is necessary to use the option **Time lapse/domain/Recalculate chargeability**, which recounts an integral IP parameter accounting data changes.

The option **Time lapse/domain/Time lapse inversion** runs IP data serial inversion mode for each implementation (in this case, the standard calculation scheme, as well as for the integral IP parameter). If the option **Time lapse/domain/Next from previous** is switched ON, as an initial approximation for each next iteration the results of previous one is used. If this option is switched OFF, as an initial approximation for each next iteration the current implementation model is used. These models are displayed in the model editor. To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.



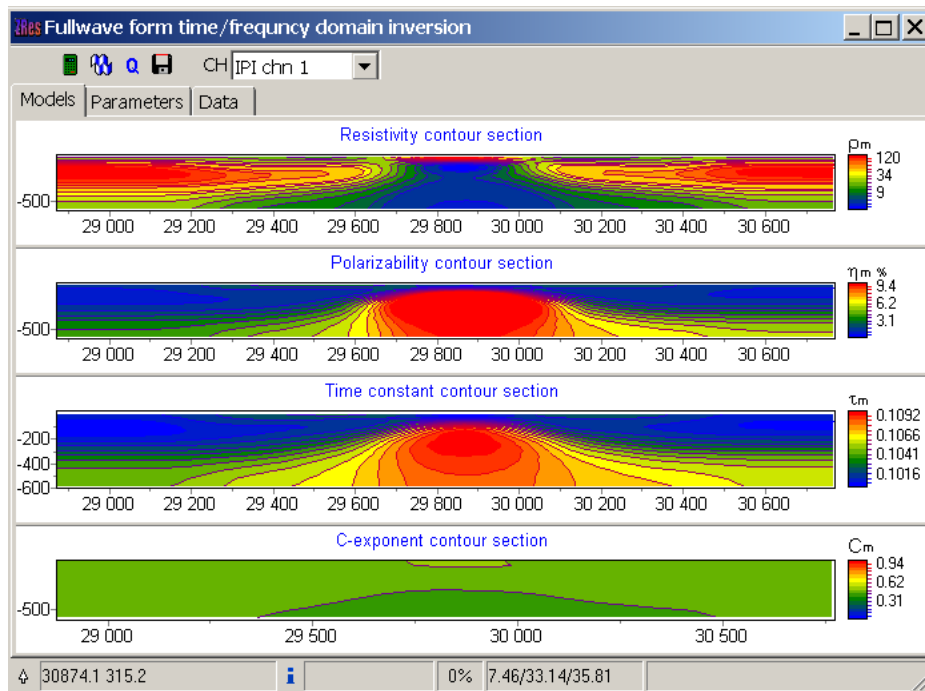


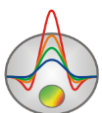
Fig. 79 The main window of the IP parameters module inversion

It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Window IP parameters full-wave inversion (Fig. 79) is run via the menu item **Time lapse/domain/ Fullwave form inversion**. The window contains three tabs. Tab **Models** displays resistivity, polarizability and Cole-Cole parameters (τ , and c), which are obtained as model inversion results.

Tab **Parameters** allows you to select parameters for which you want to carry out inversion and specify (fix) ranges of these parameters (Fig. 80). **ReBuild** button allows changing the each parameter values to the value from the table (value). If instead of the number symbol "*" are specified, the parameter value on the section will not be replaced.



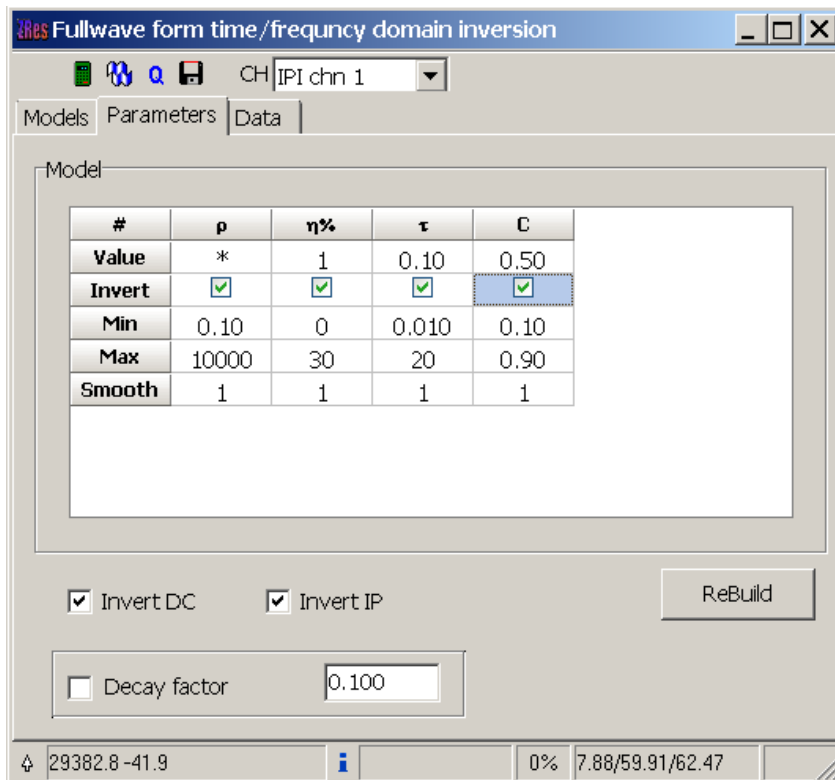




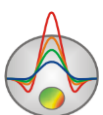
Fig. 80 Window for selecting parameters for curves IP inversion

Option **Module** (available when working in the frequency mode) indicates whether or not to select the signal module, or use phase only. As a rule, used in the frequency range module changes very little, and unlike the phase, has little effect on the inversion results.

To start inversion, press the button . This inversion occurs simultaneously on all selected parameters. Button  provides an alternative method in which the polarizability inversion occurs independently for each time (frequency) by time-lapse inversion and Cole-Cole model parameters are recovering based on obtained "real" curves.

Tab **Data** displays observed and calculated polarizability model. The drop-down list in the toolbar (available when working with all the tabs) allows switching between sections for different decay times (frequencies).

Dialog **Time lapse/domain/ Digital filter** contains temporary filter settings for the Fourier transform to the time domain, and allows you to set the length of the current impulse. *Video dedicated to the IP data inversion in the time and frequency domain, can be downloaded from <http://zond-geo.ru/zmovies/TDvsFD.exe>.*



Geoelectric monitoring data inversion

Monitoring involves measurements with the same parameters at different time. The program recognizes file with the *.z2d extension as an inputting monitoring file. The file has following structure: the first four columns contain electrodes coordinates (similar to [Main data file format](#)); next - columns, entitled res1, res2, etc. that contain Res ($\Delta U/I$) values for the first, second, etc. measurement cycles.

The following example shows the beginning of a monitoring data file by three-electrode array ([sample-file – monitoringERT.z2d](#)):

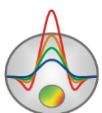
C1	C2	P1	P2	res1	res2	res3	res4	res5
0	*	2	4	5.57	5.57	5.57	5.60	5.63
0	*	4	6	1.55	1.55	1.56	1.59	1.62
0	*	6	8	0.84	0.84	0.85	0.88	0.91
0	*	8	10	0.48	0.48	0.49	0.51	0.52
0	*	10	12	0.44	0.44	0.45	0.49	0.46
0	*	12	14	0.25	0.26	0.26	0.29	0.24




Working with monitoring data, in general, is similar to the work with multifrequency IP data or temporary IP containing information about decay curve. Only instead of the polarizability values at a certain frequency or time is considered the resistivity values at a certain time.

Use the menu item **Time lapse/domain** to work with the monitoring (menu appears when you load the appropriate data file). Switching between the data and the models corresponding to different measurements times is carried out using the drop-down list on the control panel of the main window.

Option **Time lapse/domain/ Channels** runs the table in which you can enter information about each measurements implementation. An information about measurement time is inputted at the **time, s** column. Arbitrary text information, entered in the **name** column, will displayed at the drop-down list on the main window control panel (when switching between implementations).

