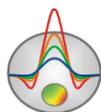


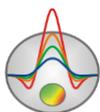
The program of two-dimensional interpretation of natural electric field method data for different types of sources.

ZONDSP2D

<i>Purpose and capabilities of the program</i>	3
<i>Installing and uninstalling the program</i>	3
<i>System requirements</i>	3
<i>Brief theoretical background</i>	3
<i>Getting started with the program</i>	6
Start model settings	7
Toolbar of the main program window.....	10
Functions menu of the main program window	10
<i>Editing the model</i>	15
Working with the block (grid) model.....	15
Polygonal modeling.....	20
Profiling graphs window	24
<i>Interpretation of field data</i>	25
Program Setup dialog box (Program Setup).....	26
Field approximation by a set of extended objects.....	31
<i>A priori information</i>	33



The substrate.....	33
Well information.....	35
<i>Visualizing results.....</i>	39
3D visualization of geoelectric models and sources across multiple profiles.....	39
Geological section editor	43
<i>Settings dialogs</i>	49
Exported image settings dialog box.....	49
Model display settings dialog box.....	50
Palette settings dialog box.....	52
Graphics Editor	53
Axis editor	57
Print preview dialog box (Print preview)	60
SP2 master data file format	61
Logging and lithology data file format	62



Purpose and capabilities of the program

ZondSP2D program is designed for two-dimensional interpretation of profile data of natural electric field (EP) method and can be used on personal computers with Windows operating system. User-friendly interface and wide possibilities of data representation allow to solve the set geological problem in the most effective way.

Installing and uninstalling the program

The **ZondSP2D** program is supplied via the Internet. It comes with this manual. You can download the latest updates of the program at: <http://zond-geo.com/>.

To install the program, overwrite the program in the desired directory (e.g. "Zond"). To install an update, simply write the new version of the program over the old one.

Before you start the program for the first time, you need to install the SenseLock security driver. To do this, open the SenseLock folder (the driver can be downloaded from the website) and run the InstWiz3.exe file. Once the driver is installed, insert the key. If everything is OK, you will see a message in the lower system panel that the key is detected.

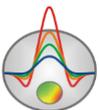
To uninstall the program, erase the program's working directory.

System requirements

ZondSP2D program can be installed on computer with Windows XP operating system and higher. Recommended system parameters: Processor P IV-2 GHz, 1 GB of memory, screen resolution 1024 X 768, color mode -True color. (You should not change the screen resolution in data mode).

Brief theoretical background

In the lithosphere there are a variety of natural electric fields, which differ in their nature, nature and scale of manifestation. Among them, a special place belongs to the electrochemical



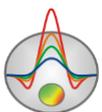
fields of natural electronic conductors, also called fields of redox or ore nature. One of the methods of electrical prospecting is based on the study of fields of this type - the method of natural electric field, which is used for prospecting and exploration of mineral deposits and mapping of certain types of rocks.

The mechanism of formation of natural electric fields over ore deposits can be represented as follows. Any electron conductor placed in an ionic medium is characterized by a potential jump at the boundary of the body, which is called the electrode potential.

If the electron conductor and ionic medium have a constant composition along their contact boundary, then the electron conductor will be an equipotential surface, the potential jump at the boundary will be constant and no external electric current will occur. A change in the composition of one of the contacting media or in the physical conditions within the contact entails a change in the jump potential. The difference in the value of the potential jump in different parts of the boundary of the ore body is a direct cause of the formation of the electromotive force (EMF) and the electric field around the body. In this case, the ore body together with the host medium can be considered as a galvanic element, the internal circuit of which is the surrounding ionic medium, and the external circuit is the body itself, which has electronic conductivity.

We can distinguish two main reasons for changes in the potential jump at the boundary of the electron conductor and ionic medium: a change in the composition of the electron conductor; a change in the composition of the electrolyte surrounding the conductor. An illustration of the first case is a polymineral deposit, such as sulfide aggregates. An example of the second case is a graphite deposit in an ionic environment.

Graphite is practically an inert electrode, i.e. there is almost no exchange of ions between it and the solution. The redox potential (ORP) of the solution plays the role of a potential-determining factor, and the potential jump characterizes the establishment of equilibrium of any oxidant with its reduced form. The oxidant takes away electrons from the electrode and shifts its potential to the positive side. The reducing agent gives electrons to the electrode and shifts the potential to the negative side. Graphite acts as a conductor of electrons from the reducing agent to the oxidizing agent. In the surrounding ionic environment, cations and anions are the current carriers. The EMF of graphite deposits is determined by the difference in ORP values of groundwater at different depths. In concentrated solutions of redox systems, sulfides also behave like an inert electrode. The main reason for changes in ORP of waters with depth is the change in



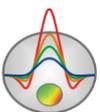
oxygen concentration, to a lesser extent - the presence of various chemical elements with variable valence.

The spatial distribution of potential-determining factors is usually such that the upper part of the ionic medium at its boundary with the conductor is negatively charged, while the lower part is positively charged. As a consequence, the observed potential at the ground surface has a negative sign.

Filtration fields occur in pressure aquifers. On the pore walls of the rock skeleton, represented largely by silicate minerals, a double electric layer is formed. The cations (positive ions) of the silicate crystal lattice are larger than the anions and therefore come to the surface. Because of this, water molecules, in which positive hydrogen ions and negative oxygen ions form an electric dipole, are attracted to the pore wall by their negative poles, forming a layer of strongly bound water. The cations move towards the liquid phase without losing contact with the crystal lattice. Other water molecules are attracted to the layer of strongly bound water, forming a layer of loosely bound water in which the molecules retain some mobility. The water molecules are oriented by positive ions into the pore. Fluid flow moves loose bound water as a whole along the pore. An excess of cations (positive charge) occurs at the exit from the pore, and their deficiency (negative charge) occurs at the entrance. Movement in the pores is laminar, the fluid is viscous, and the flow velocity is maximum along the axis of the pore. Pore diameter should allow the formation of loosely bound water in the region of high flow velocities. At constant flow, the separation of charges is stable and the field exists for a long time.

Fields of diffusion-absorption origin arise in a water-saturated porous medium at separation of charges due to different mobility of electrolyte ions of different sign and their different interaction with the electric double layer. If there is a local change in mineralization (electrolyte concentration) in the liquid, the process of diffusion - equalization of mineralization due to redistribution of ions - begins. Cations have greater mobility than anions, so they leave the area of high mineralization faster. In a porous medium, an excess of cations (positive charges) is formed at the far ends of pores from this area, and an excess of anions (negative charges) is formed near it. In addition to diffusion, there is adsorption of anions in the medium, which are attracted to the electric double layer. For the field to arise in the middle of the pore, there must be some space not occupied by bound water.

The program **ZondSP2D** allows you to solve direct and inverse problems of the method of natural field.



When solving the direct problem, the mathematical apparatus of the finite element method is used. The behavior of the electric potential inside the medium satisfies the Laplace equation. The field sources can be specified explicitly, or as a potential jump at the boundary of an electron and ionic conductor.

In the numerical solution of the direct problem, the medium model is represented by a set of triangular cells with different resistivities. The behavior of the potential inside the cell is approximated by a linear basis function.

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

To solve the inverse problem (inversion) the Newton least-squares method with regularization is used.

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

where A is the matrix of partial derivatives of measured values by section parameters (Jacobian), C is the smoothing operator, W is the matrix of relative measurement errors, m is the vector of required parameters, μ is the regularizing parameter, Δf is the vector of discrepancies between the observed and calculated values.

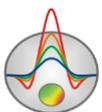
When developing the inverse problem, special attention is paid to the consideration of a priori information (weights of individual measurements, ranges of variation of parameters).

ZondSP2D uses a simple and clear data file format.

The program allows you to import and display the results of measurements by other methods, which contributes to a more comprehensive approach to the interpretation of data.

ZondSP2D program is a convenient tool for automatic and interactive interpretation of natural field method data, and can be used on IBM PC-compatible personal computers with Windows operating system.

Getting started with the program



To start working with the **ZondSP2D** program it is necessary to create a data file of a certain format, containing information about the positions of electrodes, topography and measurement results.

Usually one file contains data on one profile of observations. The text data files organized in the format of the program **ZondSP2D**, have an extension *. SP2.

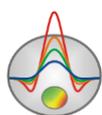
Zond data file [*.SP2]	Open a data file or a project file in Zond format.
------------------------	--

The data file format is described in detail in the [MainData FileFormat](#) section.

Alternatively, you can load data from an arbitrary text file or excel table using the option: **Import data from text/excel**.

Start model settings

After creating the *.SP2 data file, it should be loaded using the button  or its corresponding menu item. If the file is loaded successfully, the dialog box for setting the starting model appears, in which it is proposed to select the parameters of the network and the resistivity of the host medium. This model is a reference model of resistances in the inversion of natural field sources. It is also possible to call the dialog of selecting the grid parameters through the main menu of the program **Options/Mesh Constructor** (if it is required to change the grid parameters).



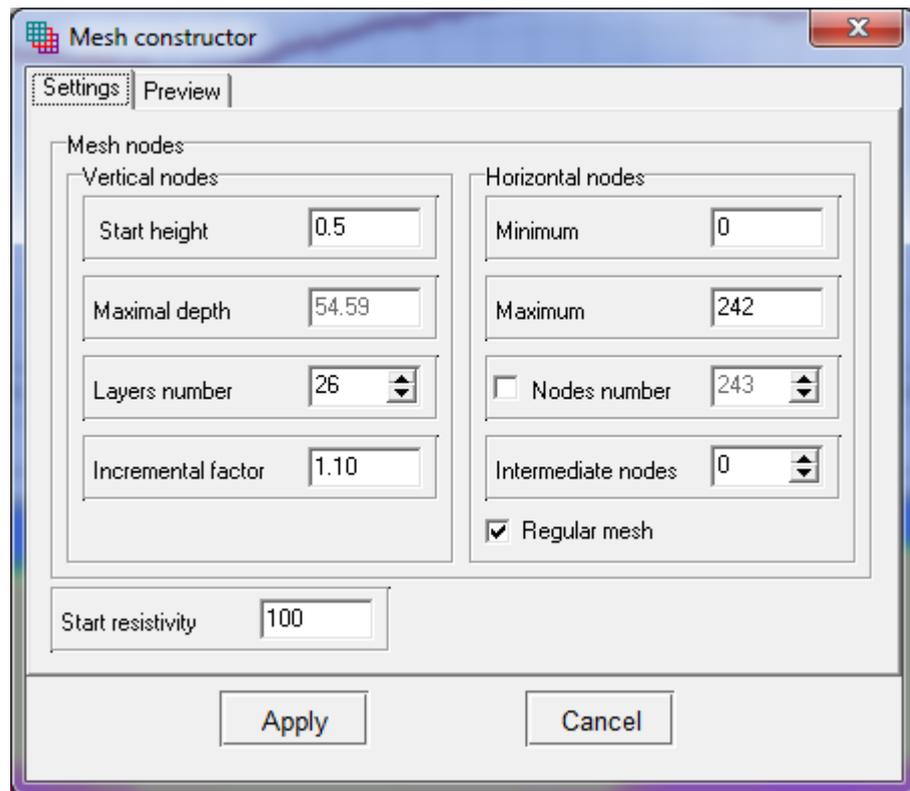


Рис. 1 Mesh constructor dialog box, Settings tab

The **Vertical nodes** area contains options that allow you to set the vertical parameters of the model grid.

Start height - sets the thickness of the first layer. This value should approximate the length of the cell and meet the required resolution.

Maximal depth - specifies the depth of the bottom layer. Note that the maximum depth should not be too great, about one-third of the profile length.

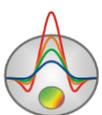
Layers number - sets the number of model layers. Usually, 12-20 layers are enough to describe a model. It is undesirable to set large values of this parameter, as it will reduce the speed of calculations.

Incremental factor - sets the ratio between the thicknesses of adjacent layers. The values of this parameter are usually chosen in the range of 1 to 2.

The **Horizontal nodes** area contains options for setting the parameters of the horizontal model grid.

Minimum - specifies the minimum coordinate of the measurement profile.

Maximum - specifies the maximum coordinate of the measurement profile.



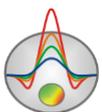
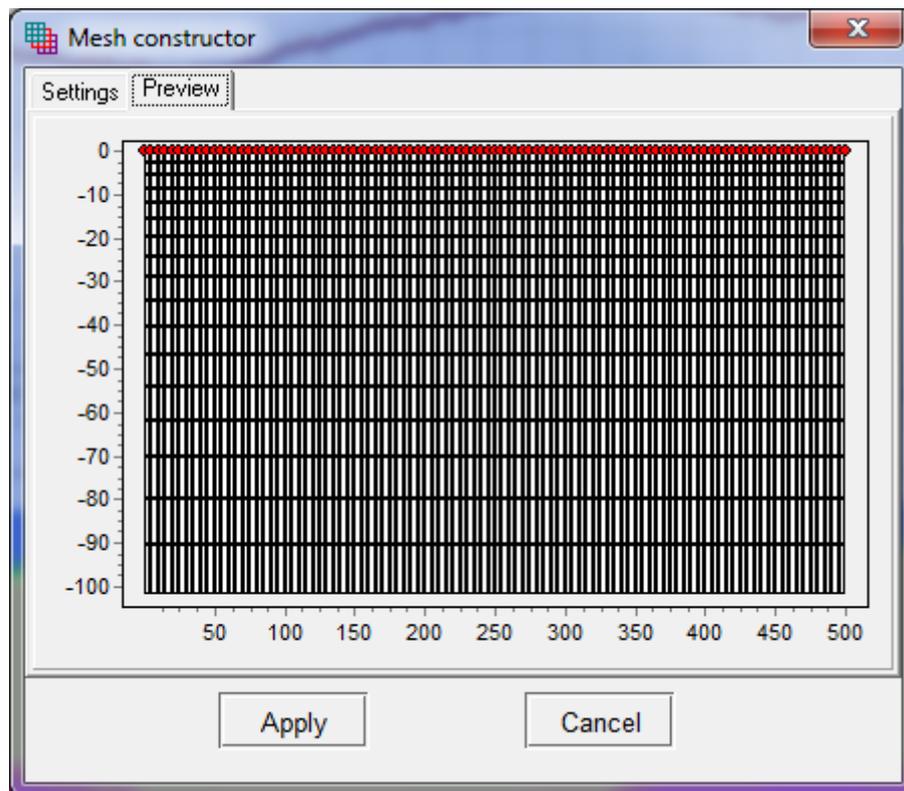
Intermediate nodes - sets the number of additional nodes between unique electrode positions on the profile (0 - 4).

Nodes number - if this option is enabled, the network is built with a uniform (horizontal) step, from the **Minimum** to **Maximum** field. The number of nodes is specified in the **Nodes number** field. This option should be enabled in case of irregular measurement network.

Regular mesh - includes an algorithm for building a horizontal network, in which additional nodes are selected from the condition of uniform partitioning. The option should be enabled in case of very different distances between neighboring electrodes (it has a positive effect on the solution of the forward and backward problem). By right-clicking on the panel labeled **Regular mesh** you can specify the step of cell partitioning along the X-axis, if the option **Nodes number** is not enabled.

Start resistivity - sets the resistivity of the start model. After creating the start model, it is desirable to load the resistivity model from a mod2d file (for example, obtained in the program ZondRes2D).

The **Preview** tab allows you to see how the network will look like (without the topography).

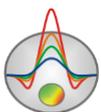


Toolbar of the main program window

The toolbar is used to quickly call the most frequently used functions in the program. It contains the following function buttons (from left to right):

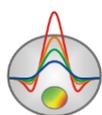
	Open a data or project file.
	Call the Save Data or Project dialog box.
	Call the Inversion Settings dialog box.
	Start the inversion procedure or stop it (if pressed again).
	<p>Run the statistical algorithm for finding natural field sources in a given section of resistances (resistances are taken into account in the calculation). The sources in this case are represented by two-dimensional extended objects of arbitrary geometry, with a linear change in the potential jump along the object. This problem is convenient to use for modeling the fields of redox nature.</p> <p>When you right-click, a pop-up menu appears with the following settings:</p> <p><i>Num of objects</i>: The number of objects with which the observed field will be approximated.</p> <p><i>"+" on top</i>: this option specifies that the positive charges must be placed on the top pole of the object.</p> <p><i>Remove results</i>: remove objects from the cut.</p>
	Block (grid) mode of resistance setting (more details) .
	Polygonal model mode for setting the model of resistances and EP sources (more details) .

