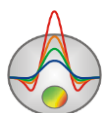


PROGRAM FOR 2D INTERPRETATION OF MAGNETIC AND GRAVITY DATA

ZONDGM2D

<i>PROGRAM FUNCTIONALITY</i>	2
<i>SYSTEM REQUIREMENTS</i>	4
<i>PROGRAM INSTALLATION AND DEINSTALLATION</i>	4
<i>DENSITY OF THE ROCKS</i>	5
<i>MAGNETIC SUSCEPTIBILITY OF ROCKS AND ORES</i>	6
<i>DESCRIPTION OF THE PROGRAM FUNCTIONAL</i>	8
The toolbar of the main window	8
Functions of the main window menu	9
“Hot” keys	14
Status panel	14
<i>WORKING GUIDELINES</i>	14
Preparing and opening a data file	14
Import and export data	16
Saving of interpretation results	20
Getting start with the program	21
Dialog of program parameters settings	24
Profiling plots (Profile)	32
Model editor	33
Working with a model	37
Dialog of model settings	38
Cell parameters setup dialog	39
Selection of medium model in inversion (modes of regular cells grid, polygonal and randomly layered medium)	39
<i>MODELING</i>	43
Modes of polygonal modeling and modeling using arbitrary layered medium	45
<i>ADVANCED PROGRAM OPTIONS</i>	50
Euler deconvolution	50
Downward continuation	51
Cell summarization dialog	52
Option «Buffer» of the main menu	53
Geological sections editor	54
Working with areal data and 3D visualization	58



A priori information input _____	63
ADJUSTING PROGRAM INTERFACE _____	67
Palette settings _____	67
Axes editor _____	68
Graph's editor _____	72
Print preview dialog _____	73
PROGRAM DATA FILE FORMAT _____	75
Data file format _____	75
Logging and lithology data file format _____	76

PROGRAM FUNCTIONALITY

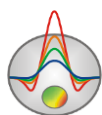
«ZondGM2D» is a computer program for 2D interpretation of profile and areal multi-level data obtained from magnetic and gravity surveys. User-friendly interface and rich options for data presentation allows solving existing geological problems with maximum efficiency.

The traditional exploration method for iron-bearing formations is magnetic survey. It studies magnetic fields of objects, which contain ferromagnetic minerals. Physical interrelation between data measured at the surface and magnetic properties at depths allows assuming presence of magnetic bodies.

In magnetic exploration, total magnetic field is measured. It is comprised of the Earth normal field, anomalous fields caused by magnetized bodies and magnetic field variations, mostly related to the solar activity. A useful component associated with a studying area is anomalous field, which can be identified, taking into account the normal field and measuring magnetic variations in the vicinity of a survey area.

The magnetic field on the Earth surface can be represented as a vector sum: $T = T_n + T_a + \delta T_v$, where T_n and T_a are normal and anomalous magnetic fields, δT_v magnetic field variations. The normal field T_n is divided into a dipole (T_d) and non-dipole (T_m) components, that is $T_n = T_d + T_m$.

The dipole field T_d , which, to a first-order approximation, is the magnetic field of the Earth, is a field of a homogeneous magnetized sphere. The difference between the dipole field (calculated) and areal and normal field (measured by satellite) is a non-dipole part T_m of the normal field. It is often named as a residual field or a field of continental anomalies (sizes of these anomalies are commensurate with the sizes of the continents). Its maxima does not exceed 30% of a dipole field value.



T_n values gradually increase from 33,000 nT at the equator to 68000 nT near the poles, a vertical component of a normal field in the North Pole area reaches 60,000 nT, changes sign to a negative when passing through the equator, and gradually decreases from 0 at the equator to -68 000 nT at the south pole of the Earth . Horizontal component has the maximum near the equator (33,000 nT) and decreases to 0 at the poles. A gradient of the normal magnetic field is about 5 nT per kilometer.

Sources of the anomalous magnetic field T_a are magnetized objects, located near the surface of the Earth. Maximum depth of magnetic rocks is about 50 kilometers. At greater depths, magnetic properties of rocks disappear due to high temperatures.

A value of the field T_a is about 10% of T_n , with an exception of the Kursk Magnetic Anomaly, where the field T_a reaches tens of thousands of nT. Anomalies of a magnetic field induced by rocks are divided into regional and local.

A variable part δT_v of the Earth magnetic field is formed by magnetic variations, whose contribution to the total field is less than 1%. The most important variations can be divided into periodic (solar-diurnal and short-periodic variations) and aperiodic (bay disturbance and magnetic storms).

All variations are induced by external (relative to the Earth) sources: interaction of charged particles with the ionosphere; electric currents propagating in stripe areas of high latitude at an altitude of 100-150 km, etc. In general, an accounting for magnetic variations is important in magnetic surveys because they are cause significant distortions in the observed data.

Program «ZondGM2D» allows solving forward and inverse problems of magnetic and gravimetric surveys (recovery of anomalous magnetic susceptibility, density and geometry of magnetic objects).

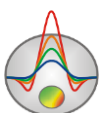
In the program, magnetic susceptibility is set in SI system ($n \cdot 10^{-5}$), density in g/sm^3 , measured values in nanotesla or milligals.

To solve an inverse problem (inversion) various versions of deconvolution methods and Newton's method with focusing regularization are used.

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

where A is the matrix of partial derivatives of measured data with respect to model parameters (Jacobian), C smoothing operator, W matrix of relative measurement errors, m the model parameter vector, μ - regularizing parameter, Δf vector of residuals between the observed and calculated values, R focusing operator.

During the development of inverse problem, special attention was given to a priori information usage (weights of individual measurements, ranges of parameters).



«ZondGM2D» uses a simple and clear data file format.

Program allows importing and visualizing data using other methods which makes data interpretation process more integrated.

«ZondGM2D» is an easy-to-use instrument for automatic and interactive multilevel data interpretation of magnetic and gravity surveys, and can be used on IBM-PC compatible PCs with Windows operating system.

SYSTEM REQUIREMENTS

«ZondGM2D» can be installed on a PC with OS Windows 98 and higher. Recommended system parameters are processor P IV-2 GHz, memory 512 Mb, screen resolution 1024 X 768, color mode – True color (screen resolution change is not recommended while working with data).

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

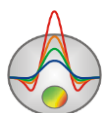
PROGRAM INSTALLATION AND DEINSTALLATION

«ZondGM2D» program is supplied on internet. Current manual is included in the delivery set. Latest updates of the program can be downloaded from website: www.zond-geo.com.

To install the program copy it from internet 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 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.



DENSITY OF THE ROCKS

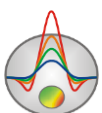
It is essential to know rock density σ which is the only physical parameter that gravity survey is based on to perform gravity surveying and especially to interpret results.

Rock density (or volume weight) is defined as its mass per unit volume. Density unit is g/sm^3 . Density is usually measured on samples taken from natural outcrop, boreholes or mines. The easiest way to measure density is to weight the sample in the air and in water and then calculate density σ . The most popular and handy device for density measurement is densitometer and it is based on this principle. Densitometer defines density within the accuracy of $0,01 \text{ g}/\text{sm}^3$ [Hmelevskoj, 1997].

In order to receive reliable and representative data it is necessary to measure large quantity of samples (up to 50). On the basis of numerous density measurements on samples from the same lithological sequence variation curve or cross-plot of σ values versus number of samples characterized by current density is constructed. Curve maximum corresponds to the most probable density value for current rock. There are gravimetric and other geophysical methods of field and borehole density assessment.

Density of rocks and ores depends on chemical-mineral composition, in other words on bulk density of solid particles, porosity, and pore filler composition (water, solutions, oil, gas). Density of volcanic and metamorphic rocks is mostly defined by their mineral composition and increases in going from acidic to base and ultrabasic rocks. Density of sedimentary rocks first of all depends on porosity, water saturation, and to lesser degree on mineral composition. But it strongly depends on deposits consolidation, their age, and depth of burial (their increase leads to density increase as well). Examples of density are given below [Hmelevskoj, 1997].

Rock	Density (g/sm^3)
Oil	0,8 - 1,0
Coal	1,0
Water	1,1 - 2
Soil	1,13 - 2,0
Sand	1,4 - 2
Clay	2 - 2,2



Sandstone	1,8 - 2,8
Limestone	2,3 - 3,0
Salt	2,1 - 2,4
Granite	2,4 - 2,9
Gneiss	2,6 - 2,9
Gabbro	2,8 - 3,1
Basalt	2,7 - 3,3
Peridotite	2,8 - 3,4
Copperpyrite	4,1 - 4,3
Magnetite, hematite	4,9 - 5,2
Density of the upper parts of the crust (average)	2,67

MAGNETIC SUSCEPTIBILITY OF ROCKS AND ORES

Magnetic susceptibility χ is the main magnetic property of rocks which characterizes the degree of magnetization of materials and rocks.

It is dimensionless number in SI system. But in practice it is measured in 10^{-5} SI units. It ranges from 0 to 10 units for different rocks. Minerals and rocks are divided in three groups by magnetic susceptibility: diamagnetic, paramagnetic, and ferromagnetic. Diamagnetic rocks have very low magnetic susceptibility (less than 10^{-5} SI units) which has negative value (their magnetization is directed against magnetizing field). Many minerals and rocks such as quartz, mine salt, marble, oil, ice, graphite, gold, silver, lead, copper, etc. are diamagnetic [Hmelevskoj, 1997].

Paramagnetic rocks have positive magnetic susceptibility with low values. The majority of minerals, sedimentary, metamorphic, and volcanic rocks are paramagnetic.

Ferromagnetic minerals (e.g. magnetite, titaniferous magnetite, ilmenite, pyrrhotite) have very large χ values (up to several millions of 10^{-5} SI units).

Magnetic susceptibility of the majority of rocks depends largely on the presence and percentage of ferromagnetic minerals in their composition.

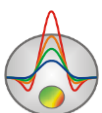
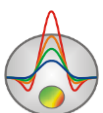


Table below gives χ values of some rock-forming minerals and rocks. It can be seen from the table that ferromagnetic minerals are strongly magnetic. Ultrabasic and base rocks are characterized by the highest magnetic susceptibility values among other volcanic rocks, acidic rocks are magnetic and weakly magnetic. Magnetic susceptibility of metamorphic rocks is lower than the one of volcanic rocks. Sedimentary rocks with the exception of some sandstones and clays are almost non-magnetic.

Mineral, rock	$\chi 10^{-5}$ (SI units)	
	range	average
Quartz	-	10
Calcite	7 – 12	-
Gypsum	-	12
Coal	-	25
Sphalerite	-	750
Hematite	500 - 50000	6000
Pyrrhotine	10^3 - 10^7	150000
Ilmenite	$5 \cdot 10^5$ - $5 \cdot 10^6$	10^6
Magnetite	10^6 - 10^7	$5 \cdot 10^6$
Limestone	25 - 3500	300
Sandstone	0 - 20000	400
Gneiss	100 - 20000	-
Granite	0 - 40000	2000
Diabase	1000 - 15000	5000
Gabbro	1000 - 100000	60000
Basalt	30 - 150000	60000
Peridotite	90000 - 200000	150000
Sedimentary (average)	0 - 5000	1000
Metamorphic (average)	0 - 75000	50000
Acidic volcanic (average)	50 - 80000	8000



Base volcanic (average)	60 - 120000	30000
--------------------------------	-------------	-------

To convert χ values in CGS system which is used in the program, magnetic susceptibility in SI units is divided by 4π .











Magnetic susceptibility of para- and ferromagnetic rocks decrease with increasing temperature and almost disappears at Curie temperature which ranges from +400 to +700C for different minerals. Maximum depth of magnetic survey investigation is approximately 25-50 km. Temperature is higher than Curie point at greater depths and all occurring there rocks become essentially non-magnetic.

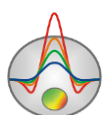
Prospecting geological structures and ores with magnetic susceptibility κ are embedded by rocks with magnetic susceptibility χ . This is the reason why similarly to gravity survey redundant or effective magnetic susceptibility $\Delta\chi$ is of interest. Value of $\Delta\chi$ can vary and be positive or negative. Magnetic anomalies appear due to variation of $\Delta\chi$ from zero [Hmelevskoj, 1997].






DESCRIPTION OF THE PROGRAM FUNCTIONAL

The toolbar of the main window

The toolbar is aimed at quick access to often used software tools. It contains the following functional buttons (from left to right):

	Open data file.
	Call save data dialog.
	Model mode
	Layered modeling mode (more).
	Polygonal modeling mode.
	Inversion parameters setup dialog (more).
	Data editor
	Work with magnetic measurements.
	Work with gravity measurements.
	Solve a forward problem.

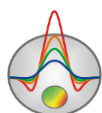


	Start (one click) or cancel (second click) inversion calculation.
	Euler decomposition (more)
	Rapid transformation of a field and construction of a pseudo-depth section.
	Undo last model modification.
	Select profile.

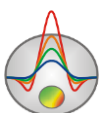
Functions of the main window menu

The following table lists menu items with their corresponding functions:

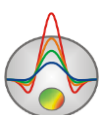
File/Create synthetic survey	Call the dialog of synthetic survey system creation (more).
File/Open file	Open data file.
File/ Add to project	Add data to the open project.
File/Save file	Save data dialog.
File/Print Preview	Call main window printing dialog (more).
File/Edit file	Open current data file in the Notepad editor.
File/Recent	Recently used files.
File/ Русский	Use Russian interface.
File/English	Use English interface.
File/Reg file	Create registration file.
File/Exit	Exit program.
Options/Mesh constructor	Call starting model setup dialog.
Options/Project information	Add project information (executor, customer, name of the survey area, equipment, etc.).
Options/Line's observation settings	Call measurements settings window (more).
Options/Set survey lines	Call survey line setup window (more)
Options/Program setup	Call inversion parameters setup dialog (more).
Options/Data editor	Call data editor dialog (more)
Options/ Modules/Downward continuation	Downward continuation of the field (more) Standard – continuation of the original field Normed – continuation of the vertical derivative.
Options/Modules/3D sections plot	Run 3D model viewer. It works with areal data (more).



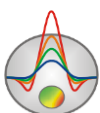
Options/Modules/Graphics map	Show graphs plan (more).
Options/Modules/Geological editor	Call geological sections editor window (more).
Options/Inversion/Set boundaries	Call dialog of geological boundaries settings (more).
Options/Inversion/Resolution	Increasing sensitivity results in increasing influence of deeper model cells.
Options/Inversion/reference model	Definition of a level, relative to which smoothness of a calculated model is determined. Available values: Median - from data average; "0" - from zero level Start model - from a starting model (useful when a priori information is available) Previous - from the previous iteration.
Options/Inversion/Focused filter	Focusing filter parameters. Own - use own filter, resulting from the inversion Load - load a model from another inversion program. Cells with low contrasts are smoothed. Other - used in joint inversion of gravity and magnetic surveys. Use a filter calculated from inversion of another method.
Options/Model/Block section	Plot model with blocks.
Options/Model/Smooth section	Display model using a smoothed interpolation palette.
Options/Model/Contour section	Display model as a contour plot.
Options/Model/Infinite borders	When switched on, the program automatically extends order objects beyond view area. Otherwise, objects size is restricted by the view area.
Options/Model/X=Z scale	Make scales along the vertical and horizontal axes the same.
Options/Model/Extend bottom	When a relief is present, this option extends lower model cells to the maximum depth.
Options/Model/Show modeler tools	Show modeling toolbar (more)



Options/Topography/Topo coefficient 1	Set distortion factor of topography with depth (0-5). 0 - relief of each following layer is the same as the previous one. 1 – relief flattens with depth, the last layer is flat.
Options/Topography/Import topography	Import relief topography.
Options/Topography/Remove topography	Remove topography data (all data will be assigned zero height).
Options/Topography/Restore topography	Return previously loaded data topography.
Options/Topography/Edit topography	Edit relief topography data.
Options/Topography/Denoise topography	Remove single spikes in topography data.
Options/Topography/Smooth topography	Smooth topography.
Options/Topography/Import 3D topography	Import a topography file of all project sections (must contain coordinates and height of observation points).
Options/Topography/Spline intermediate	
Options/Topography/Absolute coordinates	Turn the absolute height mode on (when the mode is turned off, depth is counted from the highest model point).
Options/Topography/Extended nodes	Add cells to the left and right.
Options/Boreholes/Load borehole data	Open and display a file with borehole log data and stratigraphic columns (more).
Options/Boreholes/ Create/Edit borehole data	Create/edit borehole data (more).
Options/Boreholes/ Remove borehole data	Remove borehole data.
Options/Boreholes/Set column's width	Set width of borehole columns.
Options/Import/Export/Import data text/excel	Import a data file in text or Microsoft Excel format (more)

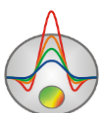


Options/Import/Export /Import model/graph	Import arbitrary data or model.
Options/Import/Export /Remove data	Remove from the project a plot derived from the imported data.
Options/Import/Export /Save selection	Save model cells selected in the model editor.
Options/Import/Export /Load selection	Open file with the selected cells, and paste it at the current cursor position.
Options/Import/Export /Extract 1d log	Save a vertical profile of the magnetic susceptibility or a density for a specified horizontal coordinate.
Options/Import/Export /Load 1d log	Paste into a model a vertical profile of a magnetic susceptibility or a density of their limits at a specified horizontal coordinate.
Options/Import/Export /Save MOD2D file	Save 2D model settings file that can later be used in others Zond programs.
Options/Import/Export /Load MOD2D file	Load 2D model settings file obtained, for example, in other Zond programs.
Options/Import/Export /Save MOD1D file	Save 1D model settings file that can later be used in others Zond programs.
Options/Import/Export /Load MOD1D file	Load 1D model settings file obtained, for example, in other Zond programs.
Options/Import/Export/Load background image	Import file format *.sec or an arbitrary image.
Options/Import/Export/Rem ove background image	Remove background image.
Options/Import/Export/Sho w background image	Show exporting image.
Options/Import/Export/Resi ze background image	Resize exporting image.
Options/Import/Export/Exp ort to Excel	Save model in Excel format.
Options/Import/Export/Exp ort model to DXF	Save model in AutoCAD format (*.DXF)
Options/Import/Export/Dra	Export model to Surfer.



w model in Surfer	
Buffer	Save a current model to the clipboard for later comparison with new models (more)
Operations/Line operations	The subset of operations executed for a single profile line.
Operations/Subtract total field	Subtract total field.
Operations/Subtract median value	Subtract median value.
Operations/Subtract user value	Subtract user-specified value.
Operations/ Subtract med in window	Subtract an average value in a current window (useful when working with data on regional profiles).
Operations/ Subtract field from height	Set and subtract a specified value if the current value exceeds it.
Operations/Reduce to pole	Reduce to the pole. Used for magnetic survey data. It allows placing anomaly over the center of an anomalous object.
Operations/Reduce to plane	Reduce to a single height level. Useful when working with airborne data. Allows avoiding false anomalies associated with relief and altitude changes.
Operations/Denoise data	Remove noisy data (single spikes).
Operations/Return to original	Back to the raw data, cancel all transformation and operations.
Operations/Areal operations	Subset of operations for areal data.
Help/About	About the program.
Help/Context	Run manual.
Help/Check for updates	Check for updates.
Help/ERROR!!! Set default values	Reload the program.

^ File of imported data should consist of two columns: a coordinate of measurement, measured value. A vertical axis of the imported data is right axis of a profile graphs plan. An imported model must be retained in a project file Zond 2D. While loading it, a new window containing a



model appears. Moving the cursor in a modeling frame, a position of an active cell is displayed on the model of the imported file.

“Hot” keys

Cursor pad /cursor in the model editor	Change of the model active cell
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 an area.
V / cursor in model editor	Delete selection.
Up/down / cursor in color scale	Change current value.
Space	Calculate forward problem.
Enter	Switch to Euler or Objects mode and back.

Status panel

Status panel is divided into a few sections which contain different information:

Coordinates of cursor and active cell.
Parameters of active cell
Additional information.

WORKING GUIDELINES

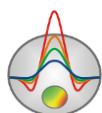
Preparing and opening a data file

To begin working with « ZONDGM2D», it is necessary to have a data file of specific format which contains information about measurement positions and measured values.

One file usually contains data obtained from one profile. Text data files created in «ZONDGM2D» format have «*.GM» extension. See **data file format** section for more details.

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

- unconventional separation symbols (use TABs and SPACEs only);
- invalid values of measured data
- It is desirable to not have more than 5000 observed data values in a single file.



There is an ability to import text files or Excel files. To do this, go to **Options/Import/Export** in the main menu and choose **Import data text/excel**. After selecting the file, a dialog box will appear (Fig. 1), in which it is possible to set a type of data for every column and data units. Thus, the imported file can be arranged in any order.

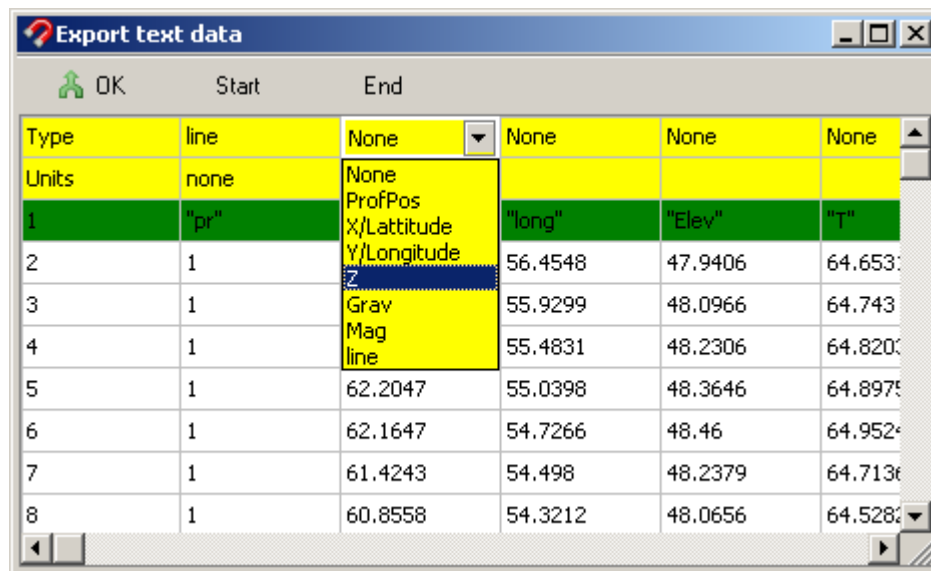


Fig. 1 **Import from text/excel** dialog box.

The first two lines with yellow fill are intended for a type of data in the column (**Type**) and units (**Units**). The data type may take one of the following values:

ProfPos - number of a measurement point

X/Latitude - horizontal coordinate/latitude

Y/Longitude - vertical coordinate/longitude

Z – height

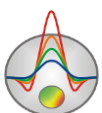
Grav - measured values of gravity field

Mag - measured values of magnetic field

line - a profile number

The first line of the opened file is displayed on the third line with a green filling color.

The keys **Start** and **End** on the panel dialog are intended to define the first and last imported rows, respectively. After selection of all necessary parameters, click **OK**.



