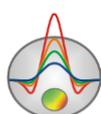


Program for two-dimensional interpretation of data obtained by magnetotelluric sounding

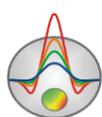
(MT, AMT, RMT)

ZONDMT2D

<i>Program functionality</i>	3
<i>System requirements</i>	5
<i>Program installation and deinstallation</i>	5
<i>Adopted definitions</i>	5
<i>Starting working and basic options</i>	6
Working guidelines.....	6
Creation and opening of data file	6
Main Window Toolbar	9
Main Menu Functions	9
“Hot” keys	15
Status bar	15
Adjusting starting model mesh settings.....	15
<i>Main data file format</i>	18
<i>Preparing data for inversion</i>	19
Options for data measurements editing.....	19
Module of quality control and data editing.....	24
Static shift correction	29
<i>Apparent parameters visualization</i>	31
Graphics plot.....	31
Pseudosection	33
<i>Input and editing topographic information</i>	34
<i>Data inversion</i>	36
Calibration of model mesh and operations with relief.....	36
Program parameters setup dialog.....	36
Inversion in layered medium	43
Incorporating a priori information in the inversion	45
Set boundaries dialog	46



<i>Model visualization modes and parameters</i>	47
<i>Interpretation results saving</i>	50
<i>Modeling and working with model</i>	51
Survey layout definition	51
Model building in ZondMT2D	52
Model editor	52
Working with block model.....	52
Cell summarization dialog	55
Polygonal modeling	57
<i>Working with multiple model in a single project</i>	60
<i>Data import and export</i>	62
Additional information visualization	62
Text (tabular) data files – import and export	66
Exporting image setup dialog	66
Borehole columns and logging data visualization.....	67
<i>Window of geological-geophysical model builder</i>	72
<i>Geoelectric models volumetric visualization using several profiles</i>	73
<i>Appendix 1: Graphics editor</i>	77
<i>Appendix 2: Graphics set editor</i>	79
<i>Appendix 3: Graphic’s legend editor</i>	80
<i>Appendix 4: Pseudosection (contour section) parameters setup dialog</i>	81
<i>Appendix 5: Axes editor</i>	83
<i>Appendix 6: Dialog of Model visualization parameters</i>	86
<i>Appendix 7: Pseudosection points editor</i>	88
<i>Appendix 8. Zdconvert program</i>	88



Program functionality

«ZONDMT2D» is computer program for 2D interpretation of profile data obtained by magnetotelluric sounding. Friendly interface and ample opportunities for data presentation allows solving assigned problem with maximum efficiency.

Basic feature of magnetotelluric field in two-dimensionally heterogeneous medium is its division into two independent parts called polarizations or modes. The main difference between E-polarized and H-polarized fields consists in the fact that in the former anomalies have inductive origin while in the latter - galvanic. In case of E-polarization, electric field is polarized along structures and for this reason it does not cross boundaries of areas with different resistivity. Anomalies are caused by excess currents induced in conductive heterogeneities or by low number of excess currents induced in nonconducting heterogeneities. In case of H-polarization, currents flow transversely to structures. Thereby they cross boundaries of heterogeneities and anomalies are caused by galvanic current flow inside them (for low-resistivity heterogeneities) or around them (for high-resistivity heterogeneities).

Problem of studying 2D fields comes to separate consideration of two tasks: for E-polarization and for H-polarization. Solution of these tasks is much easier than Maxwell equations system solution.

Maxwell equations for electric polarization T_e are given below:

$$\frac{\partial}{\partial y} \frac{1}{i\omega\mu} \frac{\partial E_x}{\partial y} + \frac{\partial}{\partial z} \frac{1}{i\omega\mu} \frac{\partial E_x}{\partial z} = -\sigma E_x \quad (1)$$

where ω - field oscillation circular frequency, μ_0 - permeability of free space.

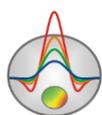
In this case medium in point neighborhood is supposed to be homogeneous with electrical conductance σ . Solution of this equation gives E_x distribution in modeling area. Two components of magnetic field can be found using E_x :

$$\begin{aligned} H_y &= -\frac{1}{i\omega\mu} \frac{\partial E_x}{\partial z} \\ H_z &= \frac{1}{i\omega\mu} \frac{\partial E_x}{\partial y} \end{aligned} \quad (2)$$

Apparent resistivity and phase for E_x and H_y are calculated using the following formulas:

$$\begin{aligned} \rho_{xy} &= \frac{1}{i\omega\mu} \left| \frac{E_x}{H_y} \right|^2 \\ \phi_{xy} &= \arg \left(\frac{E_x}{H_y} \right) \end{aligned} \quad (3)$$

In case of magnetic polarization T_m Maxwell equation has the following form.



$$\frac{\partial}{\partial y} \left(\frac{1}{\sigma} \frac{\partial H_x}{\partial y} \right) + \frac{\partial}{\partial z} \left(\frac{1}{\sigma} \frac{\partial H_x}{\partial z} \right) = i\omega\mu H_x \quad (4)$$

Solution of this equation gives E_x distribution in modeling area. Two components of electric field can be found using H_x :

$$\begin{aligned} E_y &= \frac{1}{\sigma} \frac{\partial H_x}{\partial z} \\ E_z &= -\frac{1}{\sigma} \frac{\partial H_x}{\partial y} \end{aligned} \quad (5)$$

Apparent resistivity and phase for magnetic polarization can be calculated in the same way.

Finite-element method as mathematical apparatus is used to solve forward and inverse problems. It gives best results in comparison with mesh methods.

For plane-wave electromagnetic field modeling medium is divided into triangle cells grid with different resistivity. Field behavior inside grid cell is approximated by linear basis function.

$$N(x, z) = \frac{(a + bx + cz)}{2A} \quad (6)$$

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

$$(A^T W^T W A + \mu C^T R C) \Delta m = A^T W^T \Delta f - \mu C^T R C m \quad (7)$$

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.

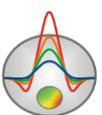
During inverse problem solution development special attention was devoted to a priori information accounting (data weights, parameters turndown).

«ZONDMT2D» has powerful system of profile data visualization, data 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 measurements' parameters, set data weights (relevance) and correct measured values in data editor. It is possible to set data weights in accordance with noise level and by total sensitivity of current measurement to medium parameters or fix those cells of the model which have parameters that almost do not influence measuring results.

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



$$S = \text{diag}(A^T A) \quad (8)$$

Research of sensitivity allows choosing optimal selection of frequencies and measurement interval in order to solve assigned geologic task.

«ZONDMT2D» uses simple and clear data file format. Apparent resistivity and phase for two types of polarization can be used as measured characteristics. Program allows importing and visualizing data using other methods which makes data interpretation process more integrated.

«ZONDMT2D» has modeling system that allows user to specify frequency set and positions of measuring points.

«ZONDMTD» is easy-to-use instrument for automatic and interactive interpretation of profile magnetotelluric sounding data and can be used on IBM-PC compatible PC with Windows system.

System requirements

«ZONDMT2D» 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).

Program installation and deinstallation

«ZONDMT2D» 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.com/english.

To install the program copy it from CD to necessary directory (e.g., 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.

Adopted definitions

Ro_a – apparent resistivity, $\text{Om} \cdot \text{m}$.

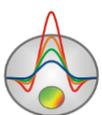
Phi - impedance phase, degrees. Positive number. Should be specified in quadrant from 0 to 90 degrees.

Frequency in Hz.

Period is inverse value of measurement frequency, seconds.

Pseudodepth – approximate depth of investigation connected with scin-layer thickness.

All geometric values are specified in kilometers.



Starting working and basic options

Working guidelines

This section briefly describes how to work with the program «ZONDMT2D». To get started after the first program run it is necessary to register the program by one of the methods specified in [Program registration](#): using a registration file, or the key SenseLock.

To take a geoelectric section (inversion mode), open the data file in the main menu **File/Open file**.

To create a data file is recommended to use a helper program Zdconvert (more details in [Appendix 8](#)). The program also supports REBOCC file format and EDI data format (more details in [«Opening REBOCC and EDI data files format»](#)).

After opening a data file, starting model settings dialog window will appear (Mesh constructor). Mesh parameters of the resulting geoelectric model are determined in the dialog window. This dialog is described in details in [«Adjusting starting model mesh settings»](#). Usually default settings can be used. After configuring the mesh press **Apply** button, and the program goes on to working mode.

The program main window consists of three parts. The upper - is section or graphics of measured parameters, the middle – section or graphs of calculated parameters. The bottom - a model environment.

The next step is data processing and editing, including filtering, coordinate system rotation, input correction to compensate for static shift etc. This option is implemented by the tool **Options/MT Editor** and is described in [«Module of quality control and data editing»](#).

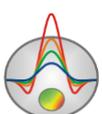
Before starting inversion it is necessary to set inversion parameters, using dialog Program Setup. It is available in the Main Menu **Options/Program setup** or using the button  on the toolbar. More information about setting inversion parameters in the "[Program parameters setup dialog](#)".

To start the inversion it is necessary to press the button  on the toolbar. After the calculation, it is possible to estimate values of relative discrepancy on the program status bar. It is also possible to visually compare observed and calculated data using dialog of viewing sounding curves (more in [«Sounding curves view and edit dialog»](#)), and displaying data as [Pseudosection](#) or [Graphics](#).

After obtaining the model (the lower section of the program main window) it is necessary to save the calculation results in a ZondMT2D project format and/or one of the proposed graphical and tabular formats (more details in [«Interpretation results saving»](#)).

If the work is being carried out in the modeling mode, it is not necessary to load the data file, but create a synthetic observation system with any parameters, specify medium geometry and properties. After calculating the forward problem, results can be saved. Read more about ZondMT2D mathematical modeling in [«Modeling and working with model»](#).

Creation and opening of data file



To start up «ZONDMT2D» it is necessary to create data file of certain format which contains information about sounding locations and measuring results. Text data files created in the program «ZONDMT2D» have «*.m2d» extension. More details about data file format in [«Main data file format»](#). A single file usually contains data of a single profile.

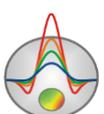
«ZONDMT2D» also supports most popular data formats: REBOCC (*.shc), EDI, Mackie (*.inp), ZondMT1D (*.mdf), J format, StrataGem, IPI2WinMT, Zonge AVG files. «ZONDMT2D» supports REBOCC (*.shc) and Mackie (*.inp) data file format. It is also possible to open raw data file in EDI format. To do that select «Other data file» in the data type window. When opening REBOCC (*.shc) or Mackie (*.inp) data file format, after selecting file, working algorithm is similar to work with standard files ZondMT format.

When opening data file, **Survey plane** dialog window (fig. 1) appears. Sounding sites location are displayed in it. It is possible to create profile lines by using dialog window tools. It is useful when working with areal data.

After setting a profile and selecting points along it, all points included in the profile will be displayed in blue. It is possible to exclude/include a point in the profile by pressing the left mouse button. If profile line does not go directly through points, the position of the sounding point projection on the profile will be displayed in green. After setting the profile it is necessary to press the button  to start inversion mode. The window of creation starting model [Mesh constructor](#) appears. Further actions do not depend on the format of the input file.

It is also possible to import data from a user-created text (table) files (Options/Import/Export/Load model/data).

An alternative to using a data file is to create a synthetic observation system, allowing to model different geological situations (more details in [«Modeling and working with model»](#)).



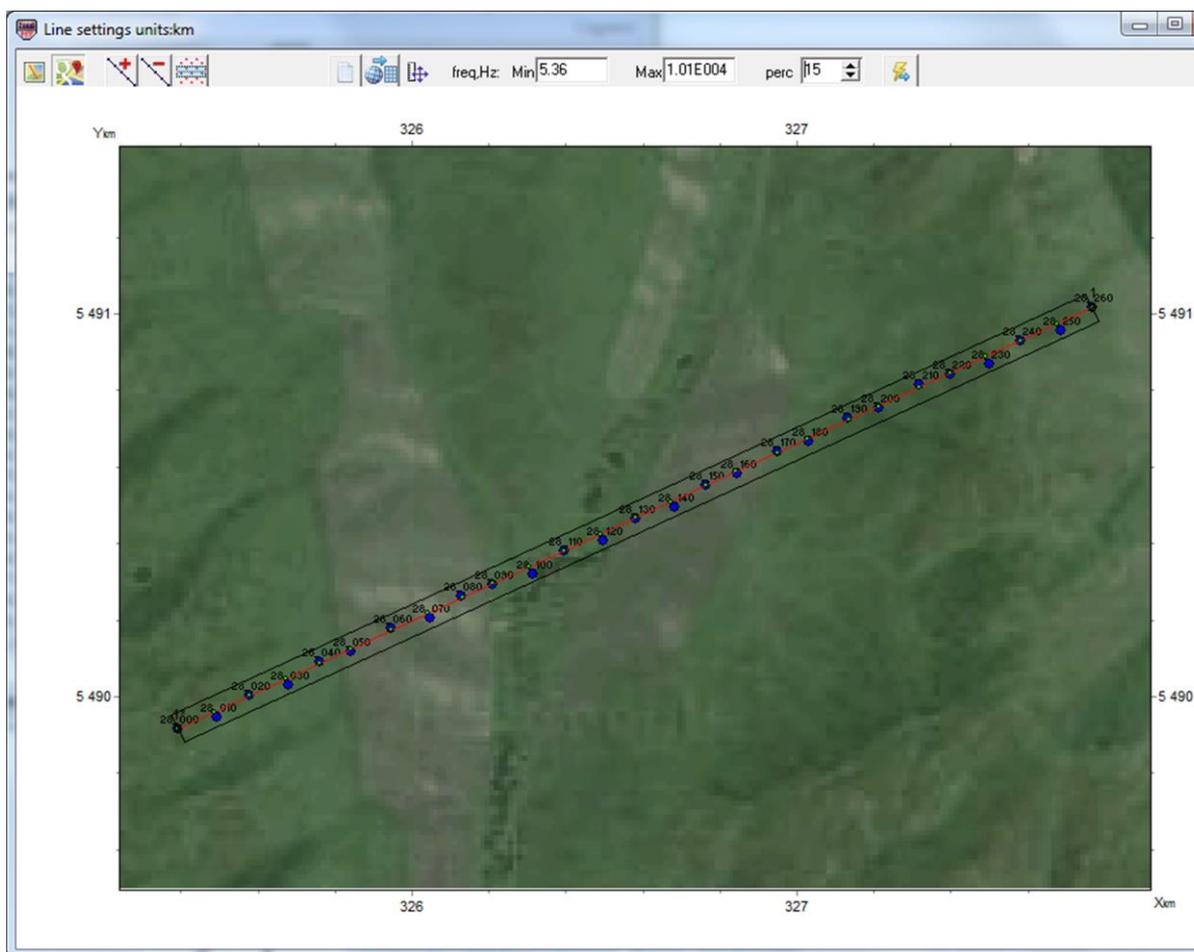
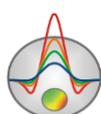


Fig. 1 **Survey plane** dialog window for working with areal data in EDI format

The window main panel contains the following features:

	Load background file (file with *.bmp or *.sec). When selecting file with *.bmp extension, dialog window for input of rectangular coordinates appears. The coordinates correspond to image boundaries.
	Load Google maps image. When working with WGS84 UTM rectangular coordinates, it is necessary to set zone number in appeared window.
	Add profile line. Press left mouse button to set profile points, use right button to set the latest point.
	Delete profile.
	Include sounding stations (located within rectangular area around selected line) in profile automatically.
	Open table to manually input or edit sounding points coordinates.
	Recalculate geographical coordinates to rectangular. When EDI files are loaded they are recalculated automatically. If beforehand file is known to contain geographical coordinates (latitude and longitude) then prior to interpretation they must be recalculated to rectangular coordinates using this button.



Frq: Min- Max	Set minimum and maximum sounding frequency respectively. Out-of-range measurements are not loaded to the program.
	Select image scale: equiaxial or maximum window filling.
Perc	Set size of sounding station automatic selection domain for profile.
	Start data inversion for selected profile.

Main Window Toolbar

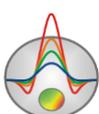
The toolbar serves to quick run of the most frequently used functions. It contains the following functional buttons:

	Open data file (it is possible to load several files).
	Save data.
	Run inversion parameters setup dialog
	Run data editor (table).
	Run window to work with separate sounding curves.
	Run window to reduce static shift.
	Turn ON/OFF mode of working with block model.
	Turn ON/OFF mode of working with layered model.
	Turn ON/OFF mode of polygonal modeling.
	Run forward modeling process.
	Start (one click) or cancel (second click) inversion process.
	Run one-dimensional inversion.
	Cancel previous step of model changing.
	Turn on magnetic polarization mode.
	Turn on electric polarization mode.
	Go to work with the impedance tensor determinant.

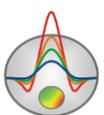
Main Menu Functions

Below the following menu functions and their meaning are described:

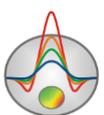
File/Open file	Open data file (it is possible to load several files).
File/Create synthetic survey	Run synthetic measurement system generation dialog. This dialog runs modeling mode.
File/Save file	Save data.
File/Edit file	Open current data file in Notepad editor.



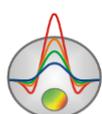
File/Print preview	Run printing dialog of model and data.
File/Run ZondMT 1D	Run current project in 1D interpretation program.
File/Recent	Open one of recently used projects.
File/Reg file	Create registration file to take temporary code (license). Administrator privileges are required.
File/Registration	Enter registration key of the program.
File/Exit	Exit program.
Options/ MT Editor	Run module of quality control and data editing .
Options/Mesh constructor	Run starting model setup dialog .
Options/Project information	Display information about loaded project.
Options/Inversion setup	Run inversion parameters setup dialog .
Options/Data editor	Run measurements editor .
Options/Geological editor	Run geological interpretation window .
Options/3D sections plot	Run goelectric models volumetric visualization .
Options/ Inversion /Set boundaries	Setting geological boundaries dialog . This dialog allows setting boundaries based on a prior data. This option helps inversion create geological model taking into account boundaries geometry.
Options/ Inversion /Long-line inversion	Inversion in “moving window” (for big data set)
Options/ Inversion /Optimization	Optimization settings.
Options/ Inversion /Resolution	Model resolution settings.
Options/ Inversion /FM interpolation	Frequency interpolation for forward problem acceleration.
Options/ Inversion /Smoothness	Smoothing settings in inversion. Options for working with smoothness, starting model, external focusing filter.
Options/ Inversion /Data errors	Select used data weights (all the unit weights, the absolute values or a relative from a file - given to the unit).
Options/ Inversion /Invert with static shift +-3	Select a static shift in the inversion. The parameter sets the maximum deviation limit of the curve on a logarithmic scale.
Options/ Data /Apparent resistivity	Display apparent resistivity values.
Options/ Data /Phase, Deg	Display impedance phases.
Options/ Data /Z impedance	Display impedance values.
Options/ Data /Tipper.r	Display real part of the tipper.



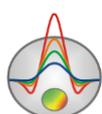
Options/Data/Tipper.i	Display imaginary part of the tipper.
Options/Data/HMT.r	Display the real part of the horizontal magnetic tensor.
Options/Data/Pseudo-section	Display observed and calculated data as pseudosection .
Options/Data/Graphics-plot	Display observed and calculated data as graphics plan .
Options/Data/Calculated data	Display calculated pseudosection in second part of window.
Options/Data/Data misfit	Display misfit pseudosection in second part of window.
Options/Data/Data weights	Display measurement weights pseudosection in the second part of window.
Options/ Model /Block section	Display model editor (model as blocks).
Options/Model/Smooth section	Display model in smooth interpolated graphic palette.
Options/Model/Contour section	Display model as contour section.
Options/Model/Resistivity	Display resistivity model.
Options/Model/Sensitivity	Display sensitivity function model as contour section.
Options/Model/Model quality	Display function of the model quality (the sensitivity normalized to residual model selection).
Options/Model/DOI index	Display research depth index as a contour section (to calculate use two-cycle inversion).
Options/Model/X=Z scale	Set horizontal and vertical scales equal.
Options/Model/ModelEditor toolbar	Display toolbar for working with block model.
Options/Topography/Topo coefficient	Set coefficient of relief distortion with depth.
Options/Topography/Import topography	Load profile topography data.
Options/Topography/Remove topography	Delete profile topography data.
Options/Topography/Restore topography	Restore profile topography data.
Options/Topography/Edit topography	Edit profile topography data.
Options/Topography/Smooth topography	Turn on mode of relief points smooth interpolation (spline interpolation). When this option is ON linear interpolation is used.
Options/Topography/Smooth topography information.	Smooth topography information.



Splined intermediate	
Options/Topography/Absolute coordinates	Display profile altitude, not the height.
Options/ Cutting /Cutting angle	Specify cutting angle for left and right corner of the model.
Options/Cutting/No cutting	Do not cut the model.
Options/Cutting/Cut by angle	Cut the model by specified angle.
Options/Model/Bound by stations	Merge horizontal cells behind observation points (to left from the first and to right from the latter).
Options/Model/Extend bottom	If relief is present this option extends bottom cells to maximum depth.
Options/ Borehole /Create/Edit borehole data	Add (edit) borehole data (lithologic columns)
Options/Borehole/Load borehole data	Open and display file with logging data and lithologic columns, and also files *.mod1d extension (1D interpretation files).
Options/Borehole/Remove boreholes	Delete logging data and lithologic columns from the project.
Options/Borehole/Set column width	Set a width of lithologic column displaying on the section.
Options/ Extra /Model smooth/rough	Run dialog of section cells merge (roughing or smoothing section).
Options/Extra/Data distribution	Run graph window of dependence on the measurements number and the frequency (histogram). It allows removing the frequencies at which few measurements were obtained, and thus speed up interpretation (more).
Options/Extra/Model histogram	Display model parameters distribution plot. Minimum and maximum values of parameter's color scale can be specified in the dialog window.
Options/Extra/Open in modeling mode (On load)	Open data file in modeling mode (option must be turned on before file is opened).
Options/Extra/Include extended nodes	Add the cells at the edges of the model (switched on before file is opened)
Options/Extra/Units	Select units for geometrical axes labels. Kilometers are usually used but meters can be more convenient for PMT survey.
Options/Extra/Reference point HMT	Select base point (from data file points) to calculate horizontal magnetic tensor.
Options/ Import/Export /Import model/data	Import arbitrary data or model.
Options/Import/Export/	Delete graph obtained from imported data.



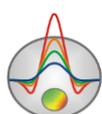
Remove model/data	
Options/Import/Export /Save selection	Save parameters selected in editor cells models.
Options/Import/Export /Load selection	Open file with selected cells and paste at current position.
Options/Import/Export /Extract 1d log	Save vertical resistivity profile for specified horizontal coordinate.
Options/Import/Export /Load 1d log	Paste (in the model) vertical resistivity profile and their limits on the specified horizontal coordinate.
Options/Import/Export/ Load background image	Import background image with *.bmp extension or SectionCor program file (*.sec).
Options/Import/Export/ Remove background	Delete background.
Options/Import/Export/ Change sizes	Change image sizes in exporting.
Options/Import/Export/ Create shaded map	Display model as Shaded map.
Options/Import/Export/ Save synthetic with noise	Save synthetic with added noise
Options/Import/Export/ Save effective data	Save apparent resistivity and phase for both modes as effective values (average in modes). Useful for joint inversion in ZondRes2D.
Options/Import/Export/ Export to ZondMT1D	Export model in ZondMT1D program format.
Options/Import/Export/ Import from ZondMT1D	Import model from ZondMT1D program.
Options/Import/Export/ Load MOD1D/MOD2D	Load model from other Zond programs.
Options/Import/Export/ Save MOD1D/MOD2D	Save model in 1D (2D) format for 1D (2D) interpretation for further models exchange between Zond programs.
Options/Import/Export/ Direct drawing in Surfer	Draw model (contour section) in Surfer program.
Options/Import/Export/ Export model to CAD	Export model to CAD.
Options/Import/Export/ Export model to Excel	Export model to Excel.
Options/ Graphics / Observed graphics	Run graphic parameters setup dialog of observed data.



Options/Graphics/ Calculated graphics	Run graphic parameters setup dialog of calculated data.
Options/Graphics/ Colorscale from PS	Copy pseudosection color scale.
Options/Graphics/ Smooth contours	Smooth model contours in the corresponding displaying mode (Contour section).
Options/Graphics/ Smoothness	Set model smoothing degree in displaying (when working in Contour section mode).
Options/Graphics/ Display every n point	The frequency of points displaying on apparent parameters sections.
Options/Graphics/ Bitmap output settings	Export image settings.
Options/ Buffer /Model	Save model in buffer or paste from it.
Options/Buffer/Open	Display all buffer models.