Functions menu of the main program window

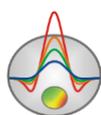


The names of the menu items and their purpose are listed below:

File/Open file	Open a data or project file.
File/Import data from text/excel	Import EP data from an arbitrary (multi-column) test file (or Excel table). The user needs to set the column names in the first row of the table.
File/Save file	Call the Save Data or Project dialog box.
File/Edit file	Open the data file used by the program in the Notepad editor.
File/Print preview	Call the Print dialog box of the main program window.
File/Recent	Allows you to open one of the recently used files.
File/Exit	Exit the program.
Options/Project information	Show or edit information about an uploaded project.
Options/Mesh constructor	Call the Start Model Network Settings dialog box (more details).
Options/Program setup	Call the Inversion Settings dialog box (see details).
Options/3D fence diagram	Call the dialog to view the volumetric model of resistances and EP sources. Works with area data (more details).
Options/Geological editor	Call the geological interpretation window of the geoelectric section (more details).
Options/Borehole/Create/Edit borehole data	Add (edit) borehole data (lithologic columns or well logs). Call the dialog of creating the borehole data (more details).
Options/Borehole/Load borehole data	Open and show the file with well log data and lithologic columns, as well as mod1d files (one-dimensional model files).
Options/Borehole/Remove boreholes	Delete logging data and lithologic columns from the project.
Options/Borehole/Set column width	Set the width of the lithologic column in the section image (in percent of the profile length).
Options/Import/Export/Background/Load background	Load a custom backing, the following formats: bmp, png, sgy, sec. The sec format is an internal Zond format and contains the image and corner coordinates.
Options/Import/Export/	Remove the substrate from the project.



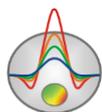
Background/Remove background	
Options/Import/Export/ Background/Change sizes	This option allows you to change the size and position of the custom backing in real time.
Options/Import/Export/ Background/Save background	Save the image of the substrate in a graphical format file.
Options/Import/Export/Load MOD1D/2D file	MOD1(2)D files are an internal Zond format. They allow you to exchange models between programs or projects. The imported model will be embedded in the current model. This option is very important in this program because it allows you to quickly load the resistance model from other programs that allow you to invert this parameter (zondres2d/zondmt1(2)d/zondip1d/zondtem1(2)d). Using this option is highly recommended before starting the interpretation.
Options/Import/Export/. Save MOD2D file	Save the current resistance model in Zond MOD2D format
Options/Import/Export/. Export model to geosoft	Export the current model to GeoSoft format.
Options/Import/Export/. Export model to SEG-Y	Export the current model to SEG-Y format.
Options/Import/Export/. Direct drawing to surfer	Build the current impedance model and source distribution in the Surfer. There can be problems if 2 versions of the Surfer are installed or the exchange libraries are not installed.
Options/Extra/. Bitmap output settings	Call the Model Graphics Settings dialog box when exporting (more details).
Options/Extra/. X=Z 1:1 scale	Set equal scales for vertical and horizontal axes of the model.
Potential's isolines	In boundary-redox mode, it plots potential isolines on top of the current model.
Graphic of potential	In most cases it is more convenient to operate with the gradient-potential data. But sometimes a large number of peaks on the graph, complicates its visual interpretation. In



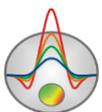
	these cases it is necessary to switch to the potential graph mode. It should be noted that if the data are imported from the program ZondRes2D and MN lines overlap each other, such recalculation can be slightly incorrect.
Convert dip to pole	If the search for sources was performed for dipole structure elements (dipole-X, dipole-Z, Volume vector), they can be recalculated and plotted as a section of effective poles.
Buffer/Model 1,2...5	The buffer allows you to store up to five resistivity models.
Help/About	About the program.
Help / Manual	Show program instructions
Help/Check for updates	Check for updates.
Help/Show news	Show Zond Software news announcements
Help/Send message to us	Send a message to the developers. The message must be typed in transliteration.

When you switch to the polygonal modeling window (the button of  the toolbar of the main program window), the following menu options become available:

Modeling / Mesh to polygon	Assign resistance values to polygons automatically. This value will be equal to the average value of all falling cells (mesh model) of the model inside the polygon. In the <i>Use mesh resistivity</i> mode this procedure will not affect the calculation results in any way.
Modeling / Polygon to mesh	Build a polygonal model into a mesh model (e.g., if the resistivity model is unknown).
Modeling / Save polygons	Save polygons to a text file.
Modeling / Load polygons	Load polygons from a text file.
Modeling / Remove all polygons	Delete user-defined polygons.
Modeling / Use mesh resistivity	If you have a resistivity model imported e.g. from the results of electrotomography, it is very convenient to use this mode. In this case, the grid model values are used instead of the specified resistance values of the polygons.



<p>Modeling / Colors</p>	<p>Sets how polygon colors are assigned.</p> <p><i>ColorScale</i> value - the color of the polygon is selected according to its resistance on the color scale.</p> <p>The <i>Transparent</i> value is for transparent polygons. This option is very convenient if you have a resistivity model, in Use mesh resistivity mode.</p> <p><i>User</i> value - the graphical parameters of the polygon are set by the user (to set the polygon you need to double-click it).</p>
------------------------------	--



Editing the model

In the program **ZondSP2D** two basic modes are provided for editing the model:

- *Block mode (mesh mode)*  is intended for editing the model cells directly (both the width or thickness of the selected cell row and the cell properties). When working in block mode, it is also possible to edit the network geometry created in the **Options / Mesh constructor** dialog box;
- *Polygonal*  is designed to define an environment consisting of a set of connected or unconnected bodies.

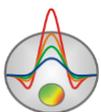
It should be noted that in any mode of model creation the means of working with a priori information are available: loading of drilling and logging data, substrate in the form of raster image, models from other programs of **Zond** package.

Working with the block (grid) model

Block model editor is used to change resistance of individual model cells or groups of cells using the mouse. Working with model cells is similar to editing a raster image in graphical editors. When you move the cursor in the model area, the bottom status bar of the main program window displays the coordinates and parameters of the active cell in which the cursor is located. The currently active cell is surrounded by a rectangle - the cursor. The selected cell is marked with a speck of white.

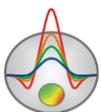
To work with the block model, there are two sets of tools in the program: the context menu (called by right-clicking in the model editing area), the remote toolbar and the color scale. The functions of the context menu and the pop-up toolbar duplicate each other to a large extent:

<i>Context menu</i>	<i>Model tools</i>	<i>Option</i>
		Zoom, change the scale of the model display.
Cell to cursor value		Use the parameter of the active cell as the current value.
Edit mode		Enable edit mode.
Selection\Free form selection		Select a set of cells within the editing area with the mouse. The area has user-defined boundaries. The same tool can be used by



		pressing the C key on the keyboard.
Selection\ Rectangular selection		Select a set of cells within the editing area with the mouse. The area has a rectangular form.
		Select a set of cells within the editing area with the mouse. The area has an elliptical form.
Selection\Magic wand		Select a set of cells within the editing area with the mouse. The active cell and the cells adjacent to it, the parameters of which are close to its parameter, are selected. The degree of closeness is set in the dialog box of model parameter settings. The same tool can be used by pressing the X button on the keyboard.
Selection\ Remove selection		Delete Selection.
Mesh options\Add column /row		Add a new vertical or horizontal boundary. The new boundary appears when you click in the selected location.
Mesh options/Remove column /row		Delete the selected vertical or horizontal boundary.
Mesh options\Resize column /row		Change the thickness of a row or column with the mouse.
Clear model		Clear the model completely. Remove all area selections and return to the homogeneous half-space with the physical properties specified when creating the starting model in "Meshconstructor" .

To the right of the model editing area there is a color bar linking the color value to the resistance value. To select the current value, click on the scale with the left mouse button, and the value is displayed below the color scale. Changing the value of a cell parameter is done with the mouse: clicking the left mouse button on the cell changes its parameter to the current value.



To quickly create a model, the program provides several modes of cell selection: rectangle, ellipse, free-form and by a certain parameter value. The corresponding options can be called from the context menu by pressing the right mouse button in the model editing area. If some area of the section is selected, then pressing the left mouse button anywhere in the selected area sets all cells of the area to the current value of the parameter.

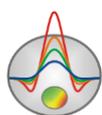
Zoom&Scroll mode (it is enabled in the context menu of model settings by pressing the right button near the Resistivity **block-section** caption in the model window) or after clicking the button  in the Model tools panel. To select the area to be zoomed in, the mouse pointer moves down and to the right with the left button pressed. To return to the original scale, the same actions are performed, but the mouse moves up and to the left.

Pressing the left mouse button while pressing SHIFT on a cell or a group of cells increases its parameter. Pressing the right mouse button while pressing SHIFT on a cell or a group of cells decreases its parameter. The percentage by which the value is changed is specified in the Model Settings dialog box. If the active cell belongs to a selection, then all operations described above are applied to the entire selection. This option works when active when the ( **Edit mode**) option is on.

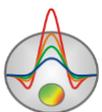
Pressing the mouse button while pressing CTRL allows you to move the highlighted set of cells within the editing area with the mouse. When you move the selection with the left mouse button pressed, the contents of the selected cells are copied to the new location. When you move the selection with the right mouse button pressed, the contents of the selected cells are cut and copied to a new location.

When you right-click in different areas of the model, contextual menus with the following options appear:

Upper area, near the "Resistivity section"	Display model mesh	Specifies whether or not a net should be imaged.
	Display objects border	Specifies whether the boundary of the object should be depicted.
	Setup	Call the Model Settings dialog box for more details .



	Display color bar	Specifies whether the color scale should be depicted or not.
	Zoom&Scroll CTRL+A	Enable zoom and scroll mode.
	Print preview	Print the model.
Color scale	Setup	Call the Model Settings dialog box.
	Set range	Manually determine the minimum and maximum values of the color scale.
	Automatic	Automatically determine the minimum and maximum values of the color scale.
	Log scale	Set the logarithmic scale for the color scale.
	Colors as histogram	Set the colors based on the distribution of model cell resistances.
	Smooth image	The mode of displaying the model with a continuous gradient palette (smooth section).
Cutting area	Context menu of work with the model	Described in the section "Working with the block model" (more details).
Vertical axis	Set maximum	Set the bottom layer depth value.
	Redivide	Set the same layer thickness for all model layers (at this scale).
	Thick mesh	Remove every other node in the vertical grid.
	Thin mesh	Add intermediate nodes to the vertical grid.
	Set maximum	Set the bottom layer depth value.
Horizontal axis	Redivide	Set the same width for cells located between unique measurement point positions.
	Thick mesh	Delete every other node of the horizontal grid (if there is no measuring point in this node).
	Thin mesh	Add intermediate nodes to the horizontal grid.



Grid model display modes

The mesh (block) model can be displayed in the form of cells (**Block section**), in the form of a gradient palette (**Smooth image**) and in the form of a **contour section**. The contour mode is set in the [modelsetup](#) dialog box by turning off the *Continuous* option (Colors tab).

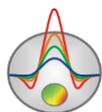
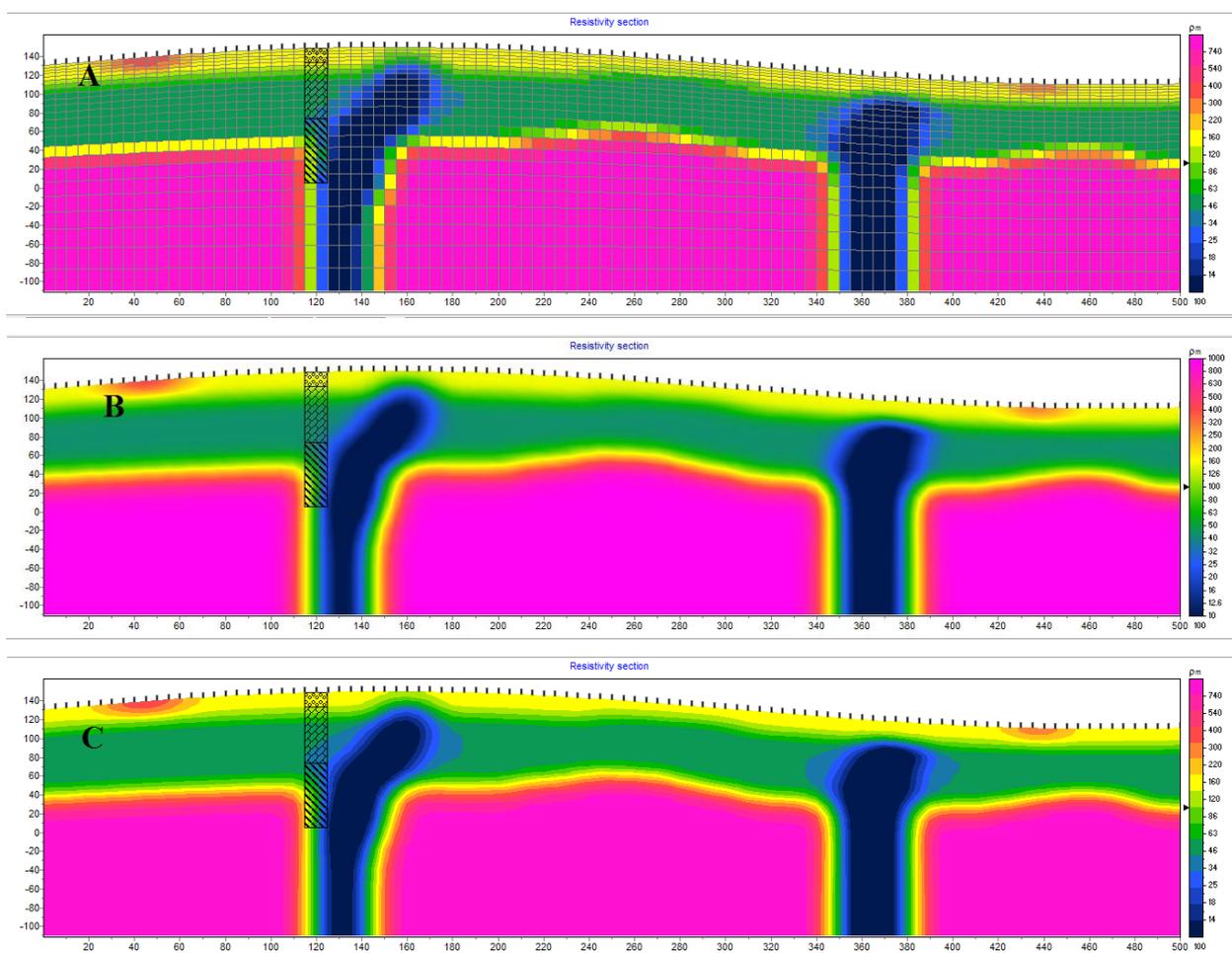


Рис. 3 Variants of the program's working window in different modes of model visualization. Where **A - Block section mode**, **B - Smooth image mode**, **C - contour section mode**

Polygonal modeling

In many situations it is more practical to set the model as a set of bodies with a polygonal section. In **ZondSP2D** polygonal modeling mode provides such an opportunity. Switching to the mode is done by clicking the button , after which the toolbar for creating a polygonal model appears in the upper left corner of the screen. If a polygonal model has already been created within the current project, it will be shown in the model window. Polygonal mode also has a very important function of limiting the area of existence of electric field sources. Also, only in this mode it is possible to simulate the natural field of redox nature (boundary-redox).

Creating polygonal models in **ZondSP2D** is similar to the process of creating polygons in vector graphics editors.

Polygonal modeling mode allows to describe both separate polygons (bodies) in a homogeneous environment and a system of related polygons (bodies). To create a polygon and edit it, it is necessary to select the necessary tool from the Polygonal modeling toolbar.

It should be noted that the main task of polygonal modeling is not the construction of the geoelectric section, but the selection of those parts of the section (borders, volumes) where sources can be located.

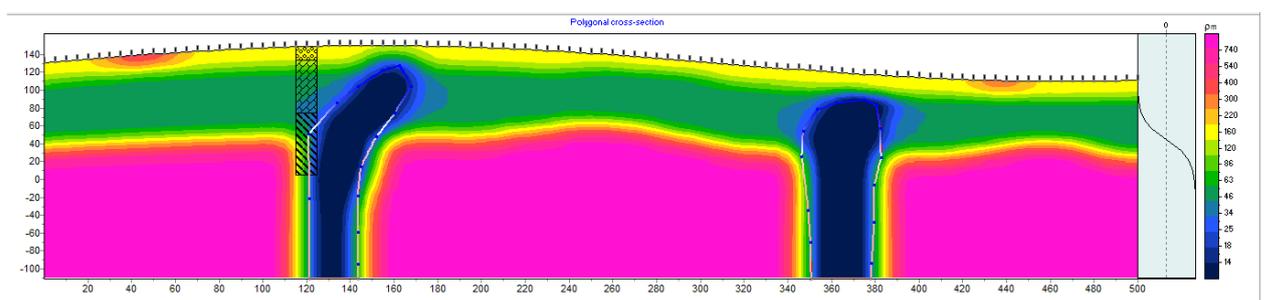
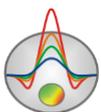


Рис. 4 Model of the environment in polygonal mode (boundary redox). The graph to the right of the model shows the distribution of the potential-determining factor with depth.



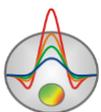
And a polygonal modeling toolbar containing the following functions:

Add polygon mode. Called by the button  on the toolbar. This mode is intended for creating a local polygon. To add a new point to the polygon click the left mouse button. Click the right mouse button to close the polygon (connect the last point of the polygon to the first one). If the creation of a local polygon is impossible (i.e. some edges intersect or another object is present in the polygon) the program doesn't allow the user to close the polygon. When creating a polygon, try not to have the points too close to each other.