There is an ability to add data to the project, which is useful when working with areal data.

Import and export data

The tab of the main menu **Options/Import/Export** contains program options for importing data of different types (raw data, data from other methods in the form of underlying image, etc.) and for exporting data to different software packages. These options allow a user to compare data from different methods, to load data of gravity and magnetic surveys of any data format, to plot quickly results obtained in other programs.

There is an ability to import text files or Excel files. To do this, choose the item **Import data from text/excel**. After selecting the file, a dialog box will appear (Fig.2), in which it is possible to set a type of data for every column and data units. Thus, the imported file can be arranged in any order.

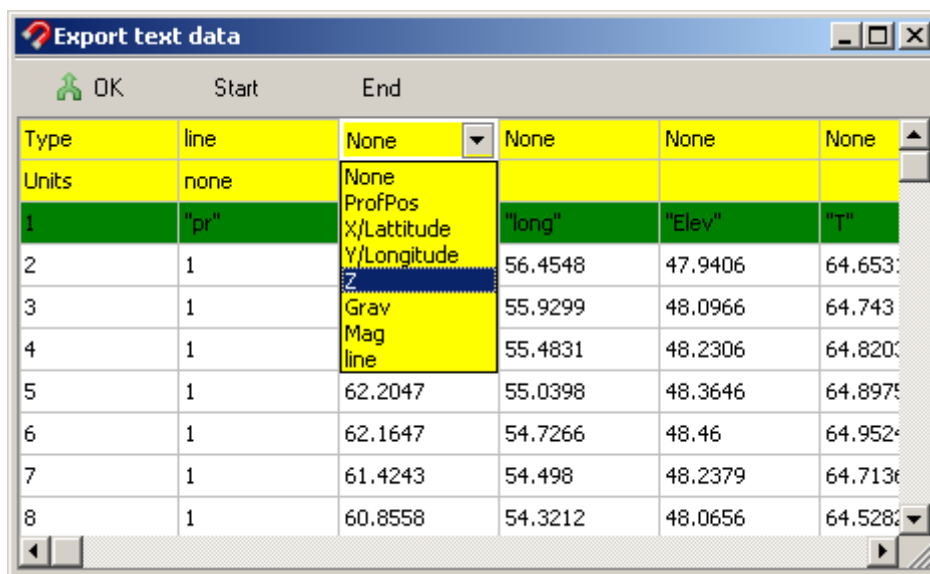


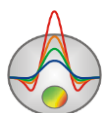
Fig.2 **Import from text/excel** dialog box.

The first two lines with yellow fill are intended for a type of data in the column (**Type**) and units (**Units**). The data type may take one of the following values:

ProfPos - number of a measurement point

X/Lattitude - horizontal coordinate/latitude

Y/Longitude - vertical coordinate/longitude



Z – height

Grav - measured values of gravity field

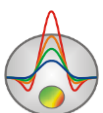
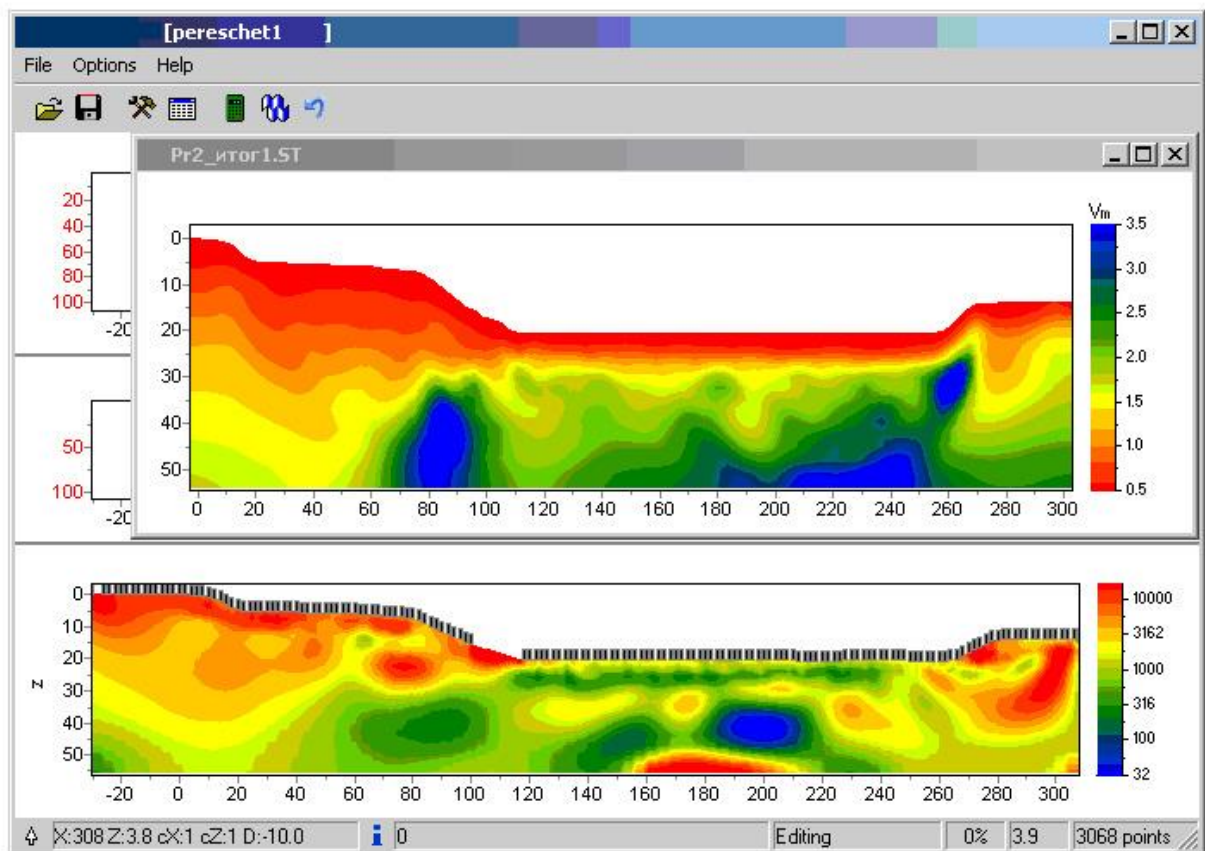
Mag - measured values of magnetic field

line - a profile number

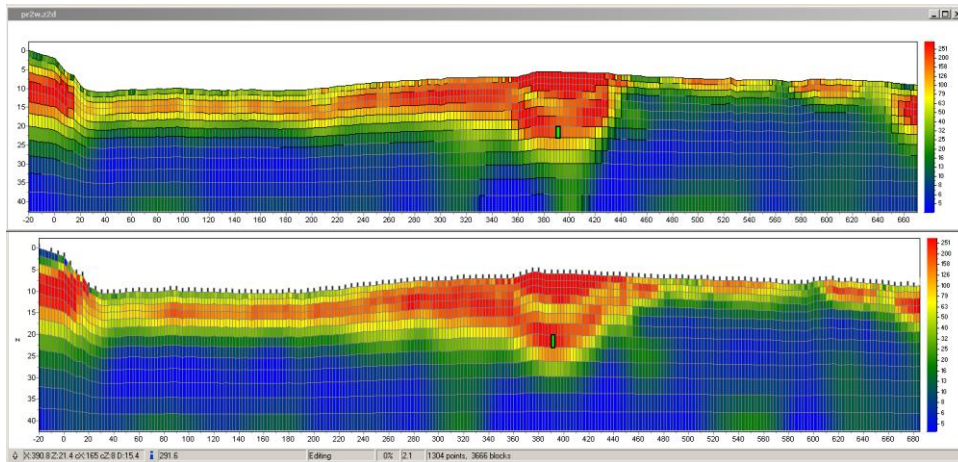
The first line of the opened file is displayed on the third line with a green filling color.

The keys **Start** and **End** on the panel dialog are intended to define the first and last imported rows, respectively. After selection of all necessary parameters, click **OK**.

Import/Export/Import model/graph, this option allows loading a model from ZOND program projects (*.mod) into separate windows. It also allows displaying a plot of any parameter in a data area, for this data should be represented in *.dat file format, where the first column corresponds to the horizontal, and the second to the vertical axis. This option can be useful when comparing results of interpretation on adjacent profiles or during integrated interpretation of data from various methods.



During cursor movement in the model editor, cursor will be displayed in all other sections, according to the size of the current cell.



Using the option **Save/Load selection**, it is possible to save or load a model tile. To save the tile, it is necessary to switch on the model displaying with blocks mode (Blocks section) and use Selection tool to extract a tile. Then, finally click Save selection.

A model tile can be loaded in the following way: Select a small area of the current model. The upper-left corner of the selected area will be considered as the place from which the tile will be placed. Call option **Load selection** and select a file name. If the selection is not available, then the fragment will be pasted from the upper-left corner of the model.

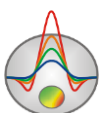
In order to save or load a vertical parameter profile for a horizontal coordinate, use the option **Extract 1d log/Load 1d log**. When saving the vertical profile it is necessary to set X coordinate in the dialog box. When loading a vertical profile it is necessary to specify the X axis range. This option can be used, for example, for incorporation of logging data, or for inspection of profile intersections.

If a priori information is available, the option **Import/export/Background** image allows using it (as an underlying image for the model editor). This can, for instance, be a geological, electrical or seismic sections, or a section from an adjacent profile.

The image can be loaded from a *.sec file in scale.

* .sec file has the following structure:

1st line – image file name;

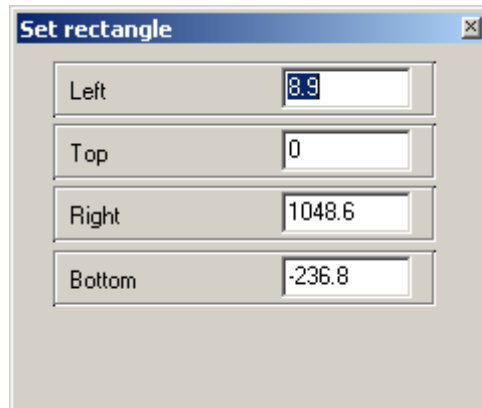


2d line – four coordinates X1 Y1 X2 Y2 of top left and bottom right corners of the image are given with space separators

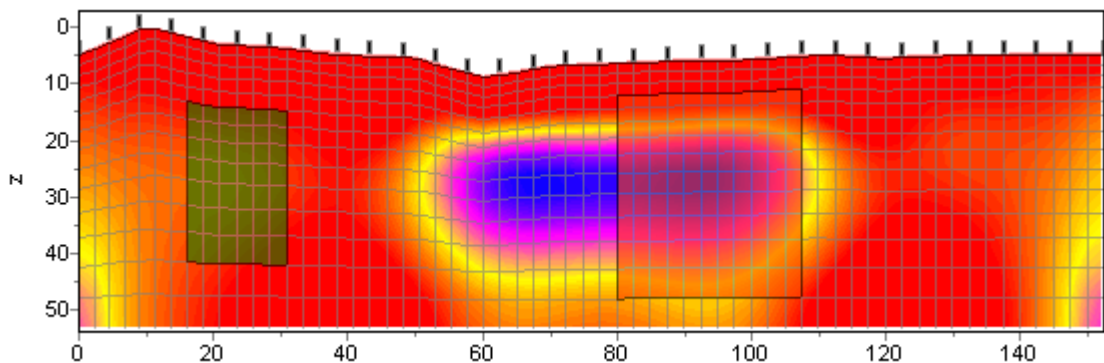
sect.emf

0 0 152.4 53.3

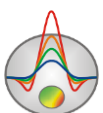
There is also an ability to load any image file. In this case, a dialog **Set rectangle** appears, allowing you to set the boundaries of the imported graphic file.

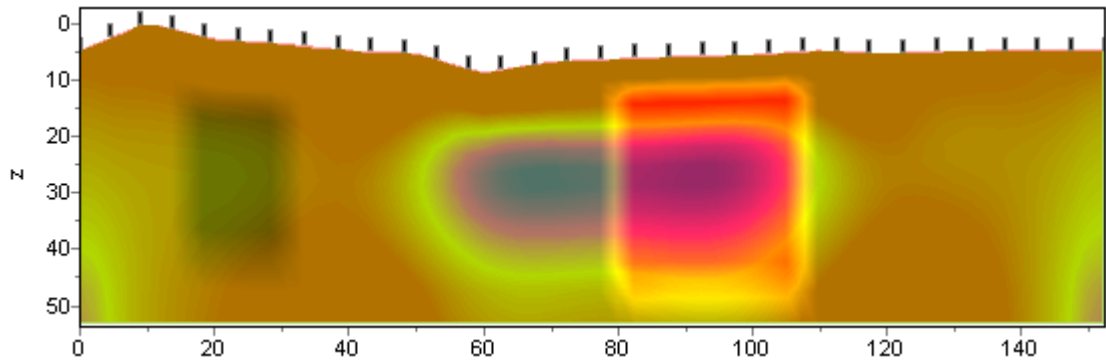


In the Blocks section mode, cells with values different from the background medium will be shown. Thus, it is possible to model anomalous objects on top of an underlying image.



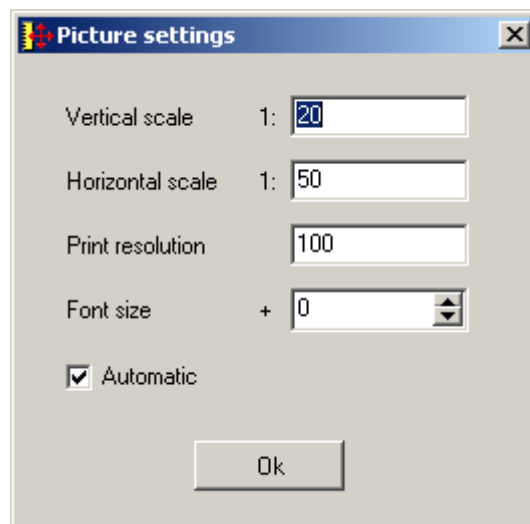
In the Smooth section mode, colors of an underlying image and the current model will be mixed and it is possible to see features of two sections simultaneously.






To remove underlying image, select the **Remove Background** option.

Output settings dialog allows adjusting vertical scale (in meters per centimeter) and horizontal scale (in meters per centimeter), outbound image resolution (in DPI), and font size.

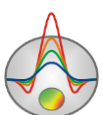


These settings are applied to a model saved in BMP format , if Automatic option is ON. Otherwise the same image as it is displayed on the screen will be saved.


The option **Export to Excel** allows saving the current model parameters in a file with Microsoft Excel format. Data in the obtained file are organized by rows, the first two rows are distance and height in meters, other rows contain parameter data depending on a model cell.

The option **Export model to DXF** is intended for exporting a model to an AutoCAD compatible file format.

Saving of interpretation results




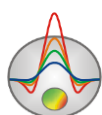
Profile interpretation result is stored in «ZONDGM2D» file format (extension *.GMP). Field data, values of relative measurement weights and current model, as well as parameters of the normal magnetic field are kept in this file. Data from the file are used for later loading and medium model creation.

In order to save result of interpretation, click the toolbar button  or the respective menu item. In the dialog box, it is also possible to select the data format for saving calculated data for the current model values of magnetic and gravity fields, and also image (Model, WorkSheet) in *.BMP format.

Zond project data	Save measured values and current medium model.
Zond project with calculated data	Save calculated values and current medium model.
Worksheet	Save three graphic window parts in BMP format.
Model	Save bottom graphic part of the window in BMP format.
Grid file	Save the current model as a grid file.
Section file	Save the current model in *.sec format.
Save data as text	Save measured, calculated data and additional information in text format.

Getting start with the program

After creating a data file «*.GM», it should be loaded using the button  or respective menu item. Upon successful file loading, the window measurement parameters selection appears and dialog of initial model settings, in which it is possible to select survey layout parameters. This dialog is also available in the Options item of the main menu. Window of measurement parameters selection is divided into two parts: **Magnetic survey** and **Gravity survey**, corresponding to settings for the magnetic and gravity fields.



The screenshot shows a software window titled "Observation settings". It is divided into two main sections: "Magnetic survey" and "Gravity survey".

- Magnetic survey:** This section is currently selected. It contains several input fields:
 - Total field, nT: 50000
 - Inclination, deg: 0
 - Declination, deg: 0
 - Profile azimuth, deg: 51
 - Magnetometer elev, m: * (checkbox is unchecked)
 - Data type: T
 - Gradient base, m: 1
- Gravity survey:** This section is also selected. It contains:
 - Gravimeter elev, m: 0 (checkbox is unchecked)
 - Data type: Gz
 - Gradient base, m: 1

An "Apply" button is located at the bottom center of the dialog box.

Magnetic survey part:

Total field, nT – magnitude of the normal magnetic field vector (T_0), in nT.

Inclination, deg – a value of the normal magnetic field inclination, in degrees (I_0). It is counted down from horizontal.

Declination, deg – a value of the normal magnetic field declination, in degrees (D_0). It is counted clockwise from north.

Profile azimuth, deg – profile azimuth, in degrees. It is counted clockwise from north
Magnetometer elev, m – a height of a magneto active sensor, in meters, relative to relief.

Data type – a type of measured data. T is the magnetic field, GrZ - magnetic field gradient

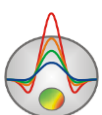
Gradient base, m – a height of a sensor above the earth's surface

Gravity survey part:

Gravimeter elev, m – a gravimetric observations height, in meters, relative to the relief.

Data type – a type of measured data. Gz – vertical component of gravity force, Grz gradient of Gz.

Gradient base, m – a sensor height above the earth's surface.

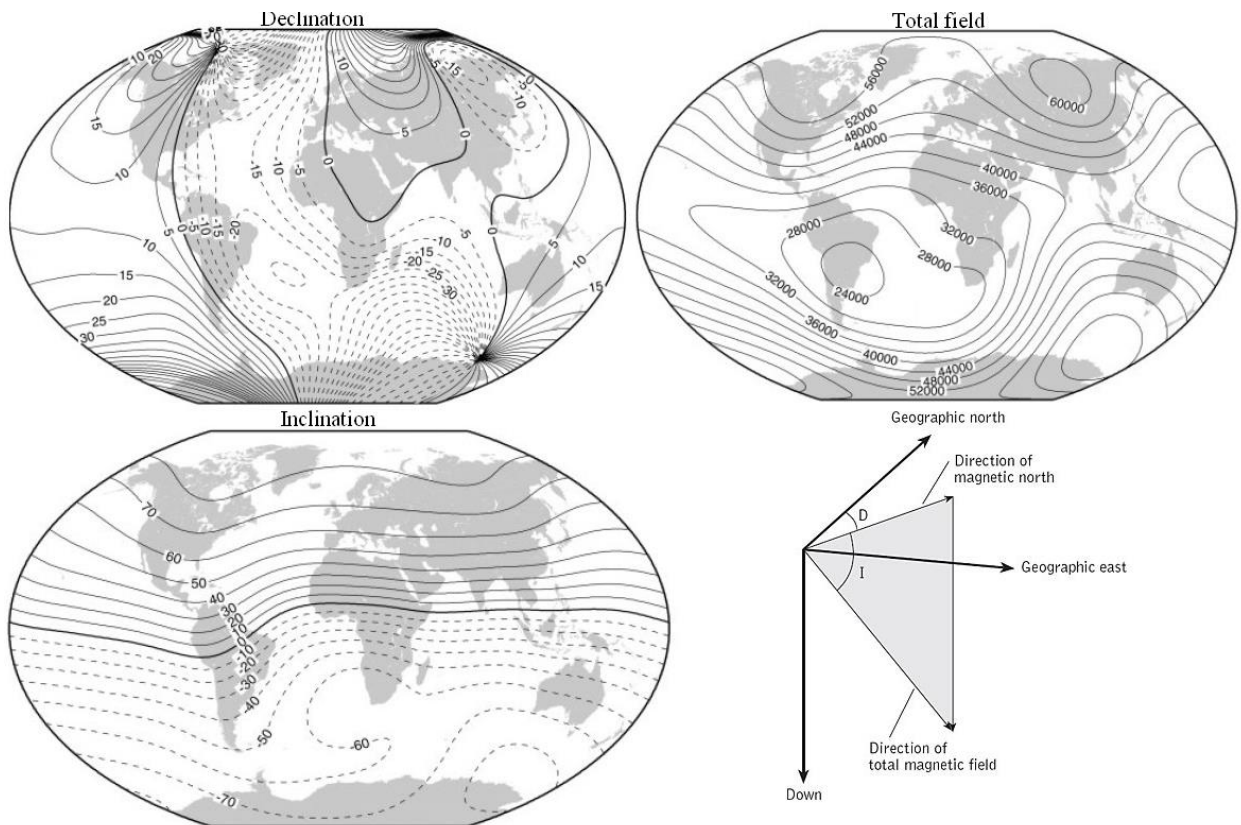


Anomalous magnetic field and directional cosines of the normal magnetic field (C_x, C_z) are calculated using following formulae:

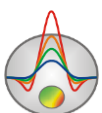
$$\Delta T \approx H_x C_x + H_z C_z,$$

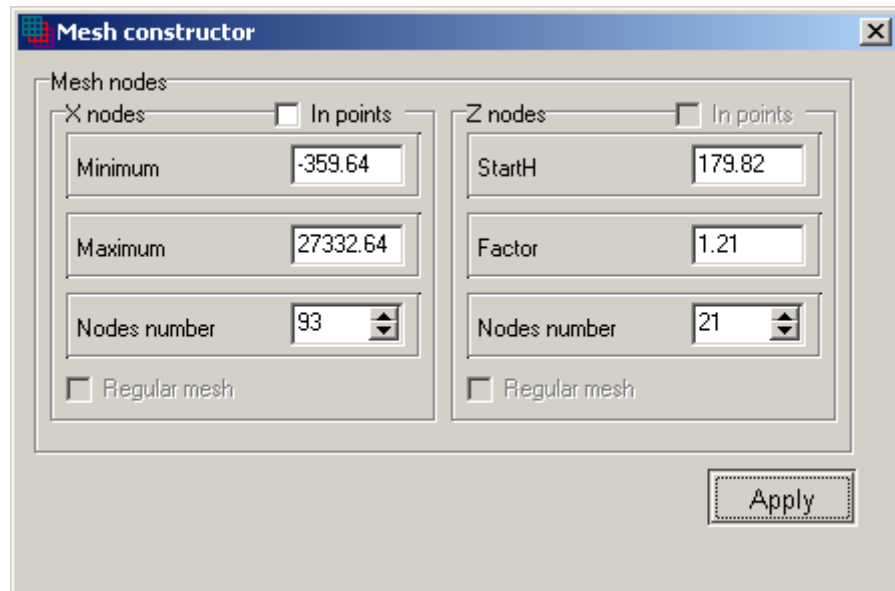
$$C_x = \cos I_0 \cos(A - D_0) \quad \text{and} \quad C_z = \sin I_0$$

Shown below are isoline maps of declination, inclination and total magnetic field in the year of 2005. The current values for specific latitude and longitude can be found on specialized sites or using GIS packages.