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, polarisability) values of corresponding block model cells (from results of preliminary inversion or modeling).
Modeling/Set values to background	Set polygon parameters (resistivity, polarisability) 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/ GraviMagnetic / Load new data	Load measurements of magnetic and (or) gravity field.
Modeling/GraviMagnetic/ Add new data	Add new data of magnetic and (or) gravity field measurements.
Modeling/GraviMagnetic/ Field settings	Set parameters of normal magnetic (gravity) field, survey parameters.
Modeling/GraviMagnetic/ Substract median grav	Substract median from the set values of the gravity field, it means to lead to the anomalous field values.
Modeling/GraviMagnetic/	Substract median from the set values of the magnetic field, it



Substract median mag	means to lead to the anomalous field values.
Modeling/GraviMagnetic/ Inversion	Start polygonal inversion of gravity\magnetic data (parameters inversion for specified parameters).
Modeling/GraviMagnetic/ Display GM window	Open window of displaying gravity\magnetic field measurements.
Modeling/Display color scale	Display color scale of resistivity\polarisability.
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 (domain with specified parameter value will be highlighted)
V / cursor in model editor	Delete selected.
Up/down / cursor in model editor	Change current value.
Space	Calculate forward problem.

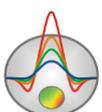
Status bar

Status bar 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.

Misfit is the most important parameter from the list. Averaged relative misfit (in percents) is calculated taking into account all measurements of current model. Other misfit type is used for inversion; it includes data of model smoothness. It can be seen during inversion.

Adjusting starting model mesh settings



When file is successfully loaded, starting model setting dialog appears (Fig. 2), in which it is possible to choose the mesh parameters and the resistivity of the starting model.

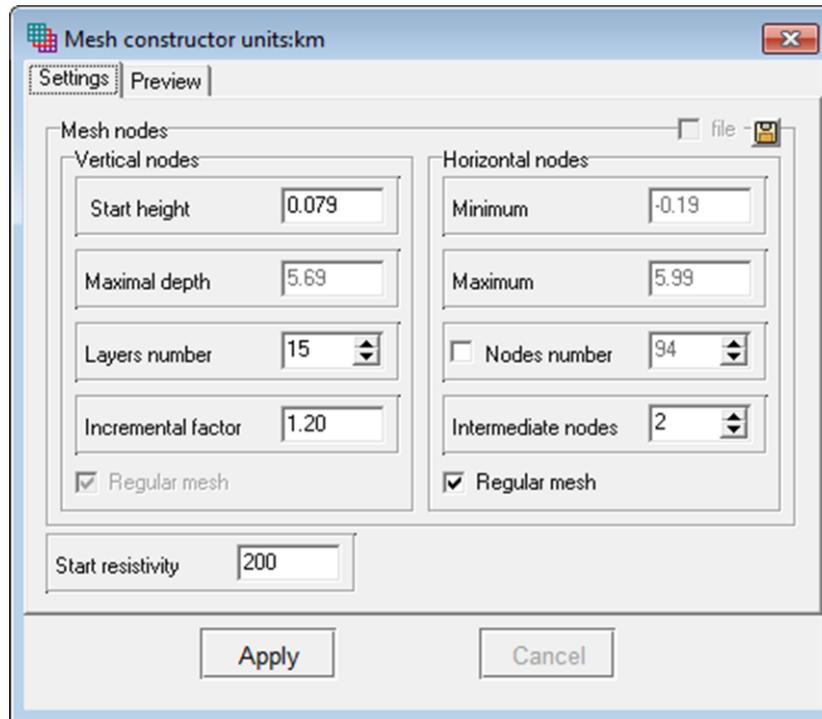


Fig. 2 Starting model setup dialog

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 unique measurement periods but does not exceed 40.
- Thickness of the next layer is 1.2 times more than the previous one.

Interpreter can change these parameters due to his concept of depth and details of investigation.

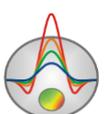
Start height – sets thickness of first layer. This value must be approximately equal to the width of the cell and necessary model resolution.

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. Maximum depth should specified proceeding from effective field penetration depth.

Layers number – sets number of model's layers. 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 – sets minimal coordinate of the profile.

Maximum - sets maximum coordinate of the profile.

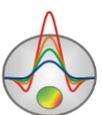
Intermediate nodes – sets number of complementary nodes between two adjacent sounding points on profile (0 – 4). It is recommended to specify 1 or 2 complementary nodes between adjacent stations.

Half-space resistivity – sets resistivity of starting model.

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 stations is very different (It is advantageous for the accuracy of forward and inverse problem solutions).

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

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



Main data file format

Program presents universal data format which contains information about coordinates and relative elevations (in kilometers) of sounding sites.

Data format of the program.

First line contains sequence of periods (in seconds) during which measurements were conducted (in increasing order).

It is followed by information about every sounding site on profile, divided in blocks described below.

Block of sounding location description

First line indicates start of sounding site description (must contain «{» symbol).

Second line contains complementary sounding parameters.

First record is sounding site coordinate on profile, second is relief excess.

Third to sixth lines contain field measurements.

Each line must start with code-key which shows to the program what type of data follows this key. Code-keys which control type of data have the following values:

First option of setting keys:

«Ro_a» – apparent resistivity.

«Phi» – impedance phase (in degrees, positive value).

In this case polarization type is unknown. Such data can be used to work in magnetic and/or electric polarizability modes. Example-file [sample_with_unidata](#).

«_w» – measurement weight.

Second option of setting keys:

«Ro_a_tm» – apparent resistivity for magnetic polarizability.

«Phi_tm» – impedance phase for magnetic polarizability (in degrees, positive value).

«Tm_w» – measurement weight for magnetic polarizability. If measurement errors are not known, program automatically sets weight “1” for every measurement. Example-file [sample_with_tmdata](#).

Third option of setting keys:

«Ro_a_te» – apparent resistivity for electric polarizability.

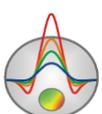
«Phi_te» – impedance phase for electric polarizability (in degrees, positive value).

«Te_w» – measurement weight for electric polarizability. If measurement errors are not known, program automatically sets weight “1” for every measurement. Example-file [sample_with_tedata](#).

Number and order of records in lines must strictly correspond to acquisition geometry described in the first line of the file. If measurements at some periods are missing, their values are replaced by «*» symbol. If there are no data about impedance phase, the whole line is excluded.

Last line indicates end of sounding site description (must contain «}» symbol)

Column of complementary horizontal grid nodes can be added after description block if necessary. Coordinate of every new node is entered after *** symbol. Complementary nodes are usually added for model extension over last stations or if there is sharp relief beyond profile. Example-file [sample_with_ext_nodes](#).



It is preferable to record stations in the same order as on profile (in increasing order of coordinate).

For correct work 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).

It is recommended total record number should be no more than 5000 in one file.

Data file can be created using Zdconvert program supplied with inversion program. Procedures of work with this program are described in [Appendix 8](#).

Preparing data for inversion

Processing magnetotelluric data is divided into three phases: viewing and editing of recorded electromagnetic field time series, obtaining transfer functions, post-processing. The first two stages are generally done in software supplied with the MT equipment. ZondMT2D allows performing the post-processing.

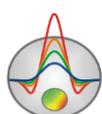
Post-processing (the final stage of data preparation for inversion) includes the following set of procedures (where appropriate): viewing, estimation of quality, editing, transfer functions smoothing, impedance polar diagrams building, the rotation of the impedance tensor, suppression of static shift effect.

These procedures are implemented in the program in the form of individual functions, which can be used in the process, and as a part of [module of quality control and data editing MT Editor](#), which is a collected in single window complete tools set for post-processing of the MT data.

Options for data measurements editing

Measurements editor

To view and edit of individual measurement values of apparent resistivity, phase and weight in a table use measurements editor. Measurements editor run by tab **Options/Data Editor**, or by pressing the button  on the toolbar. The tab includes a table that allows setting each measurement (Fig. 3).



ID	X,km	T,s	$\rho\alpha$	ϕ	err	$\rho\alpha$	ϕ	err
1	0	0.01	90.2	49.6	0	*	*	0
2	0	0.01	85.1	49.7	0	*	*	0
3	0	0.01	80.5	49.2	0	*	*	0
4	0	0.02	77.1	48.3	0	*	*	0
5	0	0.02	74.9	47.2	0	*	*	0
6	0	0.02	74	46.1	0	*	*	0
7	0	0.03	74	45.1	0	*	*	0
8	0	0.04	74.8	44.3	0	*	*	0
9	0	0.04	76.1	43.7	0	*	*	0
10	0	0.05	77.6	43.3	0	*	*	0
11	0	0.06	79.4	43.1	0	*	*	0

Fig. 3 Table of measurements editor

The table contains 9 columns:

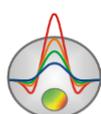
ID	Measurement index
X,km	Location of sounding sense.
T,s	Measurement period.
$\rho\alpha$	Calculated value of apparent resistivity for magnetic polarizability.
ϕ	Calculated value of impedance phase for magnetic polarizability.
Weight	Measurement weight for magnetic polarizability.
$\rho\alpha$	Calculated value of apparent resistivity for electric polarizability.
ϕ	Calculated value of impedance phase for electric polarizability.
Weight	Measurement weight for electric polarizability.

The last six columns, if it is necessary, can be edited using the keyboard. Absurd values for the apparent resistivity or phase should not be entered. Measurements weights are specified in the range from 0 to 1.

When moving the cursor in the table, the position of the active measurement is displayed on pseudosections or graphics plane.

Sounding curves view and edit dialog (Sounding curves)

Press button  in the toolbar of the main program window to run sounding curves view dialog (fig. 4). This dialog allows browsing measured (continuous line) and calculated (dashed line) sounding curves for TM and TE modes for each sounding site. The dialog also allows browsing calculation results of 1D forward problem for local model of active station.



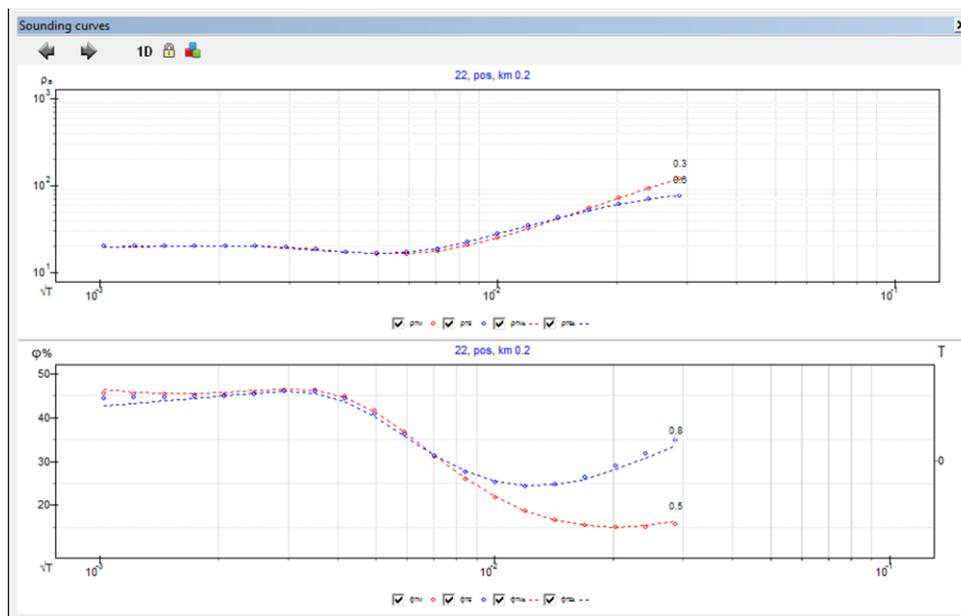


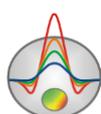
Fig. 4 Sounding curves view and edit dialog Sounding curves

Toolbar of Sounding curves dialog contains the following buttons:

	Start browsing sounding curves of previous point.
	Start browsing sounding curves of next point.
	Calculate 1D forward problem for current point of local model (1D layered model directly under sounding site). Modeling can be useful if it is necessary to estimate degree of medium “two-dimensionality”. Sometimes it helps to find out if phase is correctly specified (in necessary quadrant).
	Fix minimum and maximum values on axes. After fixing axes limits do not vary when changing points. Axes limits can be also specified in axis parameters setup dialog.
	Open list of advanced options

Advanced options are:

Invert this station	Sets use or not current point in inversion. If data at the point has intense noise, it can be ignored in inversion.
With static shift	Draw theoretical Rho apparent curve taking into account fitted p-effect value. This option is necessary for displaying curves after inversion, if static effect was to be found for each curve.
BI-logarithmic	Set bi-logarithmic scale.
Phase axis [0-90]	Set phase range from 0 to 90.
Error gates	Display weights.



Remove mode	Turn on the option of removing points.
Select mode	Turn on the option of selecting points.
Process together	Take action for resistivity and phase simultaneously (when selecting or deleting point on a curve, the corresponding point is selected or deleted on another curve).
Calculated- Observed	Replace measured values to calculated values.
1D local field	Calculate 1D forward problem at the current point for the local model (one-dimensional layered model directly under the sounding point). Modeling can be useful when it is necessary to estimate the degree to which curves are "two-dimensionality" rather than 1D. Sometimes it helps to determine whether the phase is correctly set (in the required quadrant).
Original data	Return to raw data.

As can be seen from the additional options, window allows also data editing. To do it, use the “circle” tool. To run it, turn on **Select mode** option (to select) or **Remove mode** (to remove). Circle radius can be changed by the mouse wheel. Using the circle, it is possible to remove a point (the center button), to increase (right button) and decrease (left button) point weight in inversion. Point weight – is the reciprocal of the measurement dispersion and it is indicated in the program in the statistical goal form. The larger the bandwidth, the less influence this point has for the resulting model.

It is possible to edit the vertical and horizontal axes. To run parameters axes settings dialog right-click the mouse while holding the SHIFT button on the selected axis (see the part [Axes editor](#)).

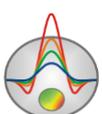
Editing data dialog Data distribution

Data distribution dialog allows rejecting frequencies with low number of measurements and thus quickening interpretation process. Dialog is accessible in the main program menu **Options/Advanced/Data distribution**.

Using this option allows estimating number of sounding points on the profile, on which data of specified period exist.

Figure 5 shows an example of using **Data distribution** function for rejection of frequencies with small amount of data. There is a table in the left part of function dialog window (fig. 5 B) which contains the following columns: N – order number of period, T – period, nTM – number of points on current period for TM mode, nTE - number of points on current period for TE mode, X – select/deselect period. Data distribution for each period is displayed as bar charts in the right part of the window.

In this example several points have data on low frequencies (fig. 5 A). These data lead to calculation time increase. In order to delete periods and decrease calculation time select them in the table and press button  in the left upper corner of the bar chart. The example (fig. 5 B) shows that low-frequency measurements observed only in several profile points were rejected.



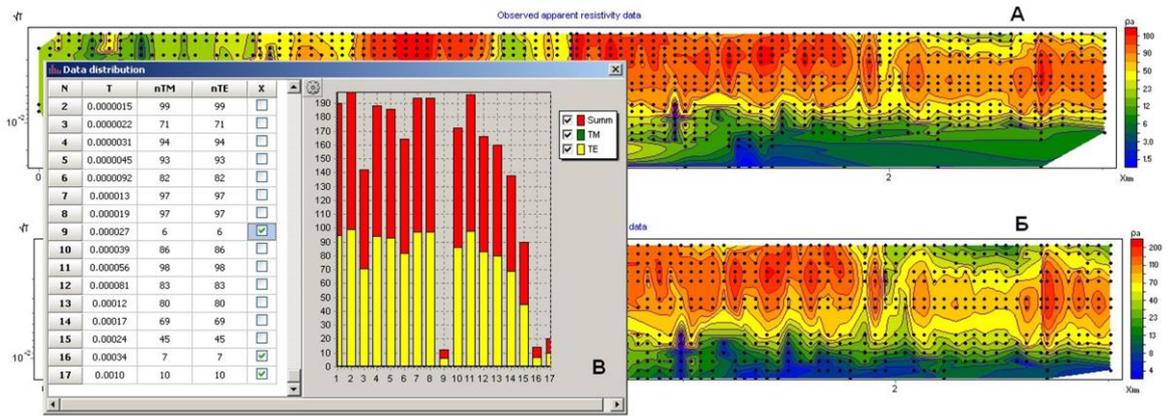
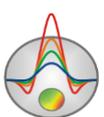


Fig. 5 Example of Data distribution function usage for rejecting unreliable data. A-B – pseudosections of apparent resistivity before and after usage of **Data distribution** function respectively, B – **Data distribution** dialog window

This dialog is an analog of Distribution mode tab [MT Editor](#).



Module of quality control and data editing

For quality control and data editing a special module is available in the ZondMT2D program. To run it, select in the menu **Options/MT Editor**. The module allows performing all basic post-processing operations of MT data.

Tab **Data** of **MT Editor** module consists of two parts (Fig. 6): the top and the bottom. The left side of them contains graphs of the selected parameter distribution along profile, the right side displays the curve for the selected parameter at the point indicated on graphs by red dotted vertical line. Parameters displayed on the top and the bottom of the window are selected, respectively, by buttons **P1** and **P2**.

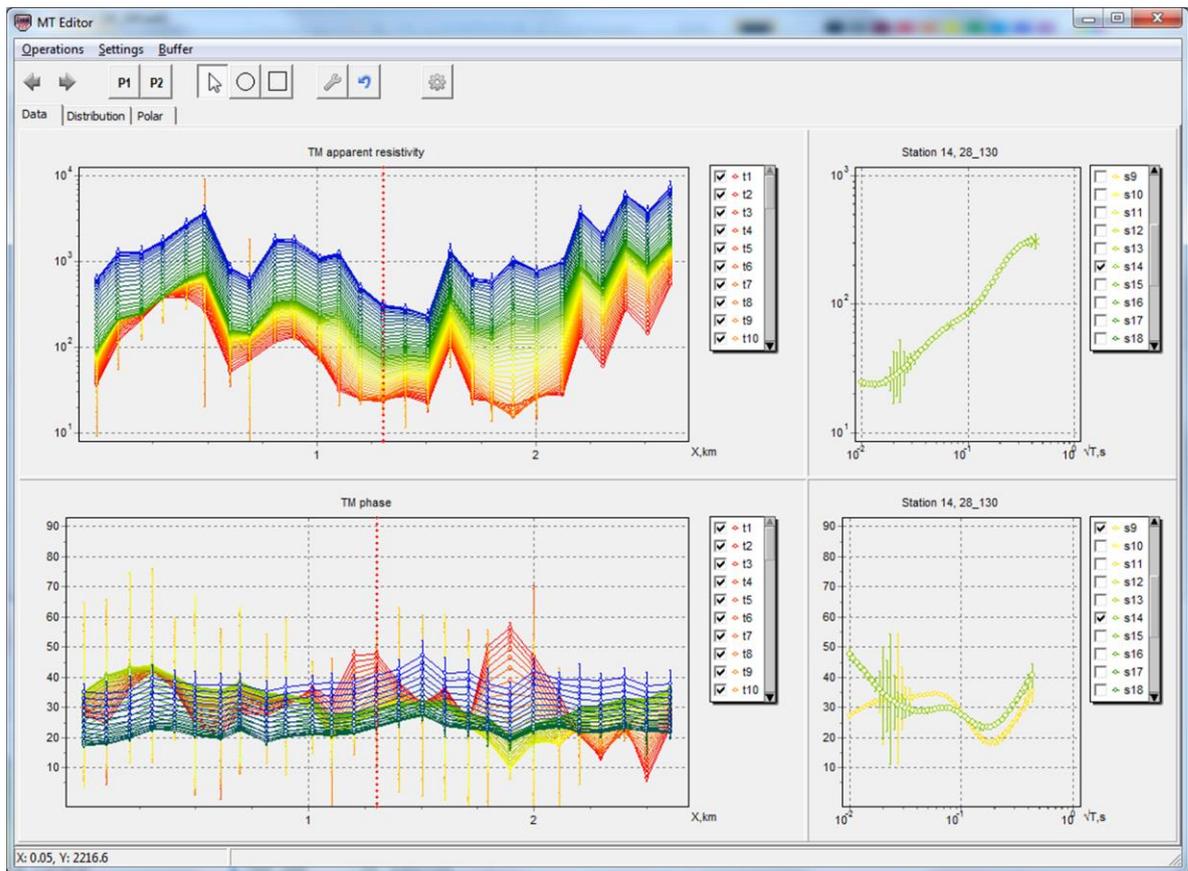
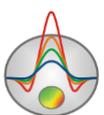


Fig. 6 Data main window of MT Editor module

In the example shown in fig. 6 in the upper window part graphs of apparent resistivity for TM mode on the profile are displayed. Each graph corresponds to an individual frequency. List of graphs is located in the right part of the window and allows disabling or adding various frequencies curves. The red vertical line defines the current point for which on the right side of the window (in the example - Station 14, 28_130) apparent resistivity curve is displayed. The points list in the right window part allows attaching graphs to other selected points. To move between the profile points, use the buttons **←** and **→** or just click on the graph point. When the option **Settings/Plots/P1 (P2) Multigraphics** is turned on, curves for all points selected with the



mouse on the graph, are displayed in the corresponding curves window (alternative way of selecting curves in the list).

To display "gates" weights on the graphs, use option **Settings/Display gates** (Fig. 6).

Use the menu options **Settings/Background data** to display initial data graphs (before editing) - **Original data**, calculated data - **Calculated data** or synthetic data for the model of one-dimensional inversion - **1D inversion**.

Parameters of lines and points are configured using the menu options **Settings/Graphics**.



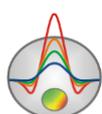
Fig. 7 Dialog of displayed parameters selection

To select points for editing on the curves and graphs, use tools  or . Afterwards, selected points can be further edited using the menu option Settings/Mode or :

Select/deselect	Turn on/off mode of points editing
Edit points	Allow editing points (moving)
Shift curve	Turn on the mode of moving the whole curve (graph). Turn on the tool  when selecting the curve.
Shift selected	Move all selected points
Edit gates	Edit weights of measurements (gates)
Remove points	Delete points
Smooth points	Smooth
Interpolate	Replace selected points by interpolation
Denoise	Suppress noise

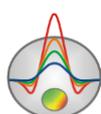
The other editing operations are made using menu items **Operations**:

Return to start data	Return to the data before editing.
Return to original data	Return to raw data
Remove selection	Deselect
TM 1D smoothing	Run 1D smoothing for TM mode data
TE 1D smoothing	Run 1D smoothing for TE mode data



TM strong smoothing	Run strong smoothing for TM mode
TE strong smoothing	Run strong smoothing for TE mode
P1 remove invisible	Don't display curve points, for which corresponding graphs are turned off (in P1 window)
P2 remove invisible	Don't display curve points, for which corresponding graphs are turned off (in P2 window)
TM correct phases (0..90)	Reduce TM phase to 0-90 range
TE correct phases (0..90)	Reduce TE phase to range 0-90
TM suppress shifts	Suppress static shift of TM mode
TE suppress shifts	Suppress static shift of TE mode
Apply inverted shifts	Apply correction of static shift obtained by inversion results (after performing inversion)
Exchange TM-TE	Replace TM mode data to TE mode data (and vice versa)
Rotate dataset	Rotate the coordinate system
Convert2impedances	Convert to impedance
Apply and close	Apply changes and exit from editor

When not working with MT-field invariants, two-dimensional model coordinate system should be oriented according to the longitudinal and transverse directions relative to the structures strike. If the profile orientation does not satisfy this condition, during data file opening, the program will report that it is necessary to rotate the coordinate system. This rotation may also be needed when selecting areal data points for interpreted profile, or if significant three-dimensional distortions exist. Appropriate assessments are based on the polar diagrams analysis (tab **Polar** is described below), and the coordinate system rotation is carried out with the option **Operations/Rotate dataset**. The rotation parameters are set in the appeared dialog (Fig. 8). It is possible determine rotation type, using **Rotation** tab (in a user-defined angle, according to tensor main axes directions, maximum Z_{xy} or phase tensor), on the **Options** tab - select a frequencies range for which the turn with the specified parameters is made.