Delete polygon mode. Called by the button  on the toolbar. This mode is intended to delete a polygon. To delete a polygon, right-click on it. The polygon outline changes its color to red when the cursor is inside the

The **Create Attached Polygon** mode. Called by the button  on the toolbar. This mode is intended for creation of polygon (additional part of the polygon) attached to the existing polygons, model edges or relief, i.e. to some linked area of the model. A connected area is a set of polygons and model edges having common faces. Click the left mouse button to add a new point to the polygon. Right-click to close the polygon (connect the last point of the polygon to the first, through the boundary of the adjoining region). The first and the last (closing) points of the polygon should be set on the outer boundary (which changes color to red when the cursor approaches) of the linked area. If the creation of a local polygon is impossible (i.e. any - edges intersect or another object is present in the polygon) the program does not allow the user to close the polygon, and deletes the created points. It should be noted that polygons attached to the left, right and bottom edges of the model have infinite extension in these directions (i.e. continue beyond the model boundaries).

Disconnect polygon mode. Called by the button  on the toolbar. This mode is intended for detaching a polygon from a set of linked polygons or model edges. Note that a polygon detached from the model edges loses its infinite extent (being limited by the model edges). To disconnect a polygon click the right mouse button on it. The polygon outline changes color to red when the cursor is inside. Then use the **Move polygon** button to move the detached part of the polygon away from the main polygon.



Split polygon mode. Designed to create two parts inside the polygon. Called by the button



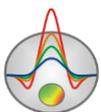
on the toolbar. This mode is intended for partitioning the polygon into two new linked polygons. The boundary of the partition is set by two points on the faces or nodes of the polygon to be partitioned. To select the first point of the boundary, click the left mouse button. A right click is used to select the second and partition the polygon. If the operation is not possible (i.e. some faces intersect or the boundary is outside the polygon) the program does not allow the user to split the polygon, and deletes the created boundary. The faces and points of the polygon change color to red when the cursor approaches.

Move polygon mode Called by the button  on the toolbar. This mode is intended for moving the unconnected points of the polygon. If the polygon has no common, unconnected points with other polygons or model edges, it is moved completely. Click the left mouse button to select the polygon to move; then the unconnected part of the polygon moves after the cursor. Click the right mouse button to fix the new position of the polygon. If the operation is impossible (i.e. some edges intersect or the polygon is in the polygon) the program doesn't allow the user to move the polygon and returns it to the initial position. The polygon outline changes color to red when the cursor is inside.

Move linked polygons mode. Called by the button  on the toolbar. This mode is intended for moving the polygon and all the associated polygons. Click the left mouse button to select the polygon to be moved; then the linked area moves after the cursor. Click the right mouse button to fix the new position of the polygons. If the operation is impossible (i.e. some faces cross or a polygon is in the polygon) the program doesn't allow the user to move the polygons and returns them to the initial position. The polygon outline changes color to red when the cursor is inside.

Add Point mode. Called by the button  on the toolbar. This mode is intended for adding a new point on an edge of an existing polygon. To add a point to a polygon, right-click on the polygon face. The polygon faces change color to red when the cursor approaches.

Delete point mode. Called by the button  on the toolbar. This mode is intended for deleting a new point of an existing polygon. Delete a point in the polygon by right-clicking it. The operation is not performed if it results in the following situations - the faces cross each other, the polygon is within the polygon, or the number of points in the polygon is less than three. The points of the polygons change the color to red when the cursor approaches.



Merge points mode. Called by the button  on the control panel. This mode is intended for combining two points into one, attaching a point to an edge of another polygon or to the edges of a model. Select the first point to merge

Magnifier mode. Called by the button  on the toolbar. Allows you to change the scale of the model display.

When working with polygonal modeling tools, remember that **all operations are terminated by right-clicking.**

Polygon settings dialog box

To change the physical properties of a polygon and its graphical display parameters, double-click the left mouse button in its center. As a result, the *Body parameters* dialog box appears.

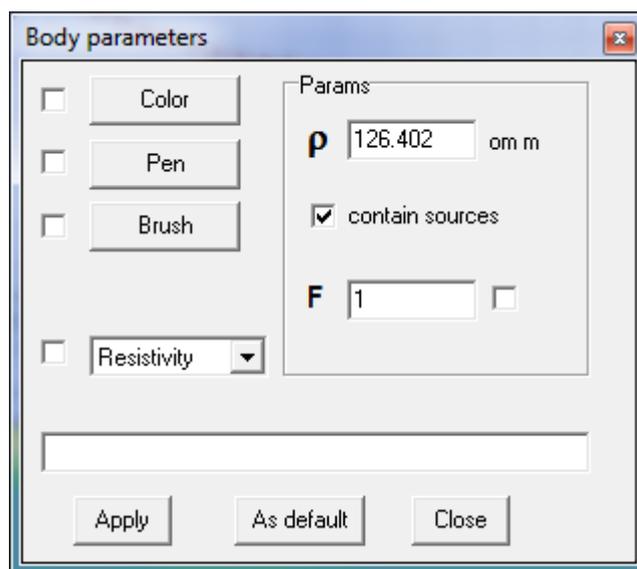
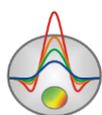


Рис. 5 Dialog window **Body parameters**

The value of ρ is the resistivity of the polygon. It is not taken into account in the use mesh resistivity mode.

When configuring the following three settings, you should set Modeling / Colors to user, in the Geological editor window it is not necessary.



Color button - calls the Select Polygon Fill Color dialog box. If this option is enabled, the selected color will be used in all polygons of the model.

Pen button - calls the dialog box for setting the polygon boundary parameters. If the option is enabled, the selected parameters will be used in all polygons of the model.

Brush button - calls the dialog box of polygon fill settings. If this option is enabled, the selected parameters will be used in all polygons of the model.

The *contain sources* option sets whether the sources can be located, depending on the type: inside or on the border of the object. If this option is off, it is considered that the object does not contain sources

F value - in boundary-redox mode, if the object contains sources, sets the value of zero potential. Depending on the nature of the contact, it can take different values. This parameter is selected during inversion in boundary-redox mode.

The next field defines the polygon caption:

The *Resistivity* value is the resistivity of the polygon

Value *None* - no signature

User text value - the value from the underlying field will be displayed on the polygon.

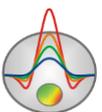
For the enabled options (ticks to the left), the selected settings will be applied to all polygons of the model.

In *boundary-redox* mode it is possible to calculate the direct problem for the given objects and the potential-determining factor. Calculation of the direct problem from the created model is performed by pressing the button ( appears when switching to boundary-redox mode) on the toolbar or by pressing the spacebar.

Profiling graphs window

By default, theoretical curves are shown as solid blue curves, experimental ones as solid red lines with circles at the measurement points. Theoretical and experimental graphs are displayed in blue Color can be changed in the dialog box of this object settings.

Axis parameters can be set in [the axis editor](#) (right-click+SHIFT on the axis).



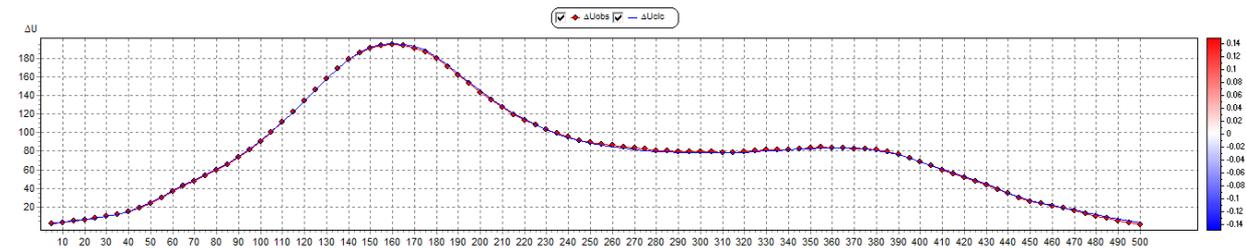


Рис. 6 Profiling graphs window

The color scale in the right part is used to set the correspondence between the intensity of the natural electric field source (of one nature or another) and the color. To call the color scale settings dialog box, right-click on it.

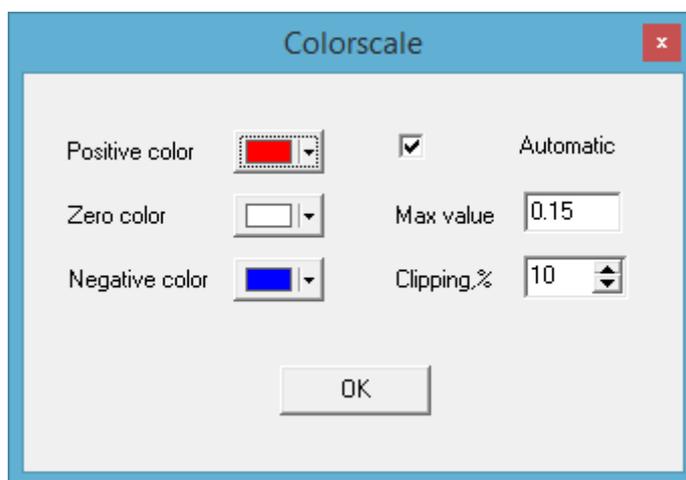
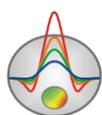


Рис. 7 Dialog of the color scale settings

In the dialog box that appears you can set: color settings for positive, negative and zero charges (Positive color, Negative color, Zero color); Automatic (automatic or user defined) option for defining the scale limits, the user maximum scale (Max value) and Clipping %, sources with intensity less than the specified percentage of the maximum will not be displayed.

Interpretation of field data

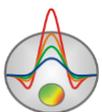


After loading the data file and setting up the start model, the next step is to select the inversion type and configure the parameters. The dialog box for setting the inversion parameters can be accessed using the button  or the **Option/Program setup** menu item.

Under the solution of the inverse problem in **ZondSP2D** we mean restoration of the potential-determining factor or distribution of sources, in a user-defined or obtained by other geophysical methods resistivity model. Depending on the type of the problem, conventionally "ore" or "hydrogeological", the type of sources and areas of the section where their presence is assumed should be selected. In the grid model mode, the sources may be located in every node of the grid, except for the surface of the section. In the polygonal mode, the sources are located only on the boundaries or inside the polygons, for which the parameter contain sources is selected, and, for volumetric sources, their positions also correspond to the nodes of the network.

Program Setup dialog box (Program Setup)

The dialog box is intended for setting parameters related to the solution of the inverse problem of the field source recovery.



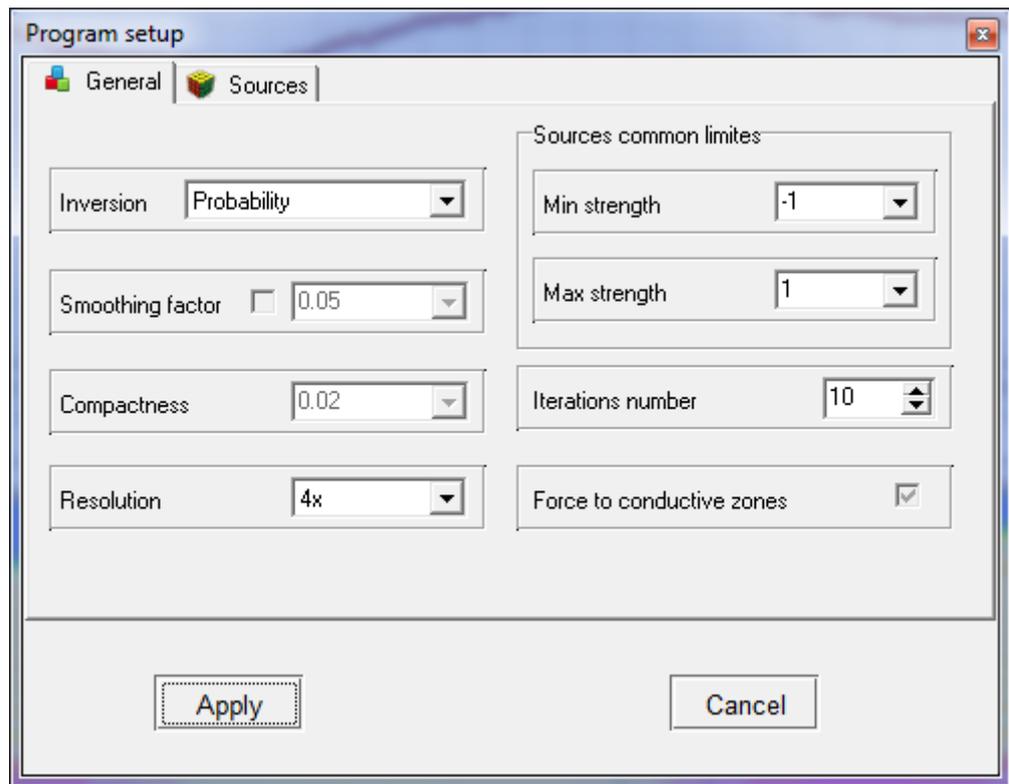


Рис. 8 Program Setup dialog box, General tab

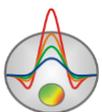
The **General** tab is used to select the basic settings of the inversion algorithm.

The **Inversion** field defines the algorithm by which the inverse problem will be solved.

Probability value is a statistical inversion algorithm based on the maximum likelihood criterion. This option is suitable for all types of sources except boundary-redox. It does not require any additional settings except the number of iterations.

The *Occam* value is a least-squares inversion using a smoothing operator. This option is suitable for all types of sources except boundary-redox. This is a more flexible algorithm that allows you to get different variants of the distribution of sources depending on the selected inversion settings.

Parametric value - includes the inversion algorithm for the problem with a small number of unknowns. It is used mainly to restore the dependence of the potential-determining factor with depth in the boundary-redox mode. But it may also be used for determining the distribution of volume sources in polygons (the source intensities in this case are the result of interpolation of values in nodes).



Smoothing factor - for Occam inversion, sets the relation between minimization of measurement discrepancy and smoothness of source distribution. With large values of the smoothing parameter, you often get large values of data inconsistency, but a smoother distribution of sources. If optimization (line-search) is disabled, the value of this parameter can be selected automatically (a check mark to the left of the option input field).

The **Compactness** field - for Occam inversion, allows to approximate the field with fewer sources, but with greater intensity, for parameter values(0.2-2).

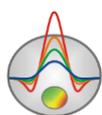
Resolution field - for Occam inversion, a parameter responsible for increasing the resolution of the inversion. The option allows increasing the influence of low-sensitivity sources (deep sources) and decreasing the influence of high-sensitivity sources (near-electrode sources). Thus, the resolution increases, i.e. the possibility to detect more deep sources

The *Min Strength*, *Max Strength* fields of the **Sources common limits** area set the limits of intensity variation for the selected source type. These values, if necessary, are selected empirically.

Iterations number - the inversion process stops when the set number of iterations is reached.

Force to conductive zones option - for Occam inversion, when this option is enabled, the algorithm will try to place intense sources in areas with higher conductivity, which corresponds to the concept of "ore" problem.

The **Sources** tab is used to set the types of sources and the potential-determining factor.



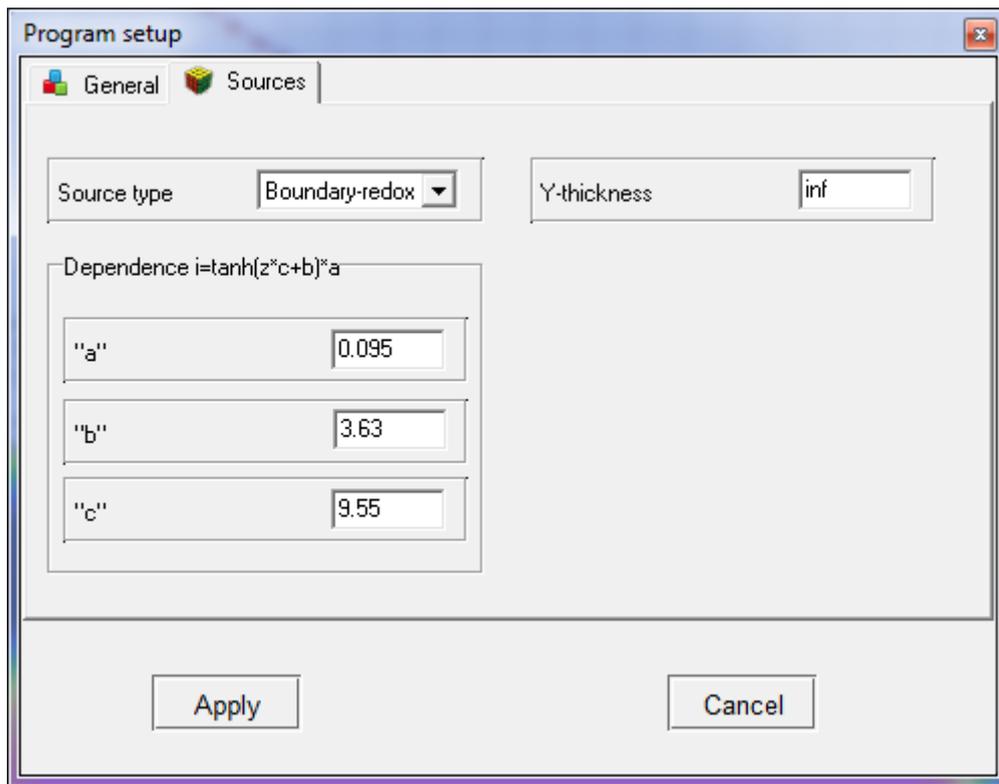


Рис. 9 Program Setup dialog box, Sources tab

Source type field - sets the type of natural electric field sources.