Dialog of initial model settings **Mesh constructor** contains following options.





Area **X nodes** contains options, which allow setting the horizontal model grid parameters.

Minimum – set the minimum coordinate of the modelling area.

Maximum – set the maximum coordinate of modelling area.

Nodes number – if the option **In points** is OFF, it sets a number of uniformly distant nodes of the horizontal grid; otherwise nodes are set at measurement points.

Regular mesh –set a uniform partition of a grid between measurement points (if the option in points is ON).


Area **Z nodes** includes options, which allow setting parameters of a model vertical grid (active for the borehole measurements).

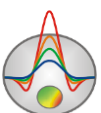
StartH – set the first layer thickness. This value need to meet the required resolution.

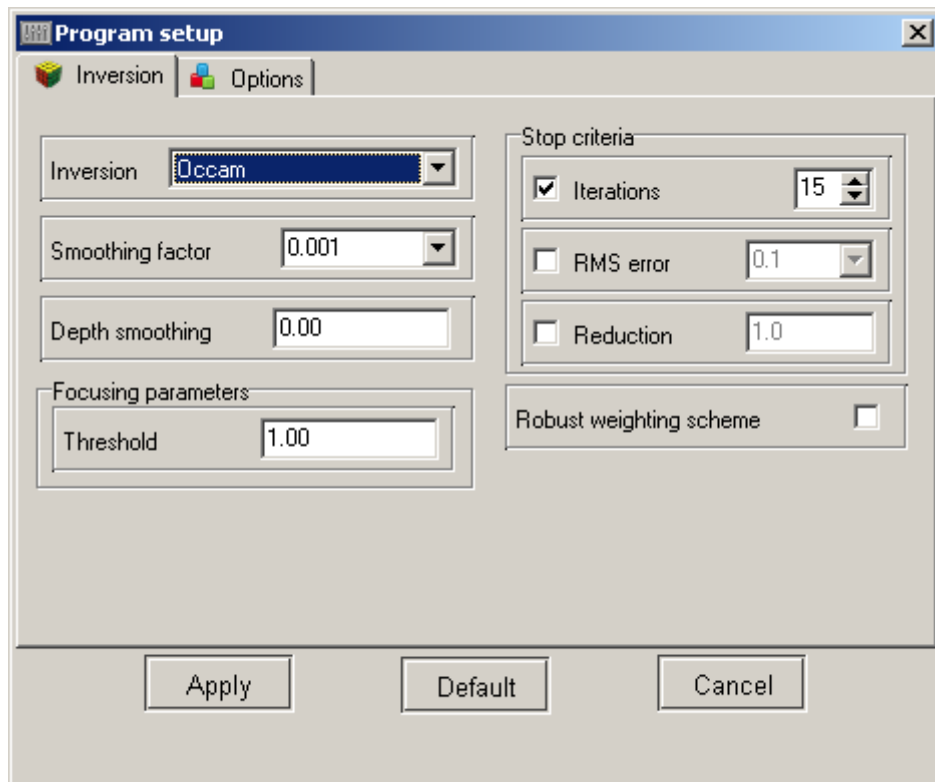
Factor – set a ratio between the adjacent layers thicknesses. Values of the parameter are generally selected in the range from 1 to 2.

Nodes number – sets a number of layers.

Dialog of program parameters settings

The dialog is available in the main menu Options/Program setup or as the button  on the main panel, and is intended to set parameters related to the solution of forward and inverse problems.

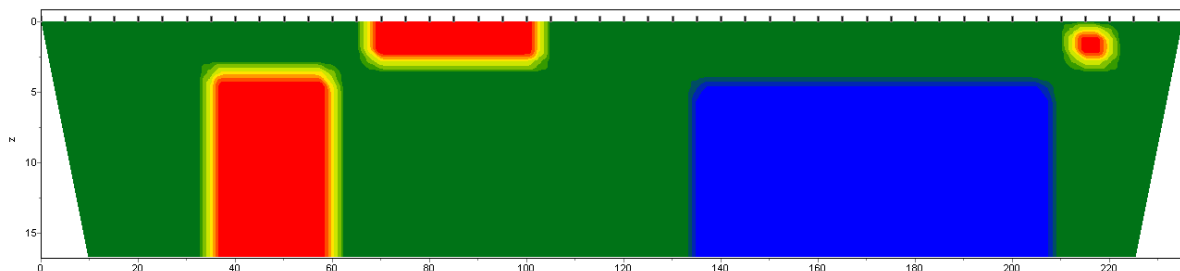




First tab “**Inversion**” is used to configure the parameters of inversion.

The option **Inversion** defines an algorithm by which inverse problem will be solved.

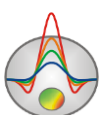
Consider various inversion algorithms, as an example take a model consisting of several blocks.

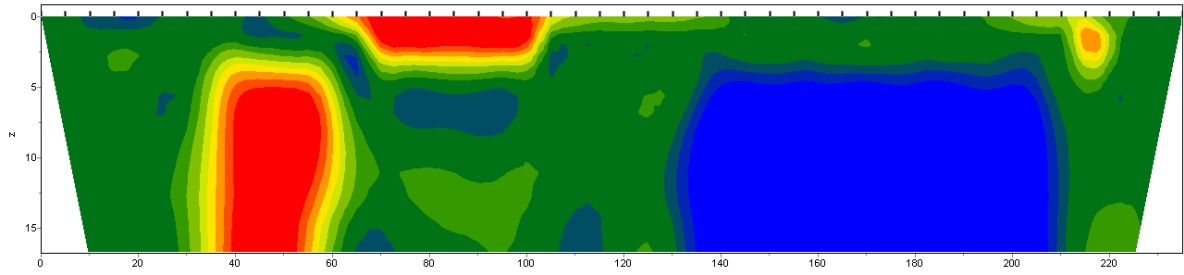


Consider various inversion algorithms, as an example take a model consisting of several blocks.

To test algorithms calculate a theoretical response for the model and add a five per cent Gaussian noise.

Smoothness constrained - inversion obtained by the least squares method using a smoothing operator. As a result of using this algorithm, a smooth (without sharp boundaries) and a stable distribution of the parameters are obtained.



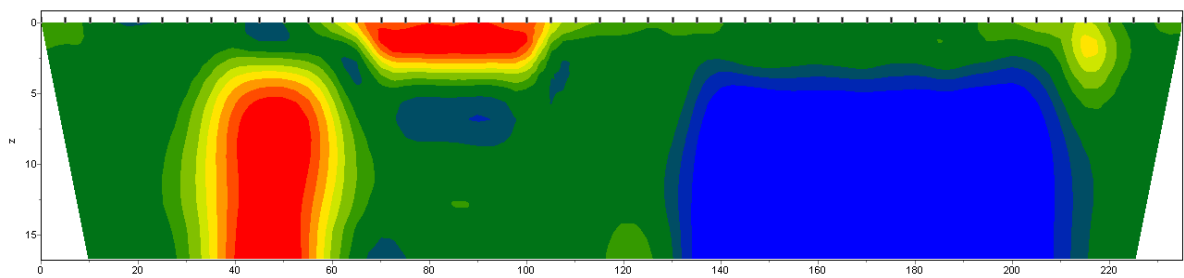


The matrix equation for this inversion algorithm looks as follow:

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

As seen from the equation, inversion model contrast is not minimized. This algorithm allows reaching minimum misfit values. It is recommended to use it in the initial interpretation, in most cases.

Occam - inversion by least squares method with using a smoothing operator and additional contrast minimization [Constable, 1987]. As a result of using this algorithm, we get the more smooth parameters distribution.

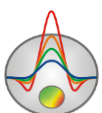
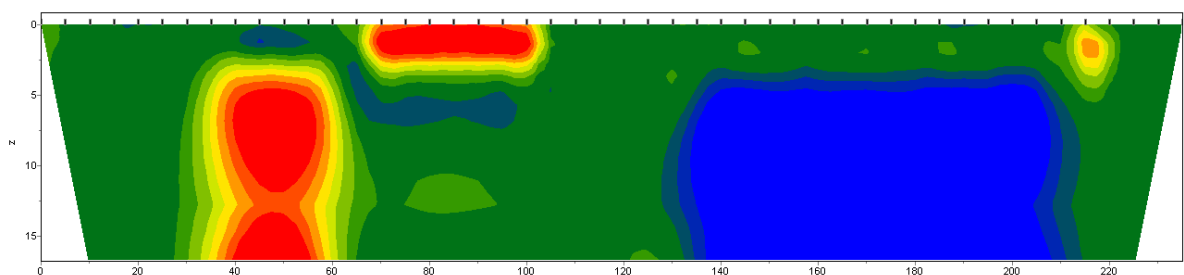


The matrix equation for the inversion version looks as follow:

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

The degree of smoothness of a resulting model is directly proportional to the value of **Smoothness factor**. It should be noted that if parameter values is too large it can lead to the misfit increase.

Marquardt - classic inversion algorithm using the least squares method with a damping parameter regularization (**Ridge regression**) [Marquardt, 1963]. For small numbers of unknown parameters, the algorithm allows to get a focused medium model.



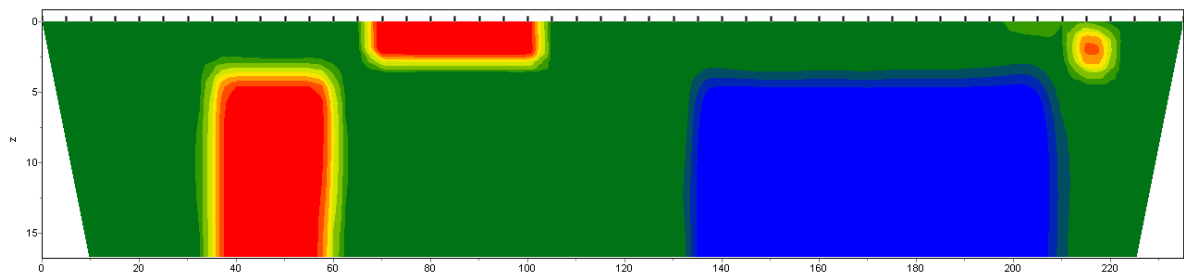
The matrix equation for this inversion version looks as follow:

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

Careless using of this inversion modification may lead to getting unstable result or to increasing root-mean-square deviation, that is algorithm divergence.

It is best to use the *Marquardt* method as refining (to reduce misfit), after carrying out the inversion with algorithms *Smoothness constrained* or *Occam*.

Focused - inversion by least squares method using a smoothing operator, and additional contrast focusing [Portniaguine, 2000]. As a result of this algorithm a piecewise smooth parameters distribution is obtained, that is a model, which consists of blocks with a constant parameter value.



The matrix equation for the inversion version looks as follow:

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

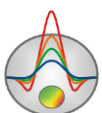
Using this inversion variant requires careful **Threshold** parameter selection. This parameter defines the threshold contrast value for adjacent cells. When reaching this value the cell parameters are not averaged between each other (that is, it is believed that a boundary between cells exists). The dependence of the degree (or weight) of two neighboring cells averaging R_i from the contrast threshold e , and contrast between these cells relooks as follow:

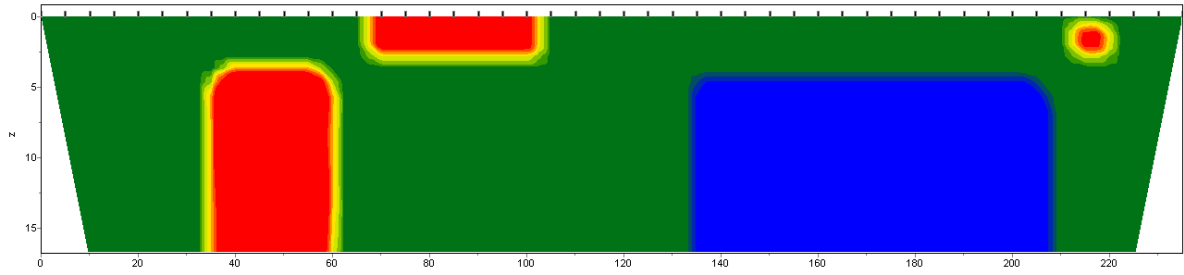
$$R_i = \frac{e^2}{e^2 + r_i^2}.$$

Blocks - selection of separate area parameters differing in the parameter value. Areas with the same parameter values are treated as united blocks.

The matrix equation for this inversion algorithm is identical to the *Marquardt* algorithm:

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

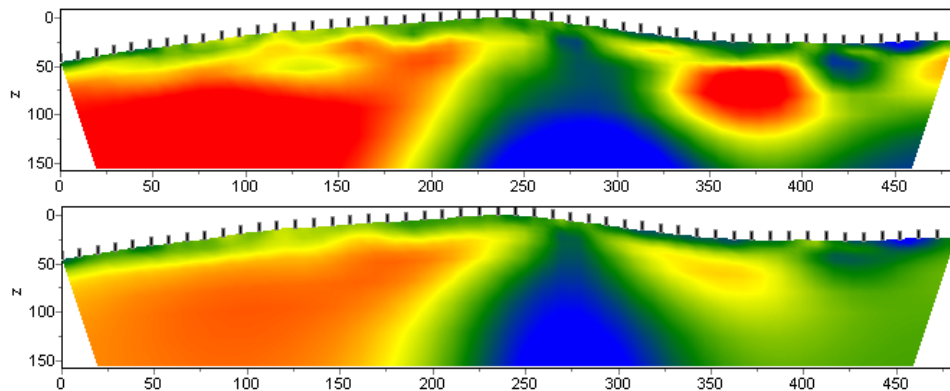




It is better to use the algorithm after results from previous methods were obtained (*Focused* is recommended). To this end, cells should be preliminarily merged using the option Cell summarization. To selecting individual blocks manually should use the model editor and extract specific areas with different parameters. Some blocks will be marked with border during work with this dialog.

Smoothing factor - sets correlation between measurements' misfit minimization and model misfit (for example, smoothness). For data with high noise levels or to get more smooth and stable distribution of parameters, relatively large values of smoothing parameter (0.5 – 2) should be selected; for high quality data values : 0.005 - 0.01 can be used. As a rule, large values of data misfit are obtained for large values of smoothing parameter. It is used in inversion algorithms **Occam** and **Focused**.

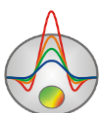
The figures below show results of inversion with **smoothing factor** values 0.01 and 1. The resulting misfit for the first model - 4.5% and for the second 6%.



The area **Stop criteria** contains criteria for inversion termination.

Iterations - if the option is ON, the inversion process is stopped upon reaching the predefined iteration number.

RMS error - if the option is ON, the inversion process is stopped upon reaching the set predefined RMS value.

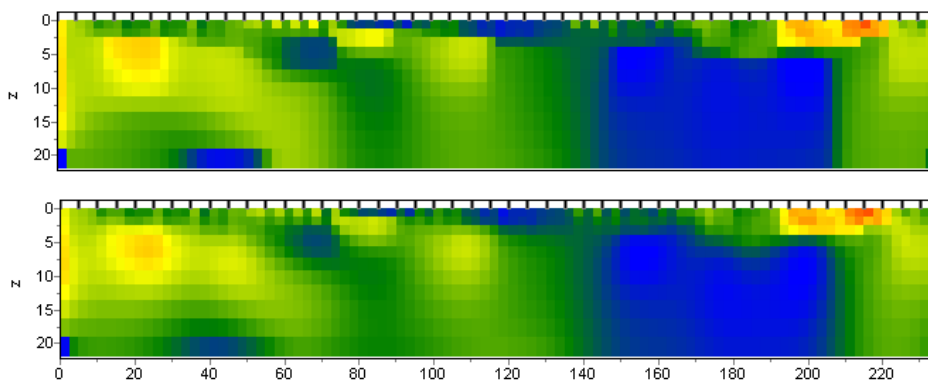


Reduction - if the option is ON, the inversion process is stopped when the residual is increased (for predefined value in %) for two consecutive iterations.

The area **Focusing parameters**

Threshold - sets a threshold contrast value of adjacent cells. When it is reached these cells parameters are not averaged between each other (that is, it is considered that the boundary between the cells exists). Values of the parameter are chosen empirically (0.1-10). Selection of a very small parameter value can lead to a divergence of the algorithm (hence, the value should be increased). Too large parameter values can lead to smooth distributions.

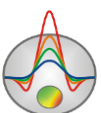
The figures below show the results of the focusing inversion with **Threshold** parameters 0.1 and 10.

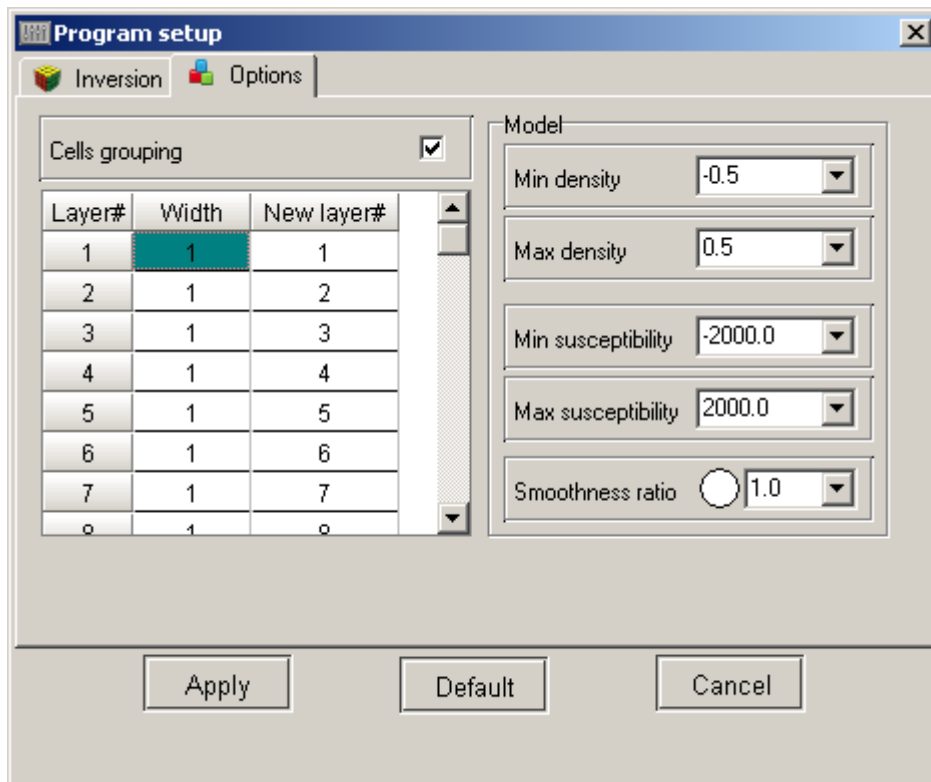


The figures below show the results of the focusing inversion with **Threshold** parameters 0.1 and 10.

Robust weighting scheme - this option should be switched ON if data contains some spikes related to the systematic measurement errors. If amount of invalid data is comparable with amount of valid measurements, the algorithm may not produce feasible results.

The second inlay **Options** is used for adjustment of additional inversion parameters.





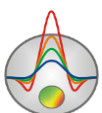
Area **Model**

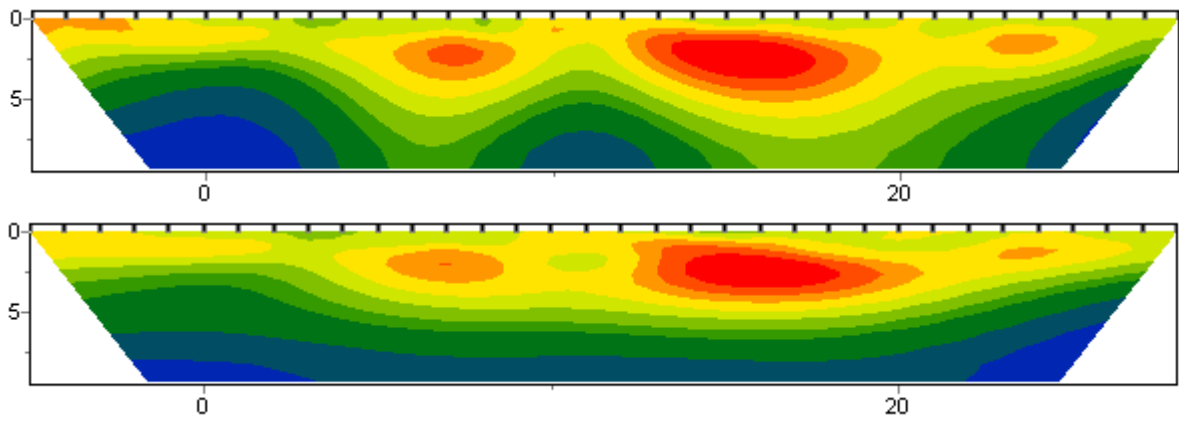
Min susceptibility, Max susceptibility - sets the limits of model parameters variation for inversion. The values of the magnetic susceptibility need to be specified in the GHS system ($n \cdot 10^{-5}$).

Min density, Max density - sets the limits of variation of the model parameters for inversion. Density values need to be specified in g / cm^3 .

Smoothness ratio –defines a ratio of smoothing in horizontal and vertical directions. For horizontally layered structures use value <1 , for vertically layered >1 . Usually, the parameter values vary between 0.2 and 1.

The figures below show the inversion results for the two values of the parameter **Smoothness ratio**: 1 and 0.3.



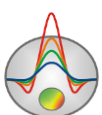


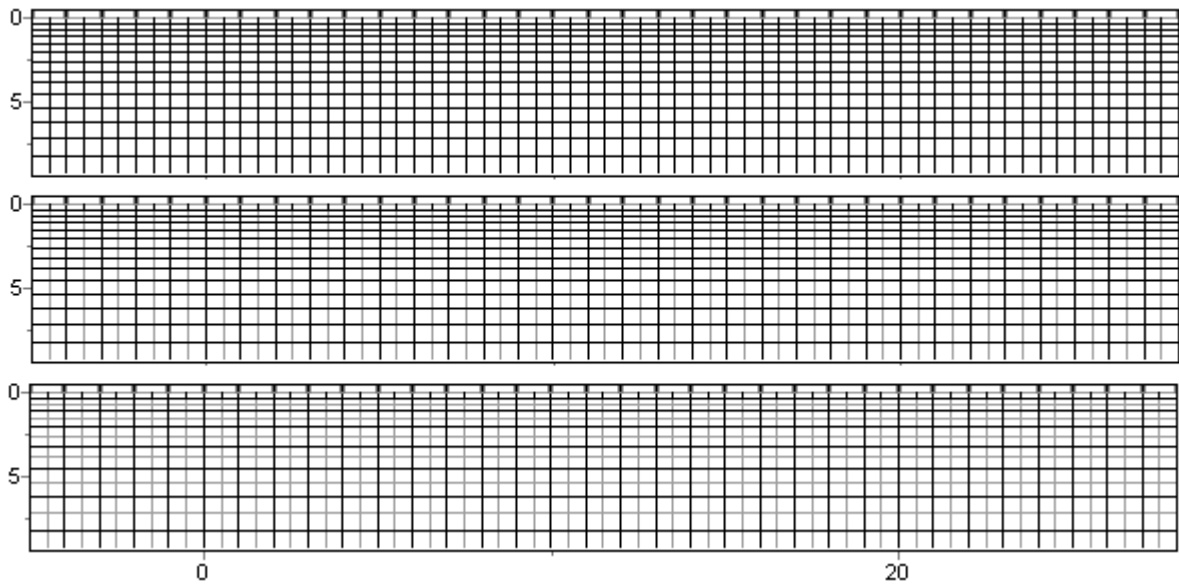
Cell grouping - use this option for large models. The option activates a table that allows to combine adjacent cells and to get smaller number of unknown inversion parameters. When the option is used, number of cells for the forward problem remains the same, but number of cells for the inversion is reduced. Ideally, number of unknown parameters should be close to the number of data measurements.