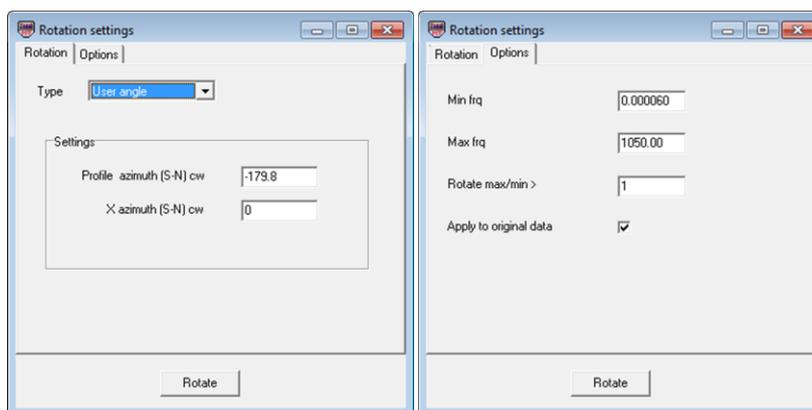


Fig. 8 Dialog of selection of coordinate system rotation parameters

Tab **Distribution** allows estimating the distribution of data on the frequency in a convenient way (Fig. 9). If there are too few data at any frequency, the data of the frequency can be removed. It will accelerate the inversion process and help avoid additional errors. Each group of columns of different colors corresponds to the period number (frequency). Each color corresponds to the parameter specified in the legend on the right part of window. To delete data for selected frequency, run menu by right-clicking on the appropriate column.

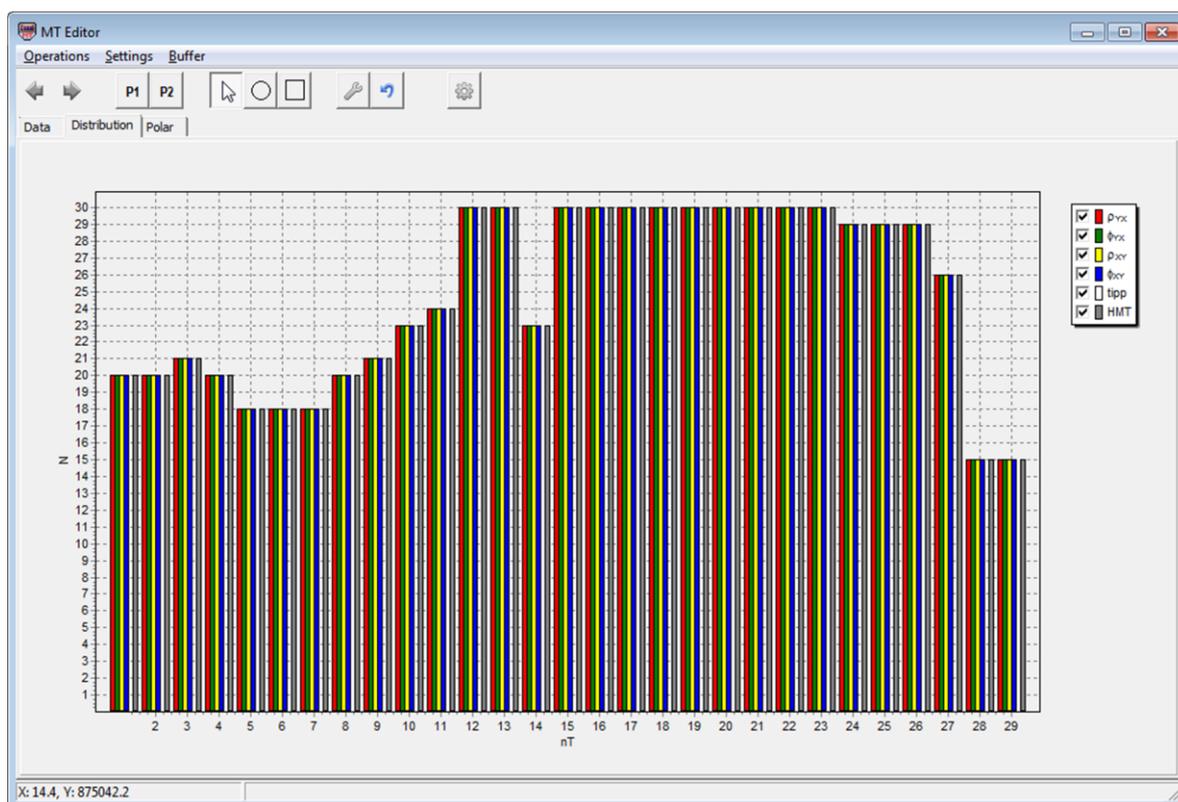
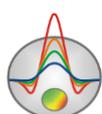


Fig. 9 Tab Distribution of MT Editor module

Tab **Polar** (Fig. 10) is performed to create impedance tensor and tensor phase polar diagrams, needed to determine the dimensions of the studying medium and the needed to rotate the coordinate system, the separation of regional and local effects etc. Polar diagrams are constructed for each point and for each frequency. Different frequencies are displayed on the



polar diagrams in different colors, as shown in the list on the left window part (if it is necessary frequencies can be turned on and off).

To run menu of parameter selection, displaying in tab **Polar**, right-click in the upper window part (in the header Impedance diagrams in Fig. 10):

Impedance diagrams	Display impedance tensor polar diagrams
Phase tensor diagrams	Display phasing tensor polar diagrams
Strike	Display effective strike
Ind arrow	Display induction arrows (tippers)
No norming	Do not normalize values (diagrams radius)
Norming for frq	Normalize by frequency
Norming for frq/st	Normalize by frequency and reduce to unit (for all frequencies radius values are equal)
Stations marks	Display points locations
Frequency range	Set frequencies range, for which diagrams will be created

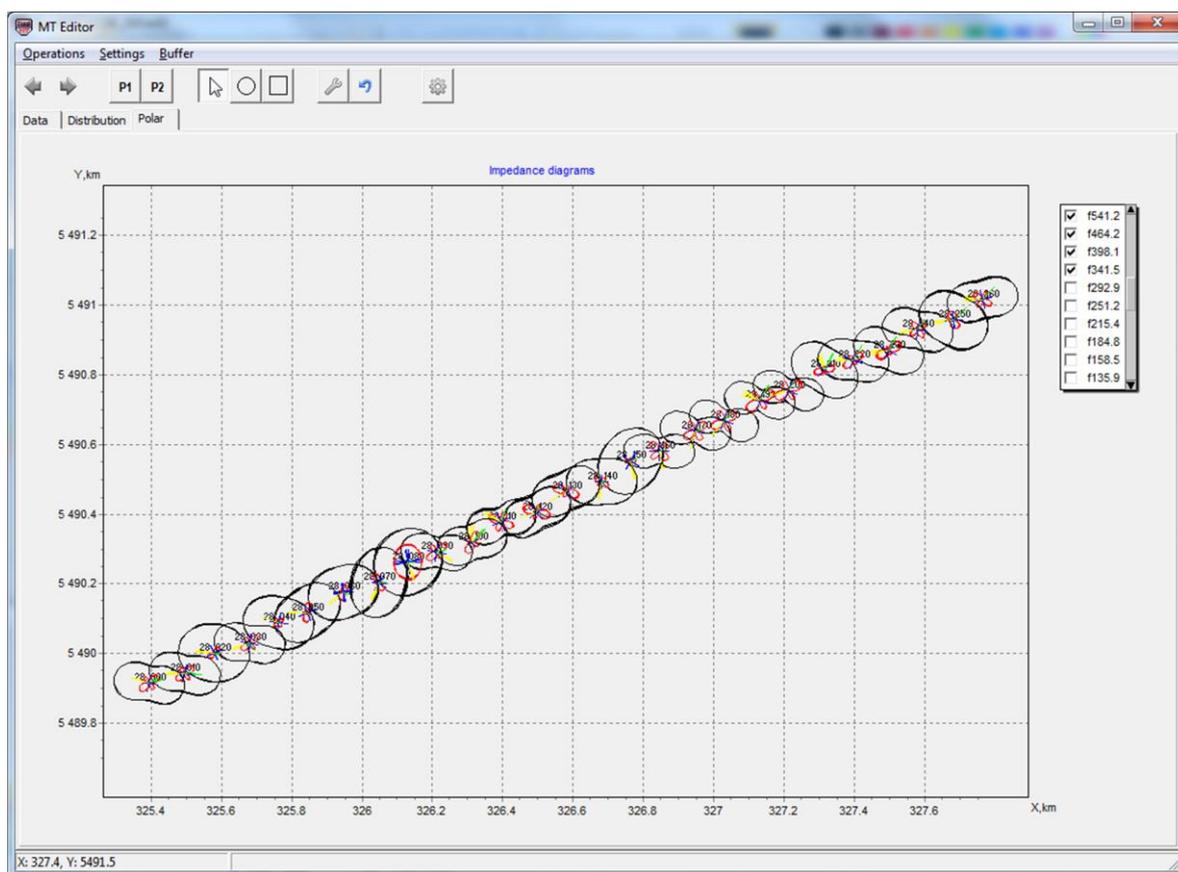
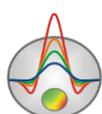


Fig. 10 Tab Polar of MT Editor window



Determination of the selected processing parameters effectiveness often can be assessed only by comparing them with each other. It is possible to store in a single project several data processing variants in the ZondMT2D. This allows carrying out the data inversion using different processing parameters, as well as returning to the saved variant or continue processing from the saved phase in a single project. Saving of the data processing results is carried out using the option **Buffer** of MT Editor window.

One can save the current processing results in four possible buffers in the pulldown list. If the selected buffer already contains data (in this case, its name will be displayed with tick), the program will ask about loading data from the buffer or rewrite the content of the current data.

It is possible to return to the raw data in the editing process (**Operations/Return to start data**) or to the data taken from the buffer before they were changed (**Options/Return to original data**).

Using the option **Buffer** of MT Editor window is similar to the procedure of working with models based on the inversion results ([Working with several models in a single project](#)).

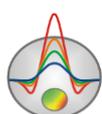
Static shift correction

Static shift caused by near-surface inhomogeneities, considerably complicate the interpretation of the apparent resistivity curves, creating false geoelectric structures. The reliability of data interpretation depends significantly on the quality of the static shift removal. There are several features (instruments) for the detection and correction of static shift in ZondMT2D program. In practice, the most efficient is to use all of these instruments, or at least to compare their efficiency on real data.

Identification and manual correction of the static shift is carried out in the [MT Editor](#). Displaying a single window with several neighboring resistivity and phase curves allows visual inspection of data for static shift effects. Using MT Editor tools, one can change the curve level or its segments, with reference to the adjacent curves, calculated curves obtained by one-dimensional inversion results or the calculated curves of the model obtained by the two-dimensional inversion.

An alternative to manual correction in MT Editor module is *a static shift correction by averaging the apparent resistivity*, with following filtration (module MT Editor, options **TM suppress shifts** and **TE suppress shifts**). This method is effective in a dense observation grid.

The most reliable way to correct static shift is to use TDEM sounding data. TDEM data is imported with the option **Options/TDEM data/Load TDEM data** (supporting formats are ZondTDEM1D and UFS) in the ZondMT2D program. Also, it is possible to use for correction VES data (imported into the project with the option **Options/VES data/Load VES data**. After data importing, it is necessary to press the main menu button **SSC** and go to the window of determining the corrections for each point (Fig. 11). In the upper left window part displays graphs of apparent resistivity on MT and TDEM, in the lower left - phase (observed and calculated), in the upper right - depth curve (observed and calculated), in the bottom right - geoelectric model of one-dimensional joint inversion.



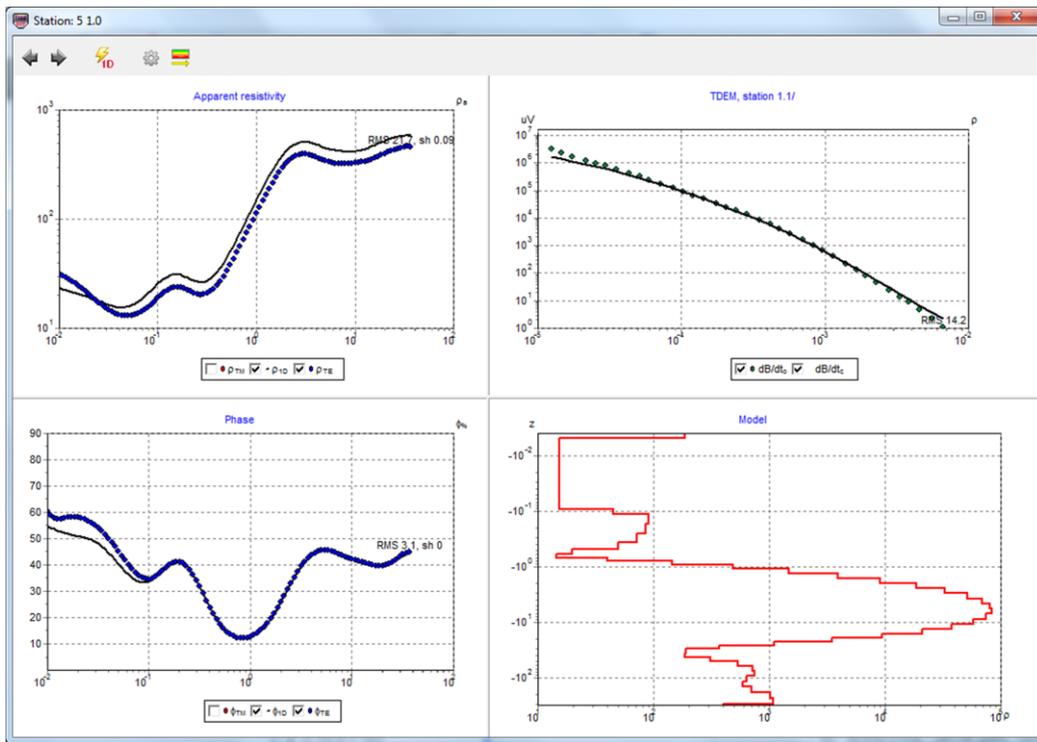
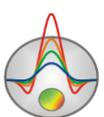


Fig. 11 Window of static shift correction based on of the TDEM and VES data

The buttons and allow proceeding to the next or previous point. The button runs the joint inversion process of MT and TDEM (MT and VES or MT, TDEM and VES). The button determines the selection of TDEM points for the considered MT point and produced action:

Rebuild Model	Reset model
Any near TDEM/VES point	Select any near TDEM point (for example, if there is only one or two points on the profile)
Nearest TDEM/VES point	Select nearest TDEM point
Invert only visible	Include to inversion only visible parameters (selected by “tick” under each graph)
Invert thicknesses	Run thickness inversion
Apply static shift TM	Apply static shift correction for TM mode
Apply static shift TE	Apply static shift correction for TE mode
Phase correction TM	TM phase correction
Phase correction TE	TE phase correction
Auto correction	When option is turned off, program ask user confirmation

After clicking the option Apply static shift TM and Apply static shift TE the corresponding corrections will be entered in the data.



The button  allows copying the model to the resistivity section. This allows setting the model obtained by one-dimensional joint inversion of MT, TEM and VES results as a starting model.

Apparent parameters visualization

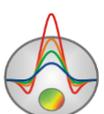
In the main window the observed and calculated parameters can be displayed as pseudosection **Options/Data/Pseudo-section**, and as graphics plot **Options/Data/Graphics-plot**.

Graphics plot

The graphics plot displays parameter values along the profile, in the graphs form (Fig. 12). Use mouse to work with graphics plot: Zooming in or moving part is performed with 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. 13A). To return to the original scale, do the same actions, but the mouse is moving up and to the left (Fig. 13B). To move (scroll) graphic, move the mouse with the pressed right mouse button.

By pressing the left mouse button on the graph point makes the following actions: hiding the rest graphics and displaying the electrodes position for the active point (until releasing the mouse button). Editing the measured values is done by moving the graph point with the right mouse button pressed.

Dialog of graphs plot settings is called from the Main Menu **Options/Graphics/Observed graphics Calculated graphics** (described in [Appendix 1](#)). To run Graphics editor click the right mouse button with pressed SHIFT key on the graph (described in [Appendix 2](#)). To run the axes editor right-click while holding down the SHIFT key on the axis (described in [Appendix 5](#)). To run legend editor click the right mouse button while holding down the SHIFT key on the legend to the right of the graph (described in [Appendix 3](#)).



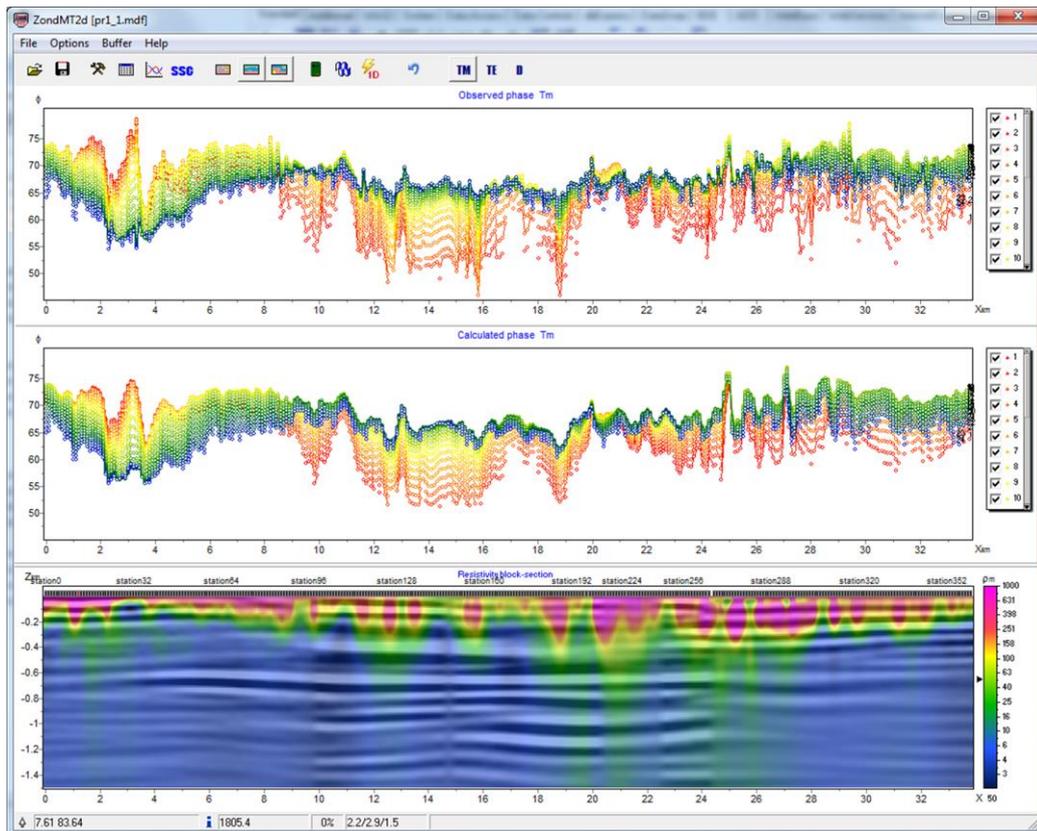


Fig. 12 The main program window. Displaying data as graphics plot

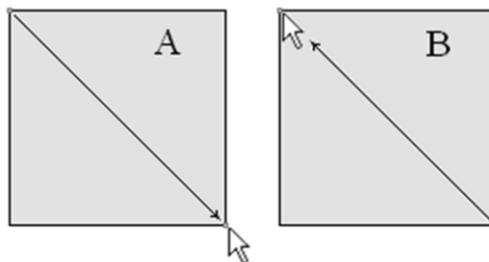


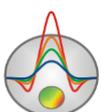
Fig. 13 The direction of mouse movement when zooming

To select one and, accordingly, the removal the other graphics click on the legend while pressing SHIFT. Two click for inverse action.

To scroll through the graphs use the mouse wheel. Select several adjacent plots (in the legend) and scroll the mouse wheel, moving the cursor to the legend. Activity graphs indexes will vary. When right-clicking on the graph point measurement will be highlighted in the table.

It is possible to exclude some measurements from processing specifying 0 as the weight in the mode of graphs data displaying. A separate measurement can be deleted by pressing the ALT key and the left mouse button on the graph point. Click the right mouse button and ALT in the graph to specify weight 0 for all graph measurements.

If the measurement weight values are specified in the input data file, it is possible to display an appropriate range of the resistivity values in the graphs (confidence interval) using the



option **Options/Data/Display/Error gates**. Confidence intervals (weights) can be adjusted in the graphics mode by pressing the left or right mouse button while holding down ALT.

Pseudosection

Pseudosection visualizes parameter distribution along profile with depth (fig. 14).

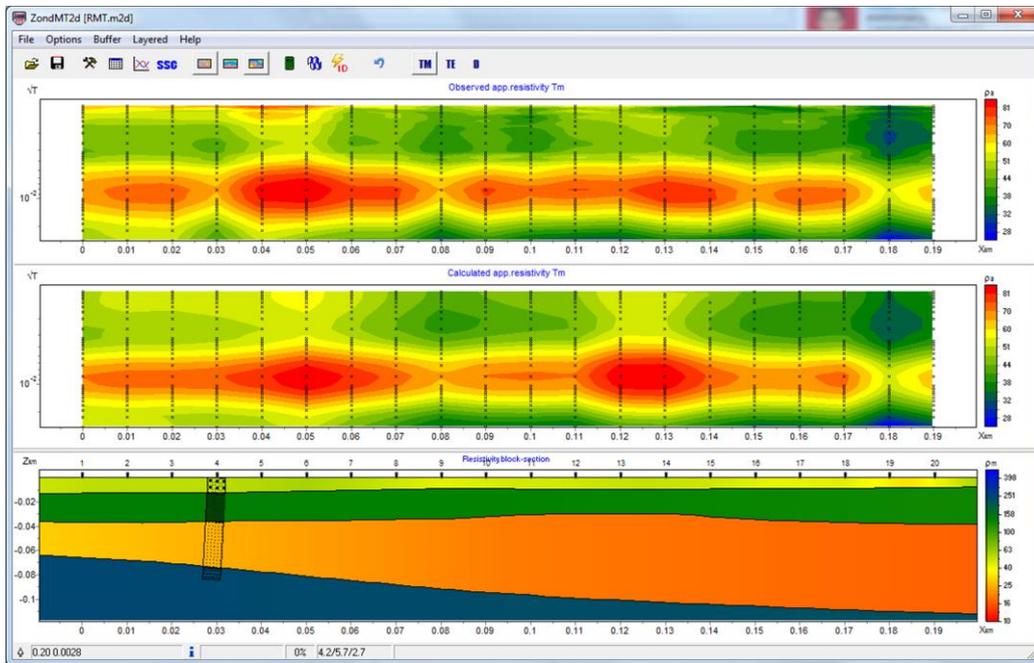


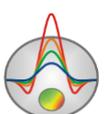
Fig. 14 The main program window. Displaying data as pseudosection

Plotting in form of contour lines has axes: measurement coordinate along profile, pseudodepth. The color scale sets ratio between the displaying value and color.

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

Log data scale	Use logarithmic scale on colour bar.
Smooth mode	Use smoothing interpolating palette/contour section.
Display grid point	Display measurement point ticks.
Display ColorBar	Display colour scale.
Setup	Run pseudosection parameters setup dialog.
Print preview	Print pseudosection.
Save picture	Save pseudosection as *.emf graphic file.
Save XYZ file	Save pseudosection as *.dat text file.
Default	Set pseudosection parameters on default.

Pseudosection parameters setup dialog is described in [Appendix 4](#) Appendix 1: Graphics editor. Axes editor is called by right-clicking while holding down the SHIFT key on the axis



described in [Appendix 5](#). Editor pseudosection points can be called by clicking the right mouse button while holding down the SHIFT key at the point (described in [Appendix 7](#)).

When displaying data as pseudosection it is possible to view the data of the individual settings using the menu **Options/Data/Display/**. When there are large measurement volumes it is possible to thin pseudosection points using the option **Display every N point**. By default, this option is turned on when loading a data file that contains more than 3000 measurements.

To display measurement weights as pseudosection in the middle program graphic section use the option **Options/Data/Data Weights** and after the inversion process relative error for each measurement by **Options/Data/Data Misfit**.

Input and editing topographic information

There are two ways to input relief - through the information in the input data file or by using the option of topographic information import of the main menu **Options/Topography/Import topography**. Importing topographical information is done from a text file that contains two columns: the distance on the profile and elevation. Option **Options/Topography/Import topography** creates a table with information from the selected text file is loaded (Fig. 15). 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 .



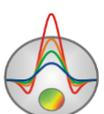
#Ind	None	Ind	None	None
1	None	1	-2.1	-7.723
2	oldX	2	-1.8	-7.952
3	newX	3	-1.5	-8.182
4	X	4	-1.2	-8.411
5	Alt	5	-0.9	-8.64
6	#Ind	6	-0.6	-8.87
7	Ind	7	-0.3	-9.099
8	0.3	8	0	-9.329

Fig. 15 Table of topographic information import

Menu item **Options/Topography/Edit topography** calls a dialog that allows editing topographic information (contained in the source data file or importing to the project) using the table.

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

When working with data obtained in the rugged area, it is recommended to use the option of topography smoothing (**Options/Topography/Smooth topography**). Turn the option on before opening a data file.