Boundary-redox value - sources of redox nature arising at the boundary of the electron and ionic conductor, due to the change in the potential jump at this boundary. In this mode of the program, sources are specified in the form of a distribution of the potential determinant with depth for the entire model and this dependence is reconstructed. The work is carried out in polygonal mode only, the current graph of the potential-determining factor is shown on the right side of the model. The color encircling of the polygons shows the intensity of the sources on the boundary.

When you switch to the *Boundary-redox* mode, the simulation ( direct problem solving) and settings of the *Dependence* area become available.

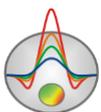
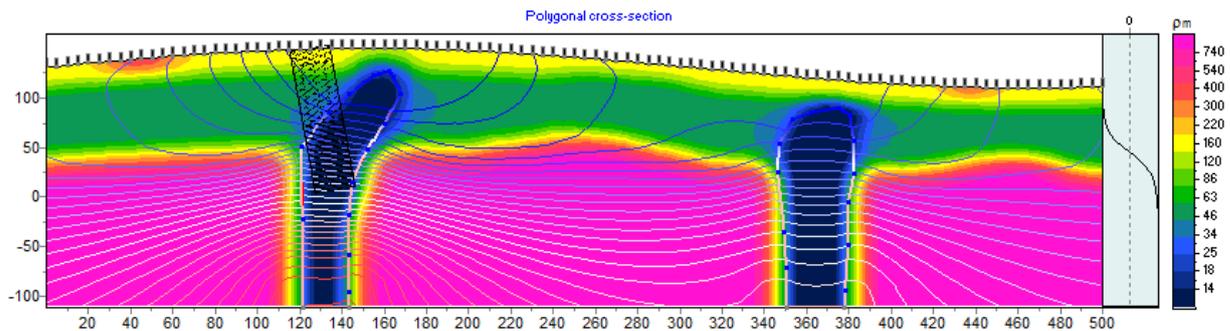


Рис. 10 Example model in boundary-redox mode

When inverting a potential-determining factor, the inversion algorithm must be chosen: *parametric*.

Volume pole value - the point source intensities are reconstructed in all (except surface) nodes of the network or nodes of the network falling into polygons (in polygonal mode). All variants of inversion are accessible.

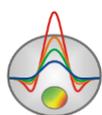
Volume dipole-X value - source intensities (horizontal dipoles) are reconstructed in all (except surface) nodes of the network or nodes of the network falling into polygons (in polygonal mode). All the variants of inversion are available. This option is suitable for working with hydrogeological tasks and reconstructing the distribution in the layer specified by the user.

Volume dipole-Z value - source intensities (vertical dipoles) are reconstructed in all (except surface) nodes of the network or nodes of the network falling into polygons (in polygonal mode). All variants of inversion are available.

Volume vector value - intensities of sources (dipoles of arbitrary direction) are reconstructed in all (except surface) nodes of the network or nodes of the network falling into polygons (in polygonal mode). All variants of inversion are available. Arbitrary dipoles are displayed in arrows, the color of which corresponds to the intensity modulus of the source. This option is suitable for working with hydrogeological problems and reconstructing the distribution in a user-defined layer.

The *Dependence* $i = \tanh(z * c + b) * a$ area appears in the *Boundary-redox* mode and is responsible for the distribution of the potential-defining factor with depth throughout the model. The dependence is defined by the formula in the area header (hyperbolic arctangent) and is shown graphically on the right side of the model. The coefficients a, b, and c can be user-defined or are in the process of inversion. The zero potentials, of each polygon, are also defined in the inversion process or can be set in the body setup dialog box.

The **Y-thickness** field is an important parameter determining the size of the body (source) in the direction perpendicular to the Y profile direction. By default, objects (sources) have infinite extent (inf) in Y direction (logarithmic potential), but in real tasks, it is necessary to limit the size of objects (in meters), based on the known geological information. The potential changes from ideal logarithmic for infinite (Y=inf) source, to point (Y=0). The logarithmic potential has more correct amplitudes for relatively small distances from the source, while the point source potential



shows a more correct attenuation character, but the amplitudes of the recovered sources will be greatly overestimated. A compromise of $Y=20-50$ метров seems satisfactory.

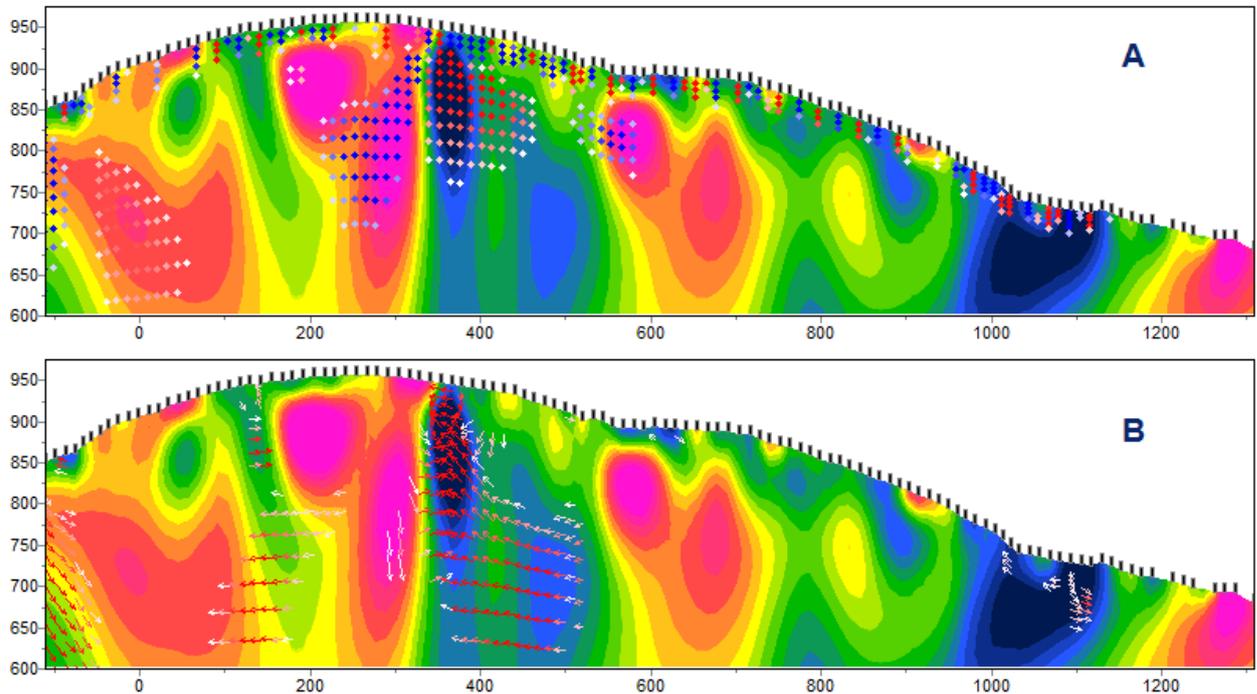
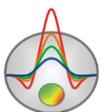


Рис. 11 Example of inversion for different types of sources. A-volume pole, B- volume vector

Field approximation by a set of extended objects

This algorithm is designed to approximate the observed natural fields by a limited set of elongated objects of different geometries with a linear distribution of sources along the incidence line. Such a formulation is typical for ore problems, where the natural field sources have a redox nature. Since the anomaly-forming objects can be very distant from each other and the problem requires a good initial approximation, a statistical annealing method is used to solve it. The search parameters are the "poles" of the objects: their geometry and intensity. The result of solving such a problem may serve as a good starting approximation when creating a polygonal model for the boundary-redox mode.



It should be noted that, as in all other cases, the solution of the problem is made taking into account the resistivity section and topography. The dimensionality of the object along the Y-axis is set in the program setup.

The main toolbar button  is used to start the search procedure. To stop the search use the ESC key.

When you right-click on this button, a pop-up menu appears with the following settings:

Num of objects: the number of objects with which the observed field will be approximated. When created, the objects will be placed evenly along the profile.

"+" on top: this option specifies that positive charges must be placed on the top pole of the object, otherwise negative charges will be placed on the top pole.

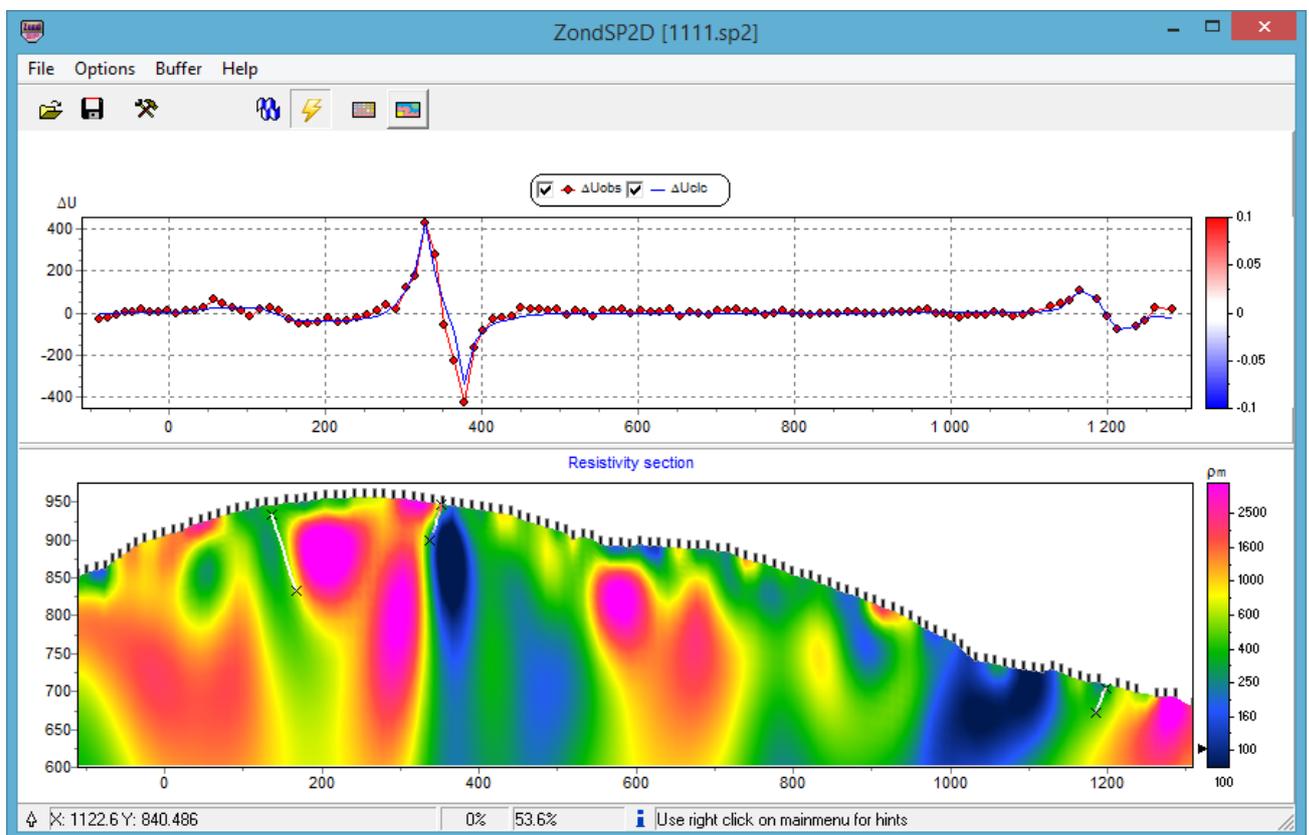
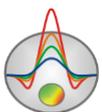


Рис. 12 Example of approximation of field data by a set of stretched objects

The objects in the program are shown as lines with a gradient fill, which determines the intensity of the sources. The fill colors correspond to the color scale shown to the right of the graphs in the upper section. When you right-click on such an object, a pop-up menu appears with options that allow you to change some settings for the next search cycle:



Lock object - lock the selected object in the next search cycle. Markers are displayed on the poles of the locked object in the form of a diagonal cross.

Lock object geometry - lock only the geometry of the selected object in the next search cycle. Markers in the form of a cross are displayed on the poles of the fixed object.

Unlock object - allows to search for intensities and geometry of the selected object.

Remove object - removes the selected object.

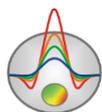
A priori information

Before performing an inversion, it may be necessary to enter existing a priori information. This may be a starting model built from geological data, data from other geophysical methods or the results of a previous inversion.

The geoelectric model (MOD1(2)D) obtained during data interpretation in another program of Zond package (for example, **ZondRes2D**) can be used as a priori data during inversion.

The substrate

If graphical a priori information is available, it is possible to use it (as a substrate for the model) using the option **Options/Import/Export/Background/Load background**. This can be, for example, a geological or seismic section, a section of a neighboring profile. After selecting the file, a dialog box for setting the image coordinates appears (see figure below), in which the coordinates of the image boundaries in the section coordinate system are specified.



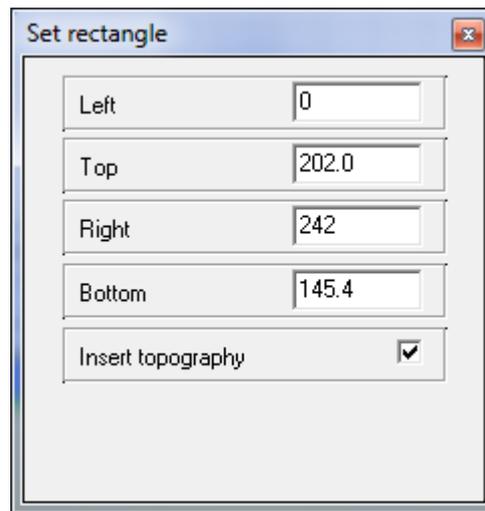


Рис. 13 Substrate settings dialog box

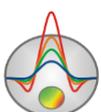
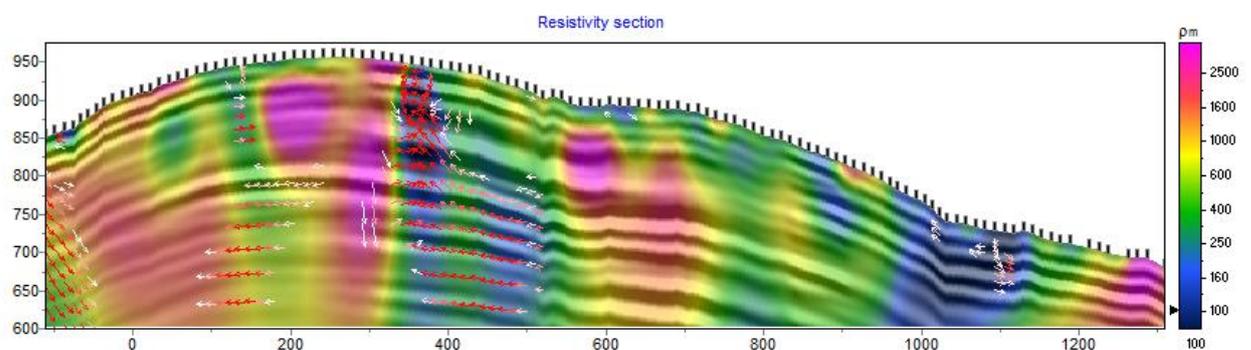
In this window you can manually set the coordinates of the upper left and lower right corners of the image.

Insert topography - allows you to change the image taking into account the topography (the upper border of the drawing will repeat the topography of the model).

The use of a substrate allows the results of other survey methods to be taken into account in the interpretation. The figure below shows an example where a seismic section is used as a substrate for a resistivity model.

In order to make the image transparent, in the model setup dialog box (right-clicking near the Resistivity section in the model window, the Setup option [\(more details\)](#), select the *transparency* mode.

The use of the substrate allows to take into account the results of other profile geophysical methods (for example, seismic survey) and geological sections along the profile when interpreting (specifying the contours of objects).



Well information

Availability of a priori information on the wells allows you to significantly increase the reliability of the resulting sections. **ZondSP2D** program has a built-in module that allows you to display well data in graphical form on the sections.

To create borehole data, select **Options/Boreholes/Create/Edit borehole data** in the main program menu. The **Add borehole data** module dialog box will appear (see figure below).