The table contains three columns: in the first (Layer) number of initial model layer is specified; in the third (ID) number of inversion layer grid is set; in the second (Width) it is necessary to set number of cells (in horizontal direction) contained in each cell of the inversion grid for a current layer. Inversion grid will be displayed in the model editor during its configuration. Double-clicking the left mouse button on a cell in the column Width allows merging cells in the horizontal direction for a current layer, where as clicking the right button does the same for the current and all deeper layers.

Double-clicking the left mouse button on a cell in the ID column allows merging cells in the vertical direction, and clicking the right button - for current and all deeper layers.

Below you can see examples of three inversion grids: in the first, inversion grid corresponds to model, in the second, from the second layer, cells are merged in groups of two, and in the third from the second layer on, cells are grouped by four cells.





Profiling plots (Profile)

This object is used to plot theoretical and experimental parameters.

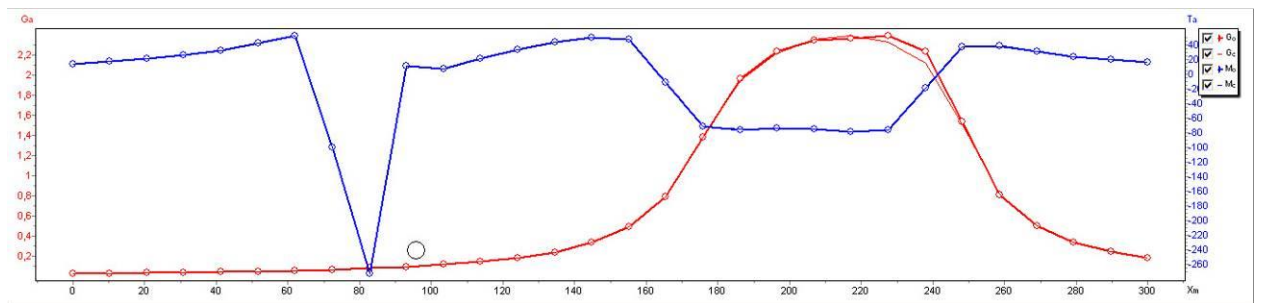
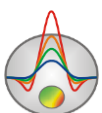


Fig. 3 The Profiling plots window

By default, theoretical curves are shown with solid curves, experimental with solid lines with circles at measurement points. Blue color is used to display theoretical and experimental magnetic plots, red corresponds to gravity data. The color can be changed in the object settings dialog.

Parameters of observed and calculated plots can be set in the plot configuration dialog ([more](#)). Axis parameters may be set in the axis editor (right button click + SHIFT on the axis).

To remove points, use the mouse wheel while holding ALT. Size of the removed region is controlled by the mouse wheel.



Program ZondGM2D also allow working with log data. In this case the window looks as follow:

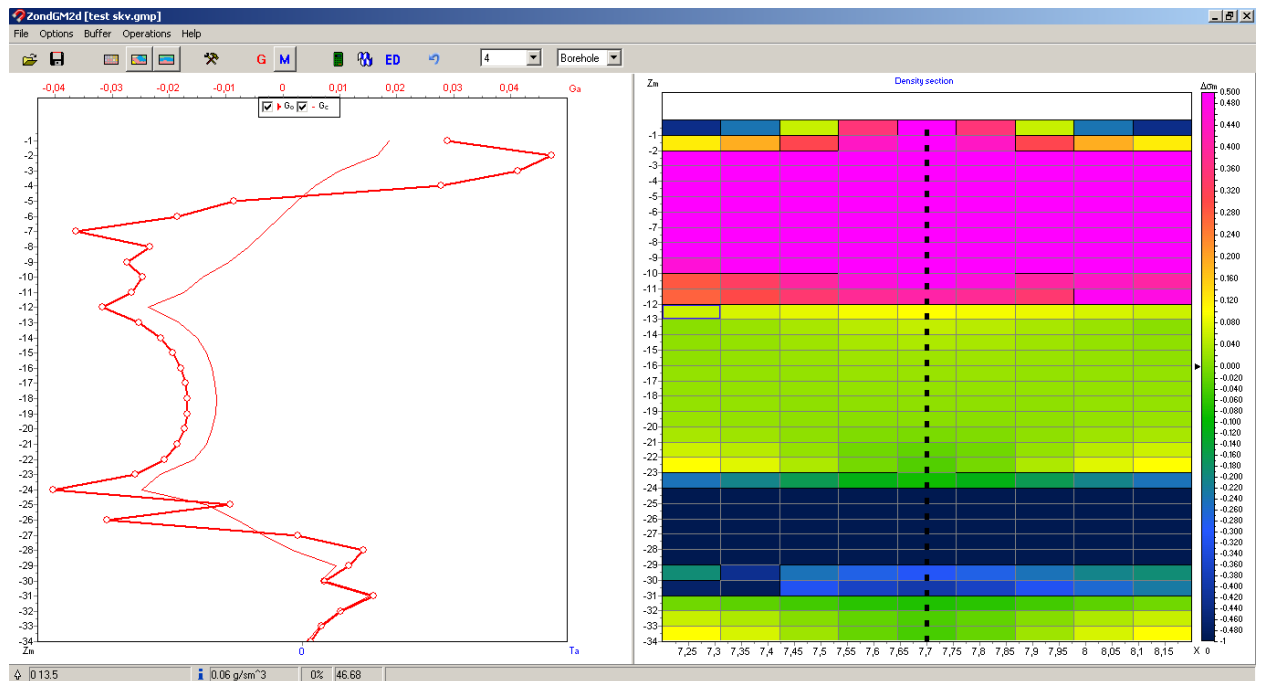


Fig. 4 The main program window when working in the log data mode.

All editing options are also available in this mode.

Model editor

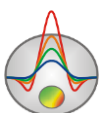
Model editor allows changing parameter values of individual model cells using the mouse. To the right from the editing model area a color scale is located, which links a color to the parameter value. To select the current value, click on the scale using the right mouse button, and its value will be displayed below the color scale.

The size and position of the color scale can be changed by pulling the slider, with the left or right mouse button pressed.

Working with model cells is similar to the pixels editing in a raster image editors. When moving cursor in the model area, on the low status panel of the main program window coordinates and parameters of the active cell in which the cursor is located are displayed. The currently active cell is highlighted by a rectangle.

The selected or fixed cell is marked in the middle by point.

The model can be displayed with cells Options/ Model/Block-section (Fig.5), with a smooth interpolated palette Options/Model/Smooth-section (fig.5B) and as the contour section Options/Model/Contour-section (fig.5C).



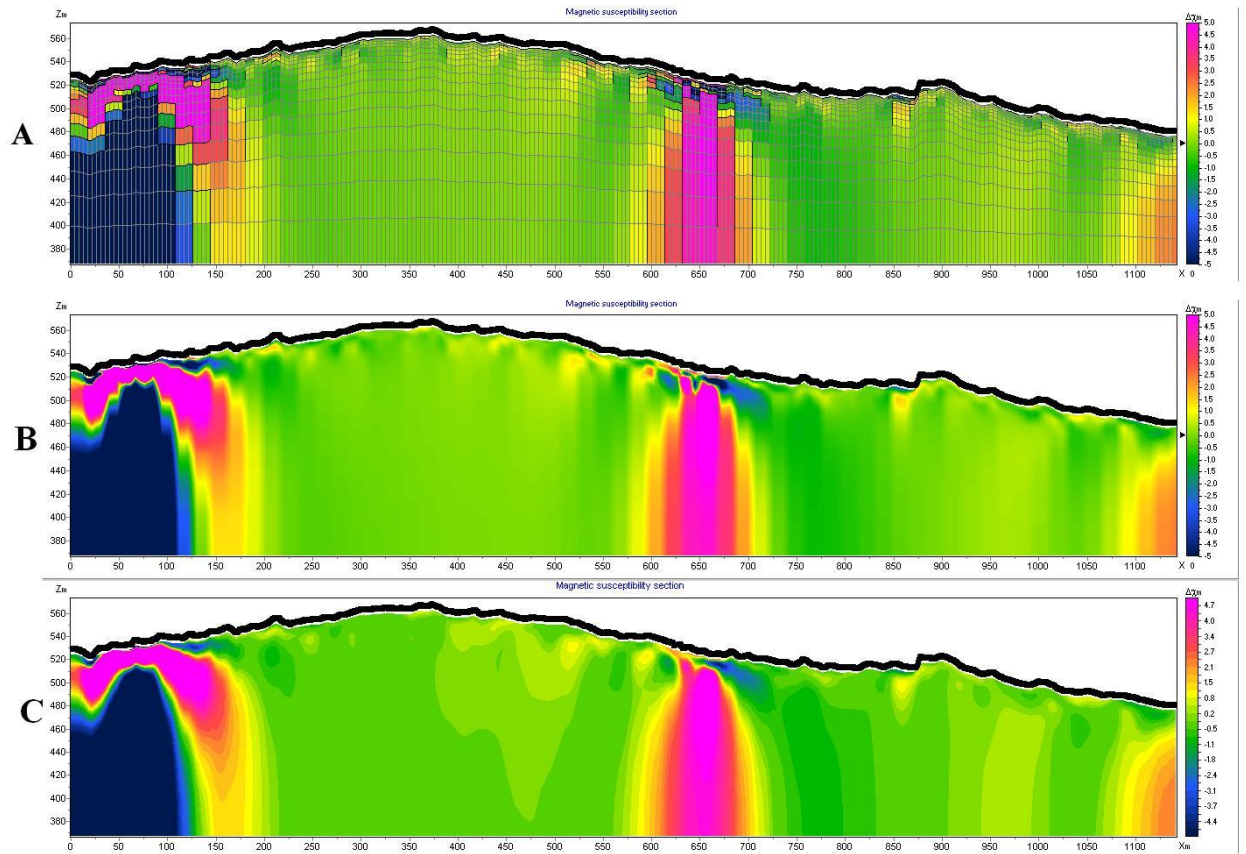
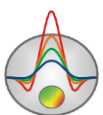
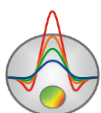


Fig. 5 Variations of the model visualization:
Block-section (A), Smooth-section (B), Contour-section (C).

Double-clicking the mouse in the area of the model editor context menu appears with the following options:



The top area	Display model mesh	Indicates whether to display model mesh.
	Display objects border	Indicates whether to display objects border.
	Display color bar	Indicates whether to display color bar
	Setup	Show dialog of model parameters setup (more).
	Zoom&Scroll	Switch on zoom and scroll mode.
	Print preview	Print the model.
Color bar	Setup	Show dialog of color bar setup.
	Set minimum	Set the minimum value of color bar.
	Set maximum	Set the maximum value of color bar.
	Set incremental factor	Set minimum and maximum values of the color bar relative to background medium value.
	Automatic	Determine the minimum and maximum values of the color bar automatically.
	Log scale	Set logarithmic scale for the color bar.
	Set cursor value	Set the current parameter value.
	Colors as histogram	Display color bar as histogram.
Vertical axis	Set maximum	Set depth value of the bottom layer.
	Redivide	Assign the same layer thicknesses for all model layers (in current scale).
	Thick mesh	Remove every second node of the vertical grid.
	Thin mesh	Add intermediate nodes in the vertical grid.
Horizontal axis	Redivide	Set the same width for cells located between the unique positions of measurement points.
	Thick mesh	Remove every second node of the horizontal grid (if this node does not





		coincide with a measurement position).
	Thin mesh	Add intermediate nodes in the horizontal grid.

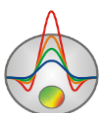
When right button clicking in the model editing area the context menu appears with the options:









Display cell setup	Call cell parameters setup dialog
Cell to cursor value	Use active cell parameter as current value
Edit mode	Turn on editing mode
Selection\Free form selection	Highlight set of cells within editing area using mouse. Area bounds is specified by a user.
Selection\Rectangular selection	Highlight set of cells within editing area using mouse. Field has rectangular form.
Selection\Elliptical selection	Highlight set of cells within edit area using mouse. Field has elliptic form.
Selection\Magic wand	Highlight set of cells within edit area 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 selection.
Mesh options\add column /row	Add new vertical or horizontal boundary. Click at a point where to create new boundary.
Mesh options\remove column /row	Delete selected vertical or horizontal boundary
Mesh options\resize column /row	Change row or column thickness using mouse
Clear model	Clear current model

Additionally, it is possible to use the option **Show modeler tools** in the menu **Options/Model**.

After selecting this option, the editing toolbar appears:

	Switch on model editing mode.
	Switch on model scaling mode.



	Select a set of cells within editing area using mouse. Field has rectangular form.
	Add new vertical or horizontal boundary. Click at a point to creat new boundary there.
	Select a set of cells within editing area using mouse. Field has elliptic form.
	Change row or column thickness using mouse.
	Select a set of cells within edit area using mouse. Field is limited by user.
	Delete selected vertical or horizontal axis
	Select a set of cells within edit area using mouse. Active cell and adjacent cells whose parameters are close to its parameter are select. Proximity is specified in model parameters setup dialog.
	Delete selection

Working with a model

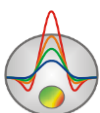
Use mouse to work with model:

Left-button click on a cell changes its parameter to the current value.

Zooming in or dragging some area is performed in **Zoom&Scroll** mode with pressed button. To select and zoom a segment move mouse cursor down and to the right, with left button pressed. To return to the initial scale do the same but with mouse cursor moving up and to the left.

Left click on a cell with SHIFT button pressed increases cell parameter. Right click on the cell with SHIFT button pressed decreases it. Percentage of value change can be set in model parameters setup dialog. If active cell is included in selection, all changes described above are applied to the whole selection.

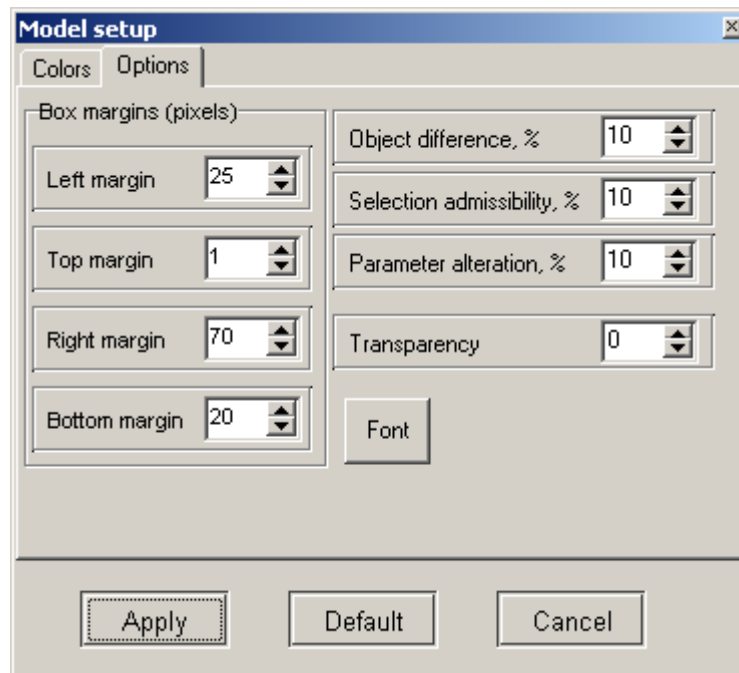
Click on the cell with CTRL button pressed to drag selected set of cells within edit area using mouse. While dragging the selection with left button pressed content of selected cells is copied to new place. While dragging the selection with right button pressed content of selected cells is moved to new domain.



Dialog of model settings

Called by right-clicking in area of model in the drop-down menu option **Setup**.

Options tab



The area **Box margins (pixels)**

Left margin - sets an indent (in pixels) for figure from the left window margin.

Right margin - sets an indent (in pixels) for figure from the right window margin.

Top margin - sets an indent (in pixels) for figure from the top window margin.

Bottom margin - sets an indent (in pixels) for figure from the bottom window margin.

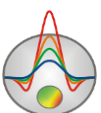
Object difference - sets the maximum value of ratio parameters of adjacent cells, above which a border between them is drawn.

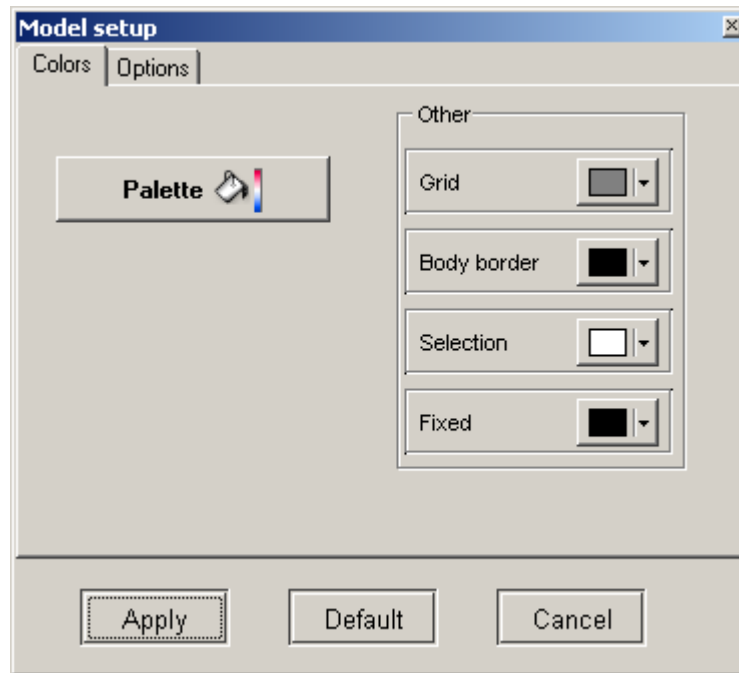
Selection admissibility - sets an acceptable level of parameter differences between adjacent cells, with which cells are a single object and they are being selected together (in the selection mode Magic Wand).

Parameter alteration - determines value of augment to parameters of the selected cells (in percentage terms relative to value of the parameter) when working in mode Edit with the bottom Shift pressed.

Font button calls font settings dialog.

Colors tab





Area **Color bar**

Palette button calls fill settings dialog ([more](#))

Area **Others**

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

Grid - sets grid color.

Selection – sets label color of the selected cell.

Fixed - sets label color of the fixed cell.

Cell parameters setup dialog

Called by right-clicking mouse in cell area, the drop-down menu option **Display cell Setup**.

This dialog serves for selecting cell parameters or highlights it.

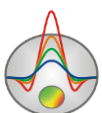
Value – sets cell parameter value.

Fixed – fixes or frees cell parameter.

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


Apply to selected – uses current settings for all selected cells if this function is ON.

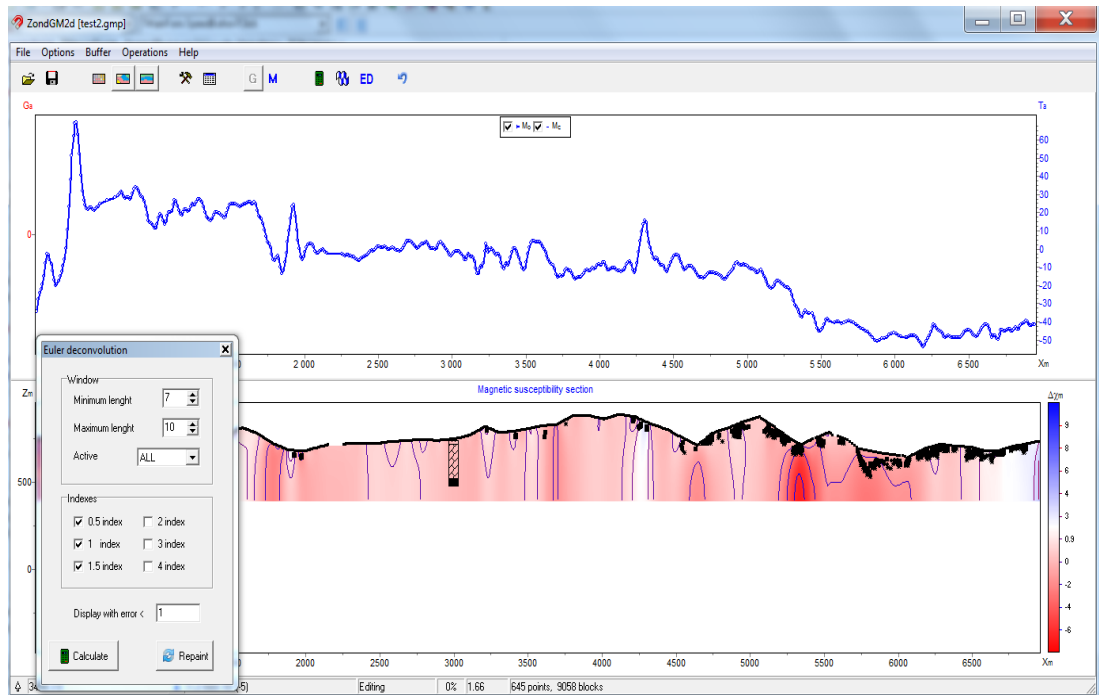
Selection of medium model in inversion (modes of regular cells grid, polygonal and randomly layered medium)



Unlike predecessors, **ZondGM2D** has three types of model medium description:

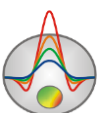
Medium is divided a regular grid of cells

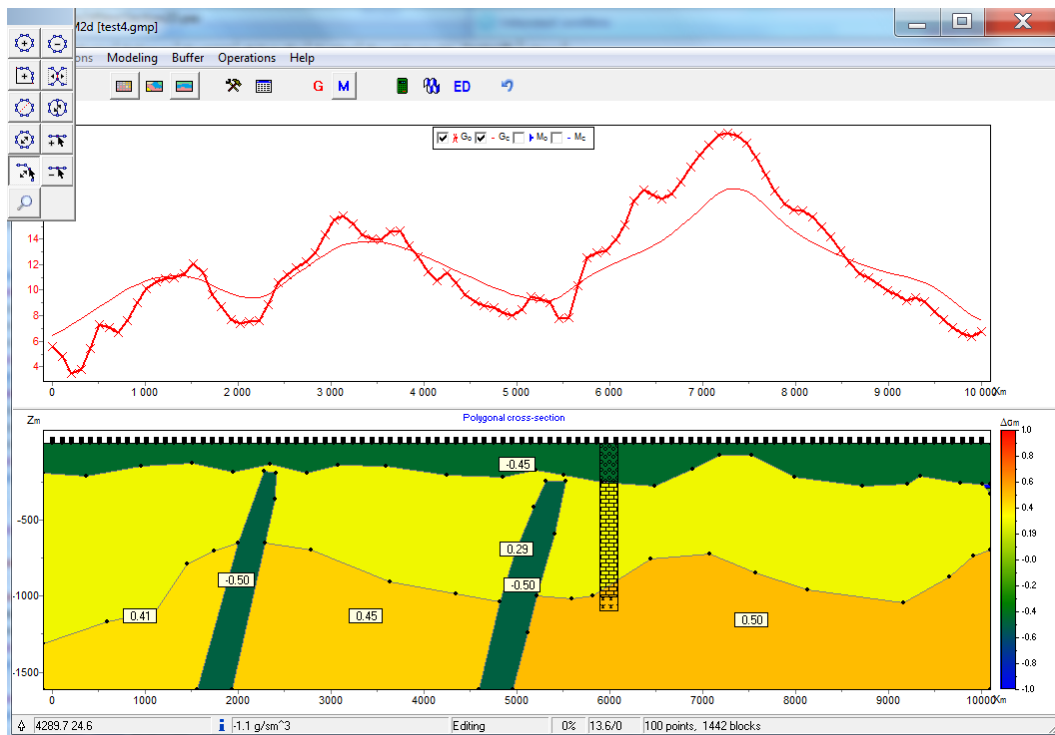
The mode is available when the button  on the toolbar of the main window is pressed. This option is more useful solving inverse problem (automatic rebuild density or magnetic susceptibility sections). Result of inversion is a good initial point for further geological and geophysical interpretation. Program options available in this mode are described in detail in the previous sections.