If it is necessary to take into account the relief forms outside the area, it is possible to add additional nodes to the model edges using the option **Options/Extra/Extended nodes**. Turn the option on before opening a data file.

When working with data it is sometimes convenient to disable the relief – by using the option **Options/Topography/Remove topography**. To restore the relief use the option **Options/Topography/Restore topography**.

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

By default, relief heights are displayed relative to zero in the model window, when the option **Options/Topography/Absolute coordinates** is turned on. Specified in the file relief heights are displayed.

When saving **Grid file** with real heights in the exported file option **Absolute coordinates** must be turned off.

Relief distortion factor with the depth (0-5) can be set in menu **Option/Topography/Topo coefficient**. 0 - the relief of each next layer repeats the previous one. 1 - relief flattens with depth, the last layer - flat (Fig. 16). Distorted depth is calculated as follows:

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

where **Topo** - relief excess, z - depth from the surface.

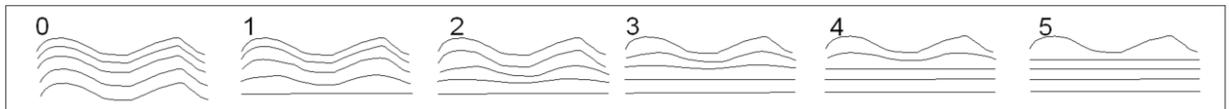
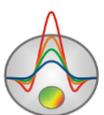


Fig. 16 Model layers distortion with Topo coefficient parameter values of 1 to 5



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 calibrate the model grid, its editing and changing of the starting model resistivity values (described in the [model creation](#)). 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**.

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

Convergence of each measurement between the observed and calculated values can be evaluated by displaying pseudosection of relative discrepancy with the option **Options/Data / Data Misfit**.

Calibration of model mesh and operations with relief

One of the most important aspects of successful forward and inverse modeling is correct selection of model mesh. Mesh must ensure exact solution for each frequency. Forward modeling is calculated on conditions that electromagnetic field in every cell of the section changes linearly. It is known that field nonlinearity degree decreases with depth but in different ways for each frequency. Geoelectric contrast of objects is an important factor that influences field behavior. That is the reason why mesh needs to be more concentrated within the bounds of these objects to receive more precise modeling results. But for discretization parameters model quality also depends on how far "infinite" boundaries that ensure fulfillment of conditions at infinite are moved away. And in its turn this depends on size of visible region of the model. In other words external mesh is set on the basis of parameters of inner, specified by user mesh.

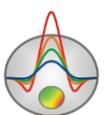
In order to raise reliability of received data model mesh calibration should be performed. Calibration is carried out for half-space model without relief. When starting model is created, press button  of forward modeling in the toolbar. A criterion of proper mesh selection is closeness of all calculated apparent resistivity values (for both modes on each frequency) to resistivity of half-space and closeness of phases to 45 degrees.

If strongly variable topography is present, it is helpful to add 1-2 complementary nodes in the upper part of vertical grid (more details in [Model editor](#)). If forward solution does not change much because of this, it means that mesh is created properly.

Program parameters setup dialog

This dialog serves to specify parameters of inverse modeling. Use **Options/Program setup** option in the main menu or button  in the toolbar to run it (fig. 17).

First tab **Inversion** serves for inversion parameters adjustment.



Inversion option selects algorithm of inverse modeling. Let us consider inversion algorithms by example of subsurface model that consists of several blocks (fig. 18). For algorithm testing theoretical response should be calculated and 5 percent Gaussian noise superimposed.

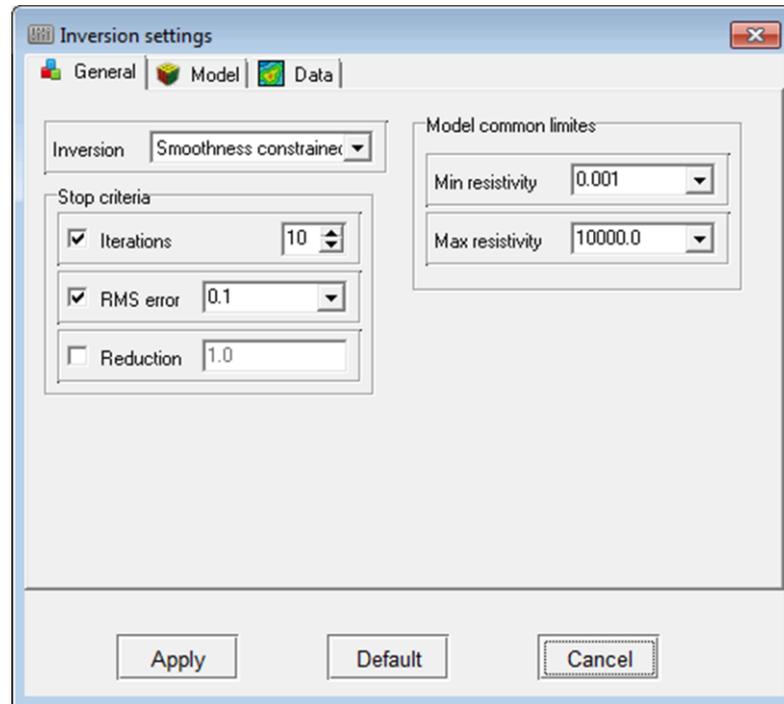


Fig. 17 Inversion parameters setup dialog, **Inversion** tab

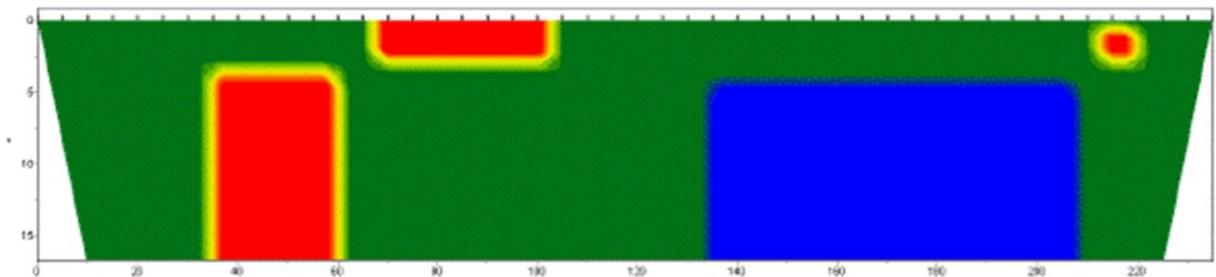
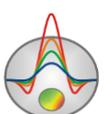


Fig. 18 Test subsurface model

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). This type of inversion is recommended in the majority of cases. Matrix equation for this kind of inversion is the following:

$$(A^T W^T W A + \mu C^T C) \Delta m = A^T W^T \Delta f \quad (9)$$



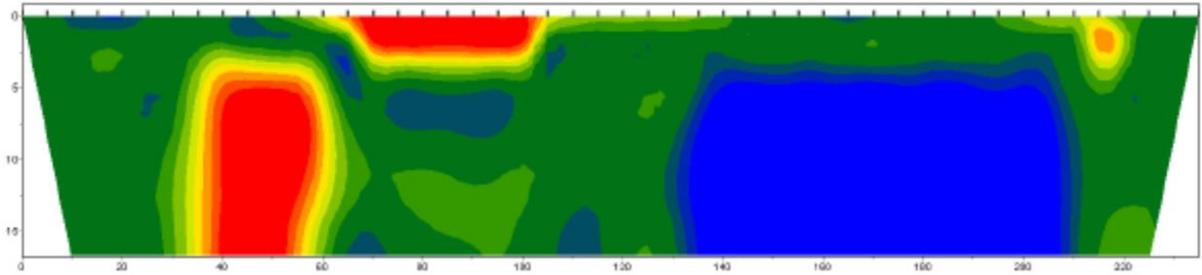


Fig. 19 Regenerated model as a result of *Smoothness constrained* inversion

Occam is inversion by least-square method with use of smoothing operator and additional contrast minimization. As a result of this algorithm the smoothest parameter distribution is received (fig. 20). Matrix equation for this kind of inversion is the following:

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

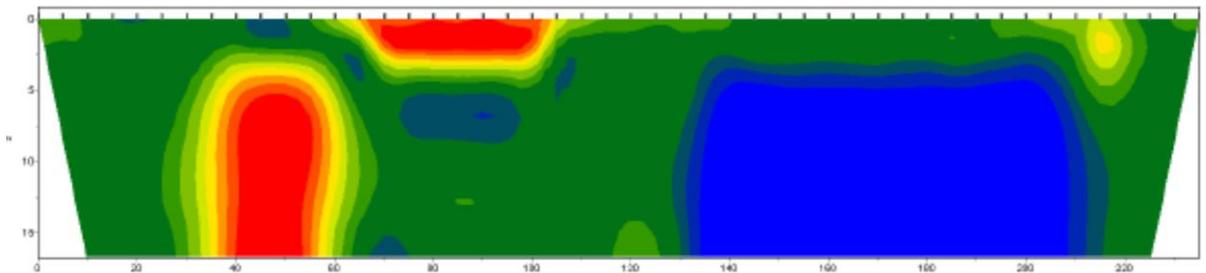
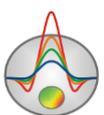


Fig. 20 Regenerated model as a result of *Occam* inversion

Marquardt – classic inversion algorithm by least-square method with regularization by damping parameter. This algorithm allows receiving model with sharp boundaries (fig. 21). 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 after *Smoothness constrained* or *Occam* inversion is performed. Matrix equation for this kind of inversion is the following:

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



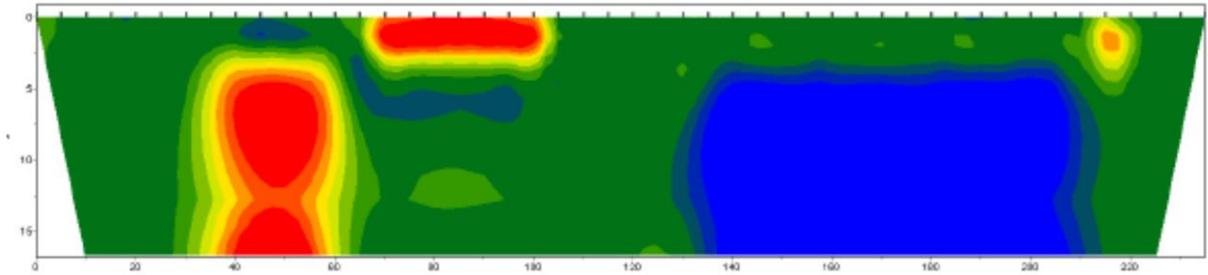


Fig. 21 Regenerated model as a result of *Marquardt* inversion

Blocks – fits parameters for certain domains which differ in resistivity. Domains with equal resistivity consider as single blocks (fig. 22). It is preferable to use this algorithm for specifying results of previous methods when necessary blocks are have already been selected.

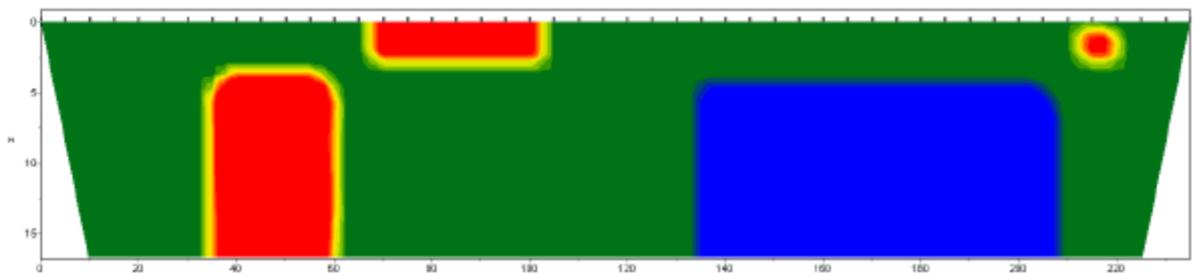


Fig. 22 Regenerated model as a result of *Blocks* inversion

Focused is inversion by least-square method with use of smoothing operator and additional contrast focusing. As a result of this algorithm piecewise smooth parameters distribution (that is model which consists of blocks with constant resistivity) can be received (fig. 23). This algorithm is sensitive to **threshold** parameter. In other words this parameter defines if adjacent cells are unified or not. If contrast of adjacent cells is higher than specified value then cells parameters are averaged weakly, if lower – strongly. Matrix equation for this kind of inversion is the following:

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

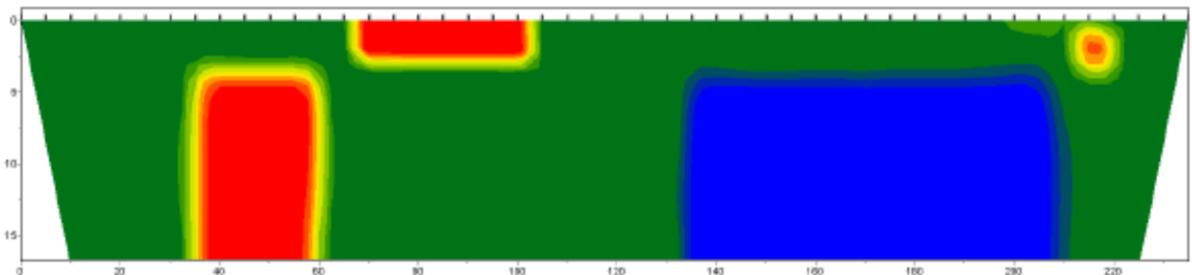
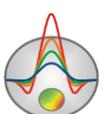


Fig. 23 Regenerated model as a result of *Focused* inversion

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).

Model field common limits (Min resistivity, Max resistivity) – set model parameters limits of variation for inversion. It is very important to constrict parameters range of variation. It makes resultant model more stable, on one hand, and allows receiving more adequate results from geological point of view, on the other.

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

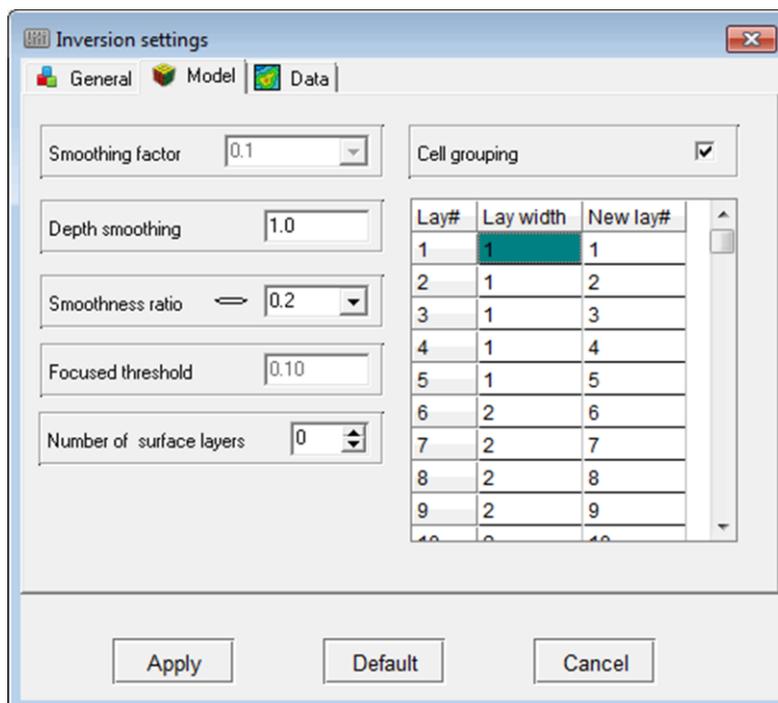
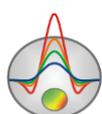


Fig. 24 Inversion parameters setup dialog. Tab Model

Option **Smoothing factor** – **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. But for model smoothness this parameter also allows stabilizing inversion process which is very important if data has considerable component of noise.

Depth smoothing - sets the attenuation factor (smoothing) with depth. The parameter value also depends on the noise level - at a high level and the appearance of the inversion



"ungeological" objects this parameter should be increased in the lower part of this section. The value is chosen empirically.

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. 25).

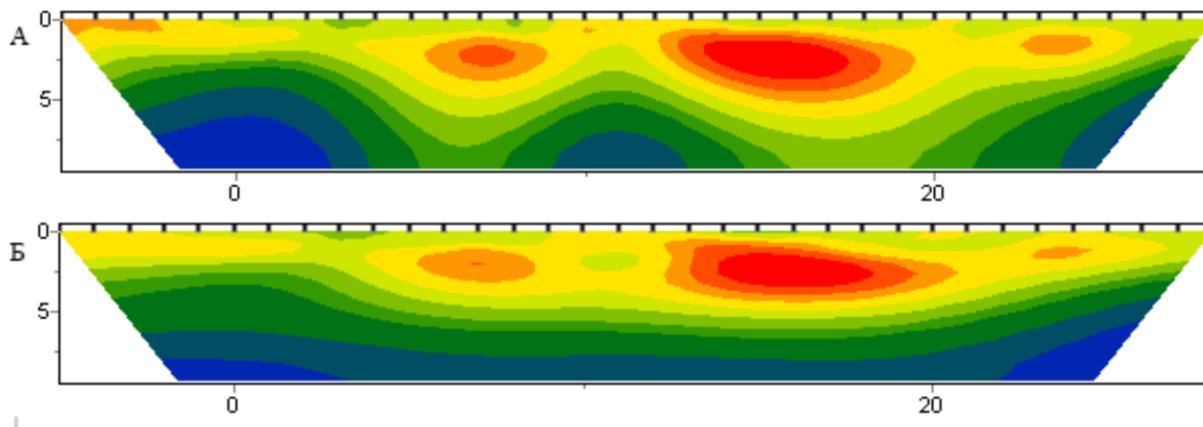


Fig. 25 Geoelectric models as a result of “smooth” inversion with **Smoothness ratio** parameter: 1 (A) and 0.3 (B)

Option **Focused 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). Small parameter value can cause algorithm discrepancy (in this case it needs increasing). Large value leads to receiving smooth distribution (fig. 26).

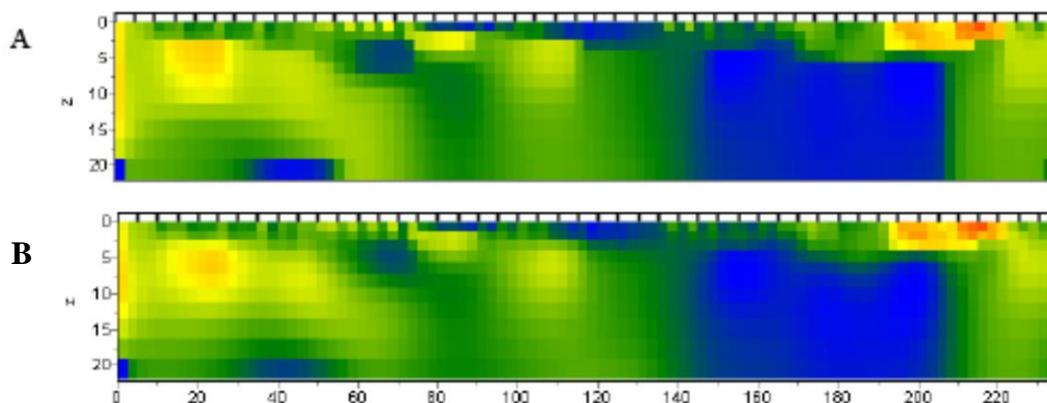
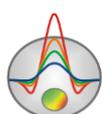


Fig. 26 Geoelectric models as a result of *Focused* inversion with **Threshold** parameter: 0.01 (A) and 0.1 (B)

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 near-surface layer and bottom part of the section is smooth after inversion.



Cell grouping – use this function for big models (when opening files with large number of data values, program automatically offers to use this function). It activates table that allows merging adjacent cells and receiving less number of determinate parameters for inversion. When using this option, the cells number for the direct problem solution remains the same, while the number of cells for the inversion is reduced. Ideally, the amount of adjustment parameters should be close to the amount of data.

The table contains three columns. Number of initial model layer is specified in the first column (**Layer**). Number of inversion mesh layer is contained 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 – to run dialog to set layers number for merging. It is applied for current and all lower layers. Merging cells in the vertical direction is carried out in a similar way when working with the column **New Layer**.

There are three examples of inversion meshes below: in the first picture inversion mesh corresponds to model one (fig. 27A); in fig. 27 B cells starting from the second layer merge by two in horizontal direction; in the third picture cells starting from the sixth layer merge by two, but from the twelfth by four in horizontal direction (fig. 27C).

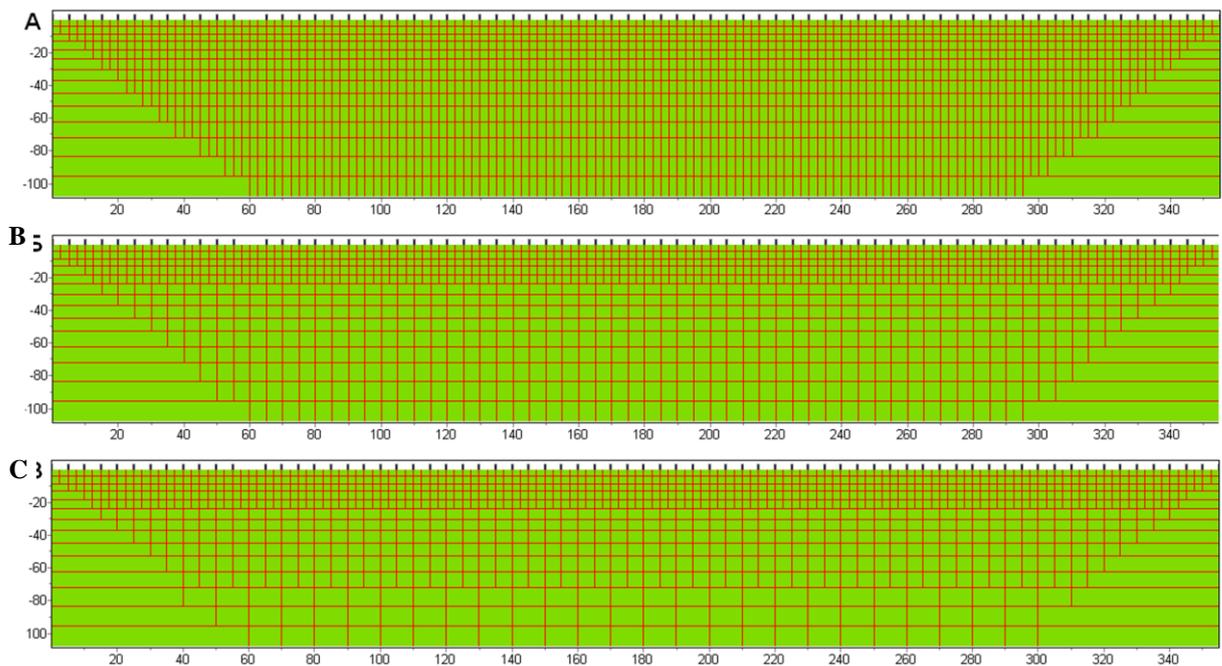
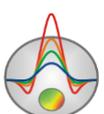


Fig. 27 Examples of inversion meshes

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 (Fig. 27 shows active cutting option for all models). Options located in the tab **Data** (Fig. 28), allow choosing which data should be used in the inversion, set the data types weight and type of data normalization.



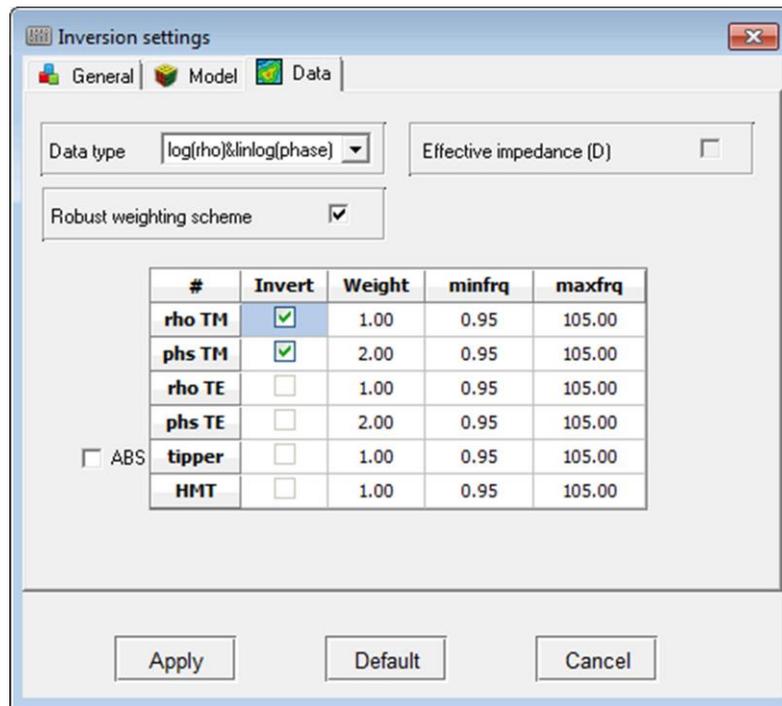


Fig. 28 Dialog of inversion settings. Tab Data

The table on the tab **Data** contains the columns, corresponding to the data type name (**#**), a field that indicates taken into account or not data in the inversion (**Invert**), the data weight (**Weight**), minimum and maximum frequency (**minfq** and **maxfq**). Option **ABS** allows using the parameter module in the inversion. When the option **Effective impedance (D)** is activated, effective impedance inversion will be made.