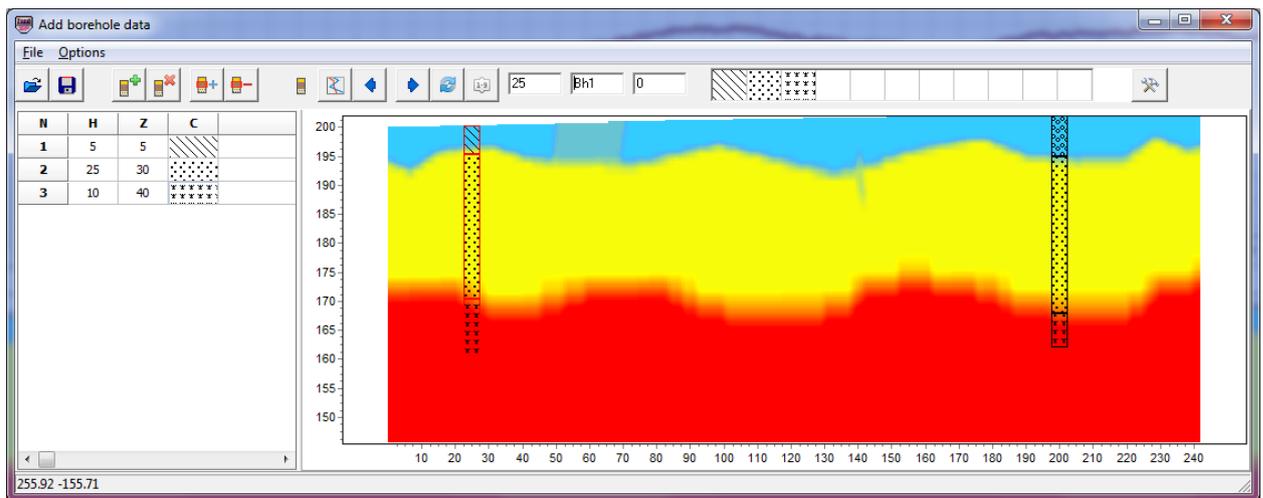
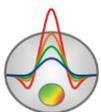


Рис. 15 The Add borehole data file dialog box (Create borehole data)

The toolbar of the dialog box contains the following buttons:

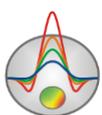
	Open the lithology file
	Save lithology file



	Create a new well
	Remove the well
	Add a layer in the well
	Remove the layer in the well
	Mode of lithological columns
	Logging data mode
	Go to the previous well
	Go to the next well
	Update the data drawing window
	Sort wells by coordinate
<input type="text" value="25"/>	Sets the horizontal coordinate (along the profile)
<input type="text" value="1"/>	Profile number <i>if the project consists of several lines</i>
<input type="text" value="Bh1"/>	Signature to the well (no more than 5 characters)
<input type="text" value="0"/>	Angle of inclination of the well in the XZ plane.
	Additional options

The module contains two main windows. On the left, there is **the Data window**, which contains a table with the following columns: **N** - sequence number of the layer, **H** - layer thickness in meters, **Z** - bottom depth of the layer in meters, **C** - fill type. In the right window, the data on the wells are displayed in graphic form.

To start creating a lithology file, click the button  on the toolbar. After that a new table will appear in **the Data window**. Use the button  to set the required number of layers. Then it is necessary to edit the table, setting values of power or depth of the bottom of each of the layers, and also to choose the type of filling according to the lithology. The dialog box of the **Pattern**



Color Editor fillings setup is invoked by double left-clicking in the **C** column of the data window (see figure below). The program offers a rich choice of lithological fills. In the **Color** option, you can select the color of the fill.

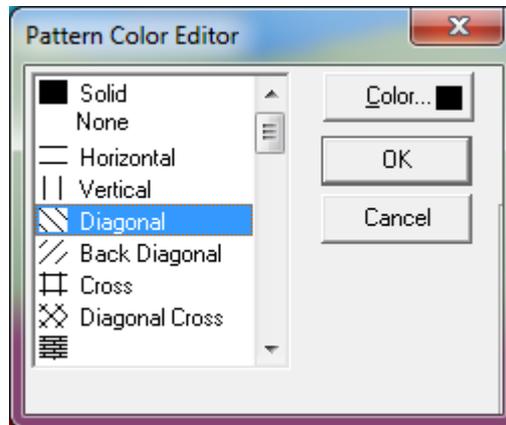


Рис. 16 **Fill editor window**

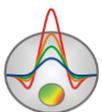
After finishing the well data input, you should press the button  and the well will appear in the graphical window. After that, it is necessary to set the horizontal and vertical coordinates of the well on the toolbar in kilometers, and then the well will be represented according to its coordinates. The active well is displayed in red color in the graphical window.

For convenient work with a large number of wells, the program has the possibility to create a palette. To create a palette, select the necessary fill in the Fill column **of the Data window**, and then right-click in the Fill area on the main program panel. In this way, you can create a set of fills, which you can then save. To do this, click the button  and choose **Save default palette**.

The saved set of fills can be used when creating a new lithology and logging data file ( - **Load default palette**).

Set borehole width, available by pressing the button,  sets the borehole width as a percentage of the profile length.

After saving the data file, several files will be created: ***.crt** - module project, which can be loaded in the program and ***.txt** - files for each well, the names correspond to horizontal and vertical coordinates. [More information](#) about the lithology file format.



To add borehole data, use the **Options/Borehole/Load borehole data** command. The borehole data will be displayed both in the section and in the area of the model editor (see figure below).

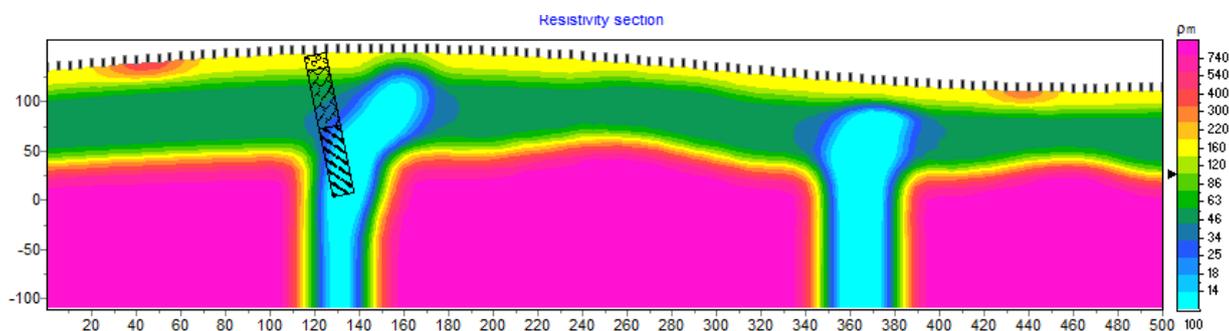
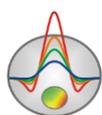


Рис. 17 **Displaying lithology data in the model**

If you have well log data, you need to prepare a file of the format described in the "[Logging and lithology data file format](#)" section and load it using the button .

The toolbar buttons of the lithology data assignment module are duplicated in the **Options** menu.



Visualizing results

3D visualization of geoelectric models and sources across multiple profiles

If several closely spaced profiles were measured on the study area, it is advisable to carry out their joint interpretation. This allows you to study the distribution of selected structures in the plan, and also facilitates the interpretation of each profile separately, simplifying the identification of the most stable elements of the model.

In the program **ZondSP2D** for joint visualization of geoelectric obtained by several profiles, the module **3D section viewer**, called with the menu item **Options/3D fence diagram**, is used. It allows to represent the obtained models in space (taking into account the relief), and also to plot the distribution of the selected parameter in the plan for a user-defined depth or absolute mark.

The volume visualization window consists of three tabs - **Lines** (setting of profile coordinates) (see figure below), **3D View** (model view window), **Options** (image settings) and Toolbar (provides access to additional parameters and the ability to load, save and export the built volume model).

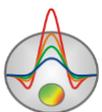
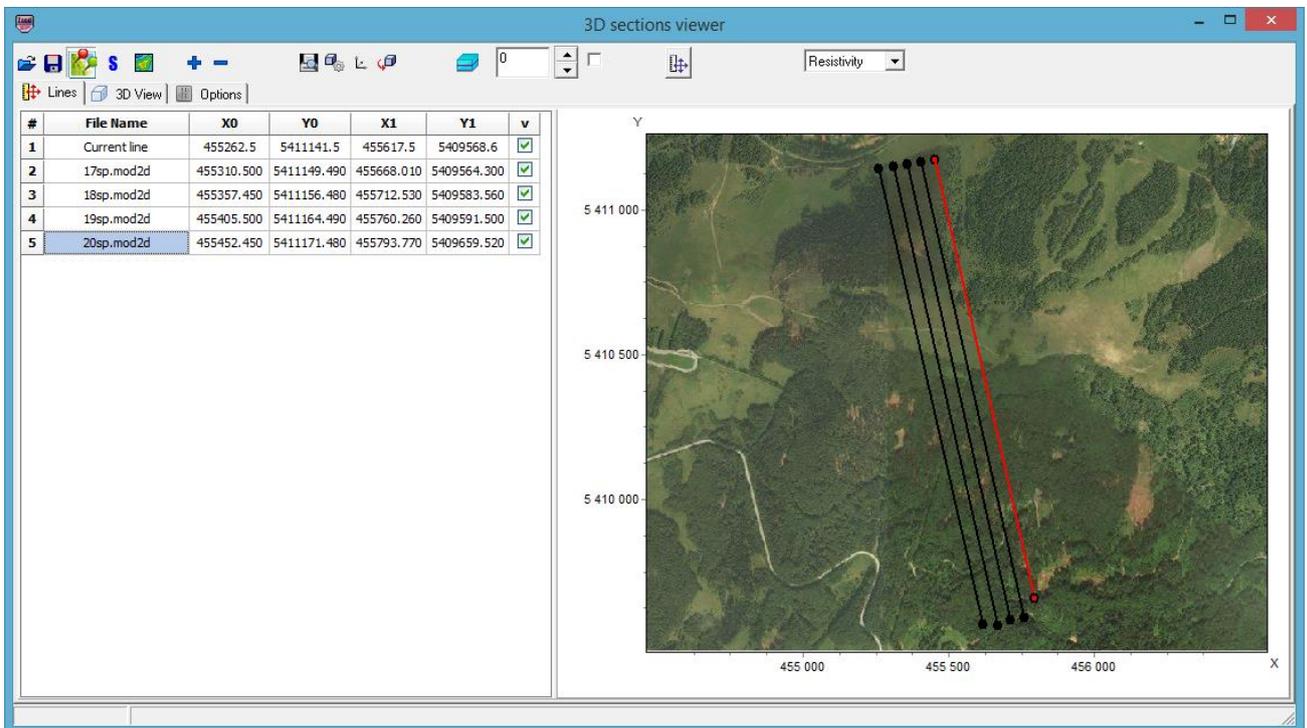


Рис. 18 The Lines tab of the 3D sections viewer window

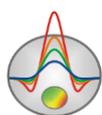
The **Lines** tab is used for adding profiles and entering coordinates. Each profile corresponds to a line in the table. To add a profile to the table, right-click an empty cell of the first column and select the necessary file. For volumetric visualization the program uses files with the extension *.mod2d, which are created automatically when you save the project in **ZondSP2D** format. You can add an empty row to the table or delete an unnecessary file from it using the buttons and   toolbar. Columns X0, Y0, X1, Y1 of the table contain rectangular coordinates of the start and end of the corresponding profile. It is necessary to note, that any mutual orientation of profiles is allowed - they can be parallel, non-parallel, intersecting or not intersecting. The plan of profiles is displayed as the coordinates are entered in the right part of the Lines tab window. The last column of the table allows to not display the selected profiles as needed.

The **Options** tab allows you to adjust the display parameters - the color scale and the scale on each of the axes. The scale ratio can also be set with the help of the toolbar  field. The field with the values contains scales of displaying on each of the axes. Clicking the button  allows you to switch to the mode of custom setting of axes scales.

- build a resistivity model for a set of profiles

- build a model of volumetric sources for a set of profiles

The 3D model itself is displayed in the **3D View** tab (see figure below).



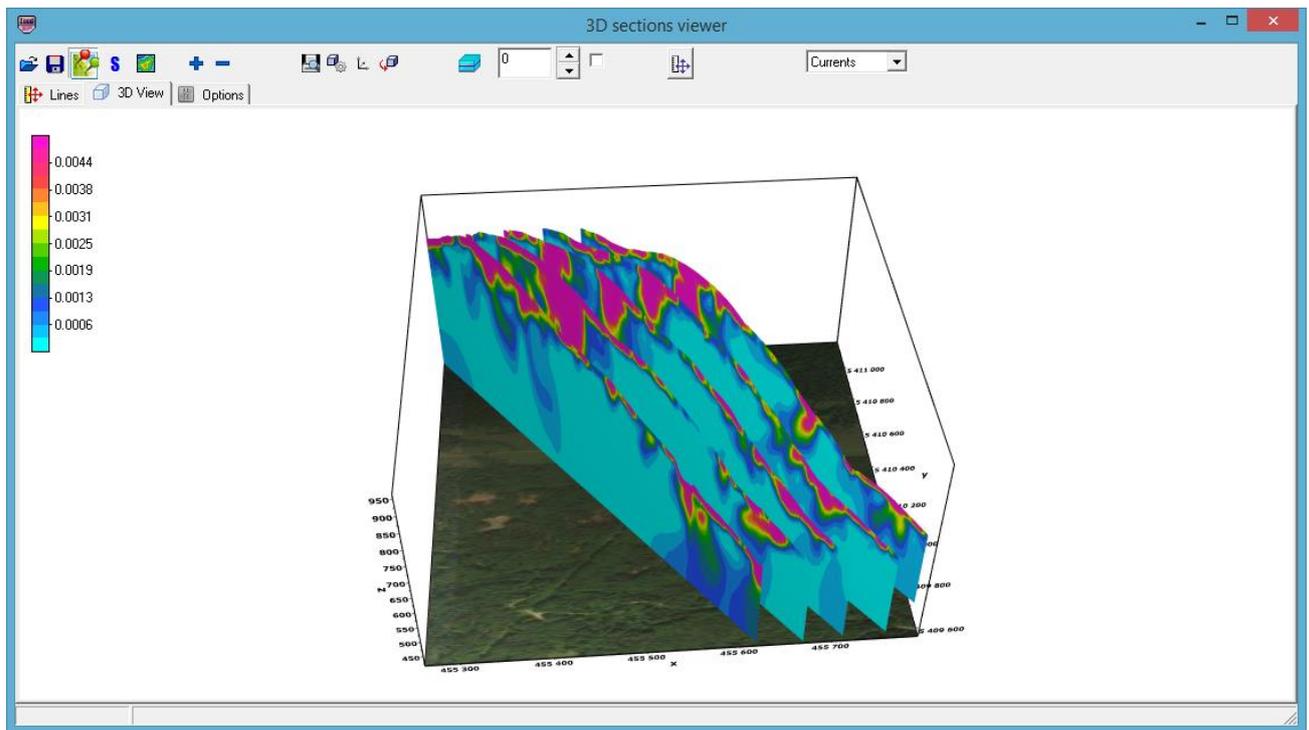


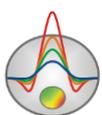
Рис. 19 The 3D View tab of the 3D section viewer window. Three-dimensional visualization

Configuration of the axes (each individually) is carried out with the help of the dialogs called by the toolbar  button. The corresponding dialog box allows you to change the properties of the axes' labels, customize the grid display, set the axes' headers, etc.

Access to settings related to the 3D model image (types of projections, scales, rotation angle, displacements of the whole model in screen coordinates, etc.) is performed with the help of the button .

You can also work with the created model with the mouse - with the left button pressed, the model can be rotated arbitrarily. The mouse wheel adjusts the scale. Pressing the button  automatically rotates the model around its geometric center.

A useful option is the ability to build a horizontal slice. The corresponding option is accessed from the toolbar toolbar set . The selection option, to the right of the input field, indicates whether the absolute height mark or depth from the surface will be used when building the slice. An example is shown in the figure below.