The medium is represented with a set of bodies with polygonal sections

This mode is available with the button  pressed. This button is located on the toolbar panel of program main window.





Polygonal approach of petrophysical medium description aids more structural approach to the data interpretation. Using this approach two-dimensional section of every body representing a section, is described as a closed polygon. Model construction is done by building a set of polygons with arbitrary geometry, with specified magnetic and density parameters. It is the most convenient option for the final stage of interpretation. **ZondGM2D** has a large set of tools to quickly create models of any complexity. Details about the set of tools are given in the part [Modeling](#).

Solution of the inverse problem for polygonal model variant means an automatic correction of polygon node positions and its petrophysical parameters. When clicking on inversion bottom  pop-up menu appears and it contains following options:

Par inversion - parameters identification (magnetic susceptibility and density) of polygons without changing the polygon nodes position

Z nodes inversion –fittingvertical position of polygon nodes

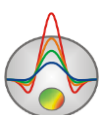
X nodes inversion – fitting horizontal position of polygon nodes


ZX nodes inversion –fitting polygons geometry

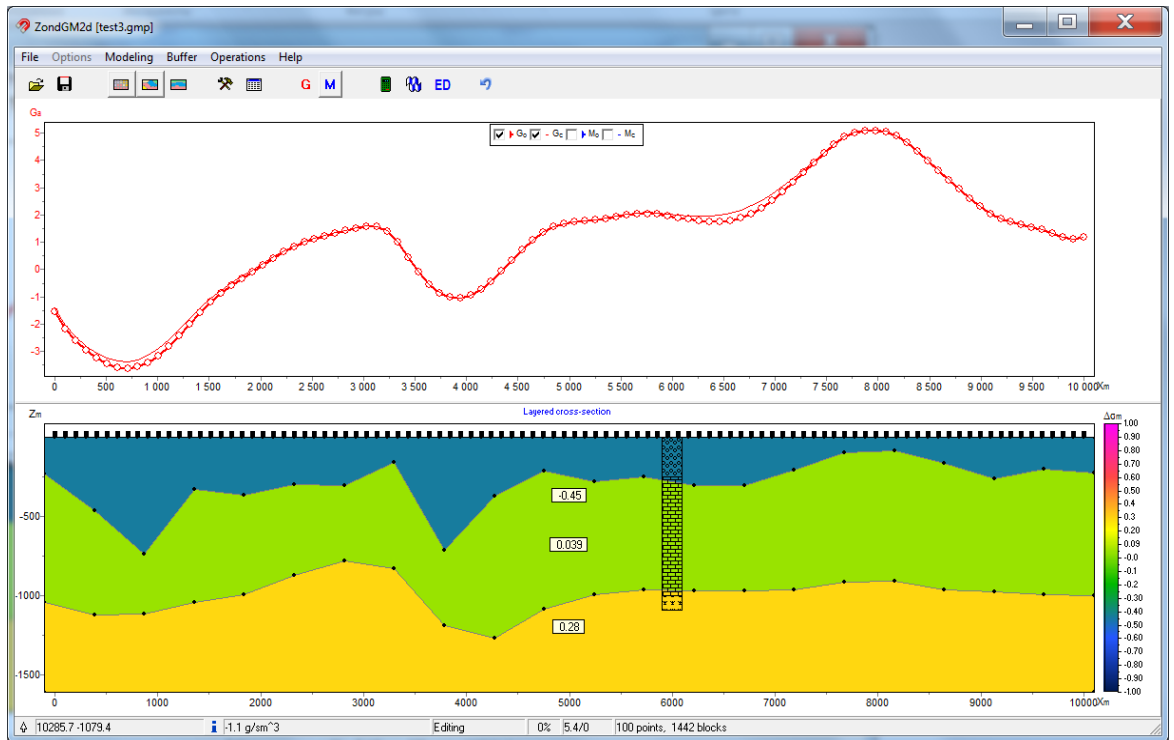
PZX inversion - fitting parameters and geometry of polygons

For this model type joint gravity and magnetic data fitting is available.

Arbitrary layered medium



This mode is available when the button  is pressed on the toolbar of the program main window.



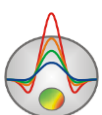
This model type is useful for data interpretation in regions with quasi-layered rocks bedding. Details about the set of options are described in the part [Modeling](#). Before starting to work, it is necessary to set starting model parameters on the tab **Modeling/Layered media setup**, where the following settings are available:

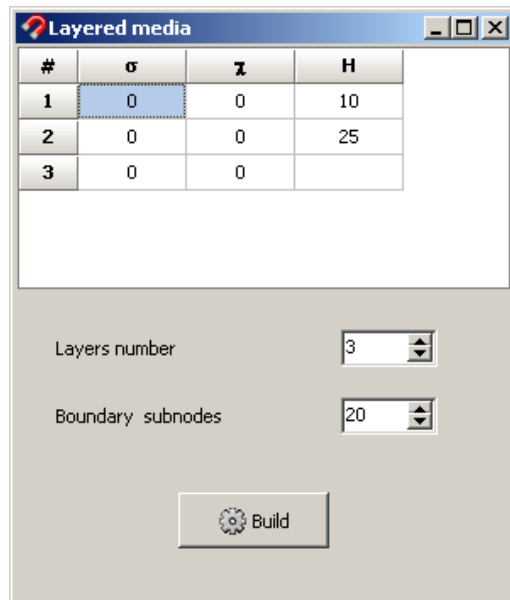
Layered numbers - number of layers


Boundary subnodes - number of nodes on each border

Build – create the specified medium model.

Parameters (density, magnetic susceptibility and thickness) for selected number of layers are set in the table.

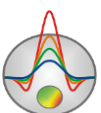




The program solves forward and inverse problems for random layered medium and allows getting densities and magnetic susceptibilities, and also geometry of layer boundaries. When clicking button , a pop-up menu appears and it is similar polygonal modeling mode. For this model type joint gravity and magnetic data fitting is available.

MODELING

Modeling is an important tool prior to the field work. It allows choosing optimal parameters for measuring system in order to solve existing geological problems. Using initial information about the survey target, interpreter can model different geological situations while planning geophysical works.



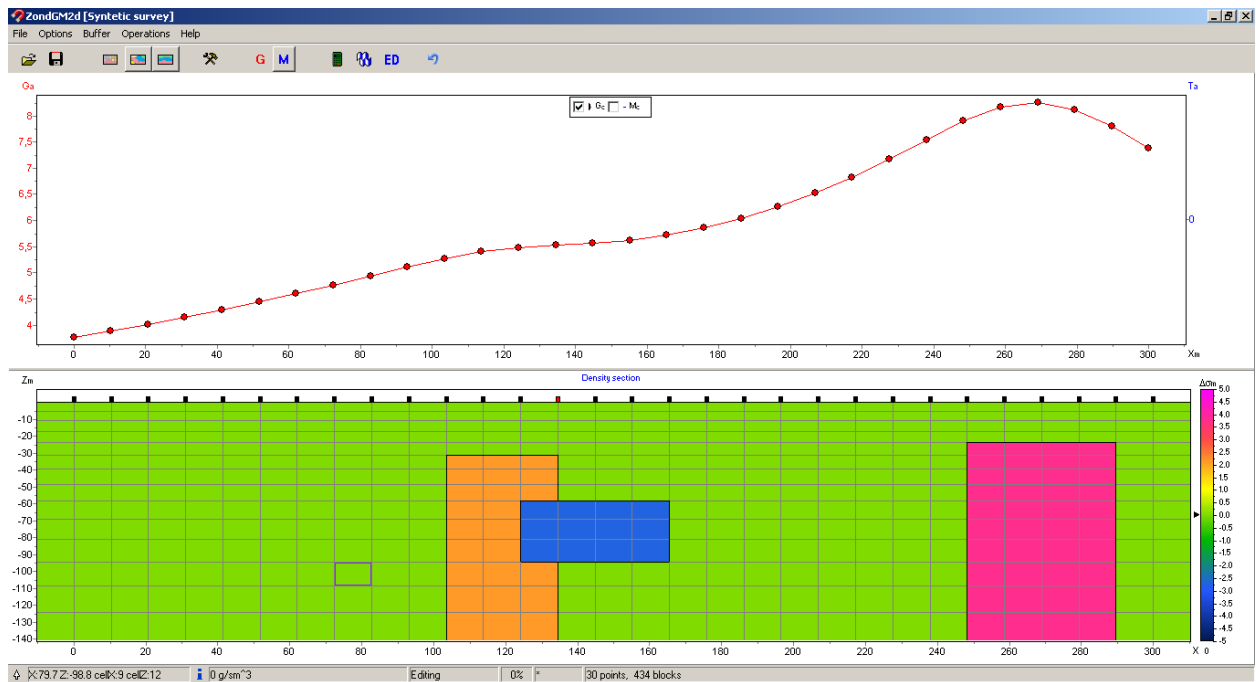


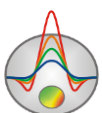
Fig. 6 Active window of the program in modeling mode

On the first stage measuring system parameters should be selected taking into consideration specifics and depth of burial of the object of interest, work conditions (relief, grounding conditions), and equipment capability. Use **File/Create synthetic survey** option to run measuring system parameters setup dialog:

Line's minimum, m - position of the first measurement point on profile


Line's maximum, m - position of the last measurement point on profile

Points number - number of profile points




Gravity and **Magnetic** options allow switching on or off calculation of the magnetic or gravity fields.


Creating of a medium model is done in the model editor - bottom graphic part of the program window during the switched on mode block-section. Read more in the chapter on [Model editor](#).


The main goals of mathematical modeling are assessment of signal level and assessment of selected surveying system resolution. In order to calculate responses from a given medium model (to solve forward problem), press the button  on the toolbar.

To recover the underlying model, that is, to solve the inverse problem for calculated from the given model data, save theoretical signals by selecting a file filter - **Zond project with calculated data**, and then open as observed. To compare restored and initial models use the option **Import model/data** (part «[Importing and Exporting Data](#)»), after the initial model was saved - **Zond model with calculated**.

Modes of polygonal modeling and modeling using arbitrary layered medium

Polygonal modeling mode is activated by button  on the toolbar of the main program window. In this mode, anomalous objects are replaced by a set of polyhedron with an infinite spread in a direction perpendicular to profile. Each polyhedron has its own geometry, magnetic susceptibility and density, for which forward problem is solved. Polygon modeling mode is used in the second stage of data interpretation. After getting a model using one of the inversion algorithms, it is recommended to switch over to polygonal modeling mode and define polygons based on the obtained model and known geology, and perform inversion again.

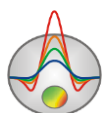
Modeling mode using polygonal-layered medium is activated by pressing button  in the main program menu. This mode is recommended to use for regions with concordant rocks bedding. Before starting to work, it is necessary to specify initial model parameters in the tab Modeling / Layered medium setup.

In order to do inversion in polygonal modeling mode and modeling using a horizontally layered medium model use the button  in the toolbar. After pressing this button following inversion options will be offered:

Parameters inversion – fitting parameters (magnetic susceptibility and/or density) for a given polygons

Z nodes inversion – fitting of vertical positions of polygon nodes

X nodes inversion – fitting of horizontal positions of polygon nodes



ZX nodes inversion – fitting positions of nodes of specified polygons

PZX inversion – fitting parameters (magnetic susceptibility and/or density) and node positions of specified polygons.

After selecting the polygon modeling mode or modeling mode using a horizontally layered medium an additional tab **Modeling** (Fig. 7) will appear in the main program menu. The tab contains the following options:

Layered media setup	Setup layered media for modeling in the arbitrarily layered medium mode
Set model size	Set model boundaries
Show background	Show the current project model as an underlying image
Remove background	Hide the current model
Get value from background	Assign average value of the current model to polygon
Remove polygons	Remove all polygons
Save polygons	Save polygons
Load polygons	Load polygons file
Export to CAD	Export polygons to AutoCAD
Display ColorScale	Display Color Scale
Colors from ColorScale	Use colors for polygons from ColorScale

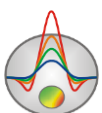
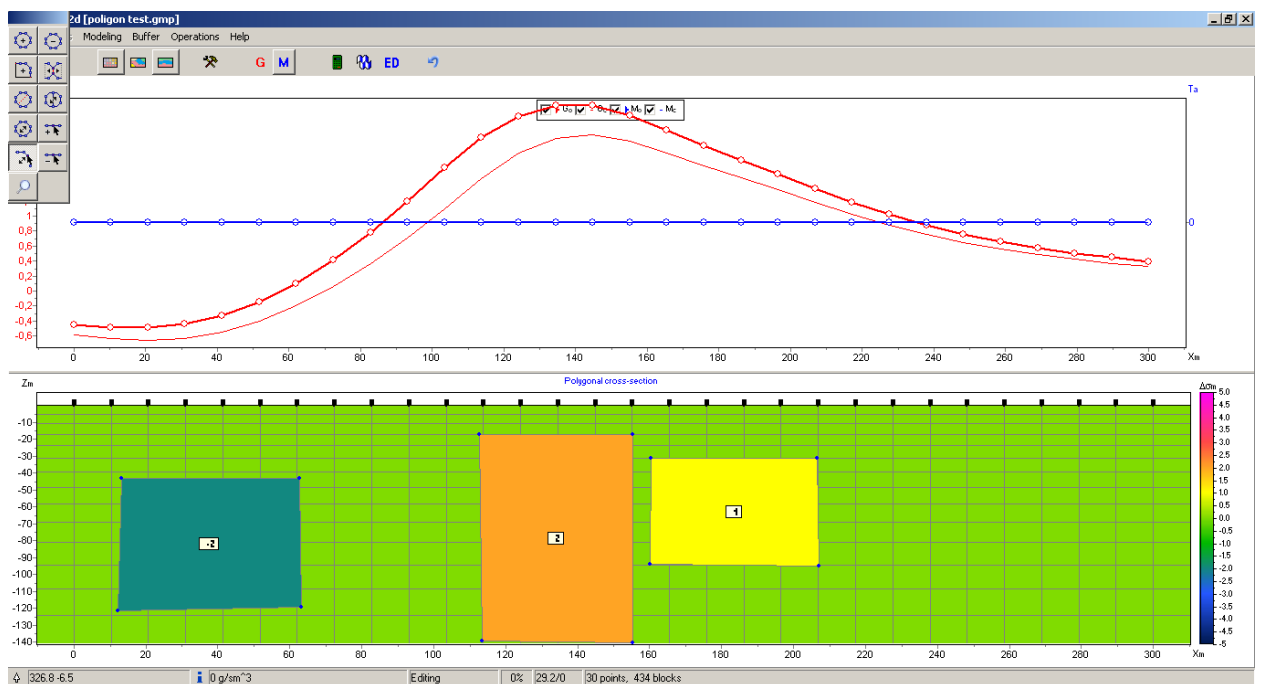







Fig. 7 Main program window in the polygonal modeling mode


Polygons creating and editing is done using the control panel that appears when the button  is clicked.

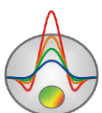
The mode **Add polygon** is called by clicking the button  at the control panel. This mode is intended to create a local polygon. To add a new point to a polygon click the left mouse button. To close the polygon (connecting the last polygon point to the first) click the right mouse button. If it is impossible to create a local polygon (that is any faces intersect, or other object exists in a polygon) program does not allow closing the polygon. When creating a polygon try to not place points too close to each other.

The mode **Delete polygon** is called by clicking the button  at the control panel. This mode is intended to remove a polygon. To delete polygon click the right mouse button on it. Color of a polygon contour changes to red when the cursor is inside.


The mode **Create a joined polygon** is called by clicking the button  at the control panel. This mode is intended to create a polygon (an additional part of a polygon), joined to the existing polygons, model borders or relief, that is to any coupled model area. Coupled area means polygons set and model borders with common faces. To add a new polygon point, click the left mouse button. To close the polygon (connecting the last polygon point to the first, across a joined area border) click the right mouse button. It is necessary that the first and last (closing) polygon points are placed on the outside border (which changes color to red when the cursor is approaching) of a coupled area. If it is impossible to create a local polygon (that is any faces intersect, or other object exists in a polygon) program does not allow user to close the polygon and remove all created points. Note that polygons joined to the left, right and bottom edges of the model have infinite strike in those directions (that is they extend beyond the model).


The mode **Disconnect polygon** is called by clicking the button  at the control panel. This mode is intended to disconnect a polygon from a set of adjoint polygons or model edges. Polygon disconnected from the model edges loses its infinite strike (it will be limited by the model). To disconnect a polygon click the right mouse button on it. Color of the polygon contour changes to red when the cursor is inside. Further, using the button Move polygon, it is possible to move disconnected polygon part from the main polygon.


The mode **Split polygon** is used to create two parts inside a polygon. It is called by clicking the button  on the control panel. The mode is intended to split the polygon into two new connected polygons. The interface is defined by two points at borders or nodes of the





polygon, which is split. To select the first border point click the left mouse button. To select the second point and to split polygon click the right mouse button. If the operation is impossible (that is any faces intersect, or border is outside the polygon) program does not allow user to split the polygon and remove the created border. Color of the borders and points of a polygon changes to red when the cursor is approaching.

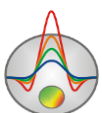
The mode **Move polygon** is called by clicking the button  on the control panel. This mode is intended to move unconnected polygon points. If a polygon has no common, unconnected with other polygons or model borders points, then it is moved completely. To select a polygon being moved click the left mouse button; after that an unconnected polygon part is moved with the cursor. To fix new position of the polygon click the right mouse button. If the operation is impossible (that is any borders intersect, or a polygon exists in other polygon) program does not allow user to move the polygon and return it to the original position. Color of the polygon contour changes to red when the cursor is inside.

The mode **Move connected polygons** is called by clicking the button  on the control panel. This mode is intended to move a polygon and all connected with it. To select a polygon being moved, click the left mouse button; after that the connected area moves with the cursor. To fix new position of polygons click the right mouse button. If the operation is impossible (that is any borders intersect, or a polygon exists in other polygon) program does not allow user to move the polygons and return them to original positions. Color of the polygon contour changes to red when the cursor is inside.

The mode **Add point** is called by clicking the button  on the control panel. This mode is intended to add a new point on a border of an existing polygon. To add a polygon point, click the right mouse button on its border. Color of the polygon borders change to red when the cursor is approaching.

The mode **Remove point** is called by clicking the button  on the control panel. This mode is intended to remove new point of existing polygon. To remove a polygon point, click the right mouse button on it. The operation is impossible in the next cases: borders intersect, a polygon is in other one or number of polygon points is less than three. Color of polygon points changes to red when the cursor is approaching.

The mode **Merge points** is called by clicking the button  on the control panel. This mode is intended to merge two points into one by joining points to the border of another polygon, or to the model edges. Selection of the first merging point.



Polygon parameters setup dialog

This dialog is used to adjust different polygon parameters. Double click on a polygon to call it.

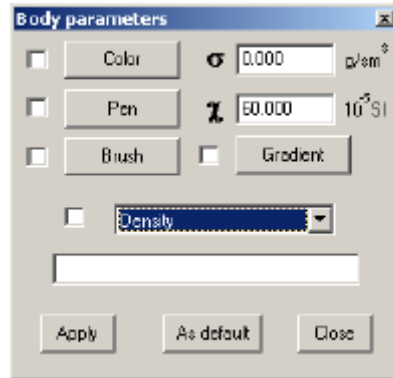


Fig.8 Polygon parameters setup dialog

Color – calls polygon filling color setup dialog. If this function is on, specified color is used in all polygons of the model.

Pen – calls polygon border parameters setup dialog. If this function is on, specified parameters are used in all polygons of the model.

Brush – calls polygon filling setup dialog. If this function is on, specified parameters are used in all polygons of the model.

Gradient – calls polygon gradient filling setup dialog. If this function is on, specified parameters are used in all polygons of the model.

σ – sets polygon density value.

χ – sets polygon magnetic susceptibility value.

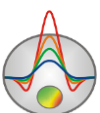
The following option specifies type of label displayed on a polygon. If this function is on, specified type is used in all polygons of the model.

Value *None* – there is no label on a polygon.

Value *Density* – polygon density value is displayed on a polygon.

Value *Susceptibility* – polygon magnetic susceptibility value is displayed on polygon.

Value *Density & Susceptibility* – polygon density and magnetic susceptibility values are displayed on a polygon.

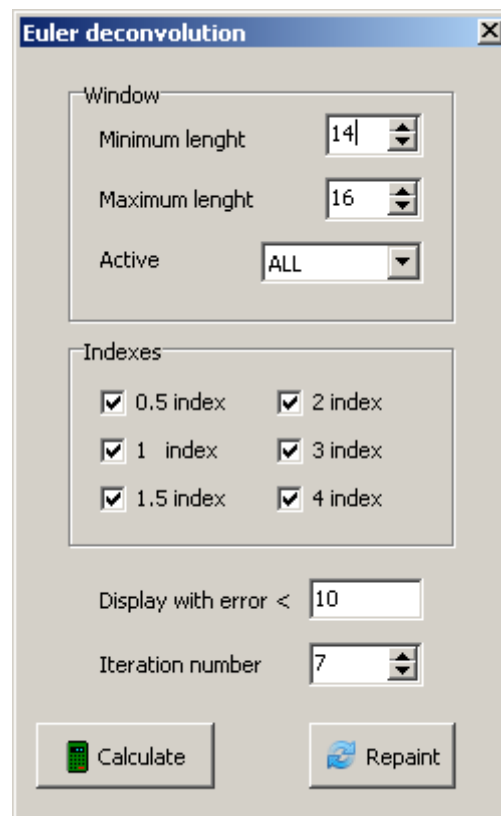


Value *User text* – value of the following field is displayed on polygon. The following option specifies type of label displayed on a polygon. If this function is on, specified type is used in all polygons of the model.