Option **Time lapse/domain/ Display channel plot** runs the window that displays the Res ($\Delta U/I$) values for each measurement point, depending on the time (Fig. 81). To move to another



measurement use the mouse (on pseudosections or graphs) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point are displayed. A separate point of the displaying curve can be edited using appearing menu items when you press the left mouse button menu on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), returned to the previous position (**Origin point**). The toolbar button  allows you to smooth the entire curve.

Option **Time lapse/domain/ Time lapse inversion** runs consistent inversion mode of monitoring data for each implementation. If the option **Time lapse/domain/ Next from previous** is switched ON, results of previous iteration are used as an initial approximation for each next one. If this option is switched OFF, the current model in this implementation is used as an initial approximation for each next iteration. Obtained models are displayed in the model editor. To switch between models corresponding to different times use the drop-down list on the control panel of the main window.

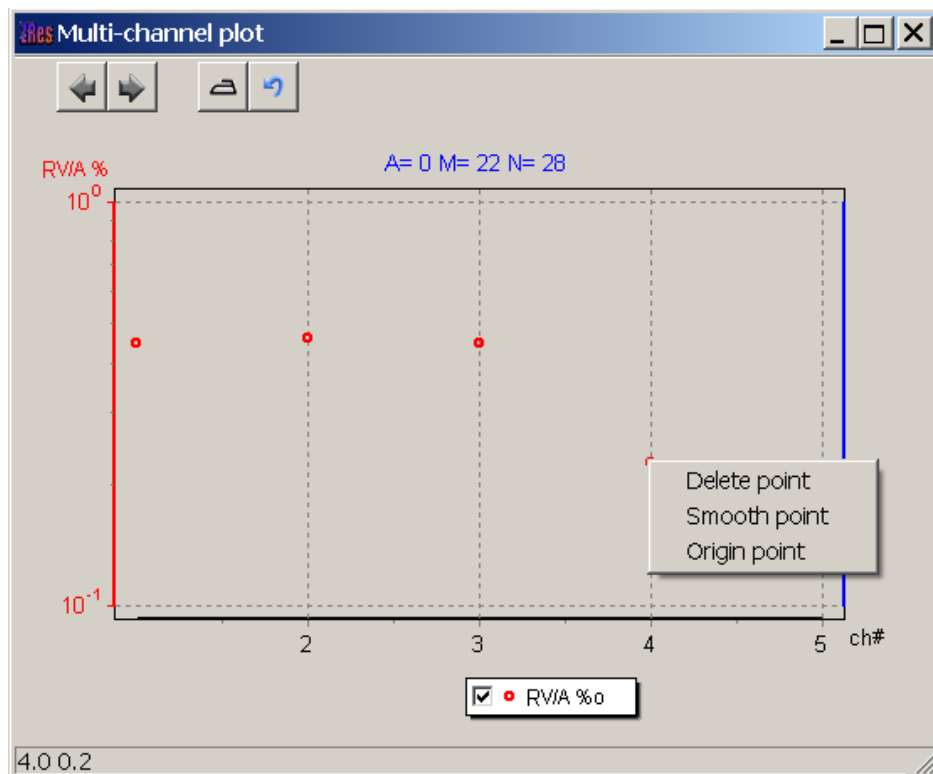
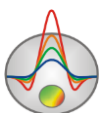


Fig. 81 Window of resistivity changes depending on the time for a single measurement point



It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Sample - video - <http://zond-geo.ru/zmovies/monitoringERT.exe>.

Magnetotelluric data inversion

If you have magneto-telluric data measurement, they can be used in inversion accounting data of resistivity method on DC. To work with magnetotelluric data measurements use the menu item **Options/MT Data**.

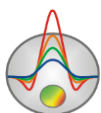
The magnetotelluric data inputting file is a file with *.m2d format. Its structure is described in the section [Magnetotelluric data import](#).

The menu items **Load MT data** and **Show MT plot** allow, respectively, loading and visualizing data of magnetotelluric measurements. Selecting **With static shift** specifies whether static shift (normalization) MTS curves. To specify MT-measurements weight use the menu item **Set weights of MT**.

If you select the menu item **Invert MT data**, the program will carry out joint inversion of MT data and the resistivity method.

Option **DET inversion** allows carrying out inversion simultaneously both longitudinal (TE), and transverse (TM) impedance.

Joint data inversion of electrotomography and PMT is shown in the movie http://zond-geo.ru/zmovies/MT_ERT.exe.

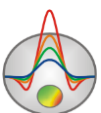


Script-files (batch processing)

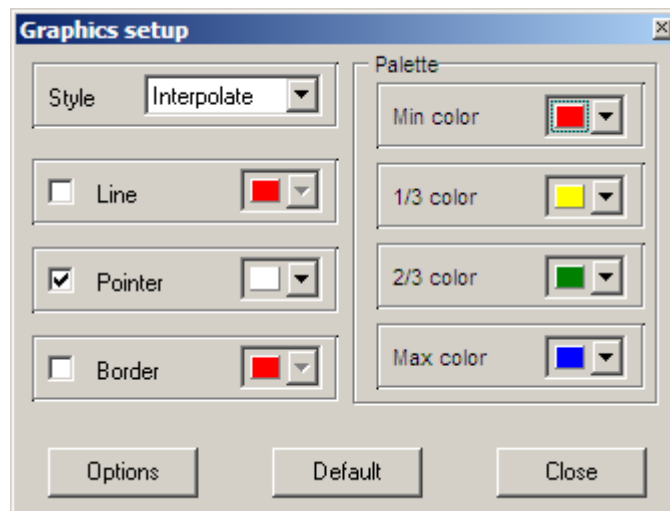
When you need to repeat the same action, they can be written to a script file. When it started, the program will perform the action automatically. For example, you can record the process of opening data file, setting the starting model, the inversion parameters, starting the inversion process and exporting results. If it is necessary to process multiple profiles data by this scheme, run the script and select all data files, for each of them the corresponding actions sequence is automatically will be applied. This tool allows you to save time when the same operations repeat and allows the user to "standardize" the process or individual working process steps with data.

To start recording a script file select the menu item **Script/Start recording script** before performing operations that need to be remembered. If script recording is currently active, the tick will be displayed on the left of the specified menu item. To stop script recording, use the menu item **Script/Stop recording script**, then it is necessary to save the script file (**Script/Save script file**).

To run the script file, it must be loaded **Script/Load script file** and using the menu item **Script/Run packet processing** select the data file for processing. When you save a file processing results in automatic mode, the results file name - the source file name with underscore addition.



Appendix 1: Graphics set editor



Graphics set editor serves for color adjustment of graphics set.

Option **Style** defines algorithm of graphic's color palette specification.

Interpolated palette is used if **Interpolate** is selected. It is created using colors specified in fields **min color**, **1/3 color**, **2/3 color** and **max color**. Value **const** sets the same color (option **color**) for all graphics. Value **random** assigns random colors for all graphics.

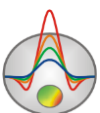
Option **Line** sets color for graphic's connecting lines. If this function is OFF palette color is used otherwise specified in **Line** field color is used.

Option **Pointer** sets color for graphic point's color fill. If this function is OFF palette color is used otherwise specified in **Pointer** field color is used.

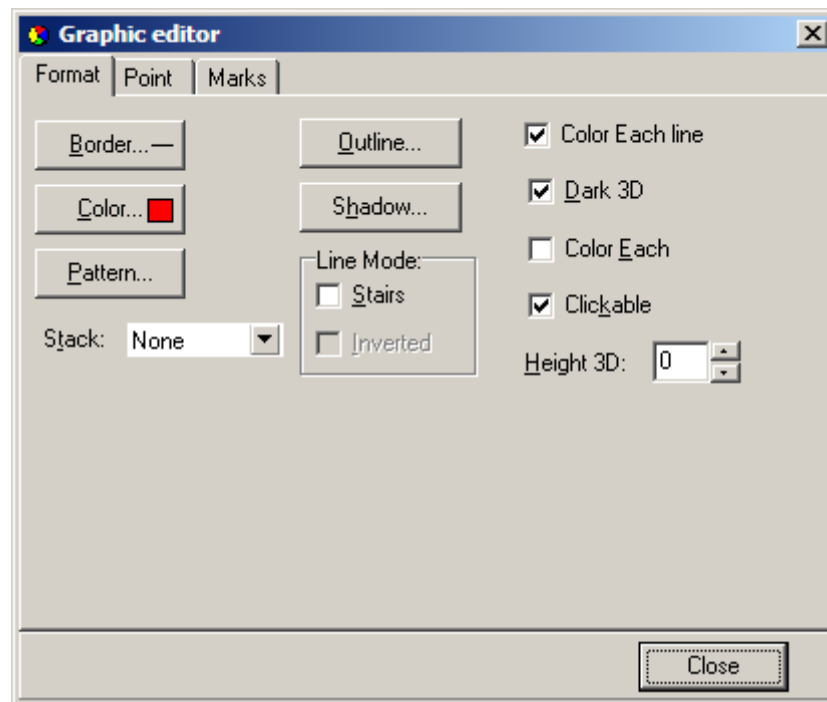
Option **Border** sets graphic point's outline color. If this function is OFF palette color is used otherwise specified in **Border** field color is used.