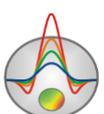
Robust weighting scheme – this option should be turned on if there are individual high deviations caused by systematic measurement errors. It is important to note that this algorithm should be used if there is no more than 30 % of rejected data. If noise constitutes the most part of data this algorithm is hardly helpful.

Data type – sets type of data normalization for inversion. $\log(\rho)\&\ln\log(\text{phase})$ is recommended. It usually leads to the best inversion algorithm convergence.

The program supports two basic modes of inversion: in the block model (if the option is turned on, the examples are shown above), and in [random-layered medium](#) (if the option is turned on, described below). It is possible to do joint inversion of MT data and the data of transient electromagnetics (**Options/TDEM data/Invert TDEM data**), as well as set results of the joint one-dimensional inversion of MT, TEM and VES as a starting model for the inversion (produced in the [SSC](#)). Option  on the main program window toolbar allows making conventional one-dimensional 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. 29).

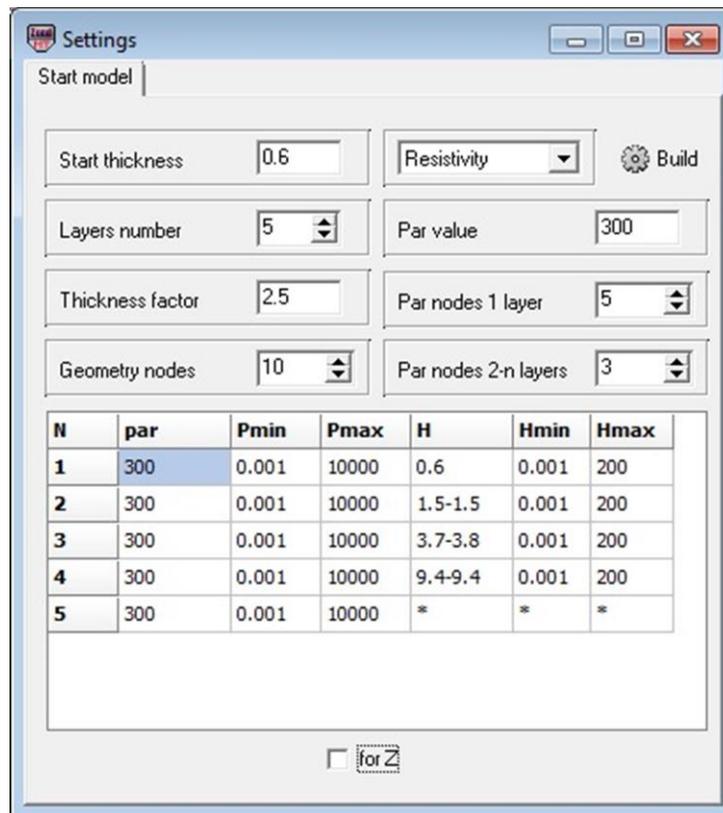


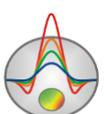
Fig. 29 Starting model setup dialog window in the mode of working in layered model

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.

Example of inversion results in the layered model is shown in Fig. 30 and Fig. 14.



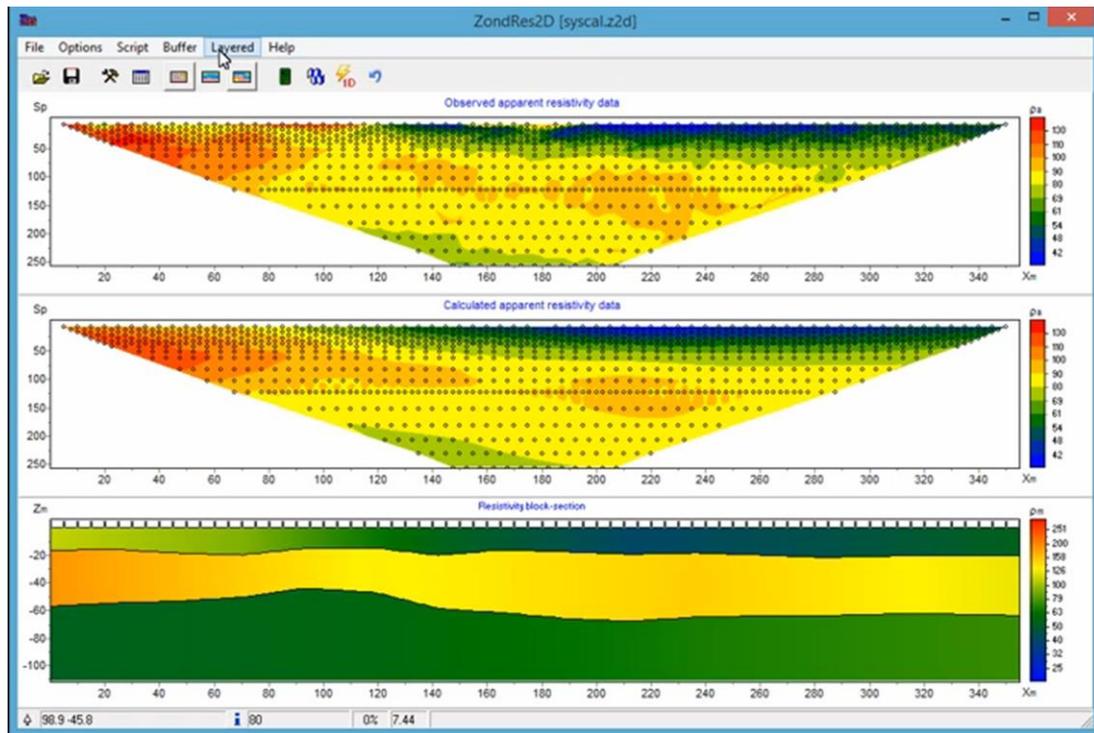


Fig. 30 Inversion results in the layered model

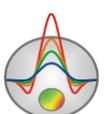
Incorporating a priori information in the inversion

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 boundary boundaries position.

By default, the starting model for the inversion is the current ZondMT2D 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 in the inversion settings window. The higher the Smoothing factor value, the less the result will be different from the prior model.

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.



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 "[Visualization of additional information](#)"

Set boundaries dialog

Set boundaries dialog can be run using **Options/Advanced** function of the main menu. It allows taking into account a priori geological information for inversion. When this tab is selected, menu with the following buttons appears:

	Enable/Disable editing boundaries mode	Turn on/off editing boundaries mode
	Add new boundary	Add new boundary
	Delete boundary	Delete all boundaries
	Save boundaries to file	Save boundaries to file
	Load boundaries from file	Load boundaries from file

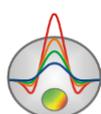
One of the most important ways to improve interpretation quality is implementation of a priori geological boundaries to inverse problem. It increases problem stability, on one hand, and decreases equivalence domain and allows receiving more consistent structure, on the other. In these domains of the model where parameters are not sensitive this is almost the only way to receive acceptable result.

It is preferable to use Occam algorithm if boundaries are implemented. Usually 1 or 2 boundaries are used, and it should be remembered that geological boundaries do not often coincide with petrophysical ones.

It is recommended to perform inversion using **Occam** algorithm before setting boundaries; then plot present borehole data on received geoelectric section. See details of logging and lithology data file creation in [«Borehole columns and logging data visualization»](#).

Boundaries should be plotted on received geoelectric section with account of borehole data and on the basis of a priori concepts of geological structure. Boundaries are set by left mouse click with enabled editing boundaries mode. Right click to close the line. It is recommended not to use many nodes when plotting the boundaries. It is advisable to plot the smoothest boundaries close to inversion mesh modes.

Inversion should be run again when boundaries are plotted to take into account specified boundaries (fig. 31).



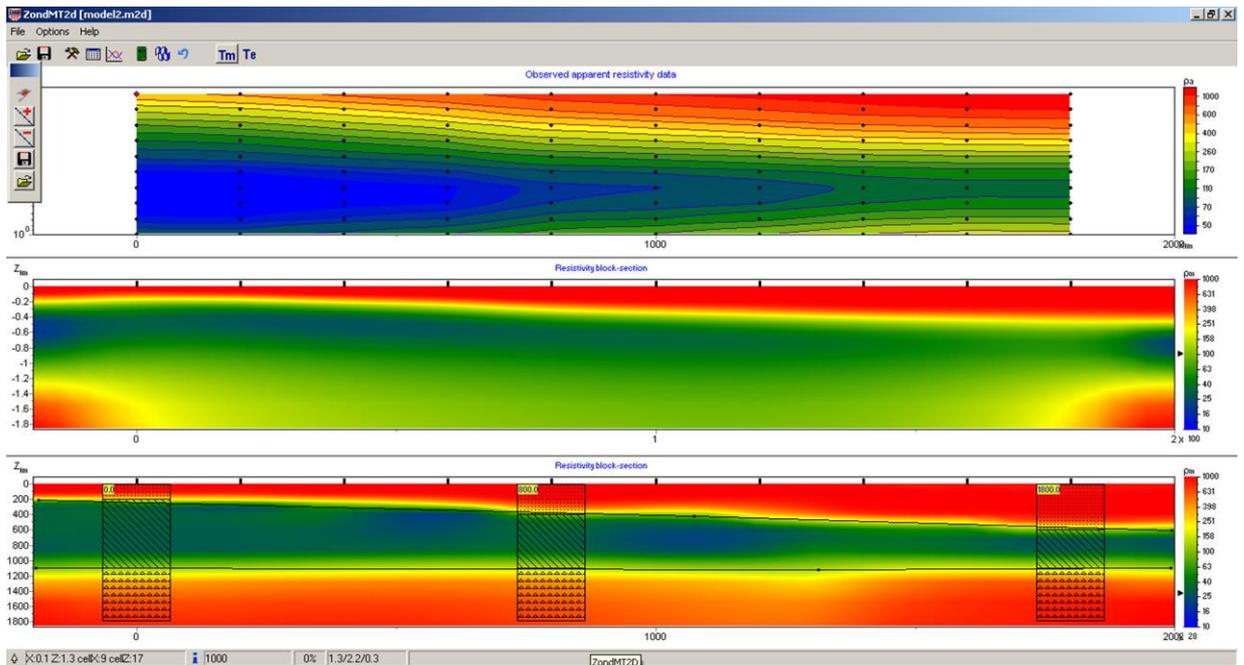


Fig. 31 Example of Set boundaries function usage. The middle section – geoelectric section received using Occam algorithm without plotting boundaries, the bottom – geoelectric section calculated after plotting boundaries

Model visualization modes and parameters

The model can be displayed as cells **Options/Model/Block-section** (Fig. 32 A), in a smooth interpolation palette **Options/Model/Smooth-section** (Fig. 32 B), and as a contour section **Options/Model/Contour-section** (Fig. 32 C).

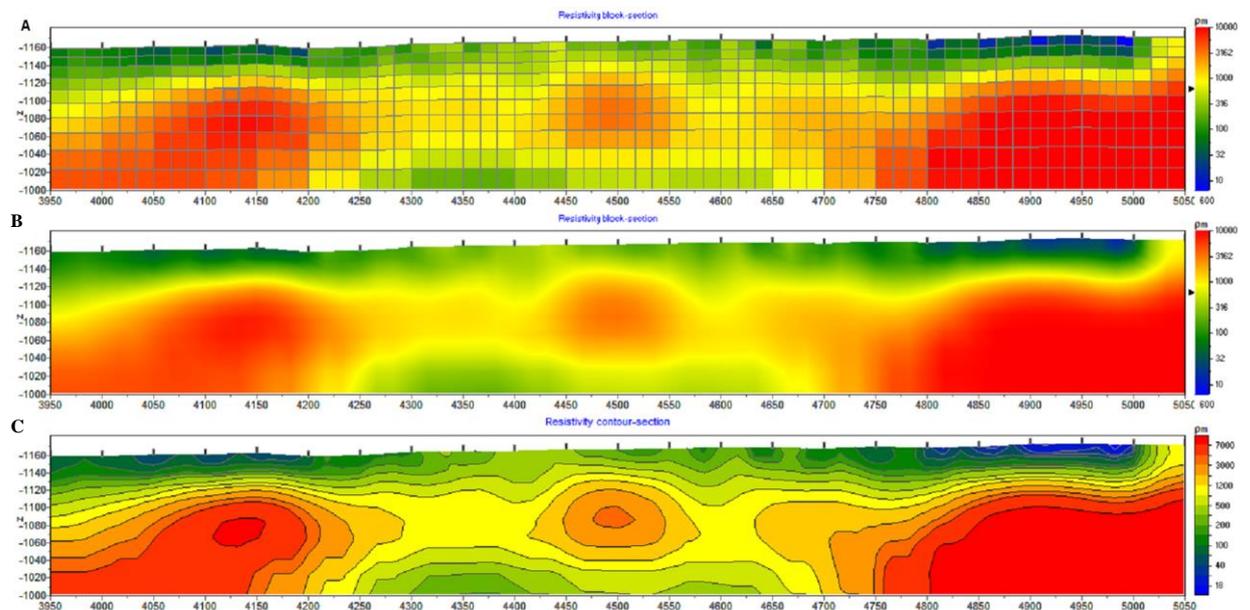
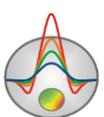


Fig. 32 Versions of displaying model: **Block-section** (A), **Smooth-section** (B), **Contour-section** (C)



When you right-click in different areas of the model editor context menu appears with the following options:

Top area near «Resistivity block-section»	Display model mesh	Enable/disable model mesh display.
	Display objects border	Enable/disable objects border display.
	Display color bar	Enable/disable color bar display.
	Setup	Run model parameters setup dialog.
	Zoom&Scroll	Enable zoom and scroll mode.
	Print preview	Print model.
Color bar	Set minimum	Set minimum value of colour bar.
	Set maximum	Set maximum value of colour bar.
	Set incremental factor	Set minimum and maximum values of color bar relative to host medium value.
	Automatic	Automatically set minimum and maximum value of colour bar.
	Log scale	Set logarithmic scale for colour bar.
	Set halfspace value	Set host medium parameter value.
	Set cursor value	Set current parameter value.
Colors as histogram	Set colors based on the distribution of the cell model resistivity.	
Section area	Working with model context menu	Described in « Working with block model »
Axes area	Model editor	Described in « Model editor »

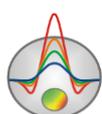
When moving the mouse cursor within working windows, local window coordinates are shown in the left section of the main window status bar.

[Appendix 6: Dialog of model parameters displaying](#) describes model parameters setup dialog **Setup** (color scale, line thickness, etc.) when working in **Block-section** and the **Smooth-section** modes. Work in the mode **Contour-section** is described in [Appendix 4: Pseudosection \(contour section\) parameters setup dialog \(contour section\)](#).

Program additional features depend on the selected display mode of the model. Mathematical modeling is more convenient to carry out in **Block-section** mode. Distribution of integral sensitivity is displayed in the contour section (**Contour-section**) - option **Option/Model/Sensitivity**. The potential distribution and the sensitivity function is displayed in (**Block-section**) or in interpolation palette (**Smooth-section**).

The edges of the model have generally worse resolution. Very often the inversion results in these areas contain many false anomalies. Option **Options/Cutting** allows hiding model edges by setting the value (in degrees) of the trim angle (**Cut by angle**, Fig. 33) or sensitivity level (**Cut by sensitivity**).

Option **Options/Cutting/Extend bottom** enables or disables the stretching of the lower model cell.



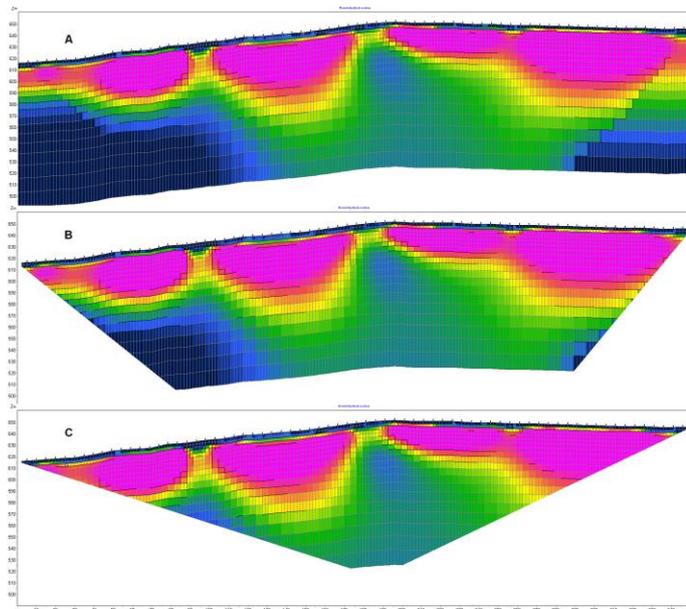


Fig. 33 Models comparison with different cut angles (Options/Cutting/Cutting angle): A - no trimming, B - angle of 30°, C - angle of 55°

Option **Options/Import/Export/Create shaded map** allows displaying model as a shaded map (shaded map, Fig. 34).

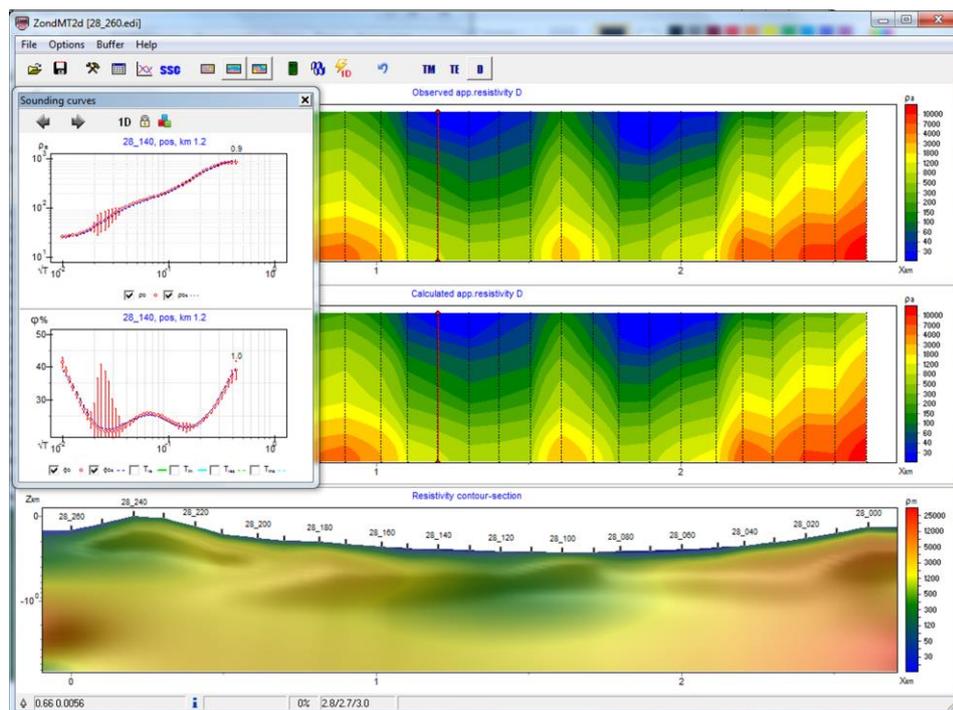
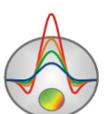


Fig. 34 The main program window with the model image in shaded maps mode (shaded map)



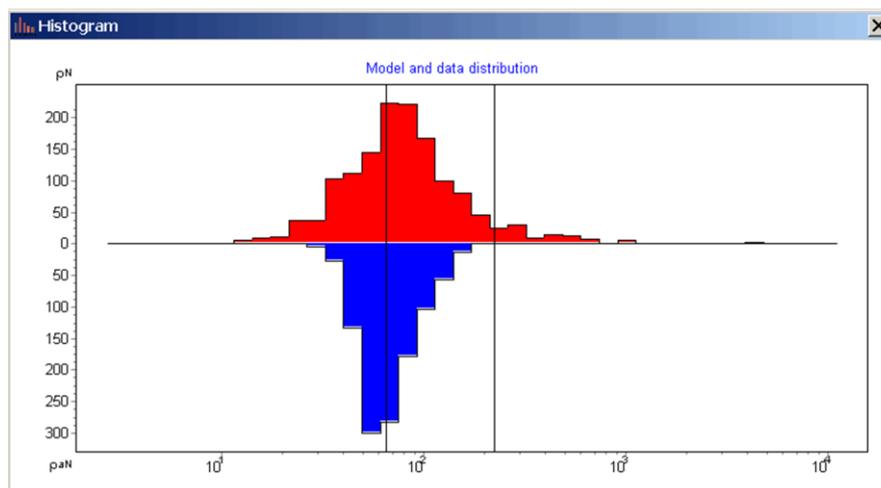


Fig. 35 Window **Histogram** - the histogram of the data values and models

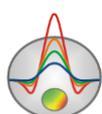
Dialog **Options/Extra/Model&Data histograms** allows to display the plot of apparent resistivity distribution in the data and distribution of the real resistivity in the model (Fig. 35). Changing of the vertical line position allows specifying the minimum and maximum of parameter color scale.

Interpretation results saving

Profile interpretation result is hold in « ZONDMT2D » file format (extension *. m2d). Field data, relative measurement weights, and current subsurface model are saved in this file. Data from the file are used for further load and subsurface model creation.

Use button  in the toolbar 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 values and images (Model, WorkSheet) in *.BMP format (more details in «[Exporting image setup dialog](#)»). There a few options to save data:

Zond MT project data	Save observed data and current subsurface model.
Zond MT calculated data	Save calculated data.
Zond MT observed data	Save observed data.
Zond MT Model with calculated	Save calculated data as observed data and current model.
Worksheet	Save three graphic section of the window in BMP format.
Model	Save bottom graphic section of the window in BMP format.
Program configuration	Save program parameters.
Grid file	Save current model as grid-file in DAT format. In addition for further convenience bln file which contains characteristic relief points is saved.
Section file	Save current model in Section format.



Modeling and working with model

Modeling 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 survey layout 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 ZondMT2D allows modeling numerous practical applications of MT-method 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 modeling. ZondMT2D allows creating observation systems 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 the survey layout in the program comes with the option **File/Create synthetic survey** (Fig. 36).

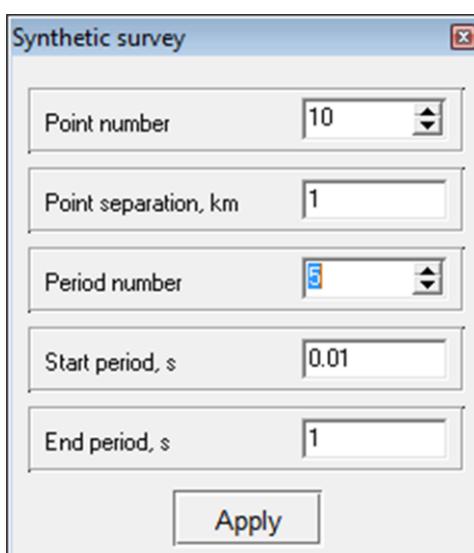
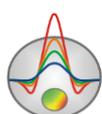


Fig. 36 Dialog of synthetic survey layout creating

Point number determines the number of observation points on the profile, Point separation - the distance between the observation points in kilometers, Period number - the number of frequencies (periods) of the sounding signal, Start period and End period - respectively, the smallest and the largest period of the sounding signal. After setting the parameters and clicking the button , dialog of grid settings **Mesh constructor** appears (more details in [«Starting model setup»](#)).



An alternative to creating measurement configuration directly in ZondMT2D 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 modeling for the measurement configuration, which is used in field research, as well as set the survey layout with all sorts of settings.

For the further parameters changing of the observing system before opening the measurements file it is necessary to go to the modeling mode: **Options/Extra/Open in modeling mode**.

Model building in ZondMT2D

Geoelectric medium creation produced in the model editor - lower section of the graphical window. The program ZondMT2D 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 ZondMT2D programs. These tools are described in the [«Data import and export»](#).

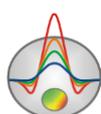
Model editor

Right-click on the vertical and horizontal axes in the model editor field options to edit the grid created when working with dialog **Mesh constructor** appears.