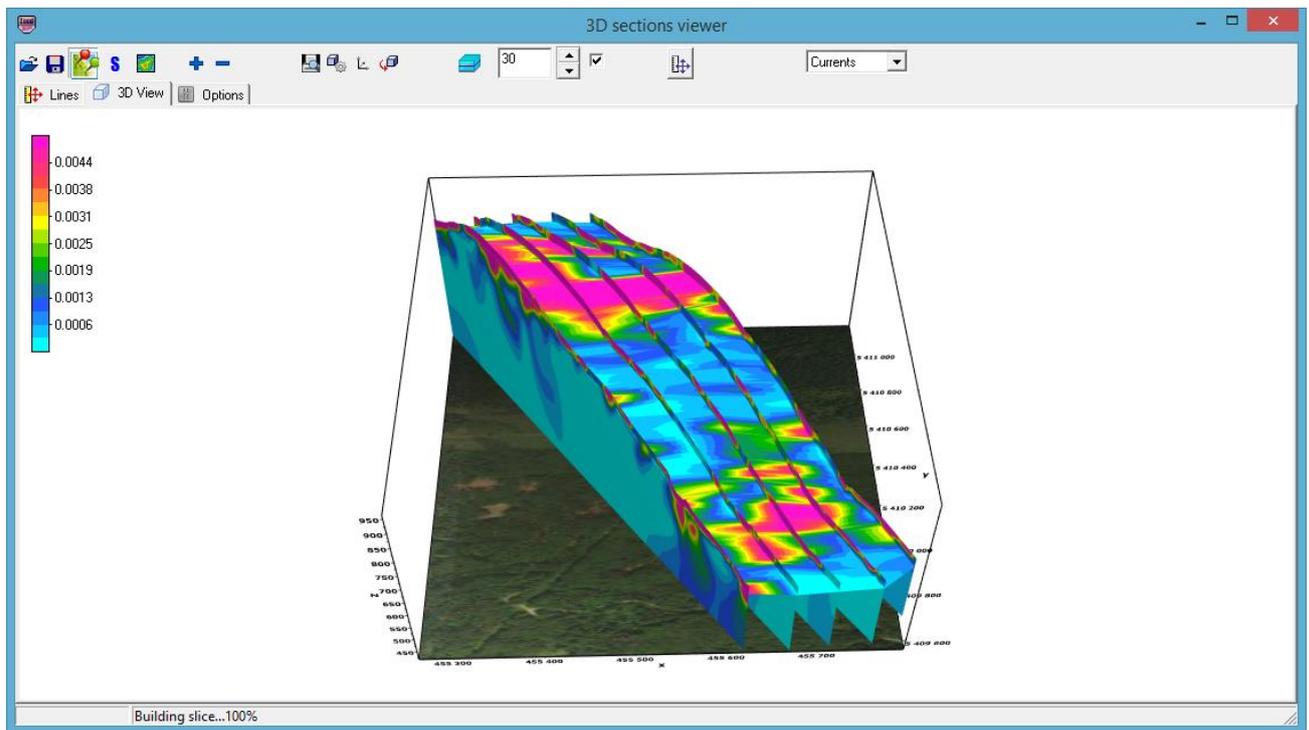


Рис. 20 The 3D View tab of the 3D sections viewer window. Resistivity map at a given depth from the surface

 - build an XY slice for a given depth.

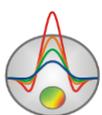
The button  allows you to load the carpet pad as a Bing map (Microsoft map service), the coordinates in this case must be in UTM format.

The button allows you to build  XY slices for a given depth by exporting the slices to Surfer.

The set of models in **the 3D section viewer** can be saved and then reopened with the and  buttons. If you select XY plane in the save dialog box, the program will create a text file *.dat for the current depth slice. This file can be used in external programs such as Surfer. When saving in Voxler 3d grid format, the program creates a text *.dat file that contains data for the entire model.

The model window can be printed by pressing the toolbar  button.

The *Options* tab of the **3D sections viewer** window



The *Color scale* area allows you to adjust the fill parameters. The **Palette** button opens the Fill Settings dialog box ([see details](#)). The *Color scale limits area* lets you set the minimum and maximum for the color scale manually or choose to define the limits automatically by checking the corresponding checkbox.

Continuous option If the option is enabled, the cut will be built using a continuous color palette, otherwise by a set of contours.

The *Boreholes* option shows well data in three-dimensional image. If there are many wells in the project, their displaying may take considerable time

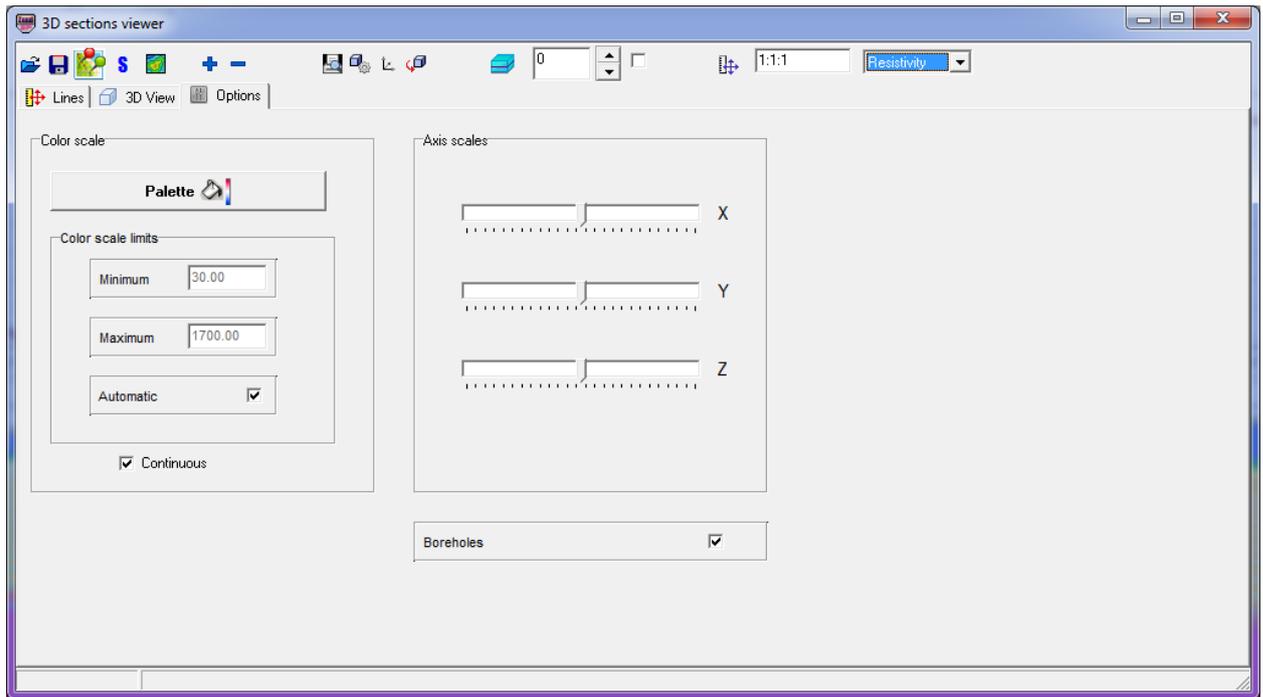
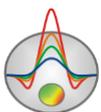


Рис. 21 The Options tab of the 3D sections viewer window. Resistivity map at a given depth from the surface

Geological section editor

To build a geological-geophysical model (to perform geological interpretation), use the **Geological model editor** window, called through the **Options/Geological editor** menu. The editor allows to interactively create a geological model based on the current project model, borehole data, data from other **Zond** programs and a priori raster information, print the resulting sections at a given scale, save and export the interpretation results.



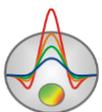
When you call the Geological Model Editor window, it displays the current project model. To load a priori information in the form of *. sec files, the menu item **File/Open section** is used to load borehole data - **Options/Load borehole data**.

The results of geophysical interpretation serve as a kind of color substrate over which the geological model is built. During model building, local objects and layers are highlighted, on which the geologic mappings selected by the interpreter are then applied. The module also allows you to display borehole data, which greatly simplifies the model building process.

Thus, the main task of the module is to quickly build geological sections based on geophysical results and further export to CAD systems.

Before starting work, it is necessary to choose the type of cut and its graphic settings very carefully. The best option is to represent the cut in the form of contours.

An example of a geoelectric section and the geological model created on its basis is shown in the following figure. The **File/Remove picture** menu item allows you to remove the substrate - the section of the parameter, on the basis of which the interpretation is carried out.



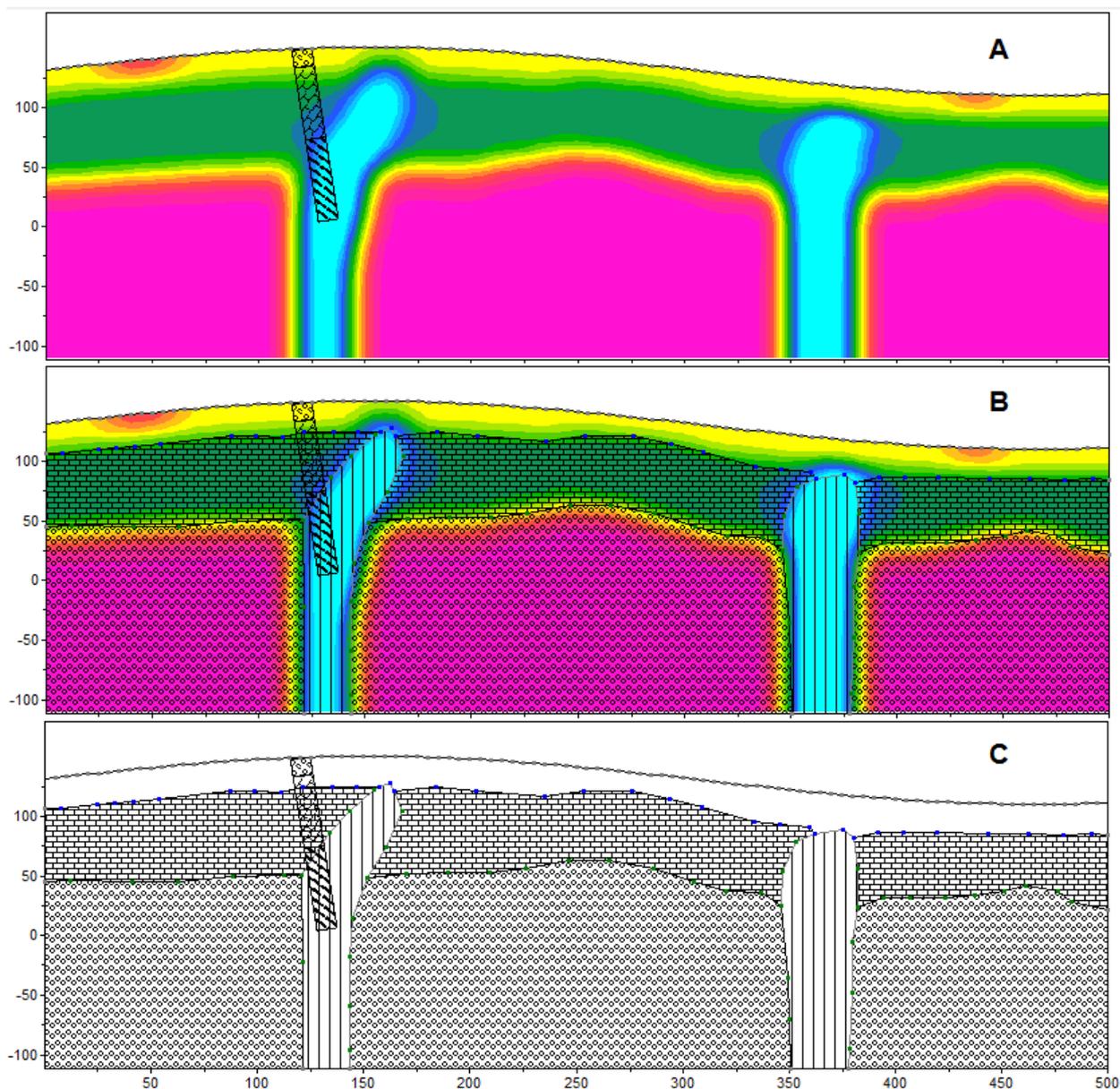
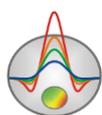


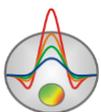
Рис. 22 Geological editor dialog window, where A - window before the work, B - window in the work mode, C - the final geological interpretation

For geological interpretation, the toolbar has two sets of buttons: one for creating polygonal bodies (with the ability to edit the mottles, fills, boundary style, etc.), the other for drawing individual lines in the section. They can be used to mark some elements of the geological structure (e.g., planes of tectonic faults) or in general to increase the clarity of the interpretive section.

The toolbar of the **Geological editor** window contains buttons for editing polygons:



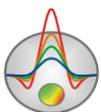
Tool	Option
	Creating a polygon. Clicking the left mouse button adds a new node to the polyline - the polygon boundary. The description of the polygon boundary is completed by right-clicking, after which the polyline is automatically closed.
	Deleting a polygon. After selecting a tool, right-click on the polygon to be deleted.
	Create a polygon adjacent to an existing polygon or boundary of the modeling area. After selecting this tool it is necessary to specify a boundary that is not adjacent to an existing polygon. The first and the last point of the boundary should belong either to the boundary of the adjacent body or to the boundary of the modeling area. The boundary assignment is completed by clicking the right mouse button. The program will select the adjacent boundary automatically or offer to select it with the help of a dialog box.
	Disconnect adjacent polygons. If a model containing adjacent polygons was created with the previous tool, this tool allows you to disconnect them to enable unlinked boundary changes, movement, and deletion of the polygon. After selecting the tool with the left mouse button, you select the polygon to be separated (click once at any point of the polygon, and its boundaries change color). Pressing the right mouse button completes the disconnection procedure.
	Divide a polygon by a straight line (create two polygons from one polygon). Click the left mouse button to indicate the first point of the line, then the right mouse button to indicate the second point. Both points should be on the boundary of the polygon to be divided.



	Move a polygon. Select the polygon by clicking the left mouse button. When you move the mouse the polygon is moved. The position of the polygon is fixed by clicking the right button.
	Move part of the polygon
	Add a node. Right-click on the boundary point where you want to add the node.
	Delete a node. Right-click on the node you want to delete.
	Move a node. Select the node by clicking the left mouse button, move it by moving the mouse, finish moving it by clicking the right mouse button.
	Disconnect connected points. This mode is intended to disconnect linked points. To disconnect connected points, right-click on the polygon. As a result of this operation instead of one connected point there appears a set of unconnected points, each belonging to its own polygon. The points of the polygons change their color to red when the cursor approaches.
	Move Point. This mode is intended for moving the point of the polygon. Click the left mouse button to select the point to move; then the polygon point moves with the cursor. Click the right mouse button to fix the new position of the point. If the operation is not possible (i.e. some edges are crossed) the program does not allow the user to move the point and returns it to its original position. Points located on the model boundary only move along the corresponding edges. Polygon points change color to red as the cursor approaches.

Double-click in the center of the polygon to open the Polygon Settings dialog box (see details in the [PolygonSettingsdialog box](#)).

Also on the control panel are buttons for creating and editing lines:



	Add a line
	Move the node
	Remove the node
	Add a node
	Delete line
	Create a polygon of two lines
	Move the line
	Save the line
	Cancel last action

The **File** menu of the **Geological editor** window contains the following functions:

Open section - allows you to load a custom file with polygons (graphic image).

Save section - allows to save the current geological and geophysical model.

Remove picture - hide the background from the editor.

Print preview - displays the Print dialog box.

The **Option** menu of the **Geological editor** window contains the following functions:

Automatic scaling - turn on the mode of automatic image scaling

Delete all polygons and lines

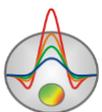
Get from modeling - use polygons from polygonal modeling mode.

Load borehole data - load borehole data from a file

Remove borehole data - remove borehole data from the editor

Output settings - call the Export Image Settings dialog box [\(more details\)](#).

Saving interpretation results



The result of data profile interpretation is stored in a binary file of **ZondSP2D** format (extension *.SP2). This file stores field data, values of relative weights of measurements, the current environment model and other information. During subsequent loading, the data from the file is used to create a model of the environment.

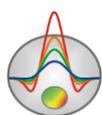
To save the result of the interpretation, click the button  on the toolbar or the corresponding menu item **File/Save file**. In the dialog box that appears, it is also possible to select the type of information to be saved: **Project data** file, image files (**Model, WorkSheet**) in the *. BMP format in the necessary scale. The variants of file saving formats are given below:

Zond project data	Save the project with interpretation results and other information.
Zond project with calculated	Save all results in the project, but replacing the observed data with calculated ones.
Zond calculated data	Save the calculated values to a text data file. For example, the results of a <i>boundary-redox</i> simulation.
Worksheet	Save the three graphical sections of the window in the BMP format.
Model	Save the bottom graphic section of the window as a BMP format. To adjust the image scale you should use the Picture settings dialog box (more details).
Grid file	Save the current model as a grid file in DAT format.
Section file	Save the current model in sec format (a graphical file with a binding of corners).
Sources file	Save the current distribution of sources as a grid file in DAT format.

The image scale can be adjusted using the Picture settings dialog box under [Options/Extra/Bitmapoutputsettings](#).

Settings dialogs

Exported image settings dialog box



The **Options/Extra/Bitmap output settings** dialog allows you to adjust the **Vertical scale** (in meters per centimeter), **Horizontal scale** (in meters per centimeter), **Print resolution** (in DPI) and **Font size** of the exported image.

These settings are applied to the model (Model)  saved in BMP format if the **Automatic** option is off. Otherwise, the image is saved as it appears on the screen.

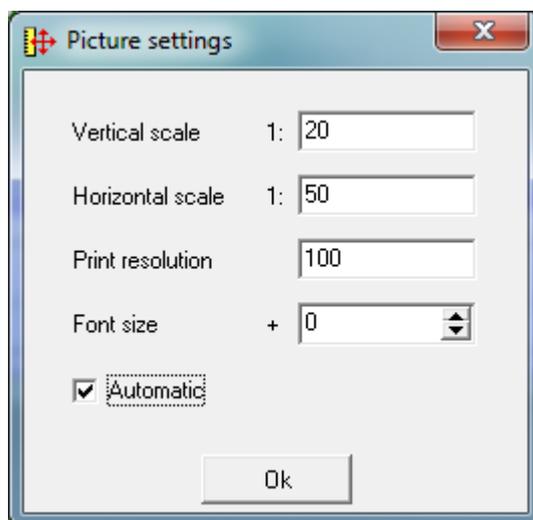
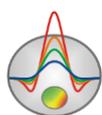


Рис. 23 **Bitmap output settings dialog box drawing is old**

Model display settings dialog box

The settings dialog for the model display is invoked by selecting **Setup** by right-clicking at the top of the model window in the area of the inscription "*Resistivity section*".