Button **Options** runs graphics setup dialog.

Button **Default** returns graphics default settings.



Appendix 2: Graphics editor



Graphics editor serves for graphic interface adjustment. Right click with SHIFT button pressed on graphic to run it.

Tab **Format** contains connecting line settings.

Button **Border** runs connecting line parameters setup dialog.

Button **Color** runs color setup dialog.

Button **Pattern** runs filling parameters setup dialog.

Button **Outline** runs graphic's connecting line setup dialog.

Button **Shadow** runs shadows setup dialog.

Tab **Point** contains plot point settings.

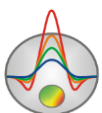
Option **Visible** is used to show/hide plot points.

Option **Style** sets point shape.

Option **Width** sets point width in display units.

Option **Height** sets point height in display units.

Option **Inflate margins** defines if image size is zoomed in according to point size or not.



Button **Pattern** runs point's color fill setup dialog.

Button **Border** runs point's outline parameters setup dialog.

Button **Gradient** runs point's gradient color fill setup dialog.

Tab **Marks** contains settings of graphic's point marking.

Tab **Style**.

Option **Visible** is used to show/hide plot point marking.

Option **Draw every** allows plotting every second, third and so on marking depending on selected value.

Option **Angle** sets point marking rotation angle.

Option **Clipped** defines whether point marking is plotted or not if it is located beyond graphic borders.

Tab **Arrows** allows adjusting arrow from marking to point.

Button **Border** runs arrow line parameters setup dialog.

Button **Pointer** runs arrowhead shape setup dialog (options in tab Point).

Button **Border** runs arrow line parameters setup dialog.

Button **Pointer** runs dialog of arrowhead form settings (tab Point options)

Option **Length** sets arrow length.

Option **Distance** sets distance between arrowhead and plot point.

Option **Arrow head** sets type of arrowhead. **None** – arrowhead specified by **Pointer** button is used. **Line** – classic thin arrowhead is used. **Solid** - classic thick arrowhead is used.

Option **Size** sets arrowhead size if classic arrow is used.

Tab **Format** contains graphic settings of marking frame.

Button **Color** runs frame background color selection dialog.

Button **Frame** runs frame line setup dialog.

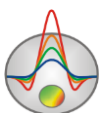
Button **Pattern** runs background parameters setup dialog.

Option **Bevel** sets frame type: usual, elevated or submerged.

Option **Size** sets elevation or submergence level.

Option **Size** rounds frame corners.

Options **Transparent** and **Transparency** sets frame seamlessness degree.



Tab **Text**:

Button **Font** runs marking font setup dialog.

Button **Outline** runs marking letter outline setup dialog.

Option **Inter-char spacing** sets letter spacing for marking text.

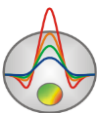
Button **Gradient** runs gradient fill for marking text setup dialog.

Option **Outline gradient** specifies if gradient fill is used in outline or interior of letters.

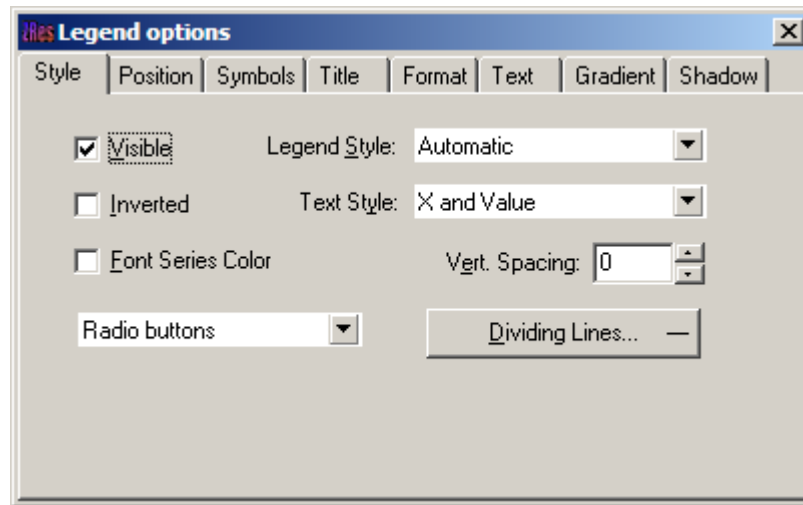
Button **Shadow** runs marking text shadow setup dialog.

Tab **Gradient** contains gradient fill settings for frame around markings

Tab **Shadow** contains shadow settings of frame around marking.



Appendix 3: Legend editor



Editor allows adjusting graphic and legend interface. Right click with SHIFT button pressed on legend to the right of the graph to run it. Pop-up window with set of tabs will appear.

Tab **Style** contains settings of legend display, allows choosing data label format and showing boundaries between legend labels and so on.

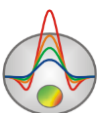
Tab **Position** serves for choosing legend position relative to graphics plan.

Tab **Symbols** sets legend symbols display parameters.

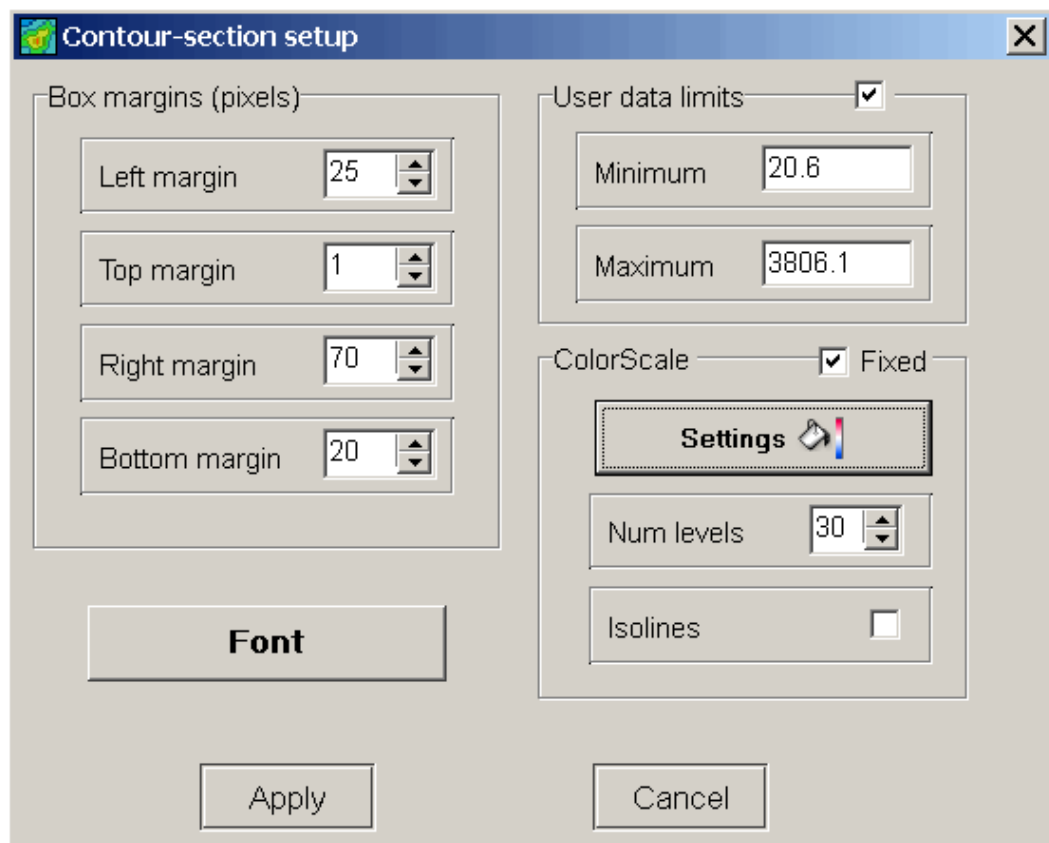
Tab **Title** specifies legend name and allows adjusting its format.