Vertical axe	Log scale	Set logarithmic scale to the vertical axe.
	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 axe
	Thin mesh	Add intermediate nodes in vertical grid
Horizontal axe	Redivide	Set equal width of cells situated between unique positions of sounding sites
	Thick mesh	Delete every second node of horizontal grid (if there is no sounding site in this node)
	Thin mesh	Add intermediate nodes in horizontal grid

Working with 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



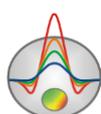
coordinates and parameters of the active cell in which the cursor is located. The currently active cell is surrounded by a rectangle - the cursor. The selected or fixed cell is hatched with white or black points.

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 creat 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.

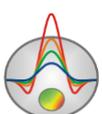
To increase the cell parameter left-click the while holding SHIFT on the cell or cells group. Right-click on the cell while holding SHIFT to decrease the cell or cells group parameter. Percentage of parameter value changing sets in the dialog of model parameters setup. If the active cell is selected, 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. 37).

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.



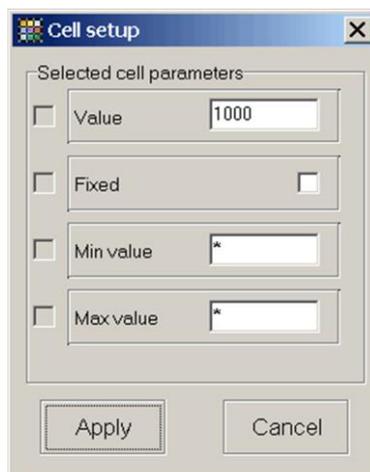


Fig. 37 Dialog window of cells parameter setup **Cell setup**

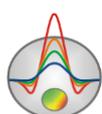
To calculate the response of apparent parameters from a given geoelectric model (to solve the forward problem), press the button  on the toolbar.

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 **Bufar**, it allows working with several models at the same time («*Working with several models in a single project*» part).

Cell summarization dialog

Cell summarization (Options/Advanced) option serves to smooth (divide it into blocks) or roughen current model. Block model can be used for inversion of **Blocks** type. Selection of parameters for each block is performed then. Roughening process represents a peculiar kind of rasterization of smooth model, in other words dividing blocks into domains with constant parameters. Result of focusing inversion is the most convenient for rasterization.

In **Blocks** mode (fig. 38) 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.



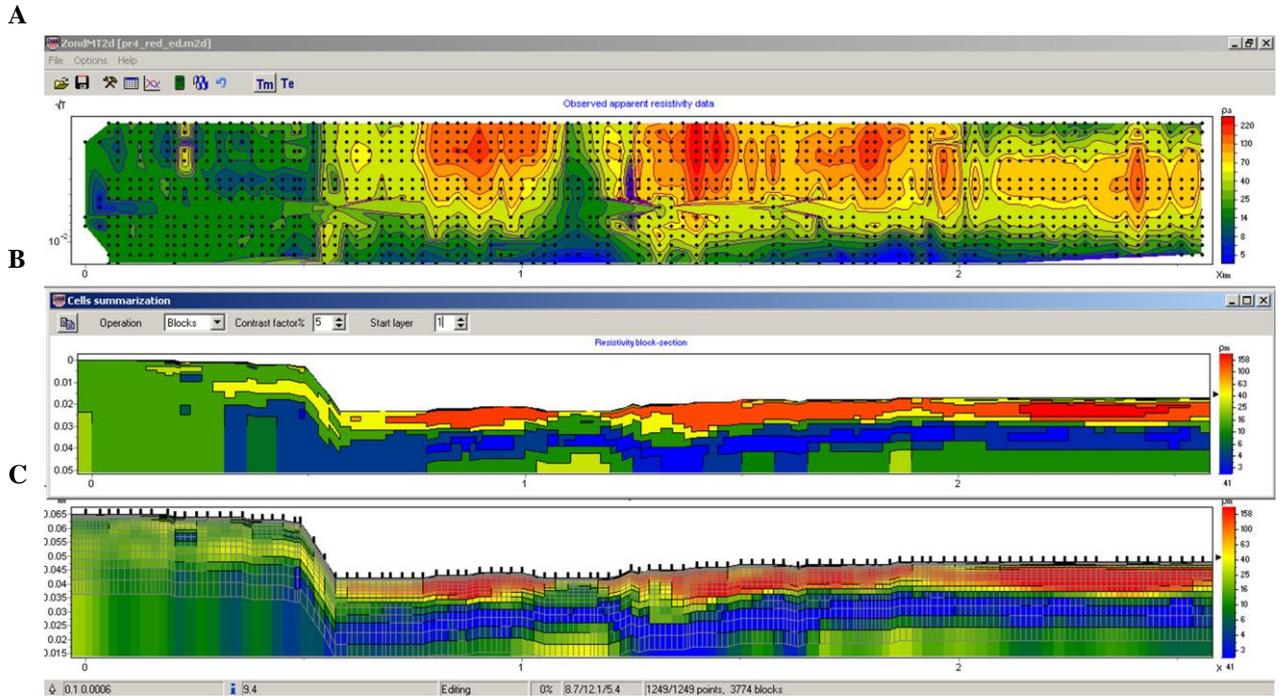


Fig. 38 Field data (A) and example of dividing geoelectric model (C) into blocks (B)

In **Smooth** mode (fig. 39) cell parameters are averaged (in other words the model is smoothed) depending on **Smooth factor**. **End layers** option sets layer number in which this operation stops.

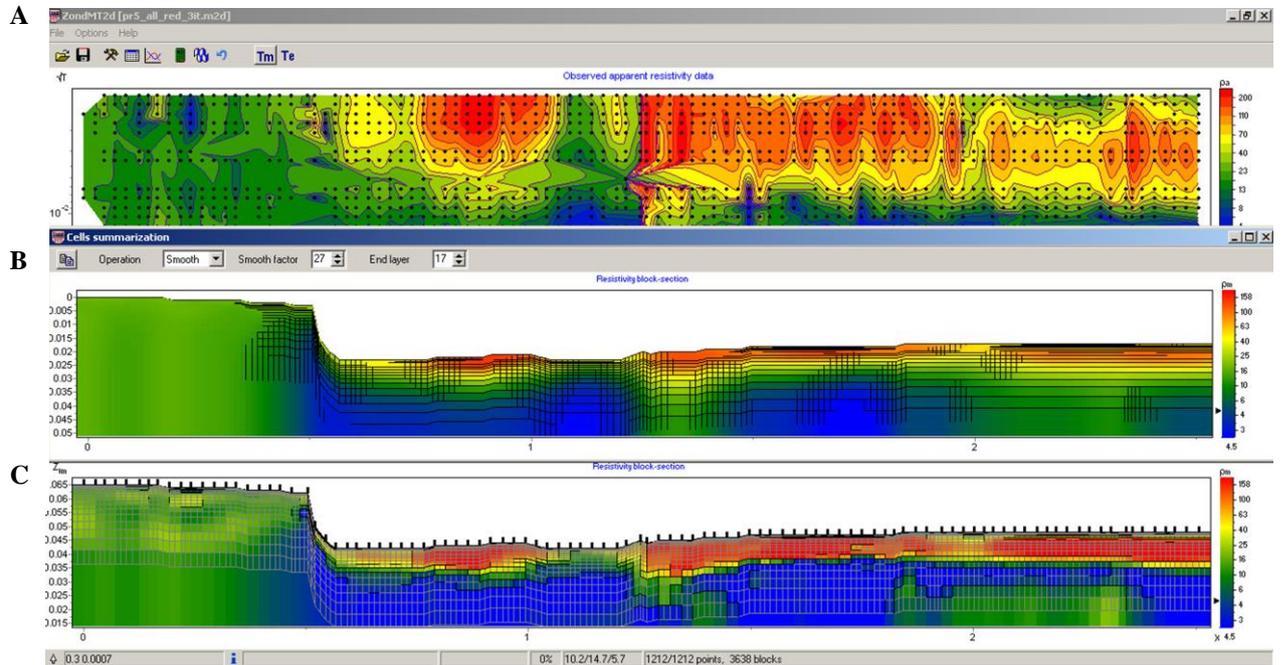
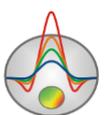


Fig. 39 Field data (A), inverted model (C) and an example of smoothing model in Smooth mode (B)



Press button  in the left upper corner of **Cell summarization** dialog window to copy smoothed or roughened model to geoelectric section.

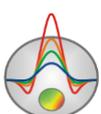
Polygonal modeling

Instead of setting model using blocks, it is more practical to define geological bodies, which have their resistivity values. Polygonal modeling mode provides this functionality within ZondMT2D. 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 ZondMT2D is similar to the process of creating closed polygons in vector graphic editors.

Polygonal modeling 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 modeling 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.
	Create polygon adjacent to the existing one or to modeling 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 modeling area. To finish boundary setting press the right mouse button. Adjoining boundary program will select automatically or ask to select by dialog.
	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.
	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.
	Move polygon. Select polygon by pressing the left mouse button. When moving the mouse polygon moves. Polygon position is fixed by right-clicking.
	Move a part of polygon
	Add node. Right-click on the boundary point where it is necessary to add a node.



	Remove node. Right-click on the node, which it is necessary to delete a node.
	Move node. Left-click to select a node, to move - move the mouse, to end moving – right-click.
	It allows zooming in models.

When using polygonal modeling 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. 40).

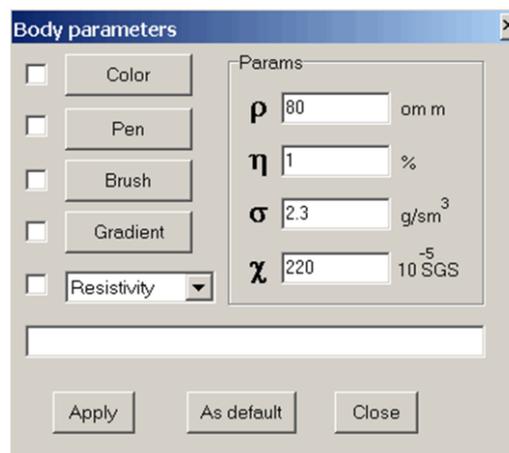
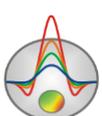


Fig. 40 Dialog window **Body parameters**

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 **Modeling/Get values from background and Modeling/Set values to background**. The first option sets block model resistivities to polygons (from the results of preliminary inversion or modeling), 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 modeling tools is presented in Fig. 41.



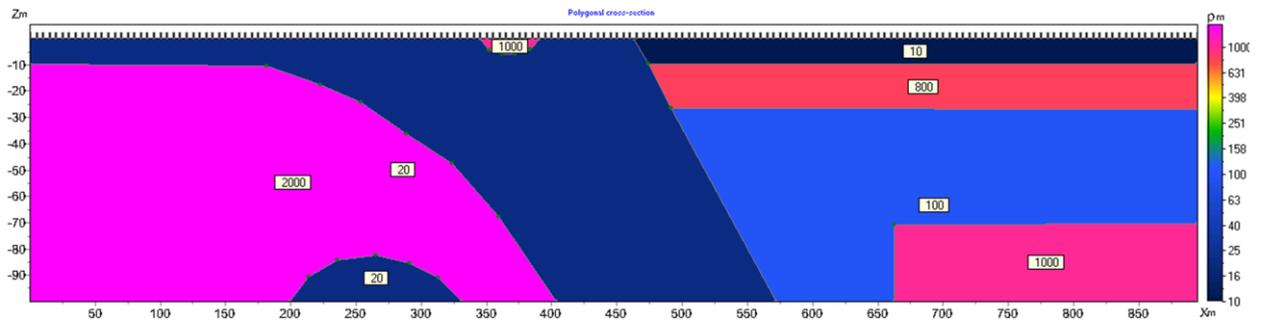
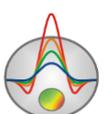


Fig. 41 An example of a polygonal model created in ZondMT2D

Polygonal models can be saved and loaded by the option **Save polygons** and **Load polygons** of menu **Modeling** 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 modeling mode, and click **Modeling/Set values to background**. After that, the calculation of the forward problem will be done already using the finer grid.



Working with multiple model 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 modeling 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 imputing the buffer model name, which will be displayed on the corresponding button in the **Buffer** list and as the title – when displaying model.

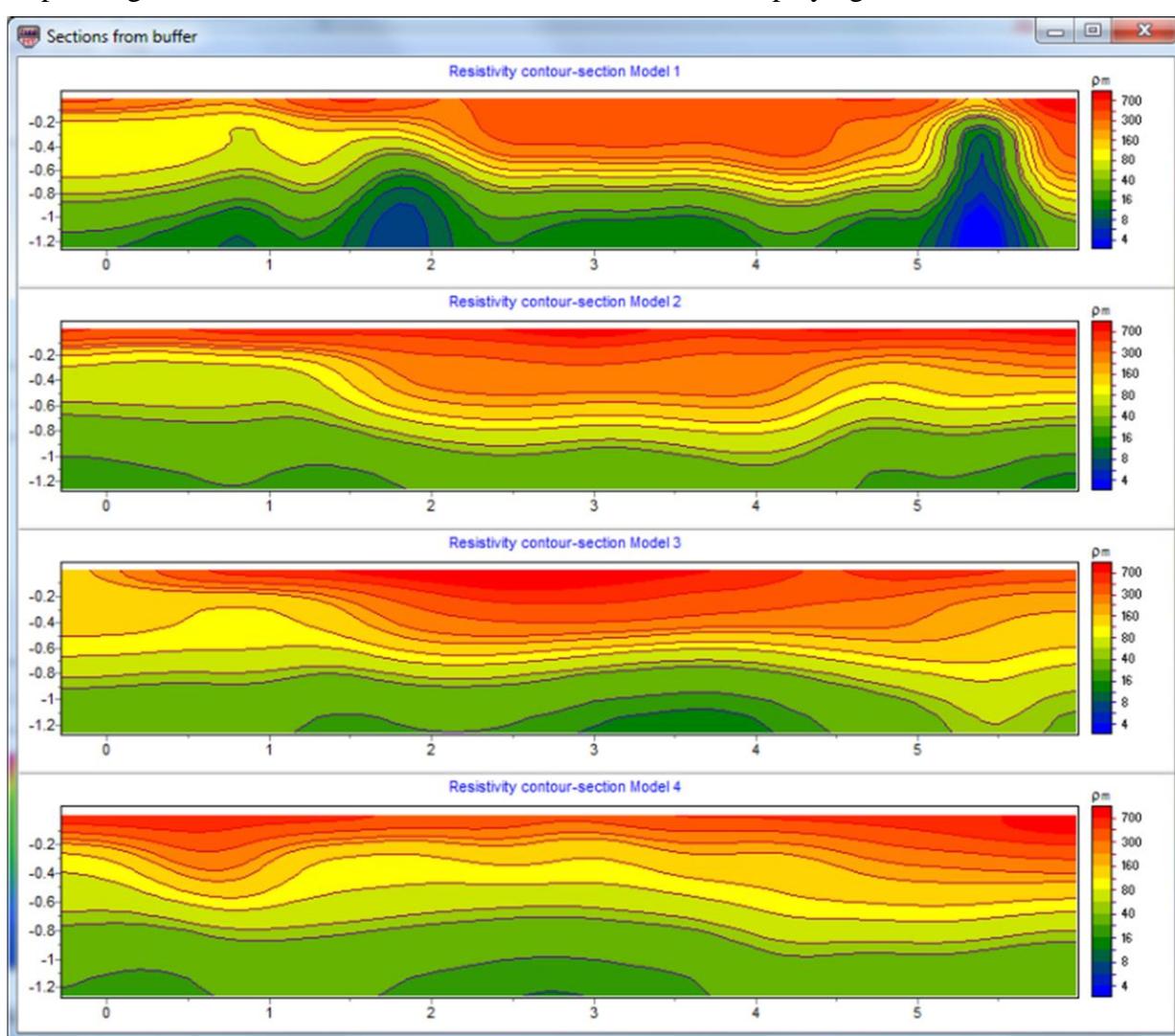
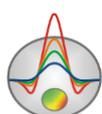
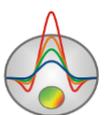


Fig. 42 Window with several models from Buffer/Open. Example of inversion of the same data using different parameters



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. 42).



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. ZondMT2D program has advanced functions to solve this problem. They are the possibility of TEM and VES data inversion, 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**, or, in the absence of source file to create the corresponding file in the program. Working with the lithological columns and logging data is described in [«Borehole columns and logging data visualization»](#).

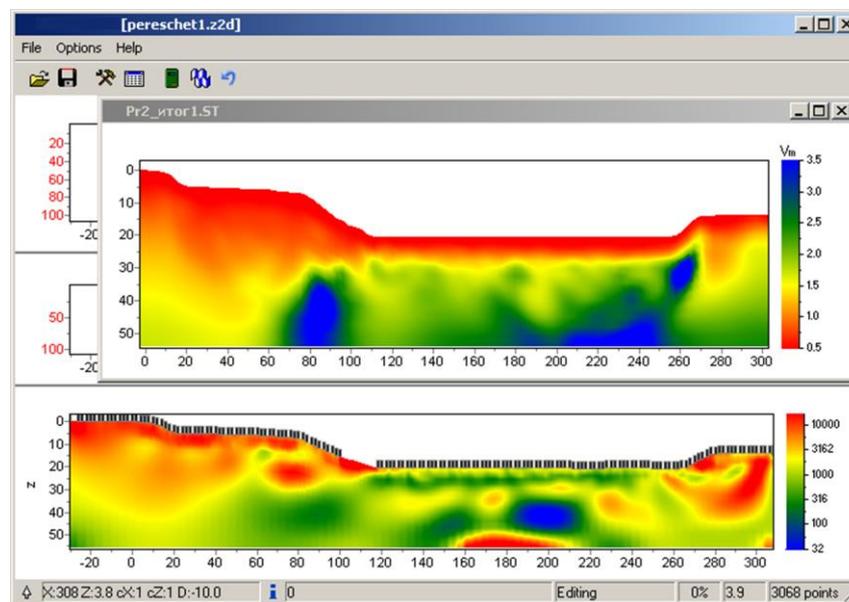
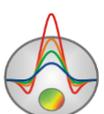


Fig. 43 Working window with importing high-speed model

Option **Options/Import/Export/Import model/data** allows loading environmental models from other ZOND programs projects (Fig. 46). The option can be useful in comparing the interpretation results on adjacent profiles or complex interpretation of the various methods. 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. 44).



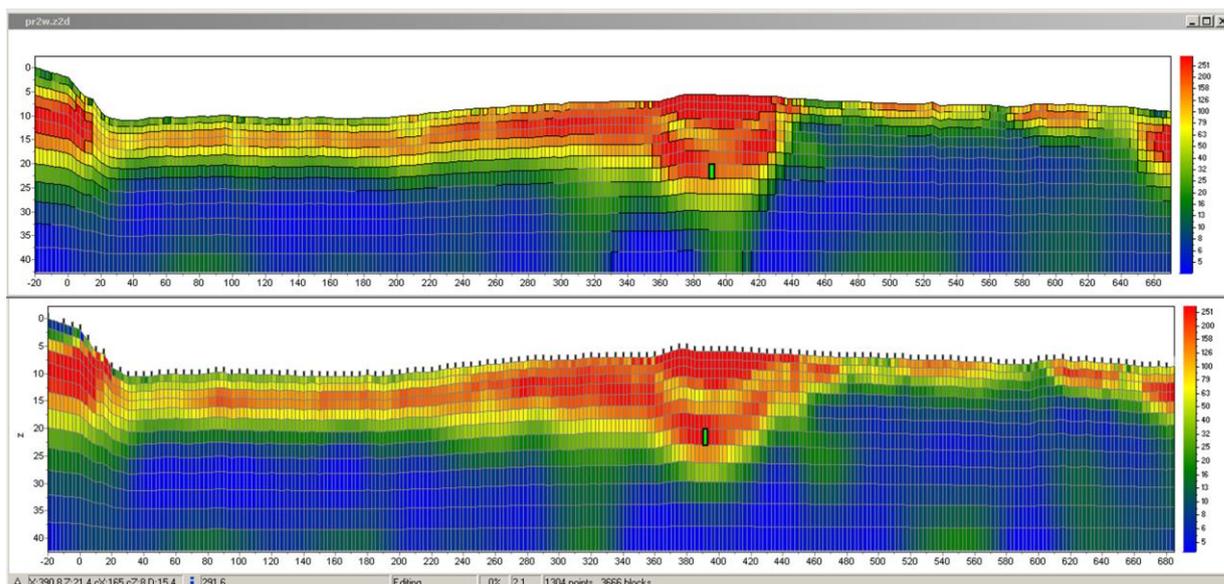


Fig. 44 Models of electrical resistivity, working, and importing as blocks with illuminated cursor

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. 45). 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.

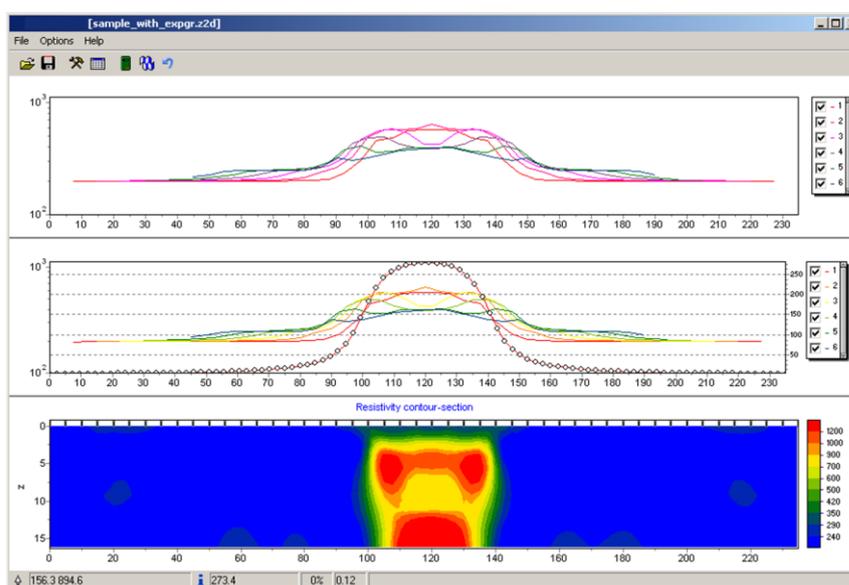
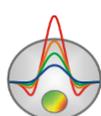


Fig. 45 Working window with importing graphics of anomalous magnetic field

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. 46), in which the image boundaries coordinates in the section coordinate system specifies.

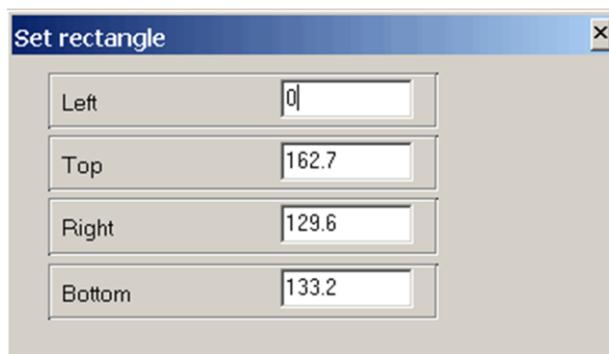


Fig. 46 Dialog of background parameters settings

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:

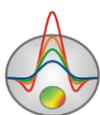
1st row - image file name;

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. 47).



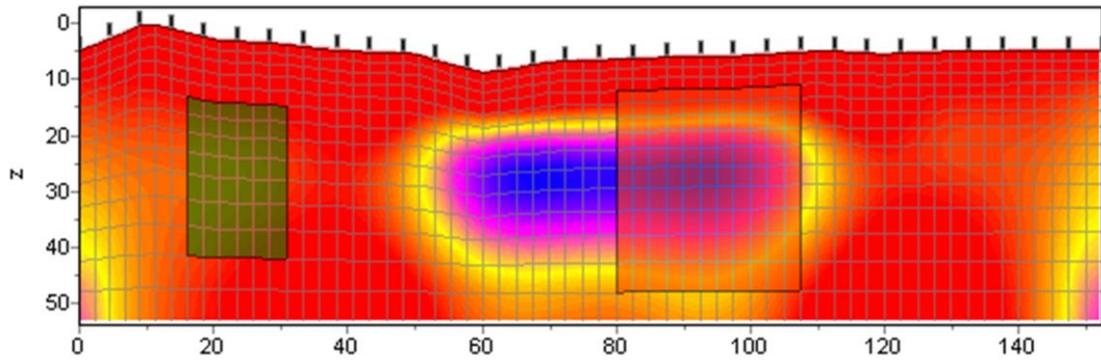


Fig. 47 Model in the mode Block-section with background

In the **Smooth section** mode it is possible to see the features of the two sections at the same time, because colors of background and current model will be mixed (Fig. 48).