Options tab



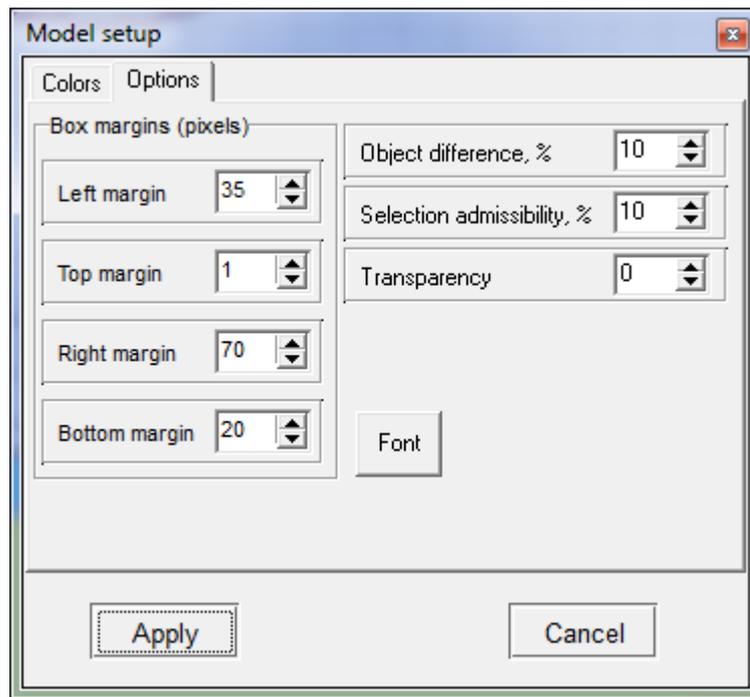


Рис. 24 Model Setup dialog, Options tab

Box margins area.

The **Left margin field** sets the margin (in pixels) of the image from the left edge of the window.

The **Right margin field** - sets the margin (in pixels) of the image from the right edge of the window.

The **Top margin field** sets the margin (in pixels) of the image from the top edge of the window.

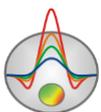
The **Bottom margin field** - sets the margin (in pixels) of the image from the bottom edge of the window.

The **Object difference, % field** sets the maximum value of the adjacent cells parameters ratio, above which the border is drawn between them.

The **Selection admissibility field, %** - establishes the permissible level of differences in adjacent cell parameters, at which cells are a single object and are selected together (in the Magic Wand selection mode).

The **Transparency** field sets the transparency.

The **Font** button brings up the Font settings dialog box.



Colors tab

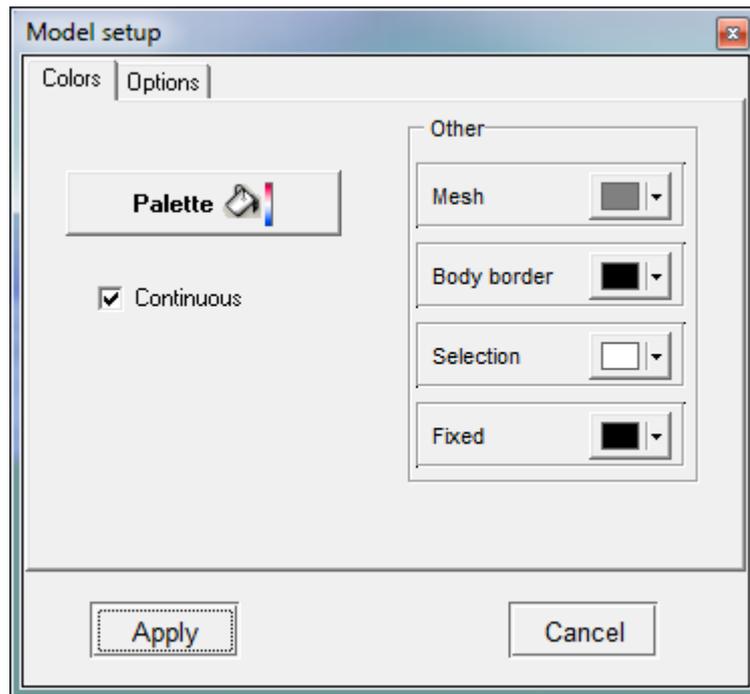


Рис. 25 Model Setup dialog, Colors tab

Press the **Palette** button to open the Palette Settings dialog box ([see details](#)).

The **Continuous** option allows to visualize cuts in a smoothed or contoured form.

Area **Other**.

The **Mesh** field sets the color of the mesh.

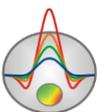
Body border field - allows you to set the color of the border between neighboring cells, if the degree of difference between them is greater than a certain limit.

The **Selection** field sets the color of the selected cell label.

The **Fixed** field sets the color of the fixed cell label.

Palette settings dialog box

The dialog is for configuring the palette of the program object and is invoked by the Palette button (see figure below). The dialog allows you to choose one of the default palettes (straight and reverse rainbow, grayscale, etc.) or create a custom scale. To add a slider to the scale, use the



right mouse button with the Ctrl key pressed. To delete a slider, use the Delete key. You can also save a custom palette using the , button  or load an existing one using the .

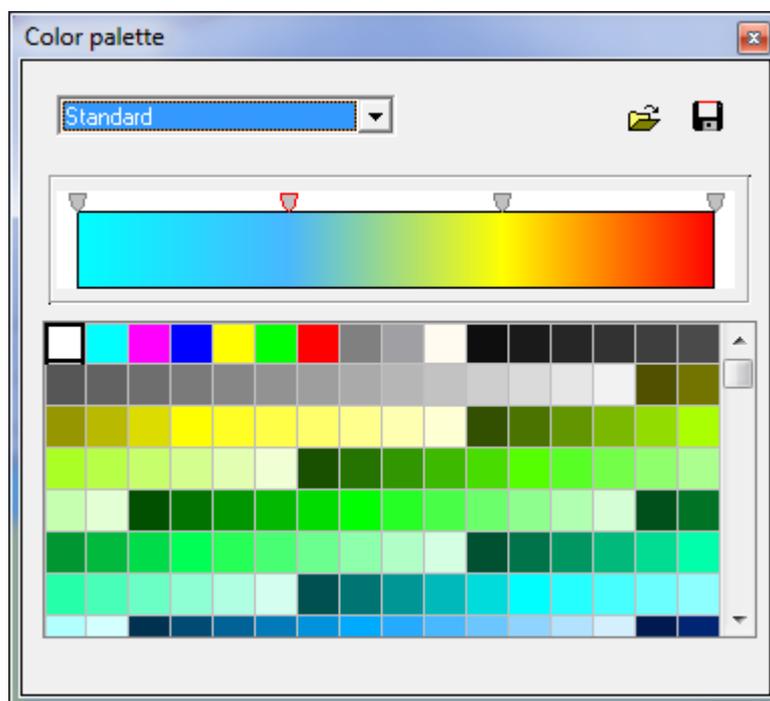
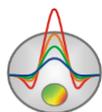


Рис. 26 Palette settings dialog box

The palette can be changed, edited, loaded from a file, and saved in a *.clr file of the Surfer.

Graphics Editor



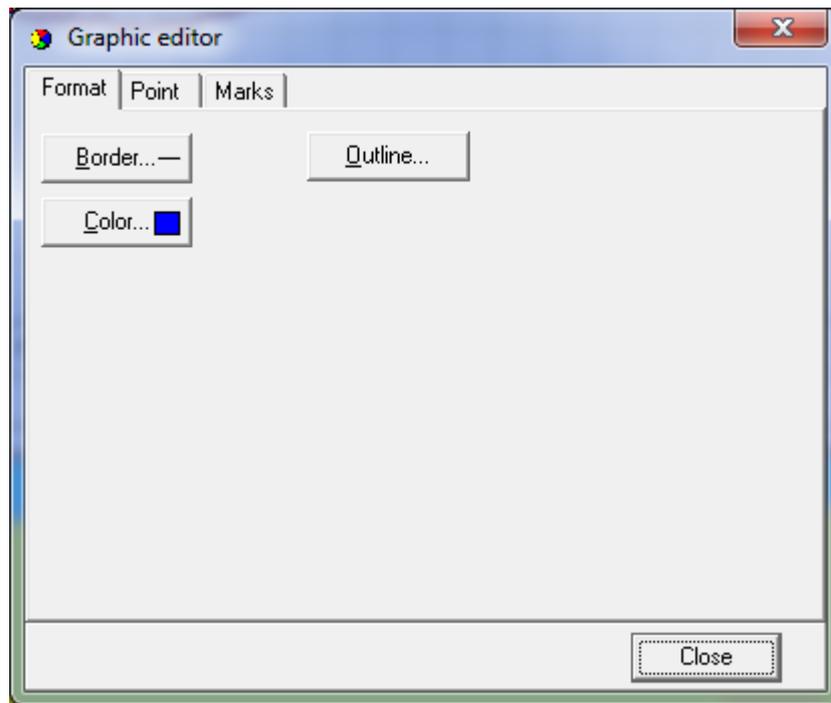


Рис. 27 Graphics Editor dialog box, Format tab

The editor is intended for adjusting the appearance of the graph. It can be called by right-clicking with the SHIFT key on the graph.

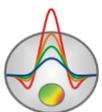
The **Format** tab contains settings for the connecting lines of the graph.

The **Border** button brings up a dialog box for setting the parameters of the connection lines of the graph.

The **Color** button brings up a dialog box for selecting the color of the graph.

The **Outline** button brings up a dialog box for setting the parameters for outlining the connecting lines of the graph.

The **Point** tab contains the settings for the graph pointers.



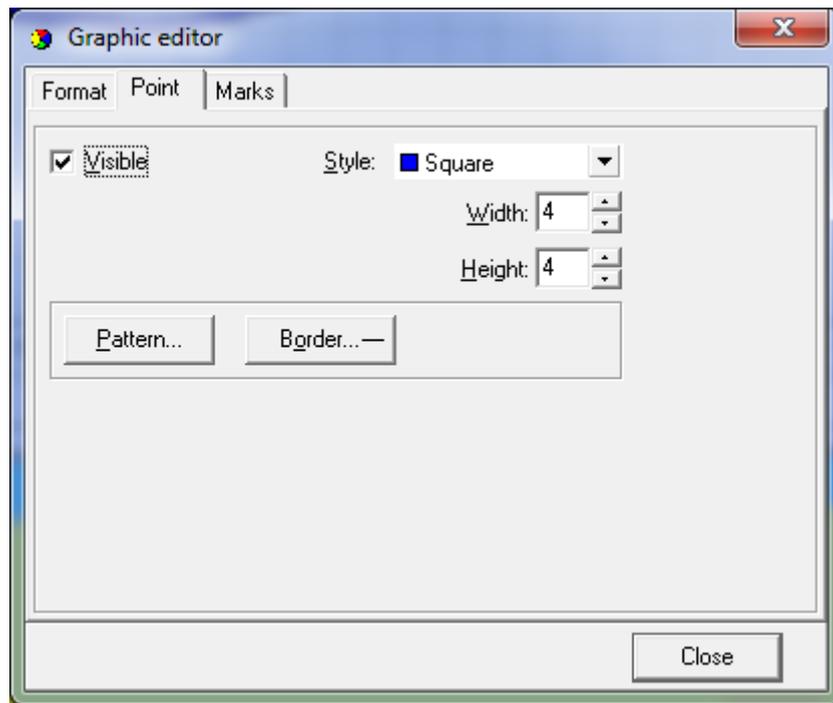


Рис. 28 Graphics Editor dialog box, Point tab

The **Visible** option allows you to show/hide the graph pointers.

The **Style** option sets the shape of the pointer.

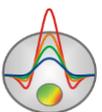
The **Width** option sets the width of the pointer in screen units.

The **Height** option sets the height of the pointer in screen units.

The **Pattern** button opens a dialog box for selecting the pointer fill options.

The **Border** button opens a dialog box for configuring the pointer outline settings.

The Marks tab contains the settings for the labels of the graph indicators.



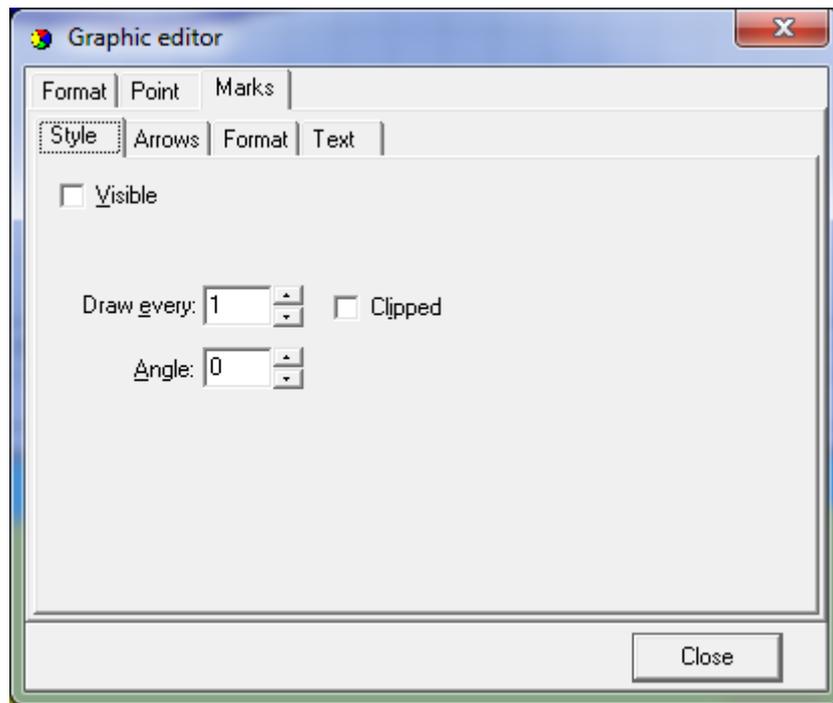


Рис. 29 Graphics Editor dialog box, Marks tab

The **Style** tab.

The **Visible** option allows you to show/hide the captions to the graph pointers.

The **Draw every** option allows you to draw every second, third, etc. signature, depending on the value selected.

The **Angle** option determines the angle of the caption text to the pointers.

The **Clipped** option specifies whether a caption should be drawn to a pointer if it extends beyond the graph area.

The **Arrows** tab is used to customize the appearance of the arrow going from the caption to the pointer.

The **Border** button brings up a dialog box for setting the arrow line parameters.

The **Length** option sets the length of the arrow.

The **Distance** option sets the distance between the arrow head and the chart pointer.

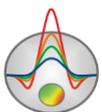
The **Format** tab contains the graphical settings for the frame around the pointer caption.

The **Color** button opens a dialog box for choosing the background color of the frame.

The **Frame** button brings up the Frame Line Settings dialog box.

The **Round Frame** option allows you to display a frame with rounded corners.

The **Transparent** option sets the degree of transparency of the frame.



Text tab:

The **Font** button brings up a dialog box to set the font for the signatures of the pointers.

The **Outline** button brings up a dialog box for configuring the outline lines of the pointer captions.

Axis editor

Many objects of the program contain coordinate axes. To customize the appearance and scale of coordinate axes, you use the Axis Editor. You can call it by right-clicking with the SHIFT key on the axis of interest.

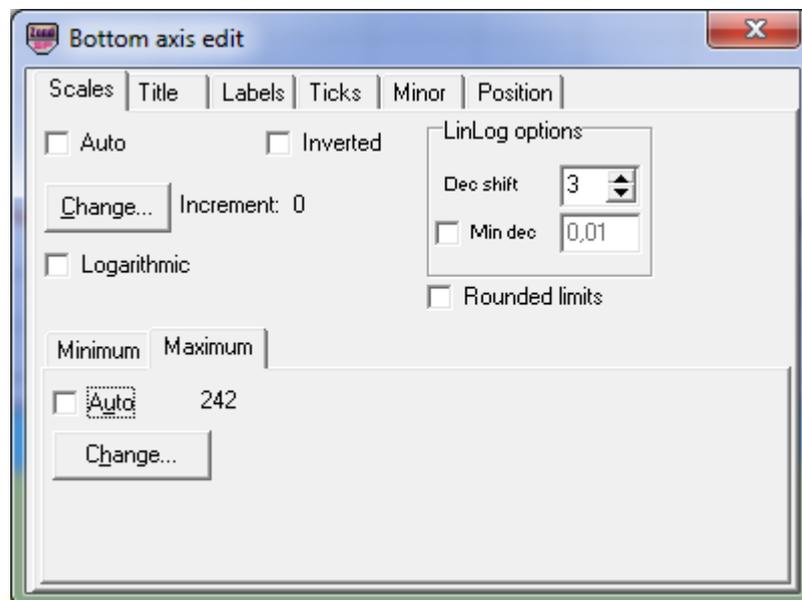
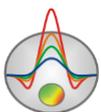


Рис. 30 Example of the lower axis editor dialog box

This brings up a pop-up menu with three options: **Options**, **Default**, and **Fix range**. The first brings up a dialog box, the second sets the values equal to the default ones, and the third fixes the axis within the current limits.