Tab **Text** serves for adjusting legend label format.

Tabs **Format**, **Gradient** and **Shadow** contain settings of legend window, its gradient fill, and shadow.



Appendix 4: Pseudosection (contour section) parameters setup dialog



This dialog serves for adjusting pseudosection parameters.

Field **Box margins**

Left margin – sets image indent (in pixels) from window left edge.

Right margin – sets image indent (in pixels) from window right edge.

Top margin – sets image indent (in pixels) from window top edge.

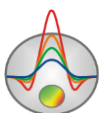
Bottom margin – sets image indent (in pixels) from window bottom edge.

Field **User data limits**

Option **User limits** specifies whether minimum or maximum data values or values from **Minimum** and **Maximum** filed are used for setting isoline sections.

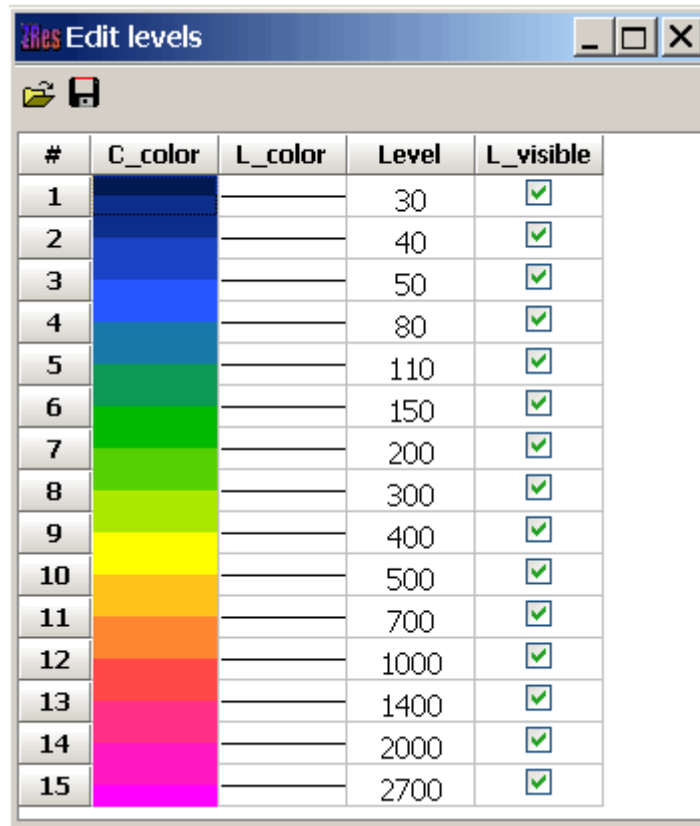
Field **Minimum** sets minimum value to specify isoline sections.

Field **Maximum** sets maximum value to specify isoline sections.



Field Color scale

Settings – sets color palette



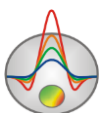
#	C_color	L_color	Level	L_visible
1			30	<input checked="" type="checkbox"/>
2			40	<input checked="" type="checkbox"/>
3			50	<input checked="" type="checkbox"/>
4			80	<input checked="" type="checkbox"/>
5			110	<input checked="" type="checkbox"/>
6			150	<input checked="" type="checkbox"/>
7			200	<input checked="" type="checkbox"/>
8			300	<input checked="" type="checkbox"/>
9			400	<input checked="" type="checkbox"/>
10			500	<input checked="" type="checkbox"/>
11			700	<input checked="" type="checkbox"/>
12			1000	<input checked="" type="checkbox"/>
13			1400	<input checked="" type="checkbox"/>
14			2000	<input checked="" type="checkbox"/>
15			2700	<input checked="" type="checkbox"/>

Dialog allows editing colors, isoline parameters, parameter values, displaying the color. It is possible to load and save color palette in *.lvl format of Surfer program.

Field **Num levels** defines number of isoline sections. Isoline section is set in uniform linear or logarithmic step depending on data type.

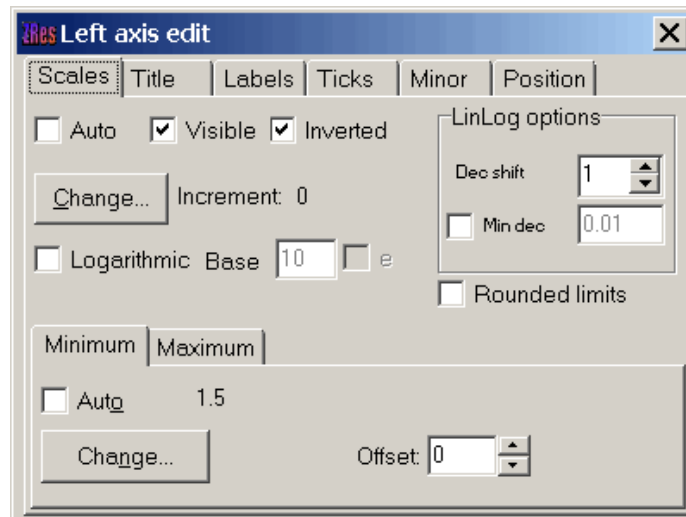
Option **Isolines** specifies whether isolines are created or not.

Button Font **runs** dialog of label font settings.



Appendix 5: Axes editor

Many objects have axes. Axes editor is used to adjust appearance and scale axes. Right click on necessary axis with SHIFT button pressed to run it.



Pop-up menu with two fields (**options** and **default**) appears. The first one runs dialog, the second sets values on default.

First tab of **Scales** dialog contains options for axes scale parameters setup.

Option **Auto** defines how minimum and maximum axis values are chosen. If this option is ON axis limits are set automatically otherwise values from Minimum and Maximum field specified by user are selected.

Option **Visible** shows/hides selected axis.

Option **Inverted** defines axis orientation.

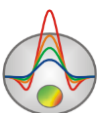
Button **Increment change** runs dialog for axis label step definition.

Option **Logarithmic** selects logarithmic or linear axis scale. In case of sign-changing scale additionally use options from **LinLog options** field.

Option **Base** sets logarithm base for logarithmic axis.

Field **LinLog options** contains options for linear-logarithmic axis adjustment. Linear-logarithmic scale allows representing sign-changing or zero containing data in logarithmic scale.

Option **Dec Shift** sets indent (in logarithmic decades) relative to maximum axis limit modulo to zero. Minimum decade (prezero) has linear scale, others have logarithmic.



Option **Min dec** sets and fixes minimum (prezero) decade value if option is ON.

Option **Rounded limits** defines whether it is necessary to round minimum and maximum axis values or not.

Fields **Minimum** and **Maximum** contain options for axis limits adjustment.

Option **Auto** defines whether axis limit is selected automatically or using **Change** button.

Option **Offset** sets percentage axis limit shift relative to its actual value.

Tab **Title** contains options for axis header adjustment.

Tab **Style**:

Option **Title** sets axis header text.

Option **Angle** sets header text rotation angle.

Option **Size** sets header text indent. If 0 value is specified it is selected automatically.

Option **Visible** shows/hides axis header.

Tab **Text**:

Button **Font** runs header font setup dialog.

Button **Outline** runs dialog for header letters' outline adjustment.

Option **Inter-char spacing** sets letter spacing in axis header.

Button **Gradient** runs gradient fill setup dialog for header text.

Option **Outline gradient** specifies if gradient fill is used in outline or interior of letters.

Button **Shadow** runs axis header shadow setup dialog.

Tab **Labels** contains options for axis label adjustment.

Tab **Style**:

Option **Visible** shows/hides axes labels.

Option **Multiline** is used for setting multiline axes labels.

Option **Round first** rounds first axis label.