ADVANCED PROGRAM OPTIONS

Euler deconvolution

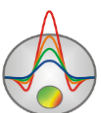
In addition to inversion, Euler deconvolution is available in ZondGM2D. The option allows getting distribution of magnetic and gravity sources (special points). This option is activated by clicking a button **ED** on the toolbar of the main program window, then the following dialog box appears:



The area **Window** is used for window calculation parameters setup. Windows Minimum and Maximum length are intended to set minimum and maximum width of the window, respectively. The field Active allows selecting windows to be displayed on the model.

The area **Index** allows selecting value of structural index, which points will be displayed. Structural indexes related to degree of field attenuation from a studied type source. For example, for a linear mass $N = 1$, for points mass and dipole lines $N = 2$, for dipoles $N = 3$. The value $N =$

50



0.5 in magnetic survey occurs in a model with an inclined step model object, and also in gravity survey for models consisting of thin layers, including dikes and sills. Points corresponding to different indexes will be displayed with different icons.

Display with error – show fitting error

Iteration number – the number of iterations

Button **Calculate** – to calculate position of singular points with selected parameters

Button **Repaint** – update image

Downward continuation

Field downward continuation is a type of transformation intended for extraction of high-frequency and low-frequency field components. Downward continuation of the measured field - Downward continuation/Standard, and vertical derivative of a field - Downward continuation/Normed are available in the program. When running downward continuation of measured field a low-frequency component has the greatest impact, when continuing a derivative high frequency has the greatest impact. (fig. 9)

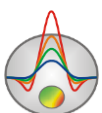
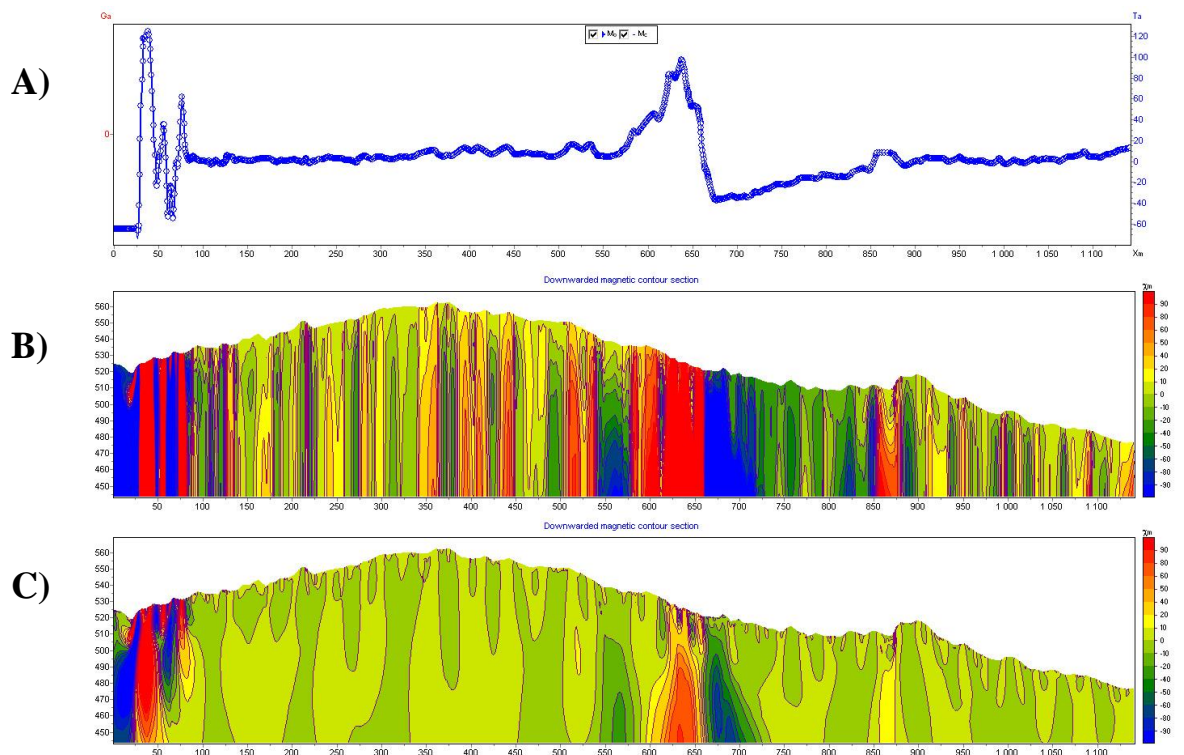
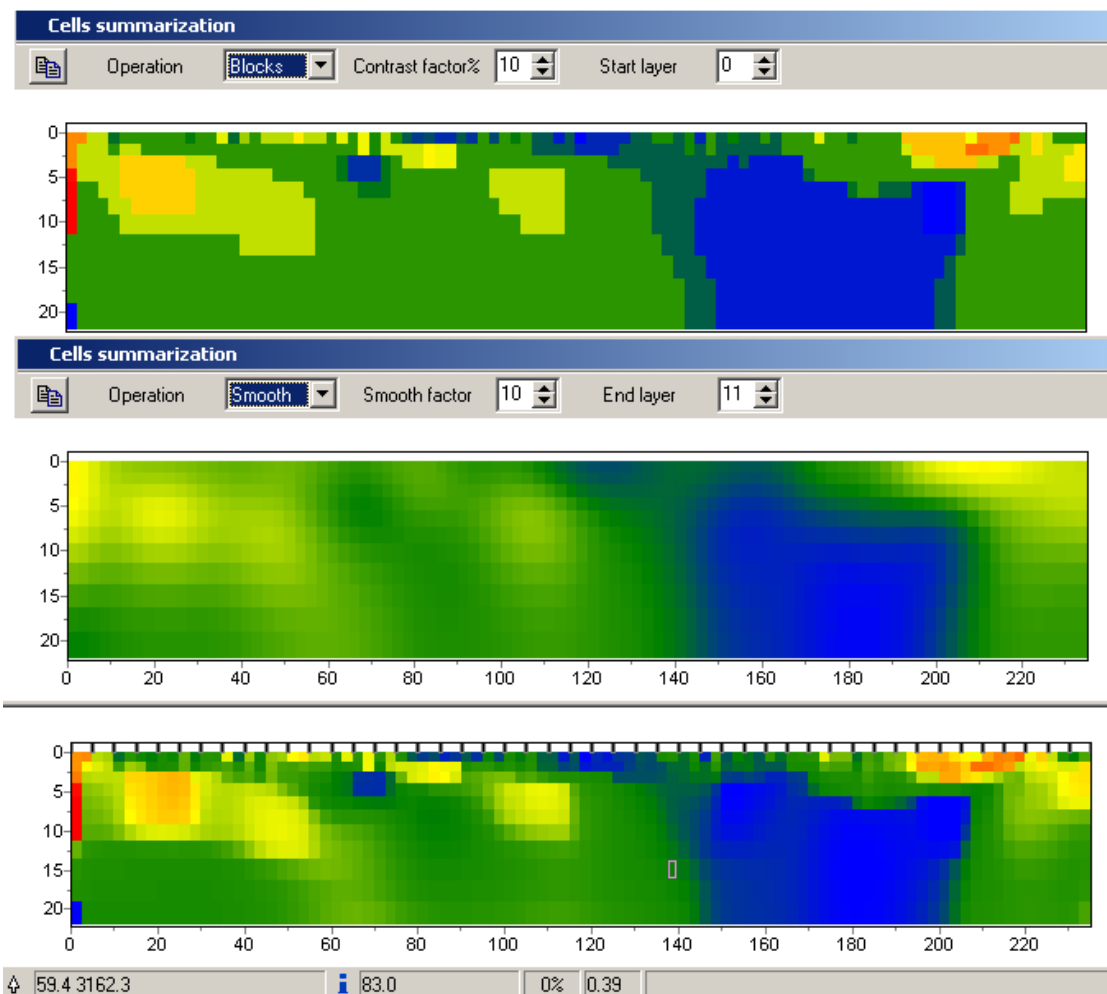


Fig. 9 a) Plot of Δg , b) the result of downward continuation of the measured field, c) Downward continuation of the vertical derivative.

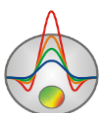
Cell summarization dialog


Cell summarization option allows smoothing or coarsening (divide into blocks) current model. Block model can be used in the inversion type Blocks. In this case, parameters for each block are selected. Before dividing into blocks is better to use the focusing inversion.



When using the Blocks mode, depending on a contrast parameter (Contrast factor), cells aggregation with similar parameters in area with a constant value are done. Start layers option specifies a layer index from which the operation is applied.

In the Smooth mode, depending on a smoothing factor (Smooth factor), model cell parameters are averaged. End layers option specifies the index of the layer up to which this operation is applied.

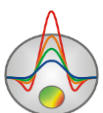
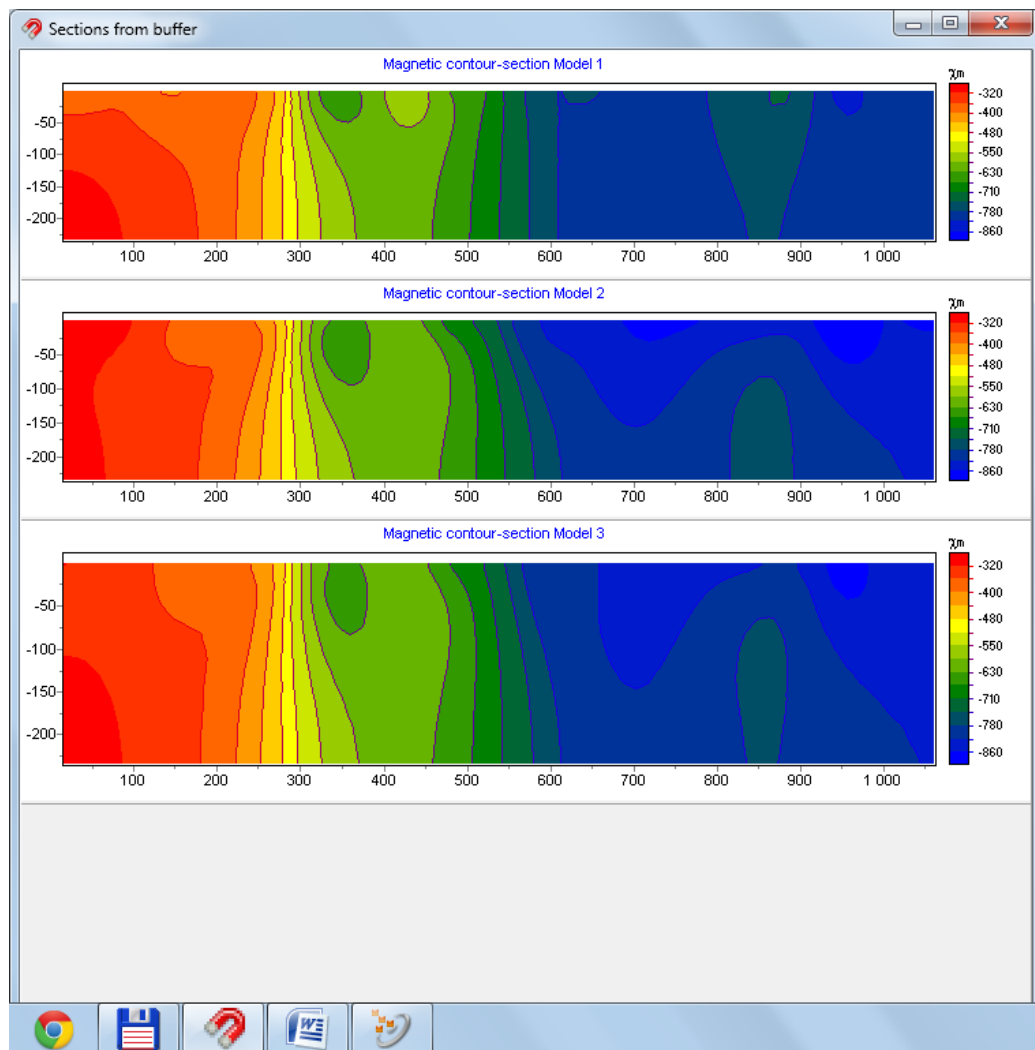


Button  copies obtained model to the model editor.

Option «Buffer» of the main menu

Buffer option of the program main menu allows comparing models, obtained with different inversion parameters and program settings (when using additional geological data, filter data, etc.). When using this option, all obtained models are displayed in using the same color palette, and scales.

After getting one of models, click on the Buffer tab of the main menu and select Model 1. The program offers to set name of a current model, in which different parameters can be stored. Then, the current model will be saved, and a new inversion can be done with different program settings and save as a Model 2. It is possible to save up to five different models. To open the window of saved models, select Buffer/Open in the program main menu.



Geological sections editor

Geological editor module allows promptly building a geological section based on the results of profile interpretation obtained in the program.

The results of the geophysical interpretation serve as an underlying image, over which geological model is built. During a model creation local objects and layers are selected. Geological texture selected by an interpreter is drawn on these local objects and layers. The module allows also displaying borehole data, which simplifies the process of model construction.

Thus, the main goal of the module is a rapid construction of geological sections using geophysical results and its export in CAD systems.

Before starting, it is necessary to select carefully a section type and its graphic settings. The best variant is to display a section as isolines.

Next, **Geological editor** is run and actual work with a section begins. Bodies and the layers are set as closed and unclosed polygons, color and texture are set to them.

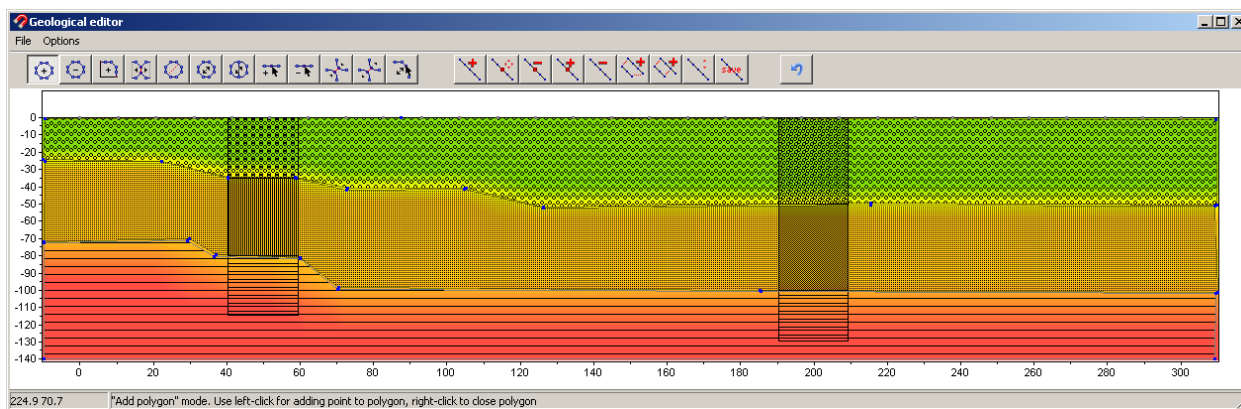


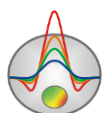



Fig. 10 **Geological Editor** window


Buttons for editing polygons and lines are located on the **Geological editor** toolbar.


The mode **Add polygon** is called by clicking the button  at the control panel. This mode is intended to create a local polygon. To add a new point to a polygon, click the left mouse button. To close the polygon (connecting the last polygon point to the first), click the right mouse button. If it is impossible to create a local polygon (that is any faces intersect, or other object exists in a polygon) program does not allow closing the polygon. When creating a polygon try to not place points too close to each other.


The mode **Delete polygon** is called by clicking the button  at the control panel. This mode is intended to remove a polygon. To delete polygon click the right mouse button on it. Color of a polygon contour changes to red when the cursor is inside.

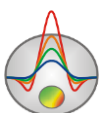



The mode **Create a joined polygon** is called by clicking the button  at the control panel. This mode is intended to create a polygon (an additional part of a polygon), joined to the existing polygons, model borders or relief, that is to any coupled model area. Coupled area means polygons set and model borders with common faces. To add a new polygon point, click the left mouse button. To close the polygon (connecting the last polygon point to the first, across a joined area border), click the right mouse button. It is necessary that the first and last (closing) polygon points are placed on the outside border (which changes color to red when the cursor is approaching) of a coupled area. If it is impossible to create a local polygon (that is any faces intersect, or other object exists in a polygon) program does not allow user to close the polygon and remove all created points. Note that polygons joined to the left, right and bottom edges of the model have infinite strike in those directions (that is they extend beyond the model).


The mode **Disconnect polygon** is called by clicking the button  at the control panel. This mode is intended to disconnect a polygon from a set of adjoint polygons or model edges. Polygon disconnected from the model edges loses its infinite strike (it will be limited by the model). To disconnect a polygon click the right mouse button on it. Color of the polygon contour changes to red when the cursor is inside. Further, using the button **Move polygon**, it is possible to move disconnected polygon part from the main polygon.


The mode **Split polygon** is intended to create two parts inside a polygon. It is called by clicking the button  on the control panel. The mode is intended to split the polygon into two new connected polygons. The interface is defined by two points at borders or nodes of the polygon, which is split. To select the first border point, click the left mouse button. To select the second point and to split polygon, click the right mouse button. If the operation is impossible (that is any faces intersect, or border is outside the polygon) program does not allow user to split the polygon and remove the created border. Color of the borders and points of a polygon changes to red when the cursor is approaching.


The mode **Move polygon** is called by clicking the button  on the control panel. This mode is intended to move unconnected polygon points. If a polygon has no common, unconnected with other polygons or model borders points, then it is moved completely. To select a polygon being moved, click the left mouse button; after that an unconnected polygon part is moved with the cursor. To fix new position of the polygon click the right mouse button. If the operation is impossible (that is any borders intersect, or a polygon exists in other polygon) program does not allow user to move the polygon and return it to the original position. Color of the polygon contour changes to red when the cursor is inside.





The mode **Move connected polygons** is called by clicking the button  on the control panel. This mode is intended to move a polygon and all connected with it. To select a polygon being moved, click the left mouse button; after that the connected area moves with the cursor. To fix new position of polygons click the right mouse button. If the operation is impossible (that is any borders intersect, or a polygon exists in other polygon) program does not allow user to move the polygons and return them to original positions. Color of the polygon contour changes to red when the cursor is inside.

The mode **Add point** is called by clicking the button  on the control panel. This mode is intended to add a new point on a border of an existing polygon. To add a polygon point, click the right mouse button on its border. Color of the polygon borders change to red when the cursor is approaching.

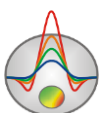
The mode **Remove point** is called by clicking the button  on the control panel. This mode is intended to remove new point of existing polygon. To remove a polygon point, click the right mouse button on it. The operation is impossible in the next cases: borders intersect, a polygon is in other one or number of polygon points is less than three. Color of polygon points changes to red when the cursor is approaching.

The mode **Merge points** is called by clicking the button  on the control panel. This mode is intended to merge two points into one by joining points to the border of another polygon, or to the model edges. Selection of the first merging point

The mode **Disconnect points** is called by clicking the button  on the control panel. This mode is intended to disconnect previously merged points. To disconnect points of connected polygon click the right mouse button on it. As a result of this operation, instead of one connected point a set of disconnected points appears. Each point belongs to its own polygon. Color of polygon points changes to red when the cursor is approaching.

The mode **Move point** is called by clicking the button  on the control panel. This mode is intended to move polygon point. To select moving point, click the left mouse button; after that polygon point moves with the cursor. To fix position of the new point click the right mouse button. If the operation is not possible (that is faces intersect) program does not allow you to move a point and returns it to its original position. Points located on model boundaries are moved only along appropriate edges. Color of polygon points changes to red when the cursor is approaching.

Polygon parameters setup dialog



This dialog is used to adjust different polygon parameters. Double click on necessary polygon to run it.

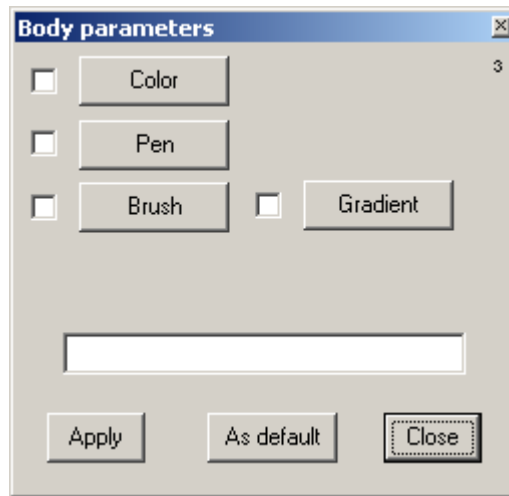


Fig. 11 Polygon parameters setup dialog

Color – calls polygon filling color setup dialog. If this function is on, specified color is used in all polygons of the model.









Pen – calls polygon border parameters setup dialog. If this function is on, specified parameters are used in all polygons of the model.

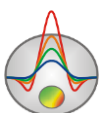
Brush – calls polygon filling setup dialog. If this function is on, specified parameters are used in all polygons of the model.

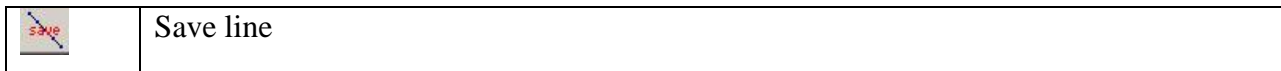
Gradient – calls polygon gradient filling setup dialog.

Signature of the field located below will be displayed on the polygon.

The buttons for creating and editing lines:

	Add line
	Move node
	Remove node
	Add node
	Remove line
	Create polygon of two lines
	Create polygon of two lines
	Move line





Menu **Options** of **Geological editor** window contains the following functions:

Automatic scaling – switch on auto zoom setup

Model setup - call dialog of displaying model area setup

Load borehole data - load lithology data

Remove borehole data - remove lithology data

Output settings - call dialog of exported image parameters setup

Export to DXF - export model to DXF file.

Working with areal data and 3D visualization

To set some profile lines select in the main menu **Options/Set lines survey**. Then a dialog box **Line settings** (Fig. 12) appears where measurement positions of an area are displayed.