The Scales tab contains options related to adjusting the scale parameters of the axis.

The **Auto** option tells the program how to determine the minimum and maximum of the axis. If the option is enabled, the axis limits are found automatically, otherwise they are set by the user in the Minimum and Maximum areas.



The **Inverted** option determines the orientation of the axis.

The **Increment change** button brings up a dialog box for setting the axis label step.

The **Logarithmic** option sets the axis scale - logarithmic or linear. In the case of an alternating axis you should additionally use the options of the **LinLog options** area.

The **LinLog options** area contains options designed to adjust the linear-logarithmic axis. The linear-logarithmic scale allows you to represent alternating or zero containing data on a logarithmic scale.

The **Dec Shift** option sets the offset (in logarithmic decades) relative to the maximum modulo limit of the axis to zero. The minimum (pre-zero) decade has a linear scale, the rest have a logarithmic scale.

The **Min dec** option sets and fixes the value of the minimum (pre-zero) decade, if the option is enabled.

The **Rounded limits** option tells the program whether or not to round the axis minimum and maximum values.

The **Minimum** and **Maximum** areas contain a set of options for setting the axis limits.

The **Auto** option determines how the axis limit is determined - automatically or set with the **Change** button.

The **Title tab** contains options related to setting the title of the axis.

The **Style** tab:

The **Title** option defines the title text of the axis.

The **Angle** option determines the angle of the axis header text.

The **Size** option defines the indent of the axis header text. When set to 0, the indentation is set automatically.

The **Visible** option allows you to show/hide the axis header.

Text tab:

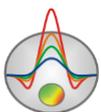
The **Font** button brings up a dialog box for setting the font for the axis header.

The **Outline** button brings up a dialog box for setting the outline lines of the axis header letters.

The **Labels** tab contains options related to configuring the axis labels.

The **Style** tab:

The **Visible** option allows you to show/hide axis signatures.



The **Offset** option defines the offset of the axis signatures. When set to 0, the indentation is automatic.

The **Angle** option defines the angle of the axis caption text.

The **Min separation %** option sets the minimum percentage distance between signatures.

Text tab:

The **Font** button brings up a dialog box for setting the font for the axis captions.

The Ticks tab contains options related to setting the main axis labels.

The **Axis** button brings up the Axis line setup dialog box.

The **Grid** button brings up a dialog box for setting the grid lines of the main axis labels.

The **Ticks** button brings up a dialog box for setting the lines of the main external axis labels.

The **Len** option sets their length.

The **Inner** button brings up a dialog box for setting the lines of the main internal axis labels.

The **Len** option sets their length.

The **At labels only** option tells the program to draw the main labels only when there is a signature on the axis.

The **Axis behind** option sets the order in which the axes and graphs are drawn.

The **Minor** tab contains options related to the setting of intermediate axis labels.

The **Grid** button brings up a dialog box for configuring the grid lines of intermediate axis labels.

The **Ticks** button brings up a dialog box for setting the lines of intermediate external axis labels. The

Length option sets their length.

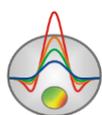
The **Count** option sets the number of secondary labels between the main labels.

The **Position** tab contains options that define the dimensions and position of the axis.

The **Position %** option sets the offset of the axis on the graph relative to the standard position (as a percentage of the graph size or screen units, depending on the value selected by the Units option).

The **Start %** option sets the offset of the axis start on the graph relative to the standard position (as a percentage of the graph size).

The **End %** option sets the offset of the axis end on the graph relative to the standard position (as a percentage of the graph size).



The **Other side** option allows you to draw the axis from the back side. If the option applies to the bottom axis, the axis will be drawn from the top.

Print preview dialog box (Print preview)

The print preview dialog box can be invoked from the **File/Print preview** main menu of the program. It is also available by double-clicking with the right mouse button in the area of any program object.

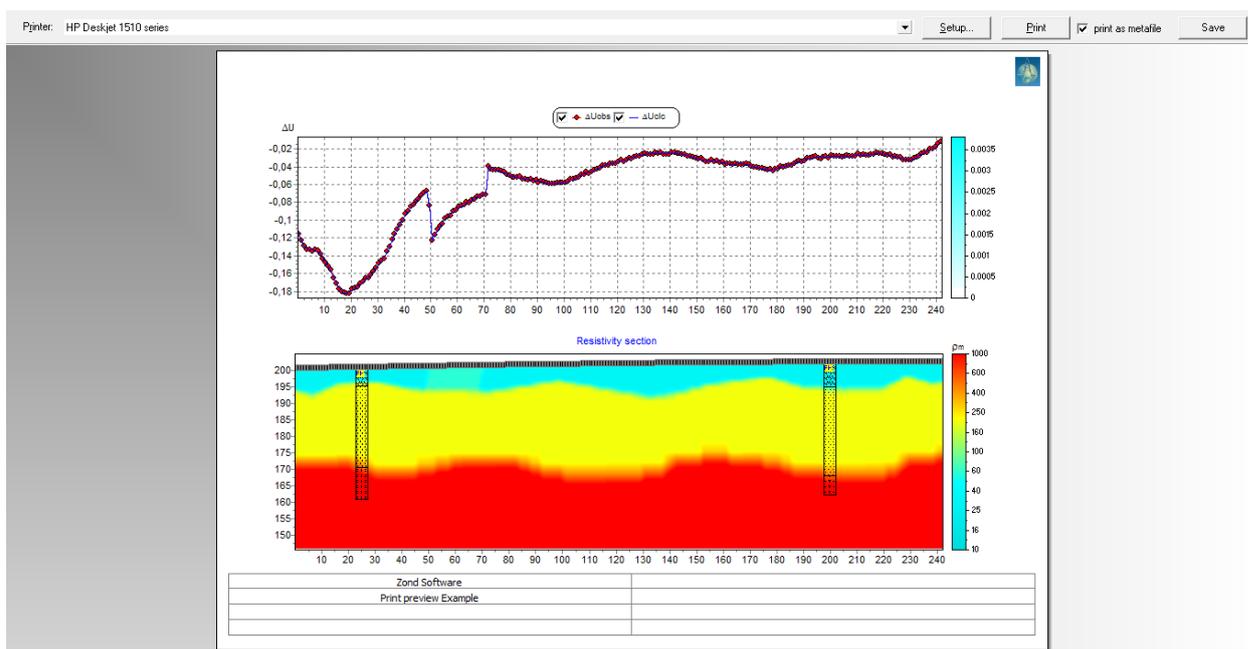


Рис. 31 Print preview dialog box

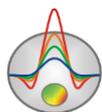
Use the left mouse button to move the print object on the sheet.

The main menu of the **Print Preview** window contains the following buttons

Printer: HP Deskjet 1510 series

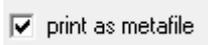
- to select the printer you want to print. In the menu that opens, you can select one of the configured printers.

- print settings button. In the window that opens, you can select paper size and orientation, print properties, number of pages per sheet, and other parameters.





- with the help of this button, after changing the necessary parameters, you can send the drawing to print.



- Send to print or save the image in vector form.



- saving to bitmap files/PNG files

The squares at the top of the sheet are for seals, stamps, or company emblems. Right-click the square and in the window that appears, select the bitmap you want to insert. The size of the square can be resized with the mouse.

At the bottom of the sheet is an editable table. To add text, right-click in the table area and type the desired text in the window that appears. You can also save all the comments in the table files by clicking the button  , or load the already saved captions by clicking the button .

SP2 master data file format

The program provides a universal data format that includes information about the coordinates of electrodes, marks of relative elevations and the actual measured values of the potential difference. Data format of **ZondSP2D data files** has an extension *. SP2.

A file usually contains several columns. The top line contains the keys that tell the program what type of data a particular column belongs to.

Possible key values are given below

p1x - horizontal coordinate of the first receiving electrode

p1z - excess of the first receiving electrode (column may be missing).

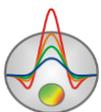
p2x - horizontal coordinate of the second receiving electrode

p2z - excess of the second receiving electrode (column may be missing).

dV - measured values of potential difference for a given pair of electrodes (if values are set in millivolts, the source intensities will be milliamps, if in volts - amps).

X - position of the measurement point on the graph (column may be missing).

Z - elevation excess at the point corresponding to column X (column may be missing).



The last two keys are used to set the horizontal position of the measurement point on the graph. If they are not set, the coordinate calculation is done automatically: for the gradient setting - the center of the receiving line, for the potential setting - the position of the moving electrode.

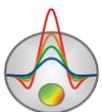
It is recommended to use a gradient setting when creating a data file (potential setting data is easily recalculated to a gradient setting). In any case, if a potential setting is suspected, the program will prompt you to recalculate the data into a gradient setting.

Example data file:

```
p1x p1z p2x p2z dV
0 131.00 5 132.00 -0.78
5 132.00 10 132.99 -0.83
10 132.99 15 133.97 -0.89
15 133.97 20 134.95 -0.96
20 134.95 25 135.91 -1.03
25 135.91 30 136.86 -1.11
30 136.86 35 137.79 -1.19
35 137.79 40 138.70 -1.28
40 138.70 45 139.59 -1.38
45 139.59 50 140.45 -1.48
50 140.45 55 141.29 -1.58
55 141.29 60 142.10 -1.68
60 142.10 65 142.88 -1.77
65 142.88 70 143.63 -1.86
70 143.63 75 144.35 -1.92
75 144.35 80 145.03 -1.95
```

Logging and lithology data file format

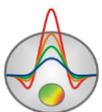
To create a lithology file it is recommended to use the built-in module of **ZondSP2D** program ([more details](#)). Logging data and lithology columns are stored in files of a certain format. The first type of file with txt extension is the actual data, well logs or lithology files. When creating a well log data file, the following file structure is used:



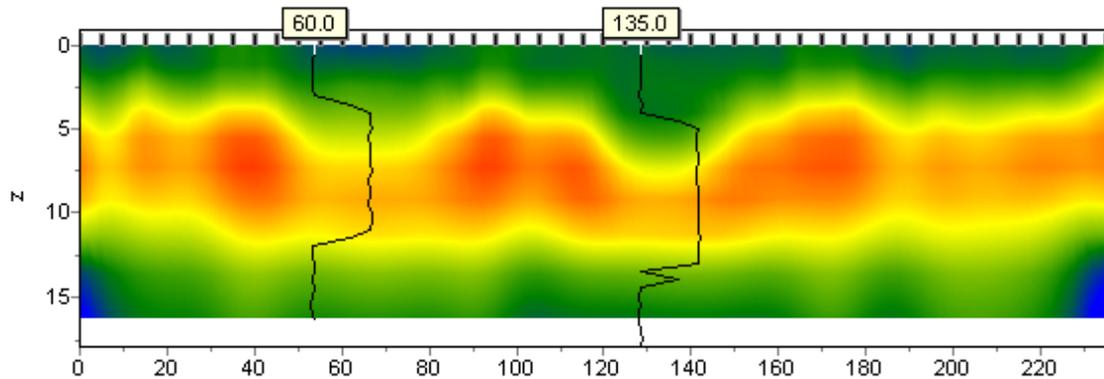
The first column contains the depth of the record point (from the ground surface), the second column contains the log measurements. The third and fourth columns contain zeros.

Below is an example of a log data file:

```
0.5118 .303539400
      1126.900238400
1.5123 .417088800
      2116.151957400
2.5117 .24088400
      3111.942417400
3.5142 .040587500
4125 .368653800
4. 5521.073056700
      5735.523259200
5.5707 .731599800
6706 .356161400
6. 5725.994562300
      7722.43362700
7. 5717.099112600
      8716.983655200
8. 5725.502401200
      9722.355171300
9. 5731.571717300
10723 .509788400
10.5726 .884498700
      11725.96203400
11. 5743.248587800
12726 .406115600
12. 5734.39988700
13727 .916630900
13.5116 .192185100
      14517.961306500
```

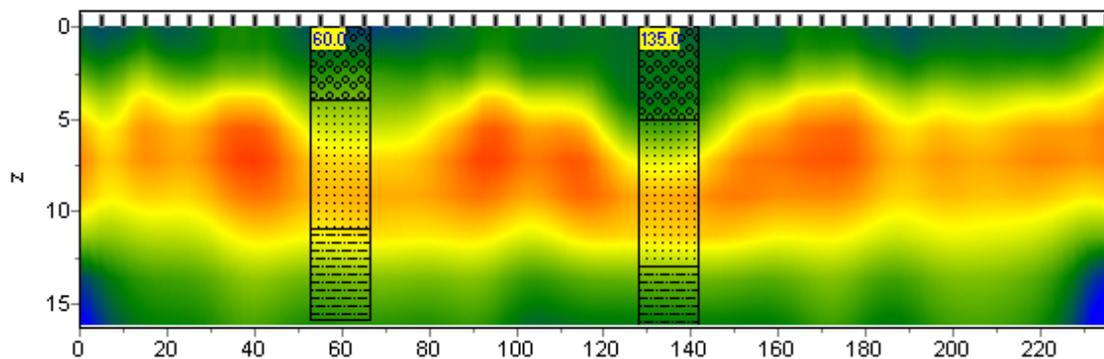


14.5125.370626400
 15111 .295247800
 15.5131 .91187900
 16107 .921730900
 16.5114 .932736100
 17134 .093919600
 17.5138 .445714300
 18129 .116510400

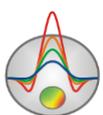


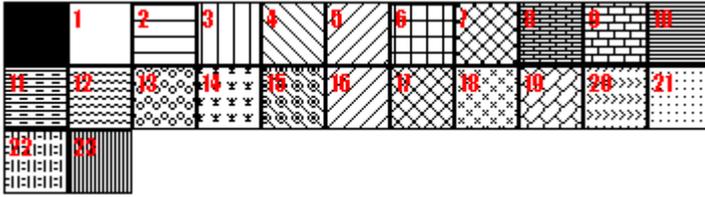
The following file structure is used when creating a file with lithological information:

The first column contains the depth (from the ground surface) of the lithological horizon. The second column should be filled with zeros. The third column the color of the layer on the lithological column. The fourth column the type of krapp on the lithological column.



Below is a list of the first 23 krapps that can be used, when creating a lithology column.





Below is an example of a lithological data file.

0 1 0 13 Roofing 1 layer
 4 1 0 13 Layer 1 sole
 4 1 0 19 Roofing 2 layers
 11 1 0 19 Outsole 2 layers
 11 1 0 27 Roofing 3 layers
 16 1 0 27 Outsole 3 layers

The second file type (*. crt extension) is a control file specifying the data type and display method. The following is a description of the CRT file structure for displaying lithologic or well logs for an arbitrary number of wells.

2280.txt. The first line is the file name of the log or lithology data

skv2280 Second line - Well's signature (will be displayed on the well)

18 2 2 1 0 1 0 0 The third line contains the control parameters -

Record 18 - coordinate of the well on the profile.

2 - image width (as a percentage of the profile length, usually 1 - 20).

2 - type of data display 0 - 3.

0 - logging data (as a graph);

1 - logging data (interpolation color column) The color scale of the section is used to display the data;

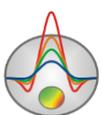
2 - lithological column;

3 - well log data (colored column) colors of the displayed data correspond to the model scale, the color on the column is selected according to the value of the model color scale;

1 - Normalization parameter of well log data 0 - 2.

0.1 - a common minimum and maximum is used for all data;

1.2 - subtract its average value from each well log;



0 - Logging method index (if several logging types are to be displayed simultaneously, you should enter indices for each of the methods) 0 - n-1, where n is the number of methods.

1 - Graphic color.

0 - Data scale is logarithmic 0, linear 1.

0 - Vertical displacement of the well relative to the ground surface.

3246.txt description of the next well on the profile

squ3246

102 2 2 1 0 1 0 0

Additional materials:

Video tutorials on youtube channel:

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

Support group in linkedin:

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

Zond demonstration projects:

ftp://zond-geo.com/.

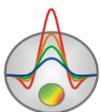
Username: download@zond-geo.com

Password: 12345

The program does not work with the USB dongle

1) The dongle driver is not installed or is not installed correctly. On some systems the dongle is detected as a HID device correctly and there is no need to install the driver, but on some systems it is not and must be installed. The link to download the driver: http://senselock.ru/files/senselock_windows_3.1.0.0.zip. In the Device Manager the dongle should appear as "Senselock Elite".

2) The free update period has expired. In this case you need to use the latest working version or buy an additional 2 years of updates.



3) Sometimes when you switch the dongle to HID mode, the system may not recognize it as a HID device. In this case you need to switch it back to USB mode with a small application which can be downloaded from the following link: <http://www.zond-geo.com/zfiles/raznoe/SenseSwitch.zip> "senseswitch.exe" is started from the cmd command:
senseswitch.exe usb