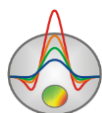
Option **Label on axis** hides labels that go beyond axis.

Option **Alternate** arranges labels in two lines.

Option **Size** sets axis label indent. If 0 value is specified it is selected automatically.

Option **Angle** sets label rotation angle.

Option **Min separation %** sets minimum percentage label spacing.



Tab **Text**:

Button **Font** runs label font setup dialog.

Button **Outline** runs dialog for label letters' outline adjustment.

Option **Inter-char spacing** sets letter spacing in label text.

Button **Gradient** runs label gradient fill setup dialog.

Option **Outline gradient** specifies whether gradient fill is used in outline or interior of letters.

Button **Shadow** runs label shadow setup dialog.

Tab **Ticks** contains options for axis main ticks adjustment.

Button **Axis** runs axis line setup dialog.

Button **Grid** runs line setup dialog for main ticks' grid.

Button **Ticks** runs external main axis tick setup dialog. Option **Len** sets its length.

Button **Inner** runs internal main axis tick setup dialog. Option **Len** sets its length.

Option **Centered** centers grid of axis ticks.

Option **At labels only** displays main axis ticks only if axis labels are present.

Tab **Minor** contains options for axis intermediate ticks adjustment.

Button **Grid** runs line setup dialog for intermediate ticks grid.

Button **Ticks** runs external intermediate axis tick line setup dialog.

Option **Length** sets its length.

Button **Minor** runs internal intermediate axis tick line setup dialog. Option **Len** sets its length

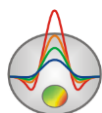
Option **Count** sets number of intermediate ticks between main ones.

Tab **Position** defines axis size and position.

Option **Position %** sets axis indent relative to its standard position on graph (in percent to graph size or in screen units depending on selected option Units).

Option **Start %** sets axis start indent relative to its standard position on graph (in percent to graph size).

Option **End %** sets axis end indent relative to its standard position on graph (in percent to graph size).

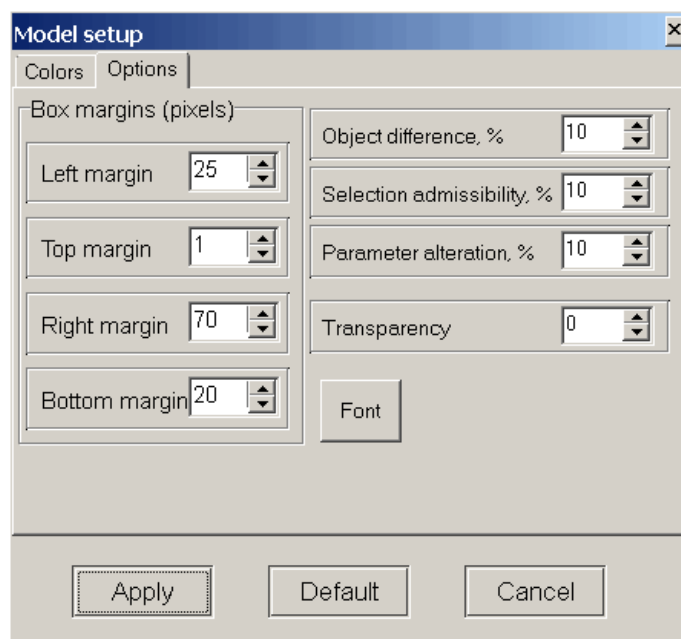


Appendix 6: Dialog of model visualization parameters

Model can be displayed as cells **Options/Model/Block-section**, in smooth interpolated palette **Options/Model/Smooth-section**, and also as a contour section **Options/Model/Contour-section**. Contour section visualization parameters setup dialog in [Appendix 4](#): Pseudosection (contour section) parameters setup dialog (contour section).

To run model displaying parameters setup dialog, select item **Setup**, when right-clicking on the top model window part in the field «Resistivity block-section».

Tab Options



Field **Box margins**

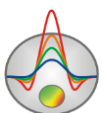
Left – sets image indent (in pixels) from window left edge.

Right – sets image indent (in pixels) from window right edge.

Top – sets image indent (in pixels) from window top edge.

Bottom – sets image indent (in pixels) from window bottom edge.

Object difference sets maximum value of adjacent cells parameter ratio, in case of exceeding it boundary between them is drawn.

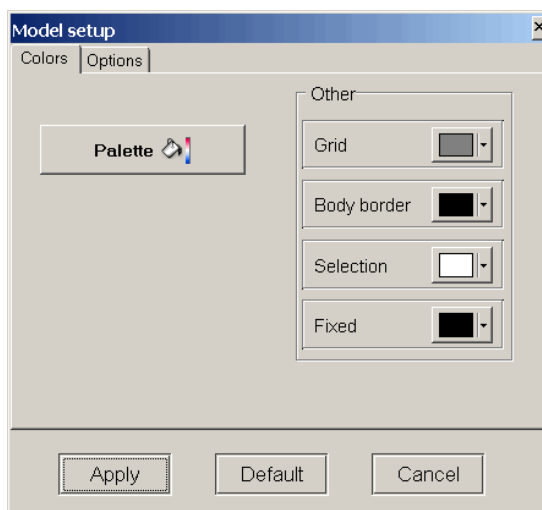


Selection admissibility sets threshold value of adjacent cells parameter difference which defines their unification and highlighting together (in Magic Wand mode).

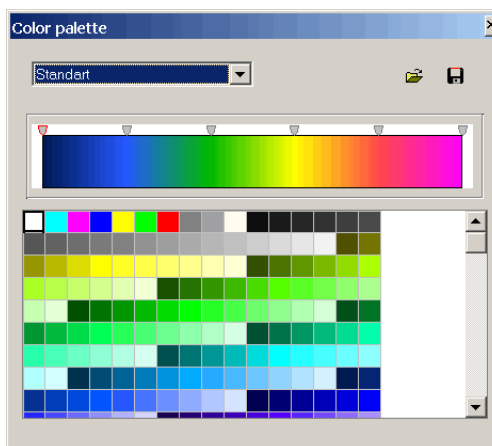
Parameter alteration sets value of selected cells parameter increment (in percent to parameter value) in **Edit** mode with SHIFT button pressed.

Button **Font** runs font setup dialog.

Tab Colors

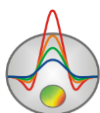


Palette - sets the color palette:



The palette can be changed, edited, loaded from a file and saved to file with extension *.clr of Surfer program.

A convenient tool for selecting the maximum and minimum resistivity values is to specify them using vertical line positions on the resistivity distribution diagram



Field **Other**

Body border - allows specifying the boundaries color between adjacent cells, if the difference degree between them is greater than the set one in the option **Parameter alteration**.

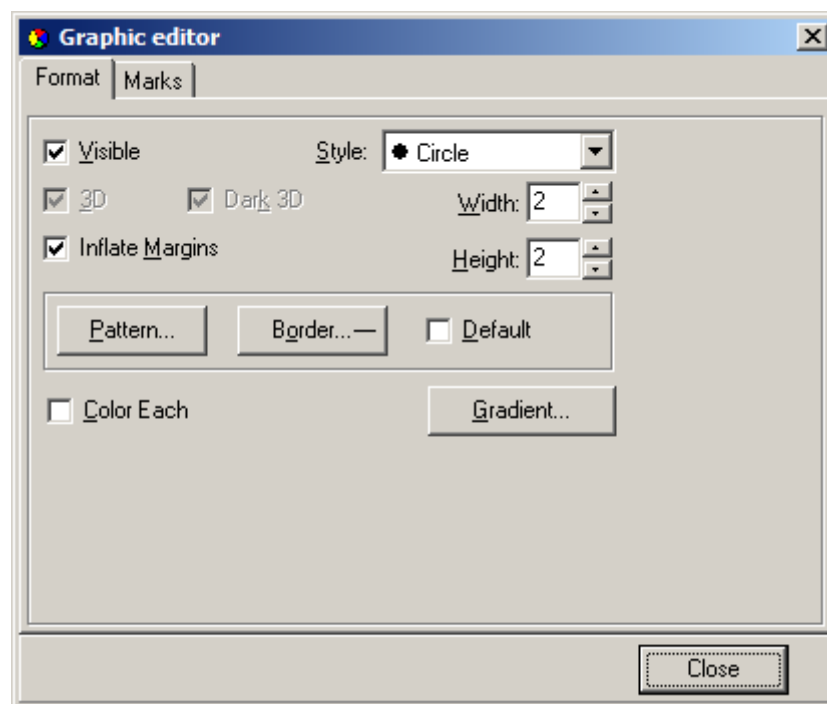
Grid - sets the grid color.