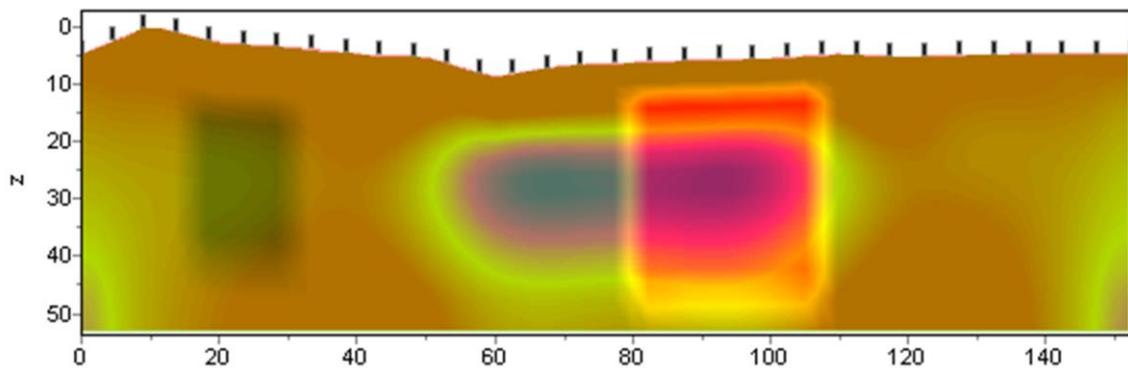


Fig. 48 Model in the mode Smooth-section with background

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 49 shows an example of working with background when using geoelectric model in the mode Smooth-section.

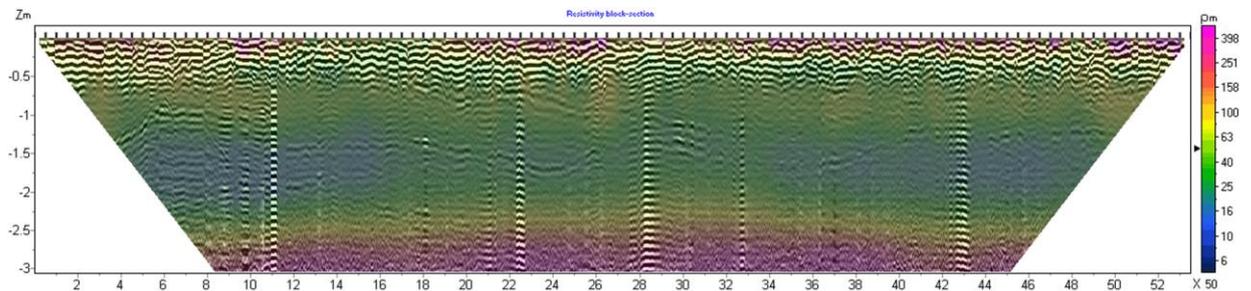
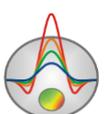


Fig. 49 Model in the mode Smooth-section with background



Text (tabular) data files – import and export

ZondST2D 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 .

Geoelectric model export to Excel tabular file

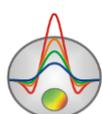
Export geoelectric model to Excel tabular file is carried out by using option **Options/Import/Export to Excel/Model**. In the creating Excel file geoelectric model is written as a matrix whose rows correspond to the horizontal coordinates of cells and columns - vertical (Fig. 54).

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

Exporting image setup dialog

One way to export the image is using the option **File/Save file/Model * .bmp**.

Dialog **Picture settings** (Fig. 50) allows adjusting the vertical scale **Vertical scale** (in meters per centimeter), horizontal scale **Horizontal scale** (in meters per centimeter), the resolution of the exporting image Print resolution (in DPI) and font size **Font size**.



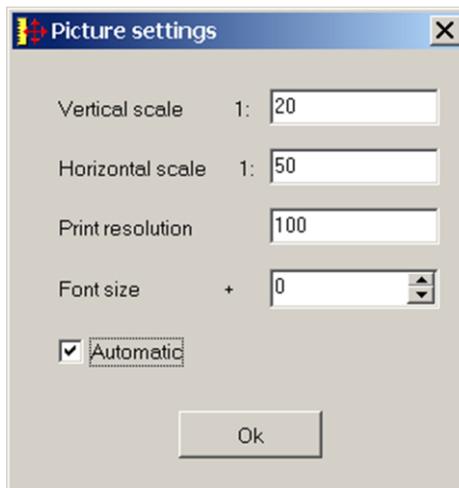


Fig. 50 Picture settings dialog window

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.

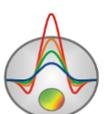
Borehole columns and logging data visualization

Borehole columns and logging data visualization

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. 51). 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 activate the mode of working with lithological columns press the button , to load logging data press the button .

To create (delete) boreholes press the buttons  and . Text field, which indicates the borehole position in the profile coordinates is located to the right of the buttons group on the toolbar. To jump between boreholes to edit them use the buttons  . The current borehole coordinate on the profile, the name of the borehole and its angle are specified on the right from keys.



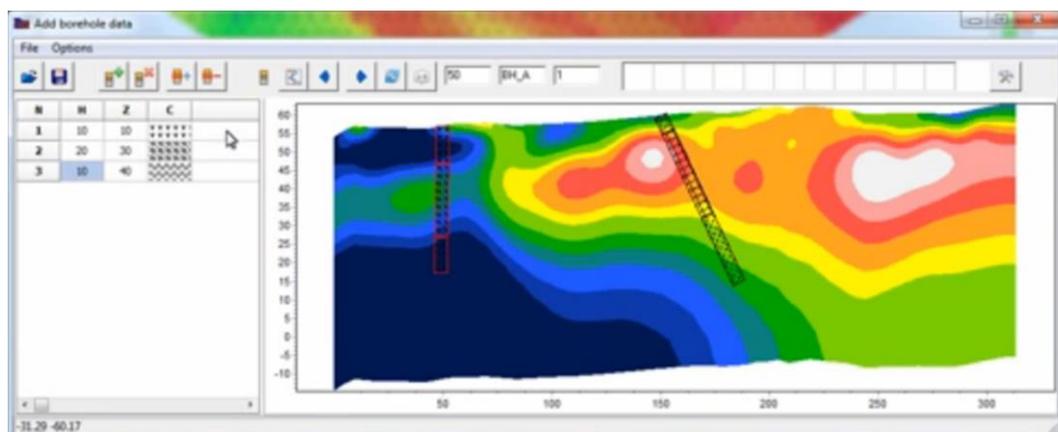


Fig. 51 Working window of working with lithological columns and logging data module

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.

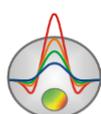
When creating or editing borehole data using described module the result is saved by program to a file of internal format with the *.bth extension.

Borehole columns description and logging data file format

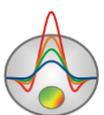
Logging data and lithological columns are stored in the files in a specific format. The first files type with the txt extension - is the actual logging and lithology data file. When creating a logging data file, use the following structure: The first column contains the recording point depth (from the surface), the second column contains the logging measurement. The third and fourth columns contain zeros.