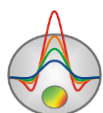
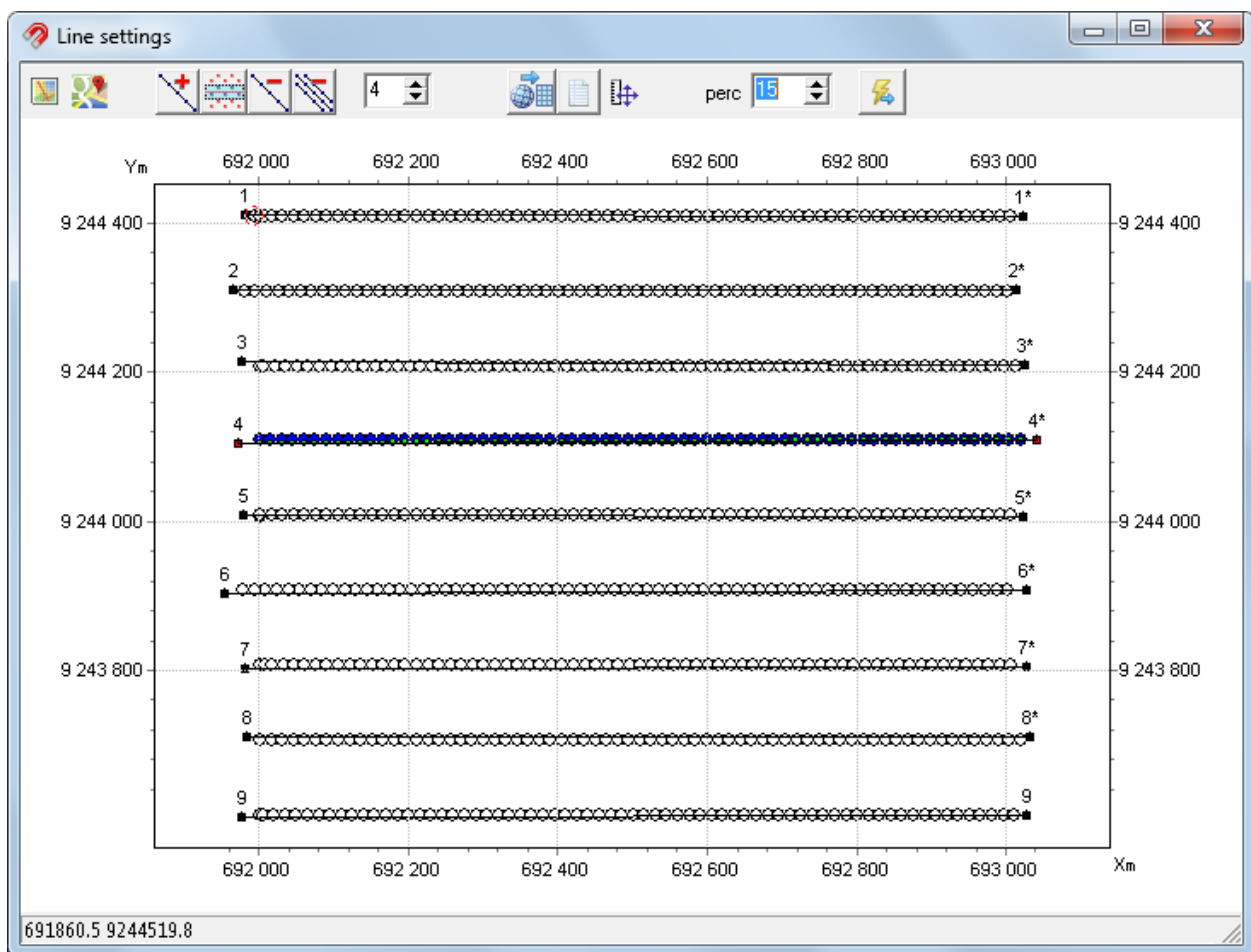












Fig. 12 Dialog box Line settings to work with areal data


The main dialog panel contains the following functions:

	Map - load and display raster map file Topography - display the relief isolines Gravity - display isolines of measured gravity field Magnetic - display isolines of measured magnetic field
	Load Google map
	Add profile line. To set profile lines click the left mouse button; to set the last point click the right mouse button
	Remove all profiles.
	Include measurement points in profile automatically. The points contained in a rectangular area around the specified line.
	Remove current profile
	Convert from geographic coordinates to rectangle. If it is known a priori geographic coordinates (latitude and longitude) were in the file, it is necessary to convert them to rectangular using this button.
	Set coordinates in a table
	Select equal axes image scale or maximum window area filling.
perc	Set size of automatic selection measured points area in a profile
	Go to the data inversion mode for selected profiles system

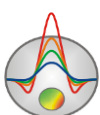
The program allows setting several profile lines simultaneously. After setting a profile setting and selecting points along it, all included profile points will be displayed in blue. It is also possible to exclude/include a point in the profile by pressing the left mouse button. If profile line does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

To display and edit coordinates of the measuring points, click the right mouse button on an interesting point. The coordinates, which can be edited, will be displayed in the appeared window show.

After profile setting it is necessary to click the button to call the data interpretation mode, and then the main program window appears. To switch between profiles use **Active line index**

window  on the toolbar of the main program window.

Option **Graphics map** (Fig.13) allows displaying graphs plan of the selected parameter (magnetic or gravity field) for all profiles. To change scales of plots, click the left mouse button in the graph area, and use the scroll wheel. It is also possible to switch on/off displaying of each profile in the legend window on the right. To select one plot and switch off other, press Shift +



left-click on the legend. To move to other plots use scroll wheel. To setup the axes parameter, click the right mouse button while pressing Shift (more).

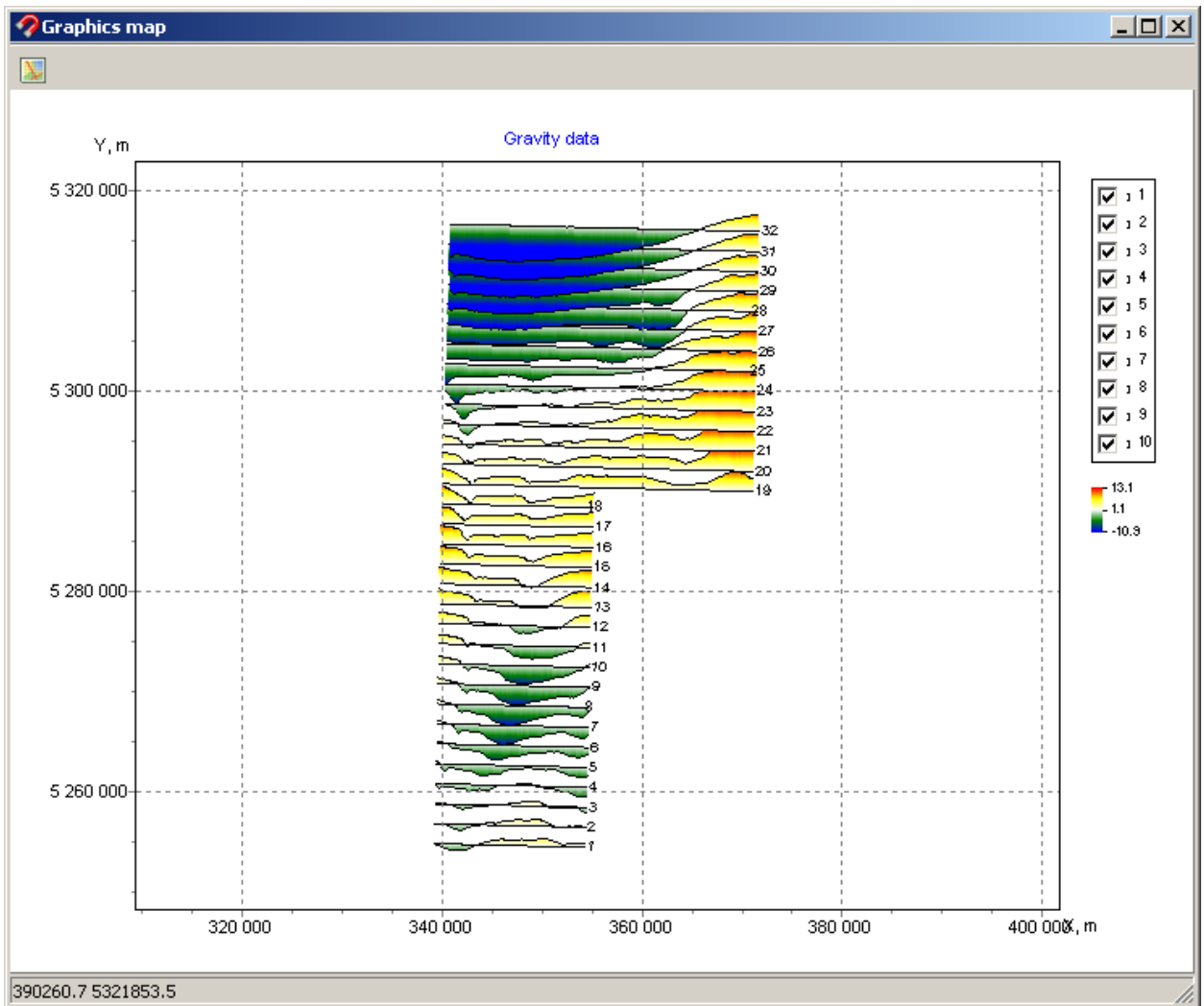
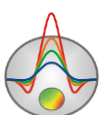






Fig.13 **Graphics map** dialog window

Option 3D sections plot (Options/Modules/3D sections plot)


This option is used for 3D visualization of the profiles interpretation results. After selecting this option, a window **3D sections viewer** appears. Window toolbar contains the following buttons:

	Show using block model
	Show using horizontally layered model
	Show using the mode of polygonal modeling
	Print preview
	Call dialog of 3D model parameters



	Setup axes parameters (more)
	Turn 3D model
	Display horizontal plane. Plan depth from the surface is set in kilometers in the right window <input type="text" value="5.0"/> <input type="button" value="↑"/> <input type="button" value="↓"/> <input type="button" value="✓"/>
	Set the same scales for all axes. The appeared window allows setting ratio of the scales for each axis.
<input type="text" value="Gravity"/>	Set data type for displaying.

The window **3D section viewer** contains three tabs:

Tab **Lines** (Fig. 14) can be used to edit profiles coordinates of the beginning and the end and also to set profiles, which will be displayed in the 3D model. A table containing names, coordinates of the end points of profiles is on the left in the window. To display a profile on a 3D model set a tick in the last column of the table (V - visible). Profiles plan is displayed on the right. The active profile is displayed in red. There is an able to edit axes properties. To do it, click the right mouse button while pressing the Shift key or button  on the toolbar. More information about axes settings is in the part [Axes editor](#).

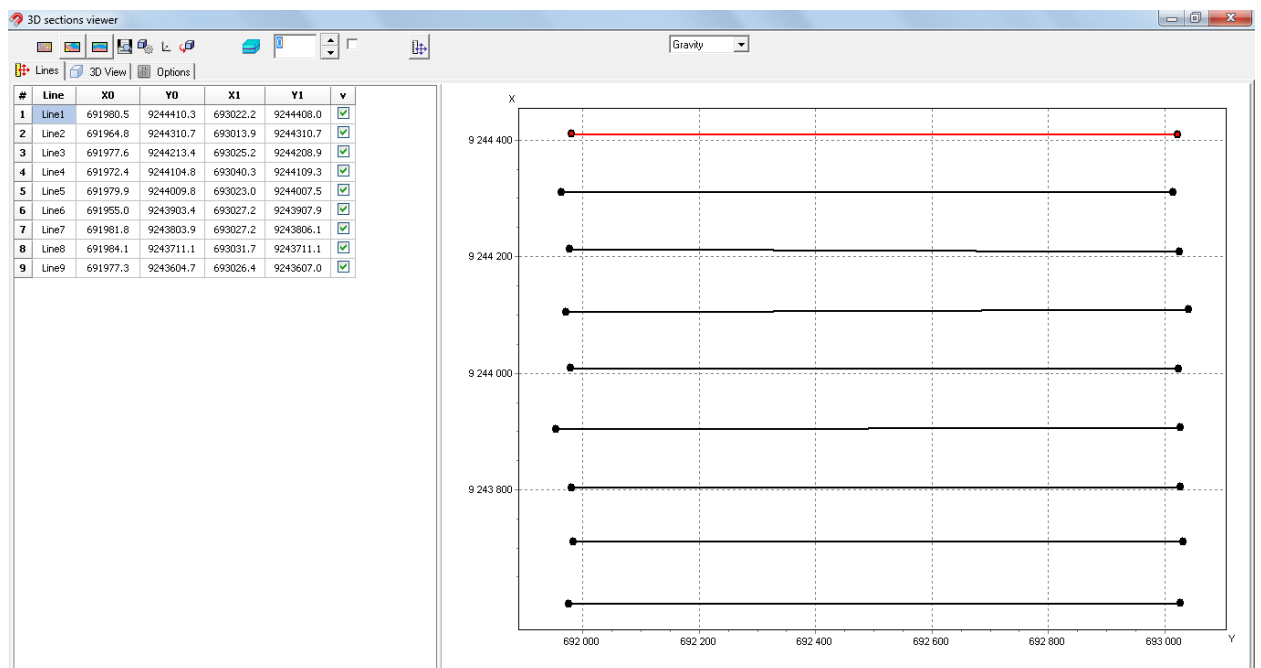
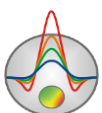


Fig. 14 Window **3D section viewer**, tab **Lines**.



Tab **3D view** (Fig. 15) can be used to view 3D models.

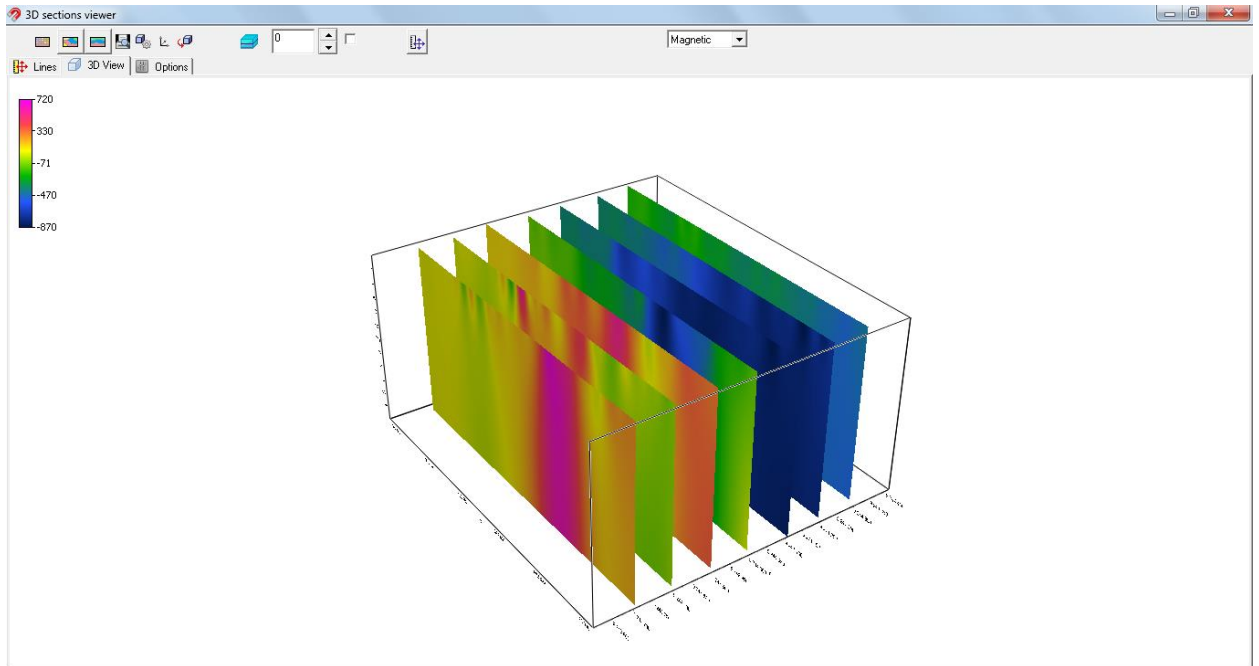


Fig. 15 Window **3D section viewer**, tab **3D view**

Tab **Options** (Fig. 16) is used to adapt image settings.

Area *Color scale* allows setting fill parameters. **Palette** button calls a dialog box of fill settings ([more](#)). Area *Color scale limits* allows setting manually the minimum and maximum for the color scale or choose automatic mode for limits setting by using corresponding check box.

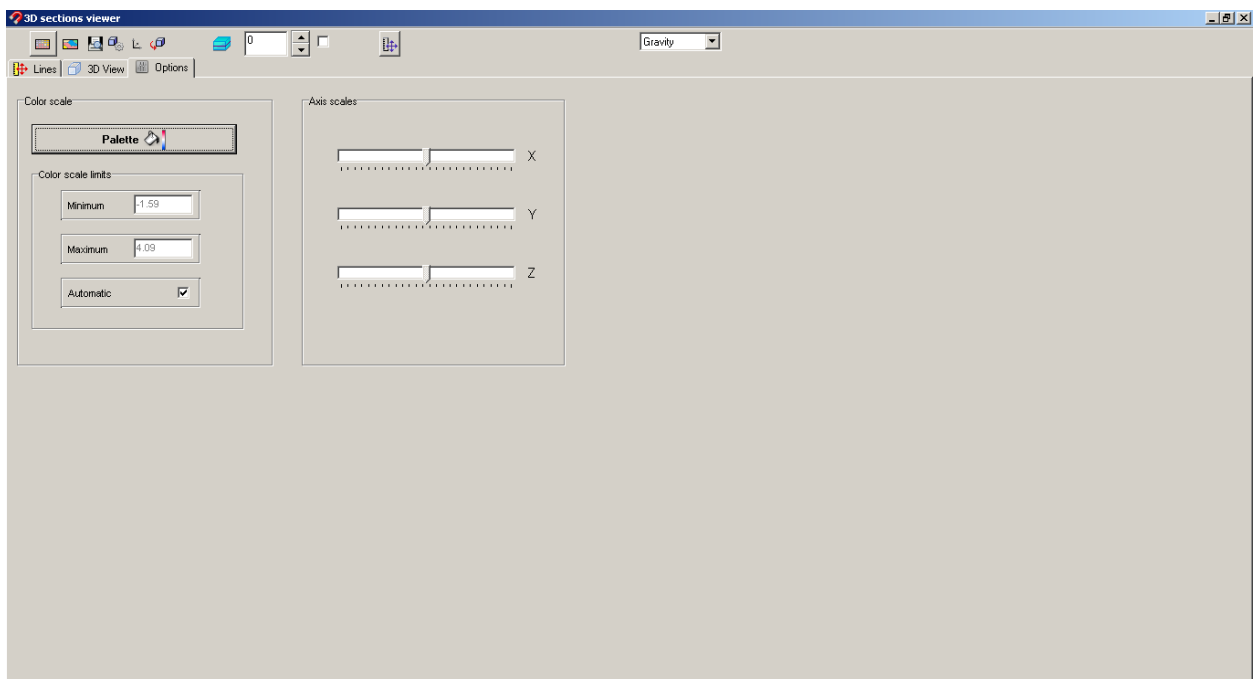
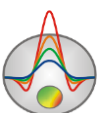



Fig. 16 Window **3D section viewer**, tab **Options**



Area *Axis scales* is intended to set axes scale. The scales are set only while the button  is pressed on the toolbar.

A priori information input

Existence of a priori information (borehole data) allows getting more reliable inversion results. Program «ZONDGM2D» has a built in module, which allows displaying a priori data in graphical form on sections.

Creation and addition lithology file

To create a stratigraphy file select **Options/Boreholes/Edit borehole data** in the main program menu. The dialog box **Add borehole data** appears (Fig. 17).

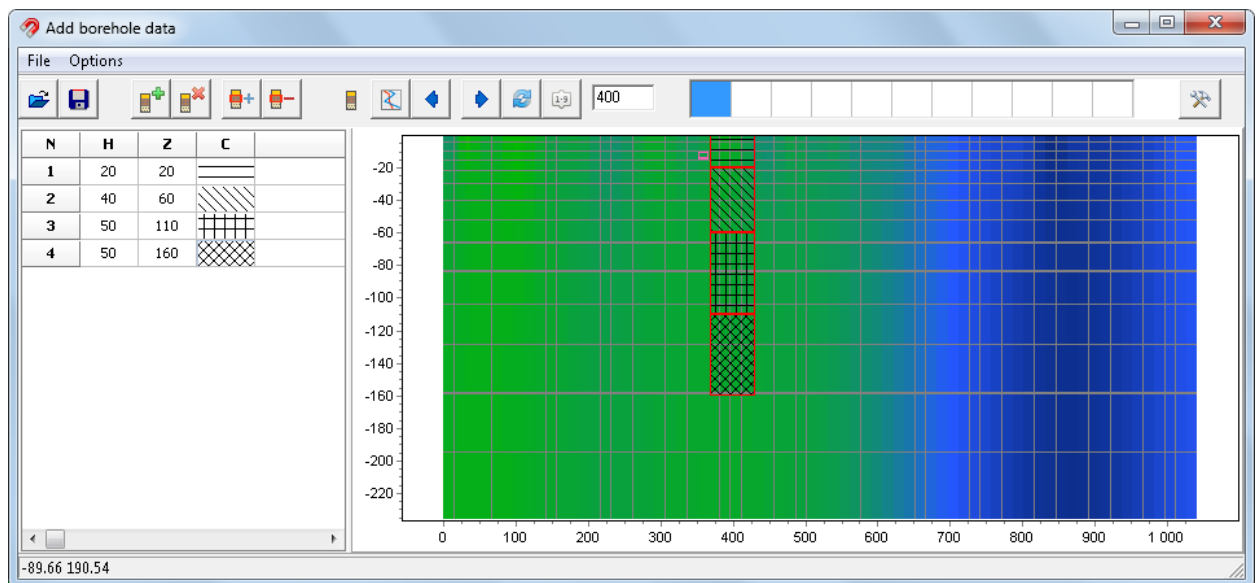



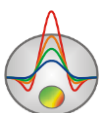













Fig. 17 Lithology file creation dialog **Add borehole data**.

Dialog box toolbar contains the following buttons:



	Open lithology file
	Save lithology file
	Create a new borehole



	Remove borehole
	Add layer in borehole
	Remove layer in borehole
	The lithology column mode
	The logging data mode
	Go to the previous borehole
	Go to the next borehole
	Update data window
	Sort boreholes by coordinate
	Select fill color of window displaying boreholes (red in the example).
	Additional options

Windows of horizontal coordinate setting (distance from the profile start) are placed on the main panel - **horizontal position**, and elevation - **Position from surface**. The horizontal and vertical coordinates are set in meters. Empty square windows are used to create a fill set.

The module contains two main windows. On the left is **Data window**, containing a table with the following columns: N - layer index, H - layer thickness in meters, Z – layer bottom depth in meters, C – filling type. Borehole data are displaying on the right window in graphical form.

To begin creating a lithology file click the button  on the toolbar. Then a new table appears in the **Data Window**. To set number of layers, click the button . Then it is necessary to edit the table: set thickness or bottom depth for every layer, and also choose the filling type according to the lithology. To call filling dialog Pattern Color Editor, click double the left mouse button in column C of the data window (Fig. 18). The program offers a rich variety of lithology fills. The option Color allows choosing the fill color.