Selection - sets the label color of the selected cell.

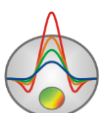
Fixed - sets the label color of the fixed cell.

If model is edited as contour section, setup dialog is similar to pseudosection parameters setup dialog ([Appendix 4](#): Pseudosection (contour section) parameters setup dialog).

Appendix 7: Pseudosection points editor



Editor serves for adjusting appearance of pseudosection points, electrode position points, and their labels. Right click with SHIFT button pressed on point to run it.



Tab **Format** contains settings of pseudosection and electrode position point appearance. Tab **Marks** contains label settings. Options in these tabs *are described in [Specifics](#)* of work with borehole data measurements

ZondRes2D allows to model and perform the inversion of *borehole measurements*. These include inter-borehole measurements, borehole-surface measurements, and their combinations. It supports both vertical and deviated boreholes.

The measurement configuration is described in the input data file of the ZondRes2D format (more details in the part «*Main data file format*»). When describing electrode measurements located in boreholes, as in the case of terrestrial measurements, planning altitude coordinates are indicated. There is no opportunity to create measurement configuration containing electrodes in borehole in the program. The corresponding data file is created in text or tabular editor ([Sample-file – sample_with_z_source](#)).

Modeling and inversion of borehole and borehole-surface data is the same as for ground-based measurements. The difference is in the method of apparent resistivity displaying - unlike conventional methods for ground-based measurements dependence of the apparent resistivity from the array factor are built here (Fig. 77). Measured and calculated data can also be viewed in a separate window via the option **Options/Data/G-Res plot**.

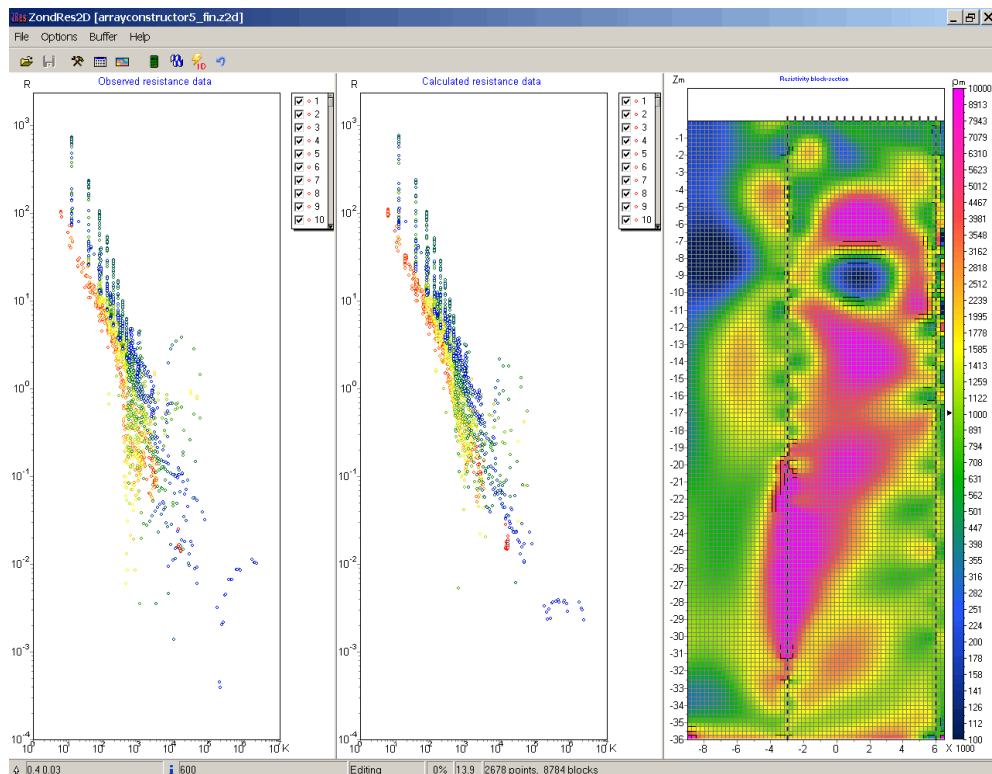
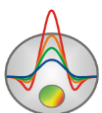


Fig. 77 Main program window when working with borehole measurements



IP parameters inversion

ZondRes2D allows you to work not only with integral polarizability values, but also with IP process parameters. In practice, appropriate investigations are carried out in one of two modes - time or frequency. When working in the time domain, measurement result for each observation point is a IP decay curve - voltage dependence (sometimes normalized by width of the measurement time window) from time on a fixed set of delay times ([sample-file – timedomain.zd2](#)). Dependences of resistivity and phases shift from frequency for a fixed frequencies set ([sample-file - frequencydomain.z2d](#)) are input data for each observation point, when working in the frequency domain.

For both modes the program allows visualizing the source data, carrying out their smoothing and performing inversion, determining for all observation points integral IP value for each delay time (frequency) or the Cole-Cole model parameters.

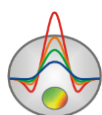
The input data file format for IP parameters inversion (extension *.z2d): the first line contains the times delay lists for time and frequencies for frequency measurement mode after the word time_ # chann. The following is a table whose rows correspond to the points of observation, and the columns - to electrodes coordinates (similar to The main data file format), to the Res ($\Delta U/I$) value and to the measurement values for the respective delay times (column headings - ipi1, ipi2 etc.) and frequencies (the column headings - mod1, mod2, etc. for module, pha1, pha2 etc. for phase).

An example of the measurement time mode data file beginning:

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

An example of the frequency measurement mode data file beginning:




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

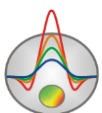


5	*	10	15	0.001	0.001	0.001	0.001	-0.005	-0.007	-0.012
10	*	15	20	0.002	0.002	0.002	0.002	-0.005	-0.009	-0.014

To work with IP parameters use the menu item **Time lapse/domain** of the main menu (menu appears when you load the appropriate data file). To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.

Option **Time lapse/domain/Channels** runs the channel setting table. It is possible to edit the values of the time delay or the measurement frequencies and headers for specified values (will be displayed in the drop-down list on the toolbar of the main window). For the some options operation it is necessary to times/frequencies were inputted previously.

Option **Time lapse/domain/Display channel plot** runs the window (Fig. 78), showing the decay curves (frequency curves) for each measurement point. To move to another measurement use the mouse (on pseudosection or graph) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point displays. A separate point of the displaying curve can be edited using the menu items. They appears, when you press the left mouse button on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), to return to the previously position (**Origin point**). The toolbar button  allows you to smooth the entire curve.



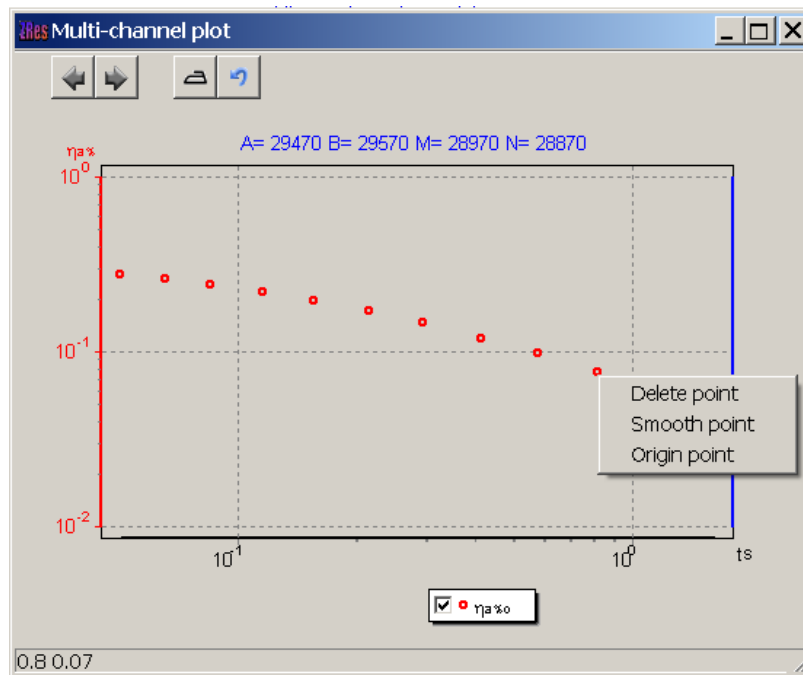
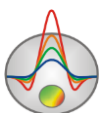


Fig. 78 The curve editing window for a single measurement point

When working in the time mode after editing curves it is necessary to use the option **Time lapse/domain/Recalculate chargeability**, which recounts an integral IP parameter accounting data changes.