Below is an example of logging data file (Fig. 52):

```
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



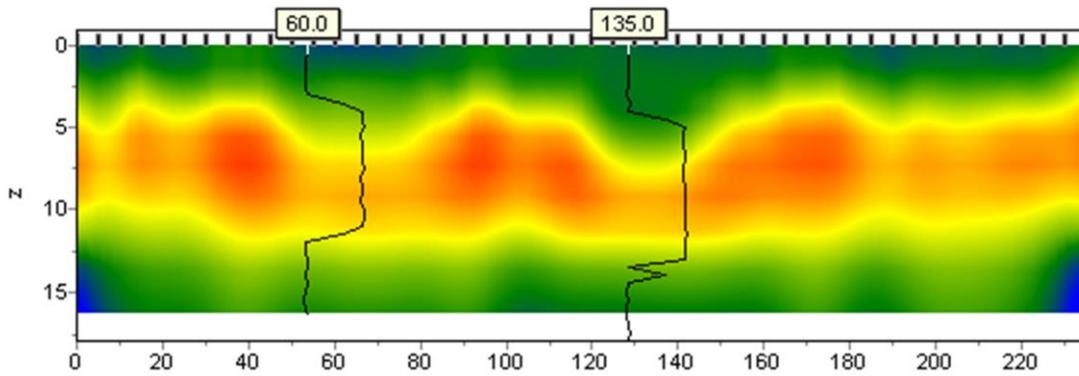


Fig. 52 Model with logging data

When creating a file with lithologic information, use the following file structure: the first column contains the depth (from the surface) of lithological horizon, the second column should be filled by zeros, the third column - the layer color on the lithological column, the fourth column – hatch type on lithological column. An example of a model with lithological column is shown in Fig. 53.

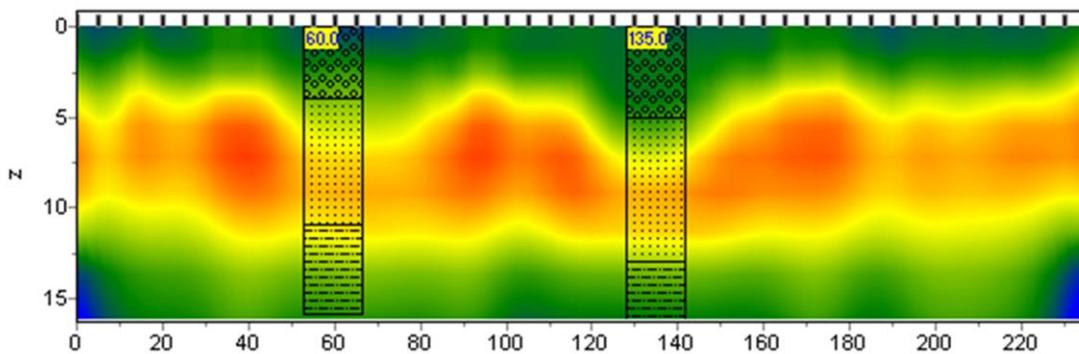


Fig. 53 Model with lithological column

Below is a list of the first 23 hatch, which can be used to create a lithological column (Fig. 54).

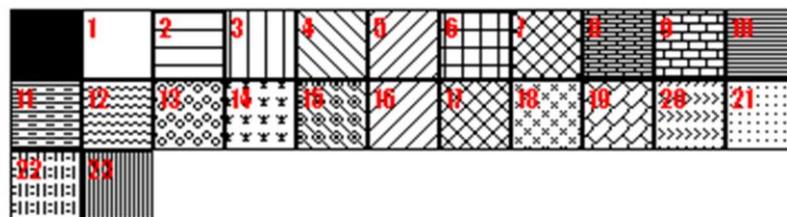
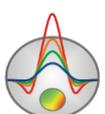


Fig. 54 Versions of lithological column hatching

Below is an example of a lithologic data file.

0 1 0 13 Top of layer 1

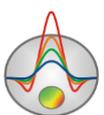


4 1 0 13 Bottom of layer 1
 4 1 0 19 Top of layer 2
 11 1 0 19 Bottom of layer 2
 11 1 0 27 Top of layer 3
 16 1 0 27 Bottom of layer 3

The second file type (extension *.crt) – is control file indicating the data type and visualization regime. The following describes the CRT file structure for displaying lithologic or logging data for any number of boreholes.

2280.txt First line – file name with logging or lithology data
brh12280 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); [File-example - carot1.crt](#)
1 - logging data (interpolated color column), section color scale is used for visualization;
[File-example - carot2.crt](#)
2 - lithologic column; [File-example - 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 ground surface.

3246.txt Description of the next borehole on profile



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. 55). To load a priori information in *.sec format use the menu item **File/Open section**, to load borehole data – use **Options/Load borehole data**.

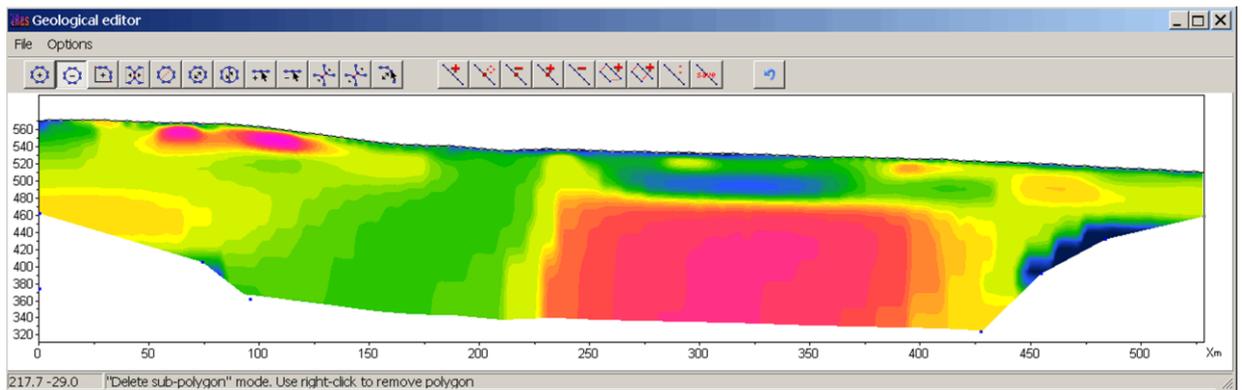


Fig. 55 Model editor window before starting work

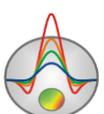
To carry out geological interpretation on the toolbar, there are two button sets: one for creating polygonal bodies (with the possibility of editing hatches, fills, boundary style, etc.), and the other - to draw individual lines on the section. They can be used to indicate certain elements of the geological 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 modeling 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. 57).

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**).



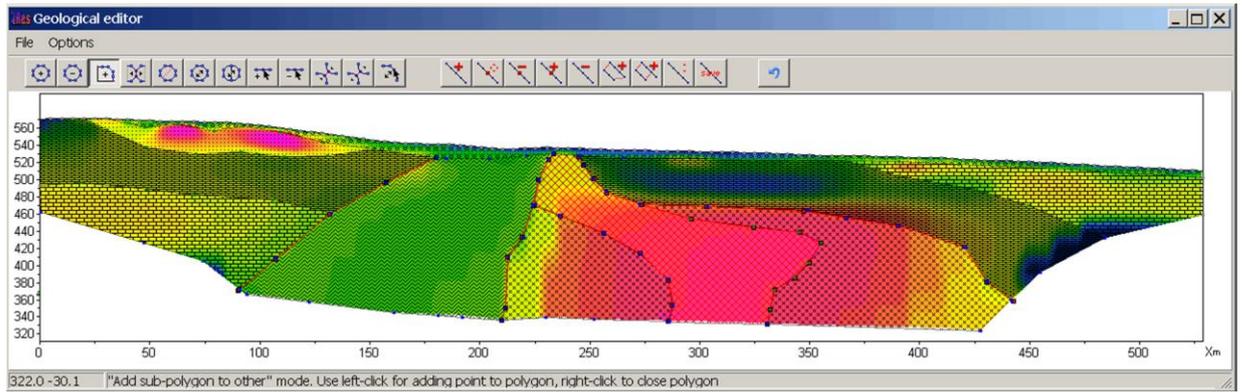


Fig. 56 Model editor window: the resistivity section and geological interpretation

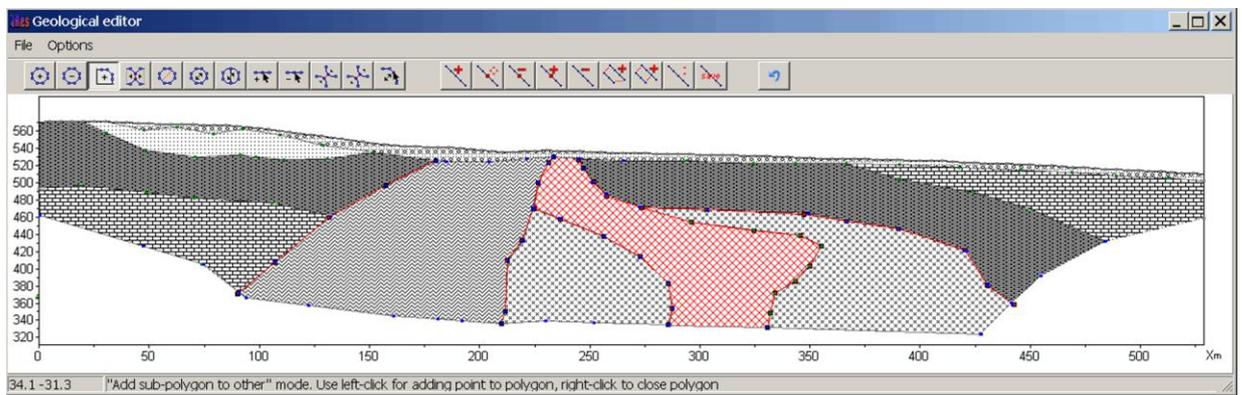


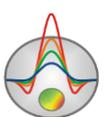
Fig. 57 Model editor window: geological-geophysical section based on the interpretation results

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).



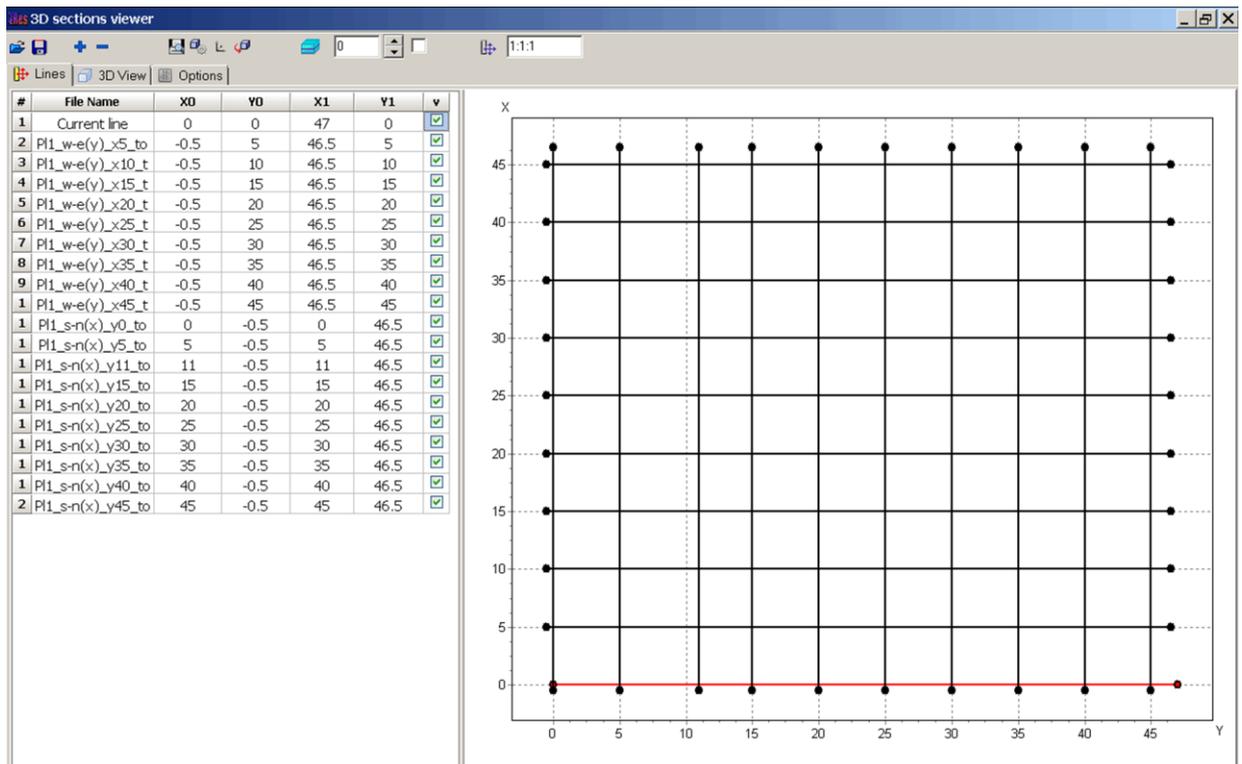
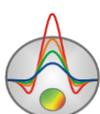


Fig. 58 Tab Lines of the 3D section viewer

To add profiles and set coordinates use the tab Lines (Fig. 58). 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 ZondMT2D. 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 to disable 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. 59).



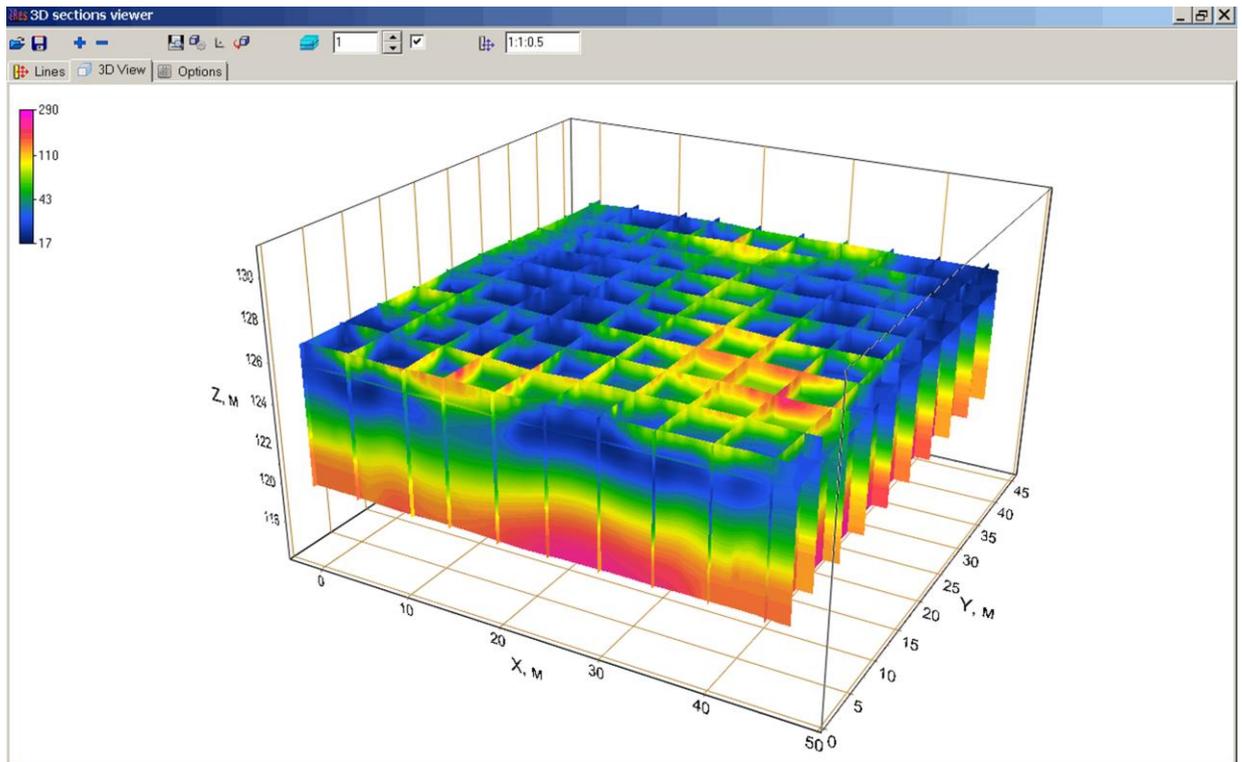
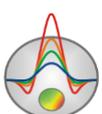


Fig. 59 Tab 3D View of 3D section viewer window. Volume visualization

Axis setting (individually) is carried out by dialogs run 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. 59 and Fig. 60.



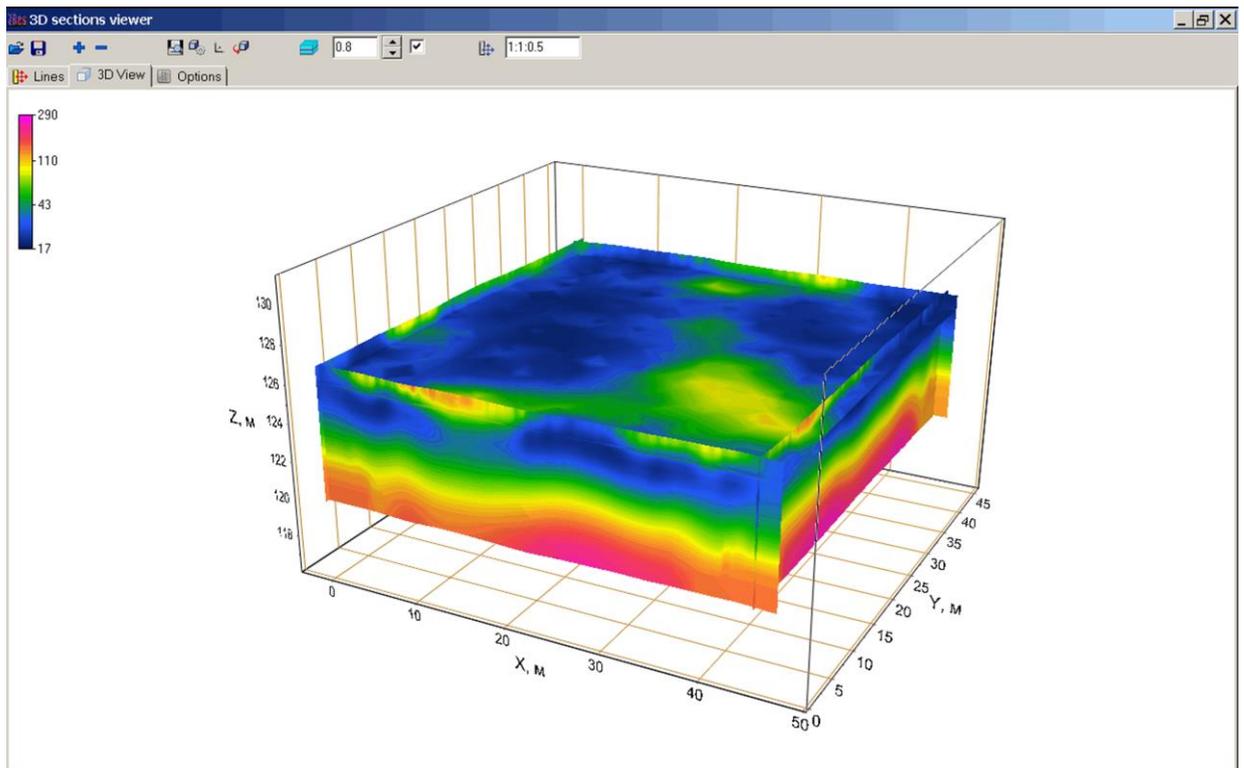
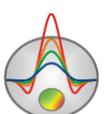


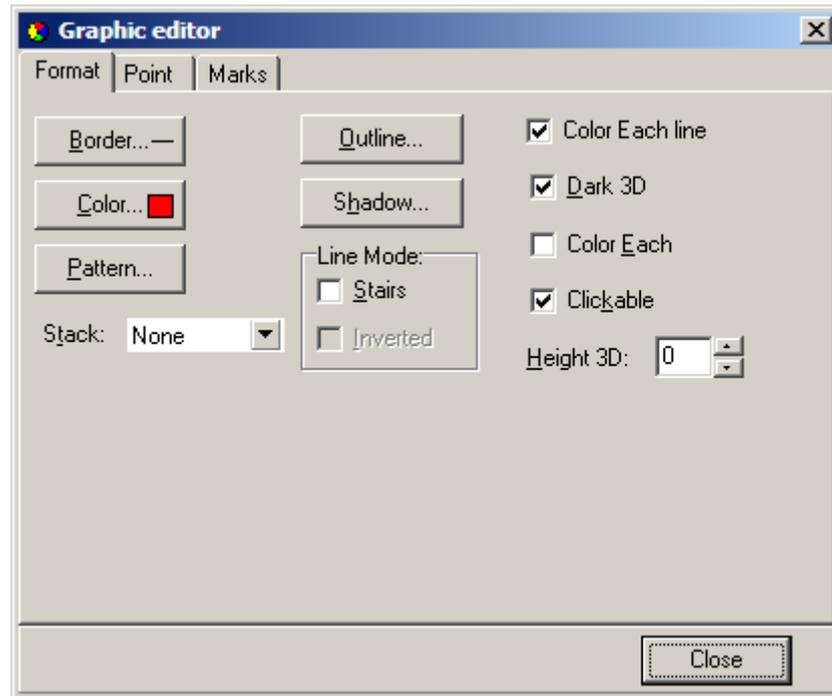
Fig. 60 Tab View 3D of window 3D section viewer. Resistivity map at a given depth from the surface

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



Appendix 1: Graphics editor



Graphics editor serves for graphic interface adjustment. Right click with SHIFT button pressed on necessary 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 pattern 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 colour fill setup dialog.

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

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

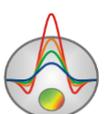
Tab **Marks** contains settings of graphic's point marking.

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).

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.

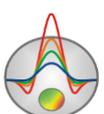
Button **Gradient** runs gradient fill for marking text setup dialog.

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

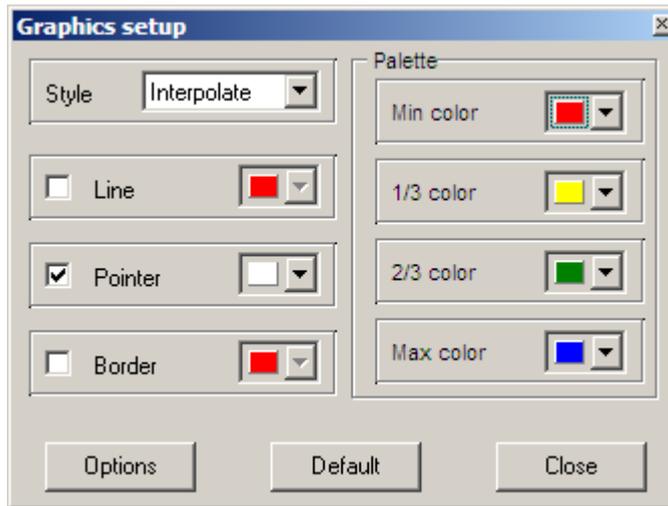
Button **Shadow** runs marking text shadow setup dialog.

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

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



Appendix 2: Graphics set editor



Graphics set editor serves for color adjustment of graphics set.

Option **Style** defines algorithm of graphic's colour 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** и **max color**. Value **const** sets the same color (option **color**) for all graphics. Value **random** assigns random colors for all graphics.

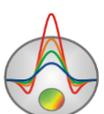
Option **Line** sets color for graphic's connecting lines. If this function is OFF palette color is used otherwise specified in **Line** field color is used.

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

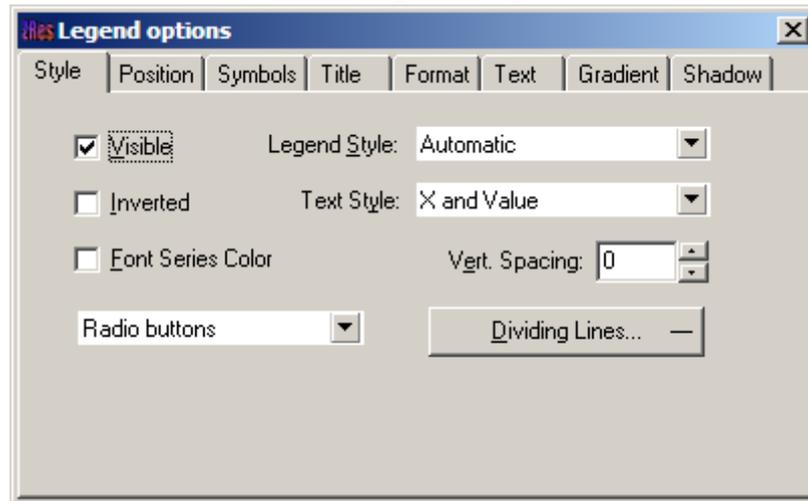
Option **Border** sets graphic point's outline color. If this function is OFF palette color is used otherwise specified in **Border** field color is used.

Button **Options** runs graphics setup dialog (see «**Graphics editor**» *for details*).

Button **Default** returns graphics default settings.



Appendix 3: Graphic's 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.

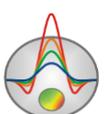
Tab **Position** serves for choosing legend position relative to graphics plan.

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

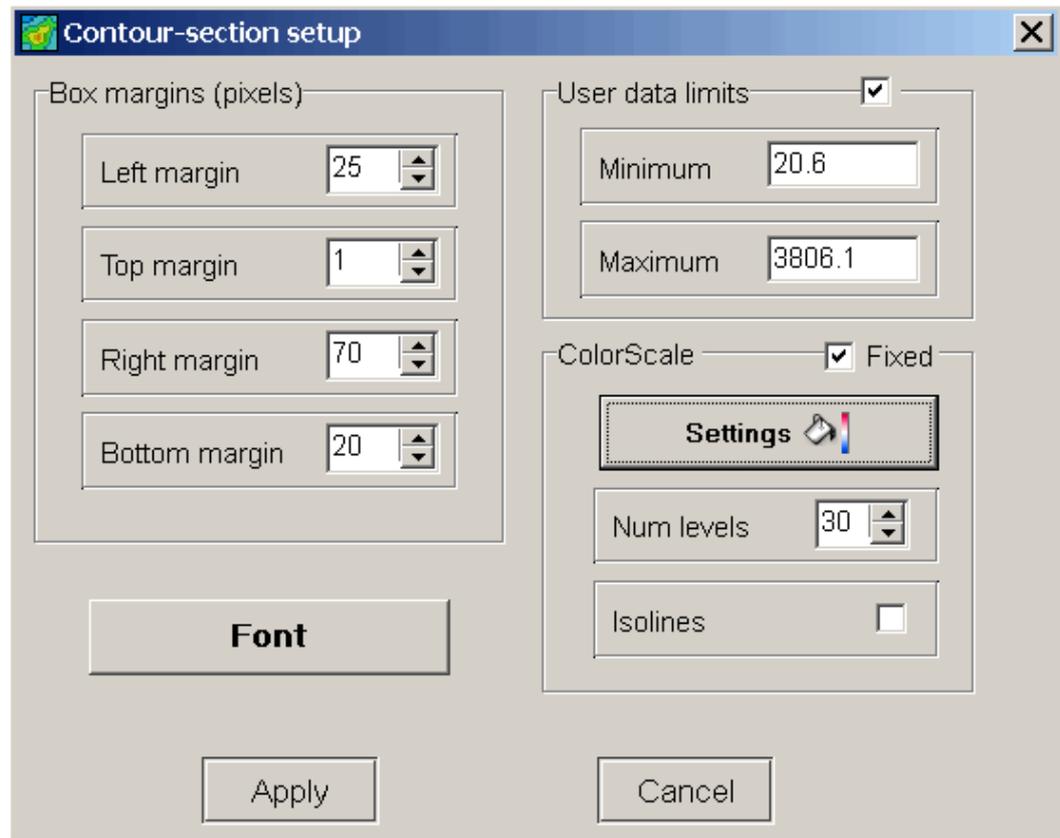
Tab **Title** specifies legend name and allows adjusting its format.

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 pseudosection parameters adjustment.

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.

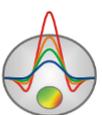
Field User data limits

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 Color scale

Settings – sets color palette



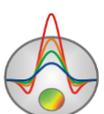
#	C_color	L_color	Level	L_visible
1			30	<input checked="" type="checkbox"/>
2			40	<input checked="" type="checkbox"/>
3			50	<input checked="" type="checkbox"/>
4			80	<input checked="" type="checkbox"/>
5			110	<input checked="" type="checkbox"/>
6			150	<input checked="" type="checkbox"/>
7			200	<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.

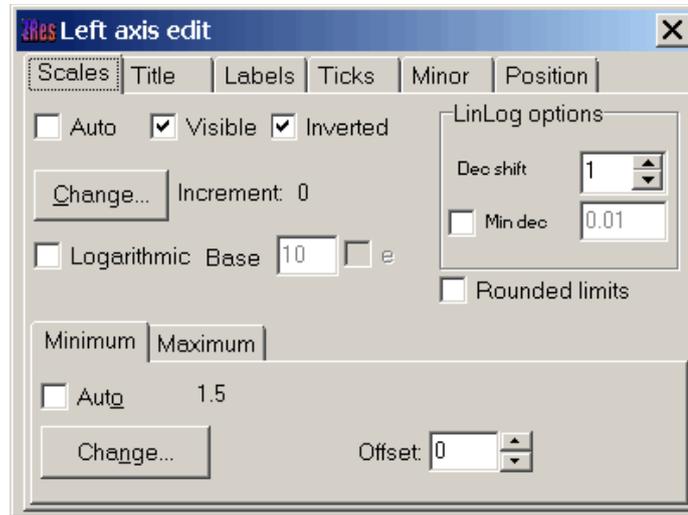
Option **Isolines** specifies whether isolines are created or not.

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



Appendix 5: Axes editor

Many program objects contain coordinate axes. Editor is used to adjust graphic and scale axes parameters. 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** filed 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.

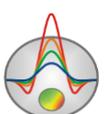
Option **Min dec** sets and fixes minimum (prezero) decade value if option is ON.

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.

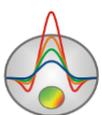
Option **Centered** centers grid of axis ticks.

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 **Len** sets its length.



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

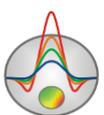
Option **Count** sets number of intermediate ticks between main ones.

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).

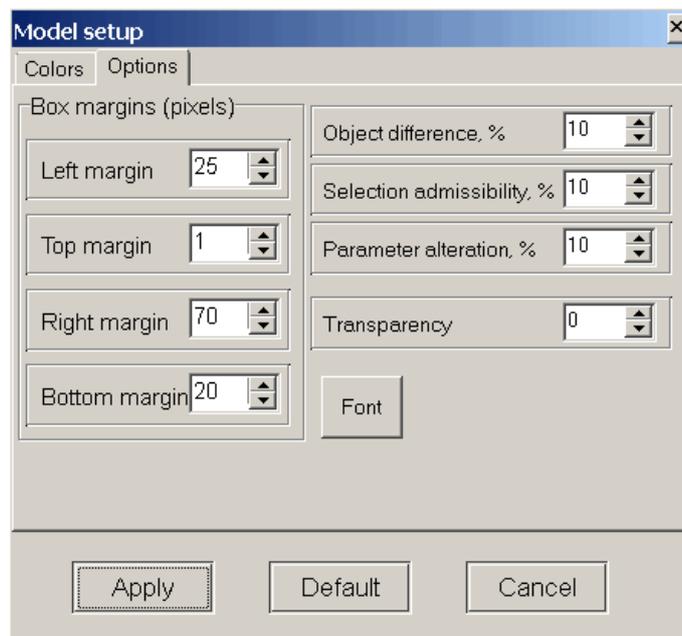


Appendix 6: Dialog of Model visualization parameters

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.

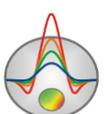
Bottom – sets image indent (in pixels) from window bottom edge.

Object difference - sets the maximum value of the ratio adjacent cells parameters, above which the boundary is drawn between them.

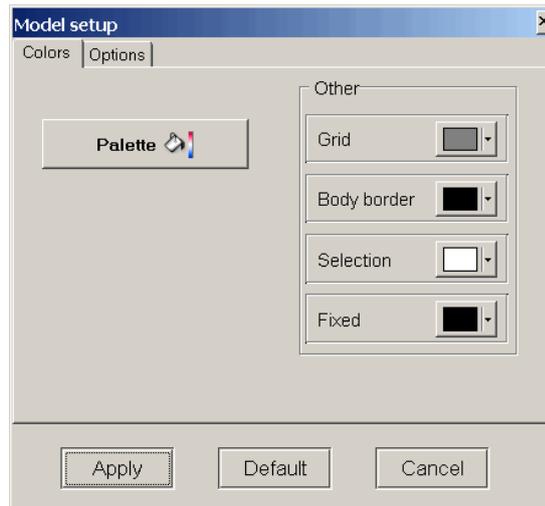
Selection admissibility - sets the acceptable level of parameter differences of adjacent cells, in which, the cell is a single entity and stand together (in the selection mode Magic Wand).

Parameter alteration - determines augment value to the selected cells parameters (in percentages relative to parameter value) when working in **Edit** mode, while pressing Shift button.

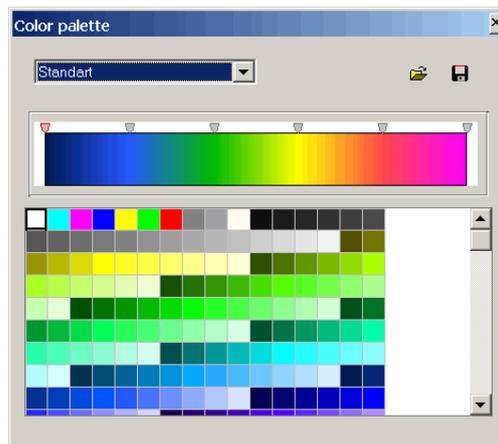
Button **Font** runs the font settings dialog.



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 (**Options/Extra/Model&Data histograms**).

Field Other

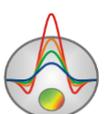
Body border - allows specifying the boundaries color between adjacent cells, if the difference degree between them is greater than the set one in the option **Parameter alteration**.

Grid - sets the grid color.

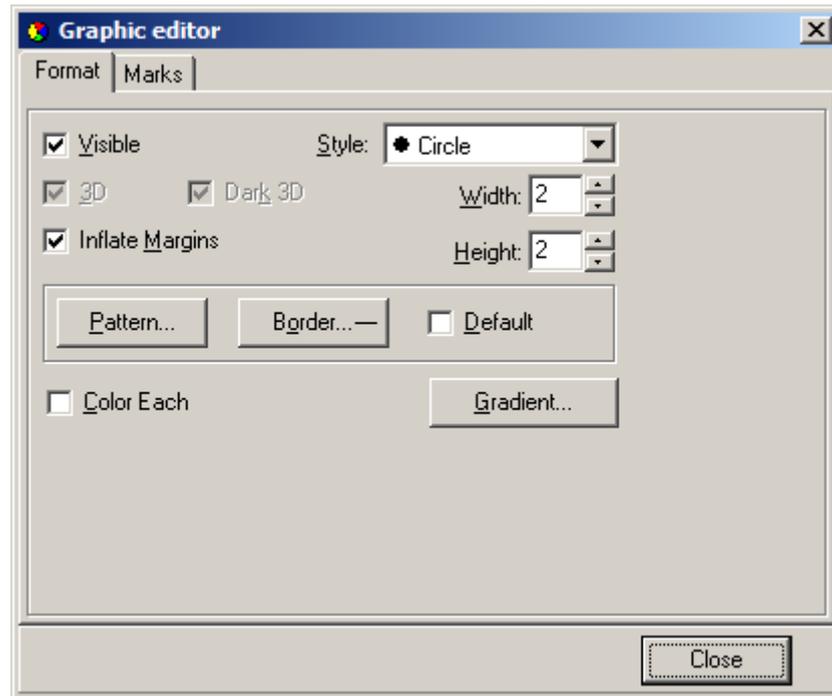
Selection - sets the label color of the selected cell.

Fixed - sets the label color of the fixed cell.

When editing the model as a contour section the setting dialog is similar to pseudosection parameters setup dialog ([Appendix 4: Pseudosection \(contour section\) parameters setup dialog](#)).



Appendix 7: Pseudosection points editor



Editor allows adjusting pseudosection points, electrode position points and their labels. It is called by right-clicking with SHIFT button pressing on the point.

Tab **Format** contains interface settings of pseudosection points and electrode position points. Tab **Marks** contains signatures settings. These tabs options are described in details in [Appendix 1](#): graphics set editor.

Appendix 8. Zdconvert program

Zdconvert program serves to create data files for «ZONDMT1D» and «ZONDMT2D» programs. Table files in txt or dat format which contain frequency, apparent resistivity, and impedance phase values are necessary to work with the program. The program has three main windows (fig. A8). List of sounding sites organized as 5-column table is located in the top right window: **Filename** – name of source file, **Name** – assigned name of the station, **Used** – include/exclude station data from new file, **pozX** – distance from the beginning of profile, **pozZ** – elevation of measuring point. The former four columns can be edited.

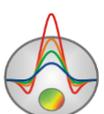
Observed and smoothed curves of apparent resistivity and phase impedance for selected station are displayed in the left window of the program. Double left click on necessary axis to run setup dialog of vertical and horizontal axes.

Main program window with three tabs is located in the bottom right corner.

Tab **Files**:

Field **Data files** – contains information about directory that contains source data files and name of resultant ZondMT1D file.

Input DIR – open directory that contains source data files.



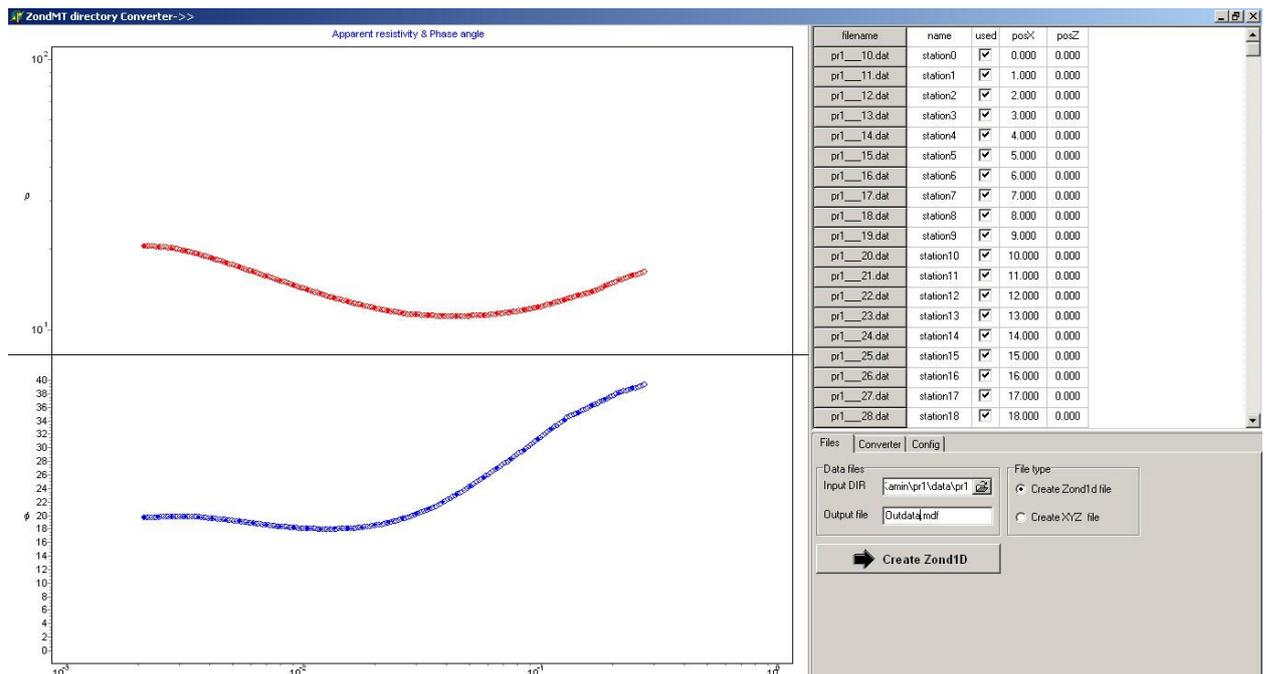
Output File – set outbound ZondMT1D (*.mdf) file name.

Filed File type – specifies type of outbound file.

Create Zond1D file – create file of 1D inversion program ZondMT1D.

Create XYZ file – this option is used to export data of parameters (apparent resistivity and phase impedance) distribution on certain frequency that is set in pop-up window. Additional column **pozY**, that can be edited, also appears in data window. It allows receiving areal parameters distribution.

Depending on selected type of outbound file **Create Zond1D** or **Create XYZ** button is located in the bottom area. After pressing this button corresponding file is created in the directory where Zdconvert program is saved.



Window of auxiliary program Zdconvert

Tab Converter:

Field **Range dividing** – specifies type of filling outbound curves with points. Automatic filling (**Automatic**) or filling in logarithmic frequency mesh (**Equal log-step**) is possible.

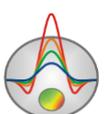
Include boundary frequencies – sets whether end points are included to outbound curves or not.

Smoothing – enable/disable smoothing curve options.

Frequencies number – number of points on resultant curve.

Press **Process** button to accept all specifications.

Three bottom fields serve to integrate two files of 1D inversion program for different polarizations in one 2D inversion file. Select 1D inversion files which correspond to TM and TE



modes in two left windows, specify file name in the right window, and press button . 2D inversion file (*.m2d) will be created in the directory that contains source 1D inversion files.

Tab **Config** is used for source files parameters setup. Specify number of columns in data file, number of column with apparent resistivity, impedance phases and frequency values in **Columns number**, **Rho column index**, **Phase column index** и **Frequency column index** fields respectively.

Specify extensions of source data files (e.g., dat or txt) in the right window.

Turn **log_rho** option on if source data file contains decimal logarithm values of apparent resistivity.

Use button  to open list of *.prd files. This list should be saved in the folder with initial data and should contain three columns: PozX, PozZ and File name (distance along profile, excess and file name respectively).

Button  is used to create list from already opened files.

Window **Step** sets step along profile between stakes in kilometers.