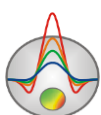








Fig. 18 Fill editor window

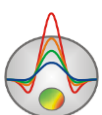
After completing entering borehole data, press a button  and the borehole appears in the graphics window. After that it is necessary to set the horizontal and vertical borehole coordinates on the toolbar in kilometers, after that the borehole will be displayed in accordance with its coordinates. In the graphics window active borehole is displayed in red.

For comfortable working with a large number of boreholes it is possible to create a palette in the program. To create a palette, select a fill on the fills column of the **Data window** and then click the right mouse button within fills area in the main program panel. In this way, fills set can be created and saved. To do this, click the button  and choose **Save default palette**. Saved fills set can be used when creating a new data file of lithology and logging ( - **Load default palette**).

The option **Set percent** is called by clicking the button  and can be used for changing scale of borehole data in graphical form.

After saving the data file, multiple auxiliary files will be created: ***.crt** - module project that can be loaded into the program «ZONDGM2D» and ***.txt** - file for each borehole, the names correspond to the horizontal and vertical coordinate. [More](#) about the format of lithology file.

To add borehole data, use **Options/Borehole/Load borehole data**. Borehole data will be displayed on both model section and in the model editor area (Fig. 19).



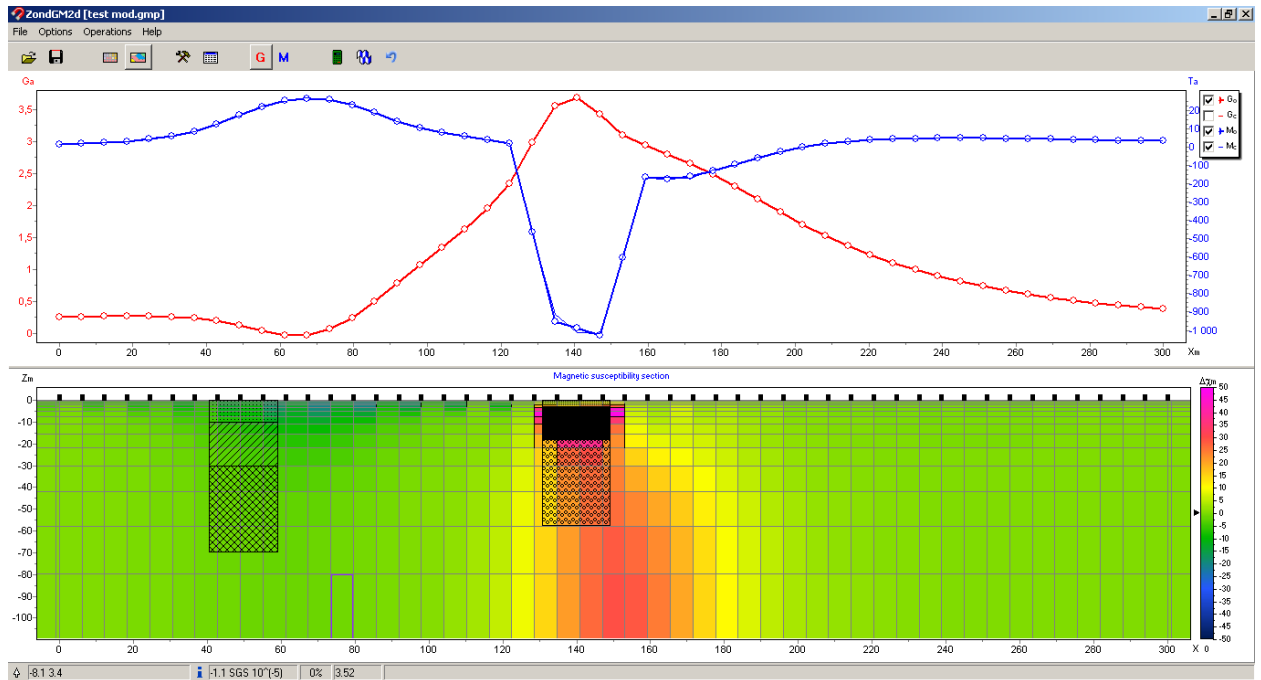








Fig. 19 Displaying lithology data in the editor section.

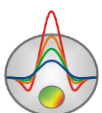
To load the available logging data of boreholes prepare a file format described in the part "[Logging and lithology data file format](#)" and load it using the button .

Geological boundaries (Dialog Set boundaries)

Dialog **Set boundaries** is available in the menu **Options/Advancied** and allows taking into account a priori geological information in the inversion. After selecting this tab, a menu appears with the following buttons:

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

Inclusion of the prior geological boundaries in the inverse problem is one of the most important methods to improve quality of interpretation. On the one hand it increases the problem stability, on the other - reduces the ambiguity and allows getting more plausible structure. In



those model areas where the parameters have very low sensitivity - it is practically the only way to get acceptable result.

When boundaries exist, it is best to use the Occam algorithm. Normally 1-2 boundaries are used and one should not forget that geological boundaries do not always coincide with the petrophysical boundaries.

The boundaries on the resulting density section or section of the magnetic susceptibility should be plotted taking into account the borehole data, or based on a priori information about the structure of the studied area. To set boundaries click the left mouse button when the edit boundaries mode is switched on. To close boundaries click the right mouse button. When plotting boundaries do not use a lot of nodes. It is desirable that the boundaries are as smooth as possible and go near the nodes of the inversion grid.

After boundaries are created, the inversion needs to be restarted, which will be calculated with the actual boundaries (Fig. 20 B).

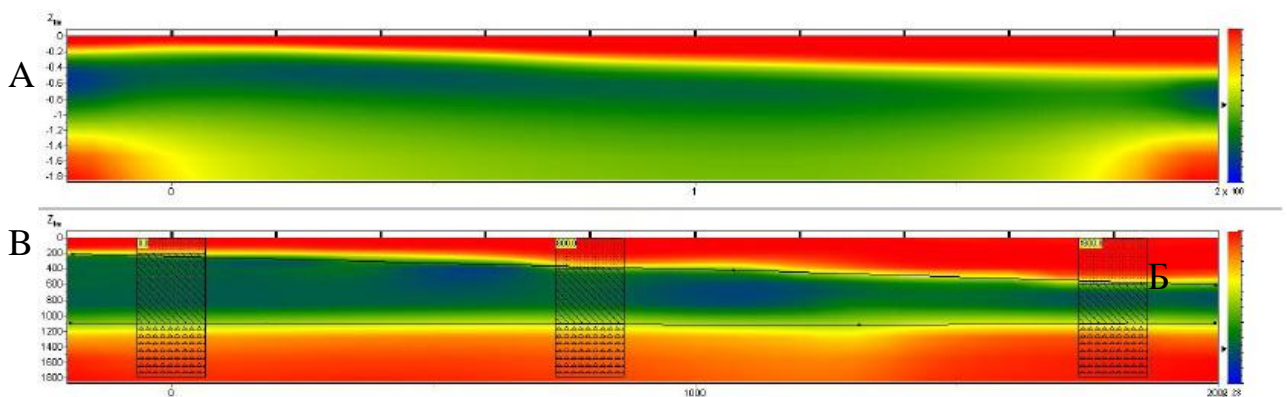


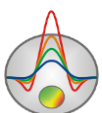


Fig.20 A section example of the inversion results without (A) and with (B) geological boundaries.

ADJUSTING PROGRAM INTERFACE

Palette settings

Dialog can be used for setting object palette of the program and is called by clicking the button **Palette** (Fig. 21). Dialog allows selecting one of the default palettes (forward and inverse rainbow, grayscale, etc.) or create a custom scale. To add a slider on the scale, click the right mouse button with pressing Ctrl. To remove a slider, press the key Delete. It is also possible to save a custom palette. To do this, click the button . To load existing ones, click the button .



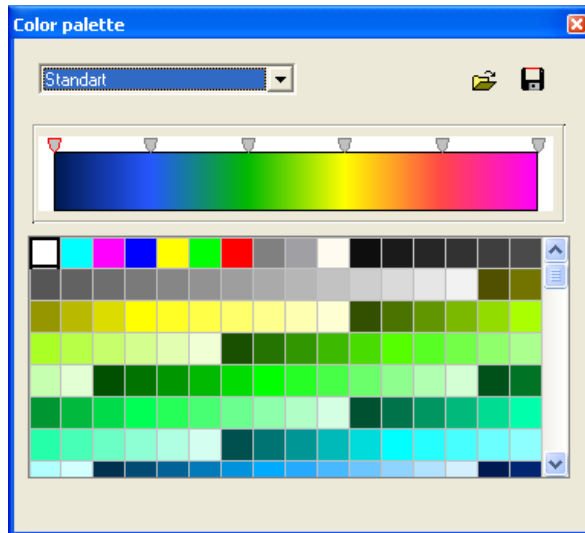


Fig. 21 Palette settings dialog.

Axes editor

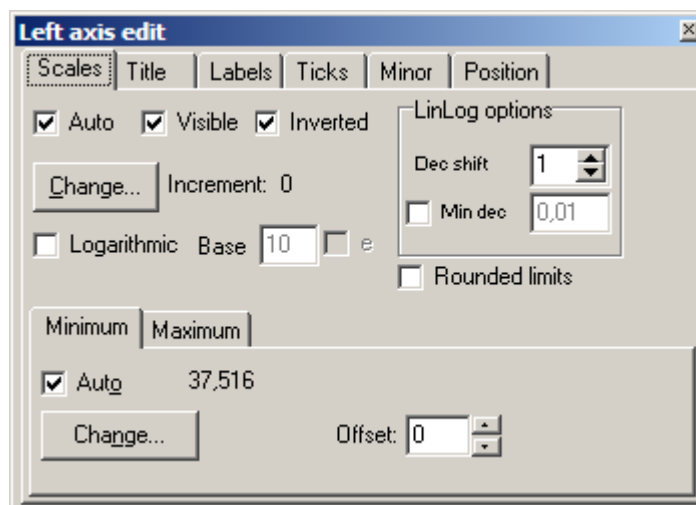
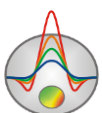


Fig. 22 Axes editor window

Editor is used for adjusting 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 Minimum and Maximum values 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** calls header font setup dialog.

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

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

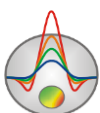
Button **Gradient** calls gradient fill setup dialog for header text.

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

Button **Shadow** calls 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** calls label font setup dialog.

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

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

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

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

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

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** calls header font setup dialog.

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

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

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

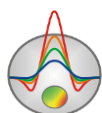
Option **Outline gradient** specifies if gradient fill is used in outline or interior of letters.

Button **Shadow** calls 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** calls label font setup dialog.
Button **Outline** calls dialog for label letters' outline adjustment.
Option **Inter-char spacing** sets letter spacing in label text.
Button **Gradient** calls label gradient fill setup dialog.
Option **Outline gradient** specifies whether gradient fill is used in outline or interior of letters.
Button **Shadow** calls label shadow setup dialog.

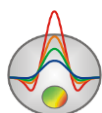
Tab **Ticks** contains options for axis main ticks adjustment.

Button **Axis** calls axis line setup dialog.
Button **Grid** calls line setup dialog for main ticks' grid.
Button **Ticks** calls external main axis tick setup dialog.
Option **Len** sets its length.
Button **Inner** calls 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** calls line setup dialog for intermediate ticks grid.
Button **Ticks** calls external intermediate axis tick line setup dialog. Option **Length** sets its length.

Button **Minor** calls 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).

Graph's editor

Graphics editor is intended for graphic interface adjustment. Right click with SHIFT button pressed on necessary graphic to run it. Pop-up menu with two fields (**Options** and **Default**) appears. The first one runs dialog, the second sets values on default.

Tab **Format** contains connecting line settings.

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

Button **Color** calls color setup dialog.

Button **Pattern** calls pattern parameters setup dialog.

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

Button **Shadow** calls 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** calls point's color fill setup dialog.

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

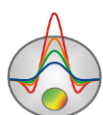
Button **Gradient** calls point's gradient color fill setup dialog.

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

Tab **Style**.

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

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



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

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

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

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

Button **Pointer** calls 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** calls frame background color selection dialog.

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

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

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

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

Option **Size** allows displaying rounding frame corners.

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

Tab **Text**:

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

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

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

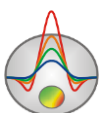
Button **Gradient** calls gradient fill for marking text setup dialog.

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

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

Print preview dialog

Print preview dialog is called in the main program menu Options/Print preview. It is also available by double-clicking on any object in the program.



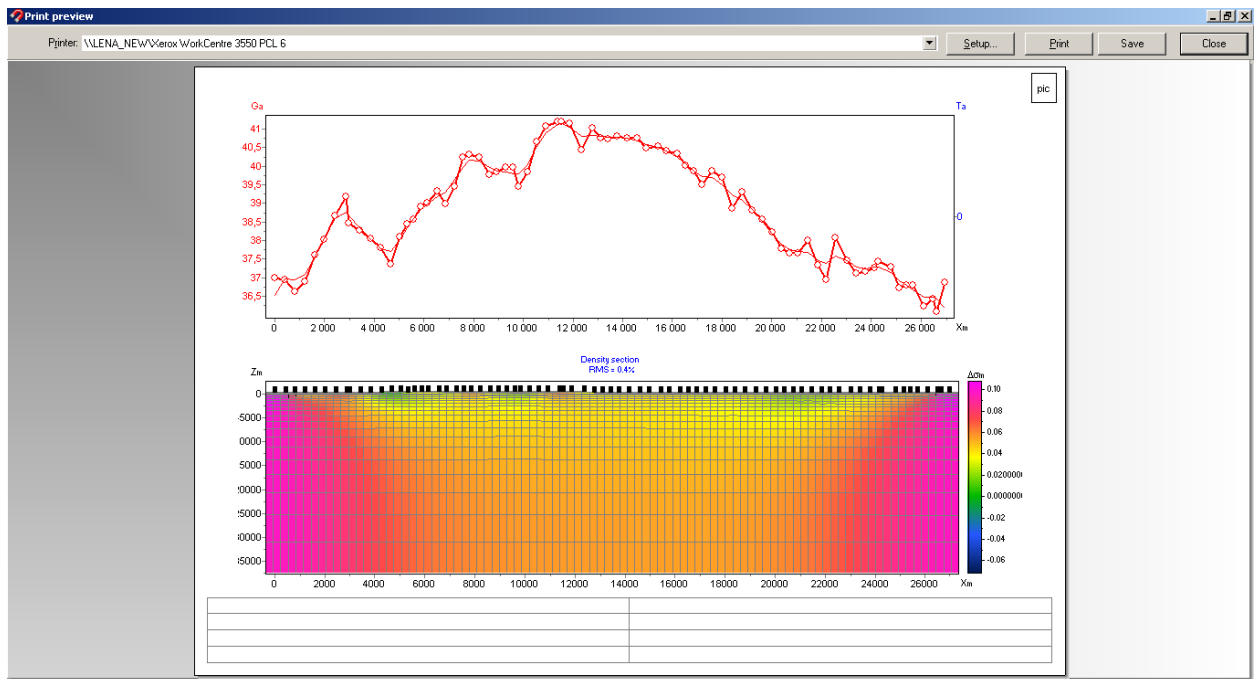


Fig. 23 **Print preview** window

To move a printing object on a sheet click the left mouse button.

In the main menu of the **Print Preview** window the following buttons are:

Printer: HP Officejet 7000 E809a Series (2) ▾

- Select Printer. In the pop-up menu it is possible to select one of the configured printers.

Setup...

- The button prints settings. In the opening window, it is possible to choose the size and orientation of the sheet, the print properties, the number of pages per sheet and other parameters.

Print

- After changing the required parameters it is possible to send a figure for printing using this button.

Save

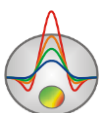
- Save in bitmap files.



Close

- Closes the Print preview window.

The squares at the top of the sheet are used for making seals, stamps or company logos. Click the right mouse button on the square and in the appeared window select a raster image to paste. Sizes of the square can be changed with the mouse.

At the bottom of the sheet editable table is located. To add text, click the right mouse button in the table and in the created window select required text. It is also possible to save all



the comments in the table files with clicking on the button , or to load already saved labels by clicking on the button .

PROGRAM DATA FILE FORMAT

Data file format

Program presents universal data format which consists of information about coordinates and relative heights of measurement points. All geometric values used by the program are set in meters.

The data format of the program ZondGM.

The text file contains the following columns:

prpos - position of the measuring point on the profile

LATITUDE - Latitude

LONGITUDE - Longitude

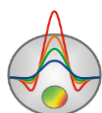
z - height

GRAV - measured value of the gravity field (in mGal)

MAG - measured values of the magnetic field (in nT)

An example of a data file:

prpos	LATITUDE	LONGITUDE	z	GRAV
479	-3.49.38.31598	-40.49.47.04385	156.645	977996.7625
478	-3.50.14.23385	-40.49.34.86847	176.749	977993.118
477	-3.50.41.81796	-40.49.12.96376	158.598	977996.16
476	-3.50.56.86075	-40.48.43.78614	153.596	977995.8746
475	-3.51.29.57703	-40.48.33.59533	163.494	977992.9616
474	-3.51.34.51079	-40.47.58.78146	150.784	977994.8251
473	-3.51.57.72425	-40.47.37.80829	173.923	977989.1426
472	-3.52.13.61573	-40.47.09.33508	151.016	977993.4726
471	-3.52.13.05078	-40.46.50.40375	164.658	977990.0839
470	-3.52.27.34988	-40.46.39.50728	170.948	977988.923
469	-3.52.44.12399	-40.46.27.26992	154.136	977992.3446
468	-3.52.54.85479	-40.46.09.14952	161.646	977990.9782
467	-3.53.10.34938	-40.45.59.98908	162.519	977990.9039



466	-3.53.27.27992	-40.46.01.57194	168.091	977990.0246
465	-3.53.45.00000	-40.46.02.08070	172.989	977989.2686
464	-3.53.50.90568	-40.45.46.82738	175.239	977988.88

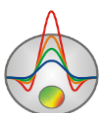
Logging and lithology data file format

To create a lithology file is recommended to use a integral program module ZONDGM2D ([more](#)). Logging data and lithological column are kept in files of a specific format. First type of files has txt extension; these files contain logging and lithology data. The following structure is used to create logging data file:

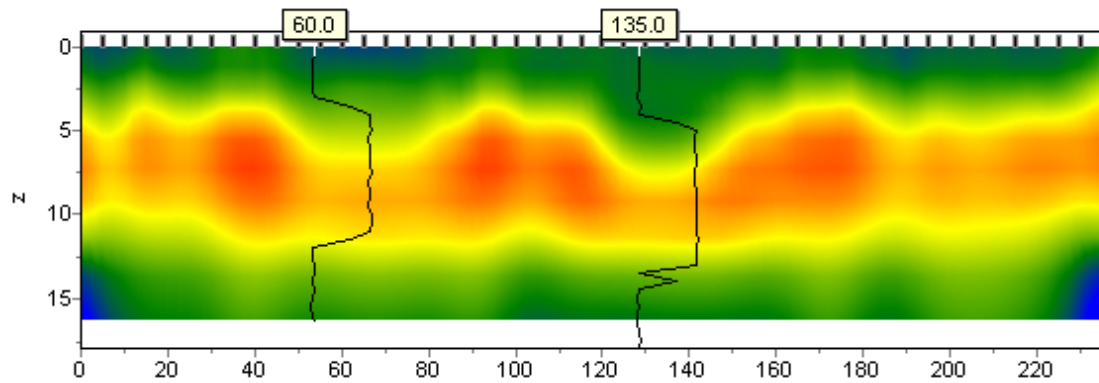
The first column contains measure point depth (from surface), second column contains log measurements. Third and forth columns are filled with zeroes.

Logging data sample-file is given below:

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

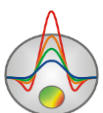
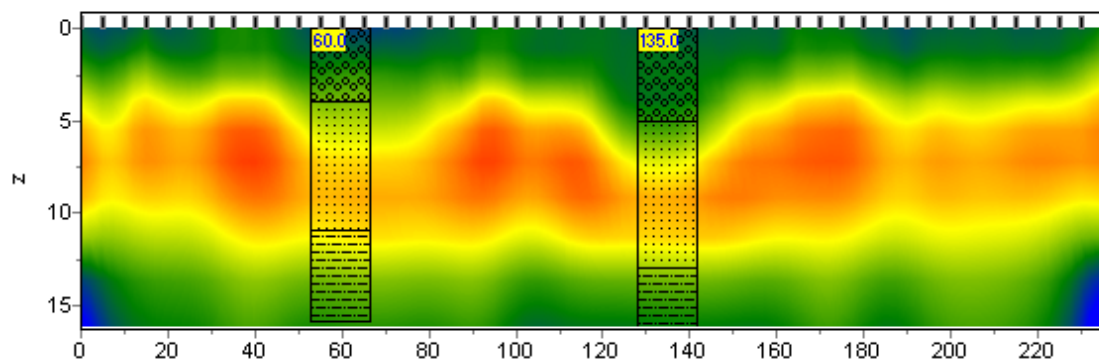


11.5	743.2485878	0	0
12	726.4061156	0	0
12.5	734.399887	0	0
13	727.9166309	0	0
13.5	116.1921851	0	0
14	517.9613065	0	0
14.5	125.3706264	0	0
15	111.2952478	0	0
15.5	131.911879	0	0
16	107.9217309	0	0
16.5	114.9327361	0	0
17	134.0939196	0	0
17.5	138.4457143	0	0
18	129.1165104	0	0

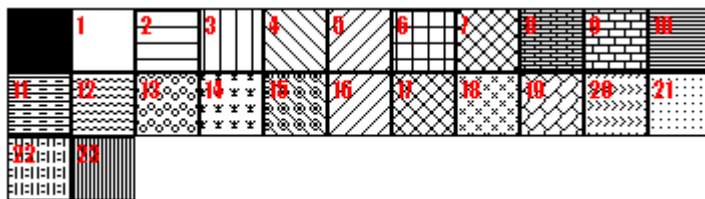


The following structure is used to create lithology data file:

The first column contains measure point depth (from surface) of lithology horizon. The second column should be completed by zeroes. The third column is the layer color on the lithology scale. The fourth column is texture type on the lithology scale.



First 23 patterns for lithologic column creation are given below.



Lithologic data sample-file is given below:

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

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

2280.txt First line – file name of logging or lithology data file

brhl2280 Second line – Borehole name (is displayed on borehole)

18 2 2 1 0 1 0 0 Third line contains control parameters -

Data record 18 – borehole coordinate on profile.

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

2 – type of data visualization 0 - 3.

0 - logging data (as graph);

1 - logging data (interpolated color column), section color scale is used for visualization;

2 - lithologic column;

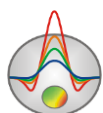
3 - logging data (color column), colors for data visualization correspond to model color scale, column colors are selected in compliance with model color scale;

1 – Logging data normalization parameter 0 - 2.

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

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

0 - Logging method index (if different logging methods are displayed indices of all methods should be specified)



0 – n-1, where n – number of methods.

1 – Plot color.

0 – Data scale is logarithmic 0 or linear 1.

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

3246.txt Description of the following borehole on profile

brh13246

102 2 2 1 0 1 0 0