The option **Time lapse/domain/Time lapse inversion** runs IP data serial inversion mode for each implementation (in this case, the standard calculation scheme, as well as for the integral IP parameter). If the option **Time lapse/domain/Next from previous** is switched ON, as an initial approximation for each next iteration the results of previous one is used. If this option is switched OFF, as an initial approximation for each next iteration the current implementation model is used. These models are displayed in the model editor. To switch between data and models corresponding to different times (frequencies) use the drop-down list on the control panel of the main window.



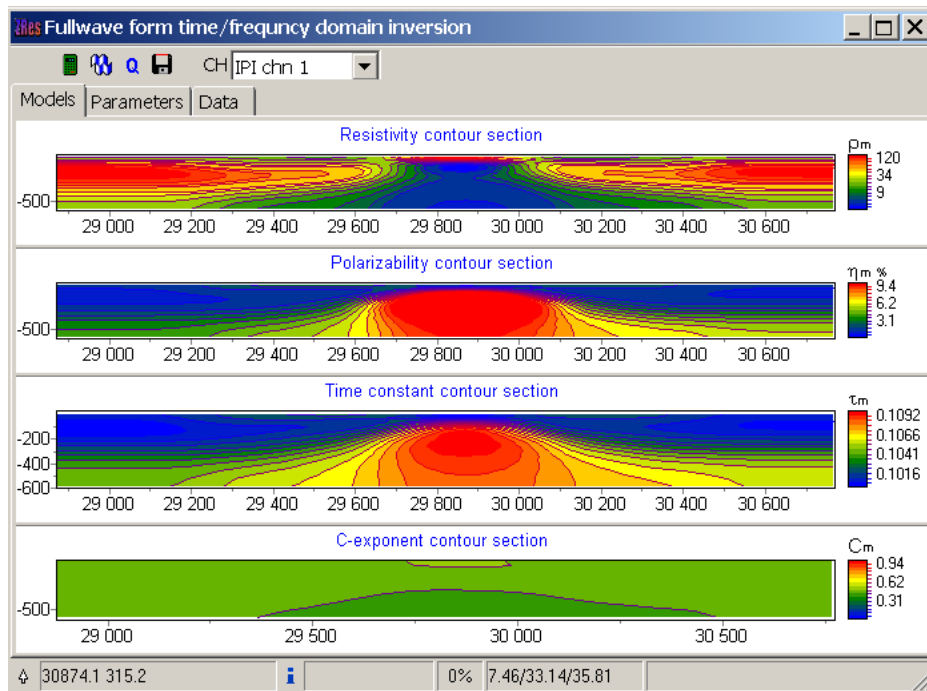


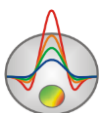
Fig. 79 The main window of the IP parameters module inversion

It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Window IP parameters full-wave inversion (Fig. 79) is run via the menu item **Time lapse/domain/ Fullwave form inversion**. The window contains three tabs. Tab **Models** displays resistivity, polarizability and Cole-Cole parameters (τ , and c), which are obtained as model inversion results.

Tab **Parameters** allows you to select parameters for which you want to carry out inversion and specify (fix) ranges of these parameters (Fig. 80). **ReBuild** button allows changing the each parameter values to the value from the table (value). If instead of the number symbol "*" are specified, the parameter value on the section will not be replaced.



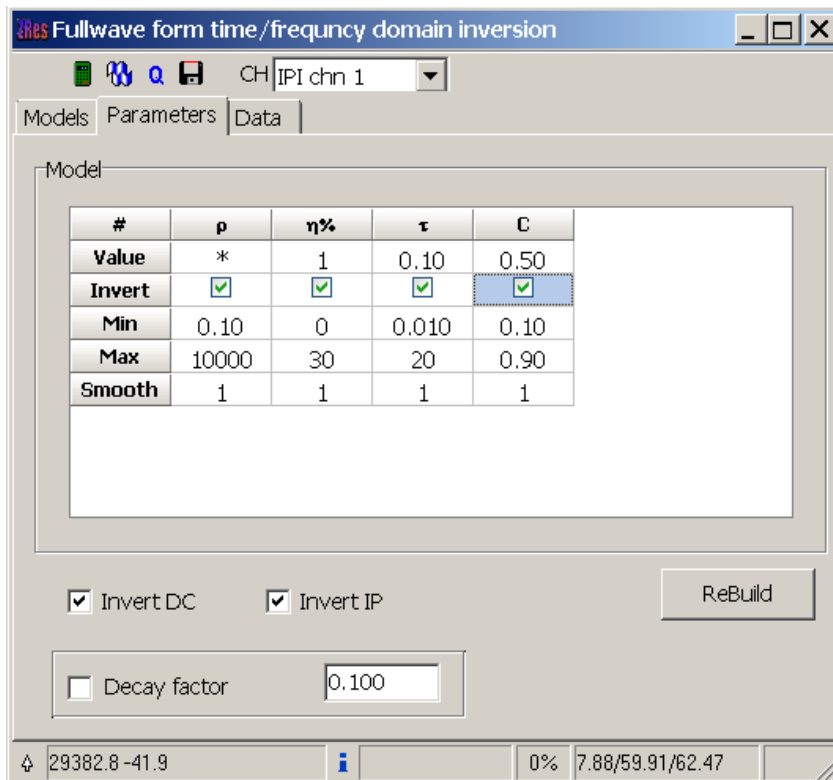




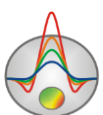
Fig. 80 Window for selecting parameters for curves IP inversion

Option **Module** (available when working in the frequency mode) indicates whether or not to select the signal module, or use phase only. As a rule, used in the frequency range module changes very little, and unlike the phase, has little effect on the inversion results.

To start inversion, press the button . This inversion occurs simultaneously on all selected parameters. Button  provides an alternative method in which the polarizability inversion occurs independently for each time (frequency) by time-lapse inversion and Cole-Cole model parameters are recovering based on obtained "real" curves.

Tab **Data** displays observed and calculated polarizability model. The drop-down list in the toolbar (available when working with all the tabs) allows switching between sections for different decay times (frequencies).

Dialog **Time lapse/domain/ Digital filter** contains temporary filter settings for the Fourier transform to the time domain, and allows you to set the length of the current impulse. *Video dedicated to the IP data inversion in the time and frequency domain, can be downloaded from <http://zond-geo.ru/zmovies/TDvsFD.exe>.*



Geoelectric monitoring data inversion

Monitoring involves measurements with the same parameters at different time. The program recognizes file with the *.z2d extension as an inputting monitoring file. The file has following structure: the first four columns contain electrodes coordinates (similar to Main data file format); next - columns, entitled res1, res2, etc. that contain Res ($\Delta U/I$) values for the first, second, etc. measurement cycles.

The following example shows the beginning of a monitoring data file by three-electrode array ([sample-file – monitoringERT.z2d](#)):

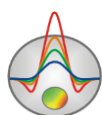
C1	C2	P1	P2	res1	res2	res3	res4	res5
0	*	2	4	5.57	5.57	5.57	5.60	5.63
0	*	4	6	1.55	1.55	1.56	1.59	1.62
0	*	6	8	0.84	0.84	0.85	0.88	0.91
0	*	8	10	0.48	0.48	0.49	0.51	0.52
0	*	10	12	0.44	0.44	0.45	0.49	0.46
0	*	12	14	0.25	0.26	0.26	0.29	0.24




Working with monitoring data, in general, is similar to the work with multifrequency IP data or temporary IP containing information about decay curve. Only instead of the polarizability values at a certain frequency or time is considered the resistivity values at a certain time.

Use the menu item **Time lapse/domain** to work with the monitoring (menu appears when you load the appropriate data file). Switching between the data and the models corresponding to different measurements times is carried out using the drop-down list on the control panel of the main window.

Option **Time lapse/domain/ Channels** runs the table in which you can enter information about each measurements implementation. An information about measurement time is inputted at the **time, s** column. Arbitrary text information, entered in the **name** column, will displayed at the drop-down list on the main window control panel (when switching between implementations).

Option **Time lapse/domain/ Display channel plot** runs the window that displays the Res ($\Delta U/I$) values for each measurement point, depending on the time (Fig. 81). To move to another



measurement use the mouse (on pseudosections or graphs) or the buttons  and , at the same time at the top window part coordinates of the feeding and receiving electrodes of considered measuring point are displayed. A separate point of the displaying curve can be edited using appearing menu items when you press the left mouse button menu on the point. It can be removed (**Delete point**), moved manually or replaced by the smoothed value (**Smooth point**), returned to the previous position (**Origin point**). The toolbar button  allows you to smooth the entire curve.

Option **Time lapse/domain/ Time lapse inversion** runs consistent inversion mode of monitoring data for each implementation. If the option **Time lapse/domain/ Next from previous** is switched ON, results of previous iteration are used as an initial approximation for each next one. If this option is switched OFF, the current model in this implementation is used as an initial approximation for each next iteration. Obtained models are displayed in the model editor. To switch between models corresponding to different times use the drop-down list on the control panel of the main window.

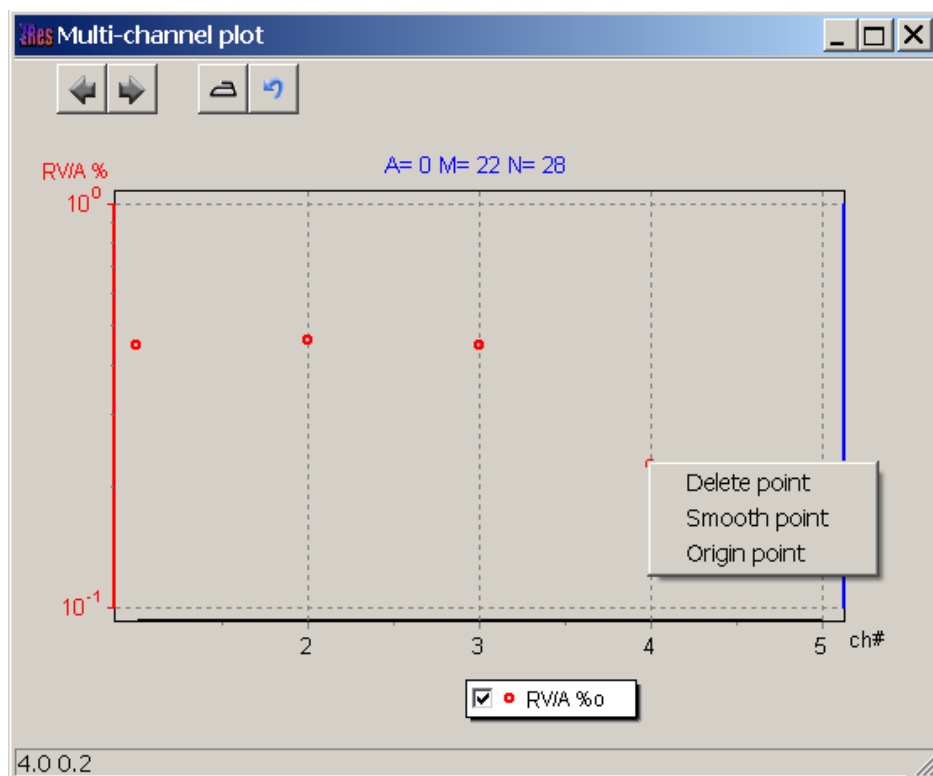
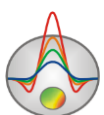


Fig. 81 Window of resistivity changes depending on the time for a single measurement point



It is necessary fewer iterations to achieve an acceptable discrepancy, because in most cases for each following implementation model of the previous implementation is used as a starting approximation. Use the option **Time lapse/domain/ Max iter number for next** to specify number of following after the first implementation iterations .

Option **Time lapse/domain/ Save time shots** saves the inversion results of IP data for all implementations in a single text file.

Sample - video - <http://zond-geo.ru/zmovies/monitoringERT.exe>.

Magnetotelluric data inversion

If you have magneto-telluric data measurement, they can be used in inversion accounting data of resistivity method on DC. To work with magnetotelluric data measurements use the menu item **Options/MT Data**.

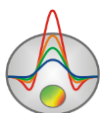
The magnetotelluric data inputting file is a file with *.m2d format. Its structure is described in the section Magnetotelluric data import.

The menu items **Load MT data** and **Show MT plot** allow, respectively, loading and visualizing data of magnetotelluric measurements. Selecting **With static shift** specifies whether static shift (normalization) MTS curves. To specify MT-measurements weight use the menu item **Set weights of MT**.

If you select the menu item **Invert MT data**, the program will carry out joint inversion of MT data and the resistivity method.

Option **DET inversion** allows carrying out inversion simultaneously both longitudinal (TE), and transverse (TM) impedance.

Joint data inversion of electrotomography and PMT is shown in the movie http://zond-geo.ru/zmovies/MT_ERT.exe.

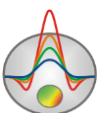


Script-files (batch processing)

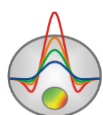
When you need to repeat the same action, they can be written to a script file. When it started, the program will perform the action automatically. For example, you can record the process of opening data file, setting the starting model, the inversion parameters, starting the inversion process and exporting results. If it is necessary to process multiple profiles data by this scheme, run the script and select all data files, for each of them the corresponding actions sequence is automatically will be applied. This tool allows you to save time when the same operations repeat and allows the user to "standardize" the process or individual working process steps with data.

To start recording a script file select the menu item **Script/Start recording script** before performing operations that need to be remembered. If script recording is currently active, the tick will be displayed on the left of the specified menu item. To stop script recording, use the menu item **Script/Stop recording script**, then it is necessary to save the script file (**Script/Save script file**).

To run the script file, it must be loaded **Script/Load script file** and using the menu item **Script/Run packet processing** select the data file for processing. When you save a file processing results in automatic mode, the results file name - the source file name with underscore addition.



Appendix 1: Graphics set editor.



Zond geophysical software
Saint-Petersburg 2001-2016

References

1. Хмелевской В.К. Геофизические методы исследования земной коры. Дубна, 1997.
2. Constable S.C., Parker R.L., Constable C.G. 1987. Occam's inversion: a practical algorithm for generating smooth models from electromagnetic sounding data. *Geophysics*, 52, 289-300
3. Dey A., Morrison H.F. 1979. Resistivity modeling for arbitrary shaped two-dimensional structures. *Geophysical Prospecting* 27, 106-136.
4. Loke, M.H., 2000. Topographic modelling in resistivity imaging inversion. 62nd EAGE Conference & Technical Exhibition Extended Abstracts, D-2.
5. Lowry, T., Allen, M.B. and Shive, P.N. [1989]. Singularity removal: A refinement of resistivity modeling techniques. *Geophysics*, 54(6), 766-774
6. Marquardt, D.W., 1963, An algorithm for least squares estimation of nonlinear parameters: *J. Soc. Indust. Appl. Math.*, 11, 431-441.
7. Portniaguine O.N., Zhdanov M.S. Focusing geophysical inversion images // *Geophysics*. 1999. V. 64. P. 874–887.
8. Tsourles, P.I., Symanski, J.E. and Toskas, G.N., 1999. The effect of terrain topography on commonly used resistivity arrays. *Geophysics*, 64, 1357-1363.
9. Xu S.Z., Duan B., Zhang D. Selection of the wavenumbers k using an optimization method for the inverse Fourier transform in 2.5D electrical modeling// *Geophysical Prospecting*, 2000, 48, 789-796.

